



HyperWorks®

## **Altair® HyperWorks®**

A Platform for Innovation®

Copyright© 1986-2015 Altair Engineering Inc. All Rights Reserved.

### **Copyright© Altair Engineering Inc. All Rights Reserved for:**

HyperMesh® 1990-2015; HyperCrash® 2001-2015; OptiStruct® 1996-2015; RADIOSS® 1986-2015; HyperView® 1999-2015; HyperView Player® 2001-2015; HyperStudy® 1999-2015; HyperGraph® 1995-2015; MotionView® 1993-2015; MotionSolve® 2002-2015; HyperForm® 1998-2015; HyperXtrude® 1999-2015; Process Manager™ 2003-2015; Templex™ 1990-2015; TextView™ 1996-2015; MediaView™ 1999-2015; TableView™ 2013-2015; BatchMesher™ 2003-2015; HyperMath® 2007-2015; HyperWeld® 2009-2015; HyperMold® 2009-2015; Manufacturing Solutions™ 2005-2015; solidThinking® 1993-2015; solidThinking Inspire® 2009-2015; solidThinking Evolve® 1993-2015; Durability Director™ 2009-2015; Suspension Director™ 2009-2015; AcuSolve® 1997-2015; AcuConsole® 2006-2015; SimLab® 2004-2015; Virtual Wind Tunnel™ 2012-2015; MDS™ 2011-2015 and FEKO™ (©1999-2014 Altair Development S.A. (Pty) Ltd.; ©2014-2015 Altair Engineering, Inc.); MDS™ 2011-2015 and VisSim™ 1989-2015.

Other Altair software applications include:

**Altair PBS Works™:** Compute Manager™ 2007-2015; Display Manager™ 2007-2015; PBS™ 1994-2015; PBS Professional® 1994-2015; PBS Application Services™ 2008-2015; PBS Analytics™ 2007-2015; and PBS Desktop™ 2007-2012; PBS Portal™ 2007-2011; e-BioChem™ 2007-2013; e-Compute™ 2000-2007; e-Render™ 2006-2010; OpenPBS® 1994-2003 and Personal PBS® 2007-2012.

**Altair Simulation Cloud Suite:** Simulation Manager™ 2003-2015; Compute Manager™ 2003-2015; Display Manager™ 2003-2015; and Process Manager™ 2003-2015.

### **Altair Packaged Solution Offerings (PSOs) Copyright© 2008-2015**

Automated Reporting Director™ 2008-2015; Impact Simulation Director™ 2010-2015; Model Mesher Director™ 2010-2015; Model Verification Director™ 2013-2015; Squeak and Rattle Director™ 2012-2015; Virtual Gauge Director™ 2012-2015; Weld Certification Director™ 2014-2015

Altair intellectual property rights are protected under U.S. and international laws and treaties. 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 in Altair software shall not be decompiled, disassembled, "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.

### **Third party software licenses**

#### **Software Security Measures:**

Altair Engineering Inc. and its subsidiaries and affiliates reserve the right to embed software security mechanisms in the Software for the purpose of detecting the installation and/or use of illegal copies of the Software. The Software may collect and transmit non-proprietary data about those illegal copies. Data collected will not include any customer data created by or used in connection with the Software and will not be provided to any third party, except as may be required by law or legal process or to enforce our rights with respect to the use of any illegal copies of the Software. By using the Software, each user consents to such detection and collection of data, as well as its transmission and use if an illegal copy of the Software is detected. No steps may be taken to avoid or detect the purpose of any such security mechanisms.

## Altair Engineering Support Contact Information

Web site [www.altairhyperworks.com](http://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
South Africa	27 21 8311500	support@altair.co.za
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

<b>HYPERMESH 14.0 RELEASE NOTES</b> .....	<b>13</b>
<b>HYPERVIEW 14.0 RELEASE NOTES</b> .....	<b>36</b>
<b>HYPERGRAPH 14.0 RELEASE NOTES</b> .....	<b>45</b>
<b>MOTIONVIEW 14.0 RELEASE NOTES</b> .....	<b>57</b>
HIGHLIGHTS .....	57
3D CONTACT .....	58
NONLINEAR FINITE ELEMENTS .....	63
ADVANCED DRIVER AND ROAD VEHICLE SIMULATION .....	66
OTHER MOTIONVIEW ENHANCEMENTS .....	69
RESOLVED ISSUES .....	78
KNOWN ISSUES .....	83
<b>HYPERCRASH 14.0 RELEASE NOTES</b> .....	<b>85</b>
<b>BASICFEA 14.0 RELEASE NOTES</b> .....	<b>90</b>
<b>AUTOMATE 14.0 RELEASE NOTES</b> .....	<b>94</b>
<b>HYPERMATH 14.0 RELEASE NOTES</b> .....	<b>96</b>
<b>HYPERSTUDY 14.0 RELEASE NOTES</b> .....	<b>98</b>
<b>HYPERFORM 14.0 RELEASE NOTES</b> .....	<b>101</b>
<b>HYPERXTRUDE 14.0 RELEASE NOTES</b> .....	<b>104</b>
<b>CFD 14.0 RELEASE NOTES</b> .....	<b>109</b>
<b>AEROSPACE 14.0 RELEASE NOTES</b> .....	<b>117</b>
<b>CRASH 14.0 RELEASE NOTES</b> .....	<b>126</b>
<b>CAD AND SOLVER INTERFACES</b> .....	<b>133</b>
<b>ABAQUS INTERFACE 14.0 RELEASE NOTES</b> .....	<b>134</b>
<b>ANSYS INTERFACE 14.0 RELEASE NOTES</b> .....	<b>137</b>
<b>CAD INTERFACES 14.0 RELEASE NOTES</b> .....	<b>142</b>
<b>EXODUS INTERFACE 14.0 RELEASE NOTES</b> .....	<b>144</b>
<b>LS-DYNA INTERFACE 14.0 RELEASE NOTES</b> .....	<b>145</b>
<b>NASTRAN INTERFACE 14.0 RELEASE NOTES</b> .....	<b>150</b>
<b>PAM CRASH 2G INTERFACE 14.0 RELEASE NOTES</b> .....	<b>153</b>
<b>RADIOSS INTERFACE 14.0 RELEASE NOTES</b> .....	<b>155</b>
<b>OPTISTRUCT INTERFACE 14.0 RELEASE NOTES</b> .....	<b>157</b>
<b>SAMCEF INTERFACE 14.0 RELEASE NOTES</b> .....	<b>161</b>
<b>OPTISTRUCT 14.0 RELEASE NOTES</b> .....	<b>165</b>
STIFFNESS, STRENGTH AND STABILITY .....	166
NOISE AND VIBRATIONS .....	175



KINEMATICS AND DYNAMICS.....	178
THERMAL ANALYSIS .....	179
GENERAL UPDATES.....	180
OPTIMIZATION .....	182
SOLVERS .....	189
NONLINEAR DYNAMIC ANALYSIS.....	190
RESOLVED ISSUES .....	191
<b>RADIOSS 14.0 RELEASE NOTES.....</b>	<b>193</b>
CRASH AND SAFETY.....	194
FORMING.....	201
BLAST SIMULATION.....	203
GENERAL UPDATES.....	205
RESOLVED ISSUES .....	214
<b>MOTIONSSOLVE 14.0 RELEASE NOTES.....</b>	<b>215</b>
SOLVER ENHANCEMENTS .....	221
ADAMS COMPATIBILITY .....	226
RESOLVED ISSUES .....	228
KNOWN ISSUES .....	232
<b>HYPERXTRUDE SOLVER 14.0 RELEASE NOTES .....</b>	<b>233</b>
HYPERXTRUDE FOR METAL EXTRUSION.....	233
HYPERXTRUDE FOR POLYMER EXTRUSION .....	235
HYPERXTRUDE FOR RESIN TRANSFER MOLDING .....	236
<b>ACUSOLVE 14.0 RELEASE NOTES.....</b>	<b>237</b>
<b>MULTIBODY SOLUTIONS 14.0 RELEASE NOTES .....</b>	<b>252</b>
3D CONTACT.....	253
<i>Pre-processing</i> .....	253
<i>Solution</i> .....	257
<i>Post-processing</i> .....	261
<i>Change in Behavior</i> .....	263
NONLINEAR FINITE ELEMENTS.....	264
<i>Pre-processing</i> .....	264
<i>Solution</i> .....	267
<i>Post-processing</i> .....	271
<i>CMS Flexible Body vs. NLFE Body: Pre-Processing Comparison</i> .....	272
<i>CMS Flexible Body vs. NLFE Body: Solution Comparison</i> .....	272
<i>CMS Flexible Body vs. NLFE Body: Post-Processing Comparison</i> .....	273
ADVANCED DRIVER AND ROAD VEHICLE SIMULATION .....	274
<i>Driver</i> .....	274
<i>Road</i> .....	275
OTHER MOTIONSSOLVE ENHANCEMENTS .....	277
<i>Geometric Stiffening for CMS Flexible Bodies</i> .....	278
<i>Compatibility with ADAMS Language</i> .....	283
<i>Resolved Issues</i> .....	284
<i>Known Issues</i> .....	288

OTHER MOTIONVIEW ENHANCEMENTS .....	289
<i>Resolved Issues</i> .....	296
<i>Known Issues</i> .....	297
OTHER AUTO SOLUTION ENHANCEMENTS .....	299
<i>Resolved Issues</i> .....	302
<i>Known Issues</i> .....	306
OPTISTRUCT ENHANCEMENTS PERTINENT TO MULTIBODY SIMULATIONS .....	307
<b>INSPIRE 2016 RELEASE NOTES .....</b>	<b>309</b>
<b>EVOLVE 2016 RELEASE NOTES .....</b>	<b>313</b>
<b>FEKO 14.0 RELEASE NOTES .....</b>	<b>318</b>
<b>ACUFIELDVIEW 14.0 RELEASE NOTES.....</b>	<b>323</b>
<b>SIMLAB 13.3 RELEASE NOTES .....</b>	<b>331</b>
<b>VIRTUAL WIND TUNNEL 13.1 RELEASE NOTES .....</b>	<b>338</b>
<b>HYPERXTRUDE 2015.1 RELEASE NOTES.....</b>	<b>341</b>
<b>SOFTWARE AND HARDWARE 14.0 RELEASE NOTES .....</b>	<b>343</b>
<b>HYPERWORKS LICENSING 14.0 RELEASE NOTES.....</b>	<b>346</b>

# HyperWorks 14.0 Release Highlights

Altair® HyperWorks® is the most comprehensive, open architecture CAE simulation platform in the industry. It offers the best technologies to design and optimize high performance, weight efficient, and innovative products.

HyperWorks includes best-in-class modeling, linear and nonlinear analyses, structural and system-level optimization, fluid and multibody dynamics simulation, electromagnetic compatibility (EMC), multi-physics analysis, model-based development, and data management solutions.

Our goal at Altair has always been to provide the best technology at the highest value to our customers, and the HyperWorks® 14.0 release is no exception.

The suite offers hundreds of new features, enabling you to continue to develop better products in shorter timescales. New and enhanced capabilities include:

- Efficient parts and assemblies management
- Extremely fast graphic engine, to handle even the largest models
- New solutions for linear and nonlinear structural analysis
- Multiphysics analysis structural, electromagnetic and fluid flow phenomenon
- Optimization-based analysis and system-level design exploration

The following sections provide a quick overview of new and enhanced capabilities included in the new release.

## Modeling and Visualization Capabilities

---

### ▼ Model Assembly

A new part-based workflow is available, bringing modularization to HyperMesh. One of its primary benefits is that it promotes flow of data from PDM to HyperMesh. Full representation management and configuration management are enabled with new data redundancy checks to ensure model integrity. The data can be read indirectly from PDM, via PLMXML, so the hierarchical structure is read and populated within HyperMesh. With the part-based workflow, the possibility of a common model approach is enabled.

### ▼ Meshing and Connectors

A new browser-based workflow for meshing has been implemented. This workflow utilizes the new "mesh control" entity. This new entity enables the definition of formal, centralized, reusable meshing scenarios.

The Connector Entity Editor (CEE) can be found in the lower part of the Connector browser and gives easy access to all important connector attributes of currently selected connectors. With the CEE it's now possible to precisely change single attributes and realize various connectors at once. All attributes are described with tooltips.

### ▼ Undo/Redo

A new history mechanism has been introduced, enabling undo/redo operations for most of the geometry and meshing operations. The functionality is particularly focused on entity creation and editing.

### ▼ HyperWorks Automate

Users with various levels of expertise can benefit from Automate. Engineers and analysts can run processes more efficiently. Methods teams and administrators can provide their customers with easy-to-use tools that incorporate best practices and advanced expertise.

Automate is a new and advanced utility in the HyperWorks Desktop environment for easily creating and executing process-based workflows. It allows users to add blocks, which can consist of several individually programmed tasks. The environment provides drag and drop functionality to arrange these blocks into a desired process.

### ▼ Performance

HyperWorks 14.0 is extremely fast, handling even the largest models, paving the road to photo-realistic visualization of FE-models. Performance increase - compared to the previous HyperWorks 13.0 version - is up to 15x for large FE models with solid elements and up to 60x for geometry models, while using less hardware memory.

## Analysis and Optimization

---

HyperWorks solver technology includes Finite-Element-Based Linear and Nonlinear Structural Analysis, design and optimization capabilities (**OptiStruct**), Finite-Element-Based Highly Nonlinear Structural Analysis under dynamic loading (**RADIOSS**), Thermal and Fluid Analysis (**AcuSolve**), computational electromagnetics (**FEKO**) 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 14.0 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:

- **Concept Design:** The greatest flexibility to impact the design of a product is at the conceptualization phase. To take advantage of this flexibility, methods aimed at driving design concepts should be employed at this stage. **OptiStruct** enhancements in this area include a new alternate optimization algorithm, 3D topology optimization for **RADIOSS** nonlinear analysis.
- **Lattice Structure Design:** A first-to-market solution for the design and optimization of lattice structures has been implemented in **OptiStruct**. Lattice structures are frequently used in designs for 3D printing. Based on well-established topology optimization technology, this novel solution is able to provide design concepts for a blended solid-and-lattice type structure.
- **Composites Layout:** In addition to the several enhancements included in **OptiStruct** and **RADIOSS** for composites design and analysis, HyperWorks now includes **Multiscale Designer**, a mathematically rigorous practical tool for the seamless integration of modeling, simulation, testing, and optimization of engineered products involving materials of multiple spatial and temporal scales.

### ▼ Broad Portfolio of Physics

An expanded number of solution types and material models broaden HyperWorks' multi-physics offering. Here are some highlights:

- **Contact:** New contact algorithms have been implemented in **OptiStruct**. Finite sliding contact has been shown to be very effective modeling gears and other sliding problems. The fast small sliding contact algorithm provides up to 10 times faster analysis while giving same accuracy as the full-fledged iterative solution. **MotionSolve** now provides a new 3D rigid-to-rigid contact which is robust and fast.
- **Large Displacement Finite Elements:** **OptiStruct** capabilities are more complete including now 1D elements like beams, bars, and bushings as well as follower forces.
- **Vibrations and Acoustics:** **OptiStruct** solution speed has been further improved. Unique techniques like One-Step Transfer Path Analysis as well as BIOT materials have been made more robust and user friendly.

- **Crash Worthiness and Impact:** Major improvements for airbags, in particular **RADIOSS** Starter capability to generate automatic Finite Volume Mesh for folded complex airbags to account for internal airbag topology.
- **Heat Transfer:** In **OptiStruct**, transient temperature at multiple time steps can now be applied in thermal-structure coupling analysis, Thermal contact has been enhanced.
- **Materials and Failure:** The library of material and failure models in **RADIOSS** has been further expanded. Thermal expansion and damage output are now possible. XFEM has been expanded to more use cases.
- **Advanced Driver and Road:** Automotive multibody simulation library for **MotionSolve** has been expanded for full vehicle simulations using advanced driver and road models.
- **Turbulence Modeling – AcuSolve** k- $\omega$  was improved to be more stable and accurate.

#### ▼ Performance

Altair HyperWorks utilizes parallel computer architectures for enhanced solvers performance:

- **High Scalability on CPU:** **RADIOSS** scalability of up 8,192 cores and beyond on a 15 million-element model has been shown. **OptiStruct** parallelism has been further developed.
- **Fast Eigen Solutions:** **OptiStruct** normal modes and frequency response analysis has been made faster through further improvement of numerical solver capabilities.

#### ▼ Computational Electromagnetics

FEKO joins the Altair family and is included as part of the HyperWorks simulation software platform. In the 14.0 release FEKO boasts numerous extensions to both its user interface and electromagnetic solvers. Highlights include curvilinear segments, complete application automation for CADFEKO, script recording and extensions to higher order basis functions (HOBf).

The GUI components have been extended to support all new features and solution techniques in the FEKO solver. In addition to supporting the new solver features, numerous other features and improvements have also been implemented.

## Value-based Business Model

---

HyperWorks stretches your software dollar through a patented, value-based license management system called HyperWorks Units, which allows metered usage of the entire suite of products. This value-based licensing model has been extended to partners' products, providing a dynamic and on-demand platform which now also includes cloud-based solutions.

When you sign up for HyperWorks, you become eligible to download our entire suite of Altair Partner Alliance solutions. We have revolutionized the way you obtain software by eliminating the hurdles that prevent you from getting what you need. So at virtually no extra cost, your team can access a growing library of solutions that complement our own technology using a single pool of recyclable units.

### ▼ HyperWorks Unlimited Solver Node

An alternative solver licensing scheme has been implemented. This new scheme is compute node-based versus the traditional licensing per job. The solver license is associated with the total number of compute nodes used simultaneously by all solver jobs. This provides a great value for massive use of HyperWorks solvers.

### ▼ Hosted HyperWorks Units (HHWU)

- Altair's innovative HyperWorks licensing model hosts 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 14.0 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 – Physical Appliance) or scale out to the public cloud (HyperWorks Unlimited – Virtual Appliance), 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, multibody 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.

Key updates for Inspire 2015 include:

- **Fasteners and Joints:** Connect multiple parts in a model using bolts, screws, pins, or sliding pins. Inspire's workflow identifies areas in the model with aligned holes to make the process easy and intuitive.
- **Contacts:** Users can now designate whether neighboring parts should be bonded, contacting, or have no contact for more realistic optimization results.
- **Gravity Loads:** G loads can now be added to a model through the Model Browser.
- **Updated User Interface:** To accommodate the new tools offered in Inspire 2015, a new ribbon style interface organizes the tools into tabs. Users are able to customize the tabs to tailor the workflow to their process.
- **Faster Geometry Functions:** Geometry functions in the program are running up to 300% faster than previous versions of Inspire.

### ▼ 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.

Key enhancements for solidThinking Evolve 2015 include:

- **New PolyNURBS toolset with *Nurbify*:** Convert a polygonal model to an organic NURBS surface with a single click using the new *Nurbify* tool.
- **Photorealistic rendering:** A completely redesigned rendering tool creates life-like images with an enhanced user interface.
- **Improved surface modeling:** Enhancements to the surface modeling toolset create a seamless experience with greater control over modeling outcomes.
- **Better user interface:** An updated user interface focuses on creating more efficient workflows.
- **Greater compatibility:** Now compatible with 64-bit Mac OS X.

## Design Exploration

---

### ▼ HyperStudy

HyperStudy 14.0 provides several exciting new features that enable users to explore designs more thoroughly and efficiently than ever before. From quickly picking your design parameters in HyperMesh to a new streamlined response definition experience, getting HyperStudy setup to run your investigation has never been easier.

The new inclusion matrix allows optimizers to utilize existing run information, increasing the optimizer's efficiency and making the most out of your data. The automatic cross-validation check can save time by calculating checks on the Fit quality without a dedicated validation matrix.

## Vertical Solutions

---

HyperWorks 14.0 extends the offer for manufacturing and engineering solutions, with new functionalities and an innovative delivery model to better capture knowledge for vertical processes.

### ▼ HyperXtrude 2015

Today's extrusion companies are forced to produce a higher mix of complex profiles in shorter product development cycles while simultaneously reducing their production costs. HyperXtrude is a simulation tool developed to analyze and validate the design of extrusion dies and process.

HyperXtrude® 2015 introduced a new CAD interface for HyperXtrude solver and takes giant leap to a direct CAD to solution approach. This approach automated and hid the meshing and data deck creation; and completely relegated it to the background. Hence, after creating the CAD model user submitted the job for analysis. HyperXtrude 2015 supports steady state and transient metal extrusion, tool deflection, and bearing optimization analysis.

### ▼ Virtual Wind Tunnel

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.

### ▼ HyperWorks Packaged Solutions

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:** 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:** Designed to reduce the simulation cycle time, freeing CAE engineers to focus on optimizing product design and performance.
- **Impact Simulation Director:** Automate the laborious, manual tasks associated with model set-up, analysis, post-processing and reporting.
- **Squeak & Rattle Director:** Offers a complete set of capabilities to streamline the entire S&R simulation workflow process from model creation to results visualization.

# HyperMesh 14.0 Release Notes

## General

### ▼ Modularization and Common Model Build

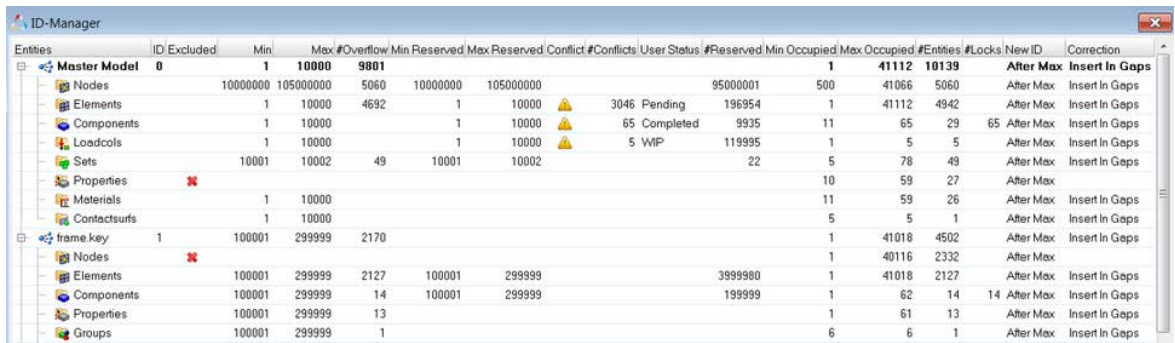
- A new part based workflow is available which brings modularization to HyperMesh. One of its primary benefits is that it promotes flow of data from PDM to HyperMesh. Full representation management and configuration management are enabled with new data redundancy checks to ensure model integrity. The data can be read indirectly from PDM, via PLMXML, so the hierarchical structure is read and populated within HyperMesh. Once the PLMXML data is read into HyperMesh, and the CAD representations are also read in, it's possible to automatically batchmesh the parts directly. Batchmeshing of parts provides the ability to mesh several representations such as Crash, NVH and Durability, each with their own meshing parameters. Once meshed, it's possible that a sub-system can be saved; that sub-system could be sent to other users, subsequently altered, and read back into HyperMesh. Any changes and/or redundancy are captured and resolved and the parts and their changes are read back into the correct hierarchical location. With the part based workflow, the possibility of a common model approach is enabled. This workflow is important for any multi-variant, multi-representation, multi-model model build. It provides the opportunity to significantly reduce model build times by generating and managing the CAE part data more effectively via a layered common model approach. The layered common model contains all representations, it can then be configured so that the appropriate model can be generated quickly. The information from PDM is inherited within HyperMesh and applied to the various entities; the PDM hierarchy is preserved, material and properties are generated from the PDM data directly plus much more.

Entities	UID	Representation	Active	CID	PID	MID	Material	Thickness	PD
Model									
Frame_Assembly_Config_A_000425_Safety	Frame_Assembly_Config_A		<input checked="" type="checkbox"/>						
CenterRail_A_000426_Safety	CenterRail_A		<input checked="" type="checkbox"/>						
LeftRail_A_000433_Safety	LeftRail_A		<input checked="" type="checkbox"/>						
RightRail_A_000436_Safety	RightRail_A		<input checked="" type="checkbox"/>						
RightInner_A_000437_Safety	RightInner_A		<input checked="" type="checkbox"/>	301	301	10	Steel	1.700000	
RightInner_A_000437_Safety.c01			<input checked="" type="checkbox"/>	301	301	10	Steel	1.700000	
RightOuter_A_000438_Safety	RightOuter_A		<input checked="" type="checkbox"/>	302	302	10	Steel	1.500000	
RightOuter_A_000438_Safety.c01			<input checked="" type="checkbox"/>	302	302	10	Steel	1.500000	
Connectors			<input checked="" type="checkbox"/>						
Center Rails			<input checked="" type="checkbox"/>						
Center Rail Connectors v2	Rail_Conns_v2		<input checked="" type="checkbox"/>						
Center Rail Connectors v1	Rail_Conns_v1		<input checked="" type="checkbox"/>						
Longitudinal Rails			<input checked="" type="checkbox"/>						
Longitudinal Rail Connectors	Rail_Conns		<input checked="" type="checkbox"/>						

- Redundant entity management is enabled which provides the ability to control which FE/geometry, material, and/or property attributes are read into the session – the user has the ability to select whether to retain the information already in session, or, select the incoming information to overwrite what's already in session.

▼ ID Management

- ID management coverage has increased to support all solvers that use include files. The supported solvers now include; LS-DYNA, RADIOSS, OptiStruct, Nastran, Pamcrash, and Abaqus.
- Reserve: This new capability enables a discreet or range of IDs to be reserved. These reserved IDs will be sacred and can be reused later if required.
- The ID management is now supported for the master model. So, if the model does not have an include structure, it is still possible to manage the IDs and renumber the model at the master model level.

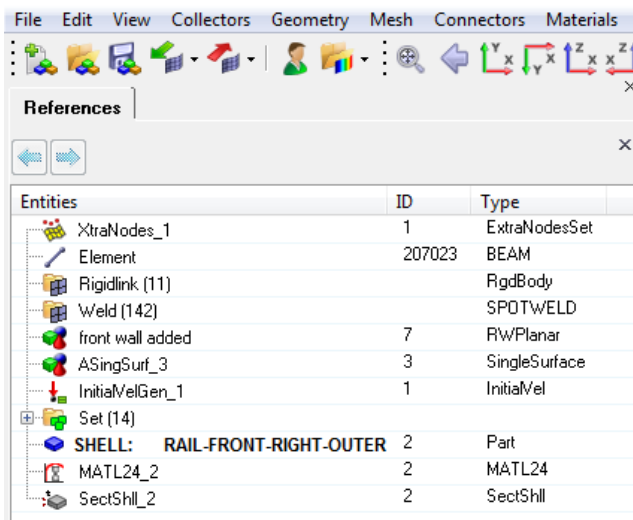


Entities	ID Excluded	Min	Max	#Overflow	Min Reserved	Max Reserved	Conflict	#Conflicts	User Status	#Reserved	Min Occupied	Max Occupied	#Entities	#Locks	New ID	Correction	
Master Model	0	1	10000	9801													
Nodes		10000000	105000000	5060	10000000	105000000				95000001	500	41066	5060		After Max	Insert In Gaps	
Elements		1	10000	4692	1	10000		3046	Pending	196954	1	41112	4942		After Max	Insert In Gaps	
Components		1	10000		1	10000		65	Completed	9935	11	65	29	65	After Max	Insert In Gaps	
Loadcols		1	10000		1	10000		5	WIP	119995	1	5	5		After Max	Insert In Gaps	
Sets		10001	10002	49	10001	10002				22	5	78	49		After Max	Insert In Gaps	
Properties												10	59	27		After Max	
Materials		1	10000									11	59	26		After Max	Insert In Gaps
Contactsurfs		1	10000									5	5	1		After Max	Insert In Gaps
frame key	1	100001	299999	2170								1	41018	4502		After Max	Insert In Gaps
Nodes												1	40116	2332		After Max	
Elements		100001	299999	2127	100001	299999				3999980	1	41018	2127		After Max	Insert In Gaps	
Components		100001	299999	14	100001	299999				199999	1	62	14	14	After Max	Insert In Gaps	
Properties		100001	299999	13							1	61	13		After Max	Insert In Gaps	
Groups		100001	299999	1								6	6	1	After Max	Insert In Gaps	

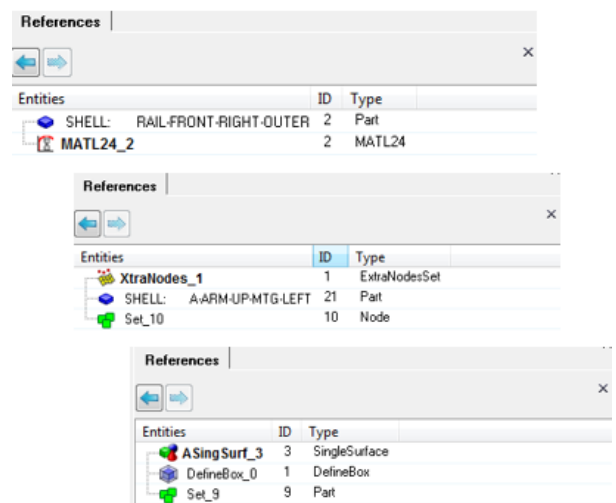
## ▼ XRef Updates

Improvements have been made to the usability of the XRef tool. It's now easier to traverse through the references and cross-referenced entities.

### A new hierarchical view offered for XRef



### Xref can be invoked multiple times while in the XRef dialog for optimal model exploration

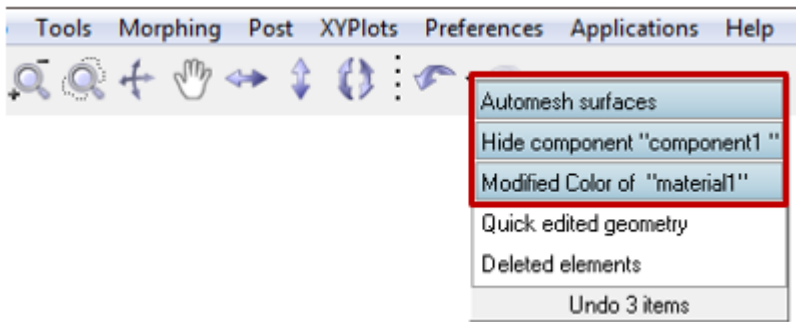


## ▼ Delete - Referenced, Empty and Unused

Within the browser, new options have been added so that it's possible to delete an entity and its referenced entities. In addition, and as per the Delete panel, it's now possible to delete empty and unused data directly from the browser.

## ▼ Undo/Redo

A new history mechanism is enabled which allows, by default, one hundred undo/redo operations. The undo/redo support is focused on entity creation and editing, most geometry and meshing operations plus more. Not all operations are supported and if one of those operations is executed then the history mechanism is reset.

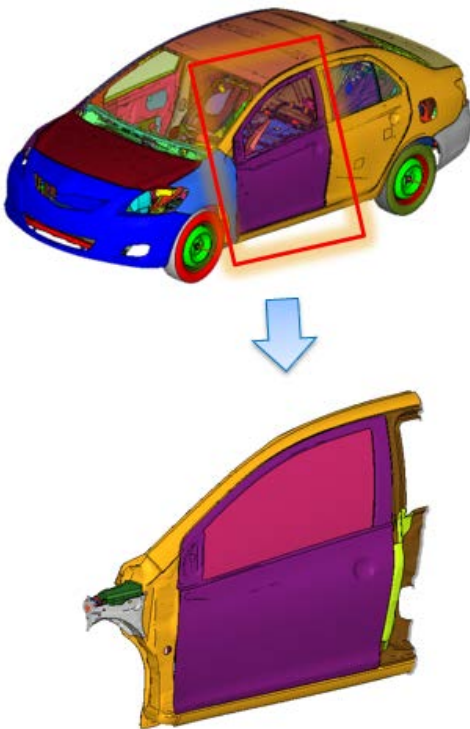


▼ **Solver Browser Updates**

For LS-DYNA, it's possible to view the data as per the Include structure.

▼ **Box Trim**

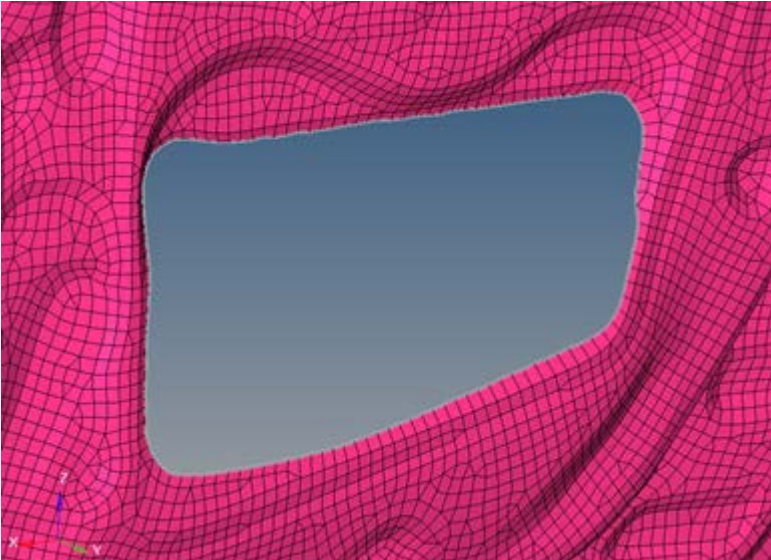
In addition to Ls-Dyna, Box Trim is now enabled for OptiStruct, Nastran and Abaqus solver profiles.





### ▼ Face and Edge Selection

A new selection mode is enabled for the selection of faces and edges. Pressure and Contact creation are the two main areas where the selection modes are available.



### ▼ Dynamic Design Analysis Method (DDAM) Interface

Dynamic Design Analysis Method is used to simulate underwater explosions. Within HyperMesh, there is a new tool to support the setup of response spectrum analysis. It is available for the Abaqus solver profile.

**Define DDAM loading values**

Ship Type: Mounting System    Deformation Type: Elastic

Submarine: Intal

Directional:    Y1: Accel    Y2: Accel    Coefficients:    Y1: Accel

Forward (D1): 0.4    1.4    20.0    10.4

Aftward (D1): 1.0    1.0    40.0    4.0

Vertical (D2): 1.0    1.0    100.0    20.0

Specify values:     Specify values   

Minimum Acceleration:    6    Gs   

Mode	Frequency	Modal Wt	Acc	Y1	Y2	Acc (D1)	Acc (D2)
1	68.9	1.94e-001	240.30	95.95	98.92	54.21	54.21
5	502.4	1.02e-002	240.40	95.99	99.79	392.21	99.79
10	827.1	6.03e-003	240.52	95.99	99.91	398.12	99.91

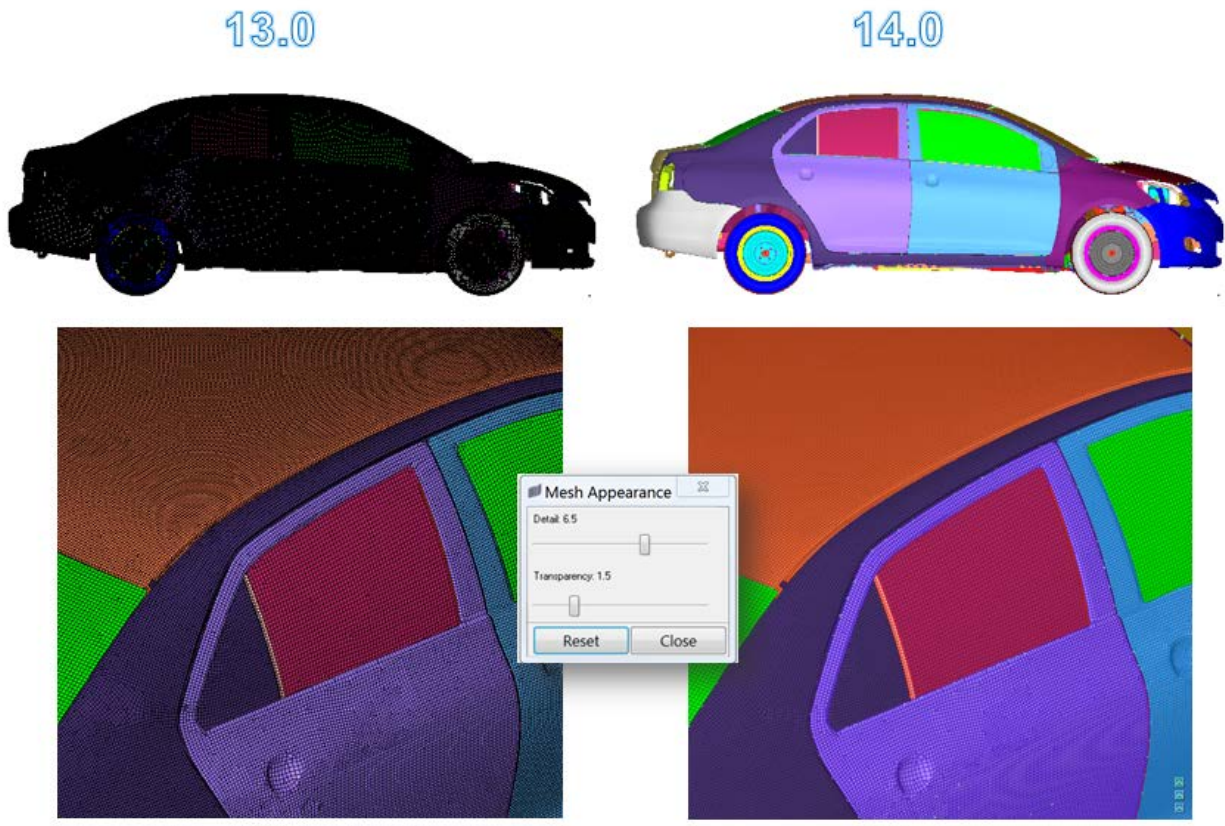
**Shock Spectra**

Direction	Wt Avg (D1 (G))
Forward (D1)	57.895
Aftward (D1)	226.186
Vertical (D2)	139.701



▼ **Improved Visualization**

Significant performance and usability improvements have been achieved by automatically controlling the amount of mesh line detail displayed. The mesh lines are drawn appropriately dependent on the zoom level of the model. For example, if the model is viewed at a global level, the mesh lines will be suppressed, when the model is viewed more closely, then mesh lines will appear. Improvements have also been made to the visualization and performance of the drawing of boundary conditions, node selection, transparency and utilizing current GPU technology to increase large model handling. A new “optimize view controls” option helps increase dynamic model handling speeds by simplifying graphics.



▼ **Updated 3Dconnexion Device Support**

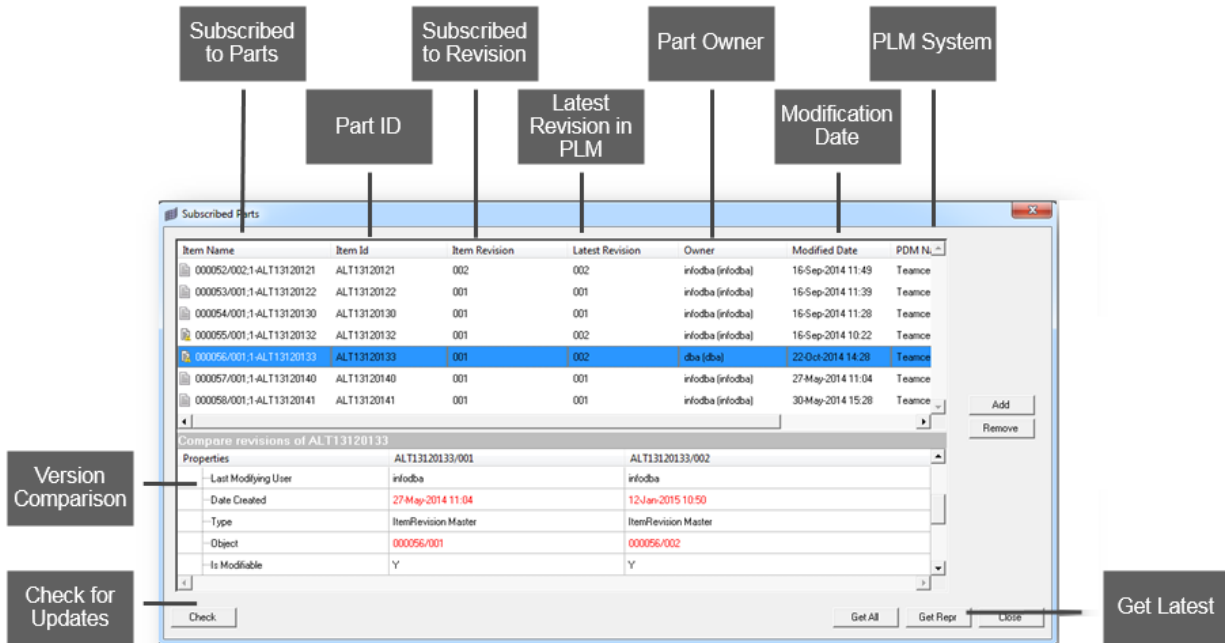
Updated support for 3Dconnexion devices, including better default settings and usability controls through the updated driver APIs within HyperMesh.

▼ **Added Better Support for Integrated GPUs on Notebooks**

HyperMesh will now try to utilize the dedicated GPU (graphics) in a laptop or notebook using combination of Intel’s integrated GPU and NVIDIA’s or AMD’s dedicated graphics cards.

### ▼ Subscribe in HyperWorks Collaboration Tools

It's now possible to subscribe to a part(s) in PDM, and if a change occurs the user, within HyperMesh, can be notified of those changes so that they can download and import the amended CAD file.



Item Name	Item Id	Item Revision	Latest Revision	Owner	Modified Date	PDM N...
000052/002:1-ALT13120121	ALT13120121	002	002	infodba (infodba)	16-Sep-2014 11:49	Teance
000053/001:1-ALT13120122	ALT13120122	001	001	infodba (infodba)	16-Sep-2014 11:39	Teance
000054/001:1-ALT13120130	ALT13120130	001	001	infodba (infodba)	16-Sep-2014 11:28	Teance
000055/001:1-ALT13120132	ALT13120132	001	002	infodba (infodba)	16-Sep-2014 10:22	Teance
000056/001:1-ALT13120133	ALT13120133	001	002	dba (dba)	22-Oct-2014 14:28	Teance
000057/001:1-ALT13120140	ALT13120140	001	001	infodba (infodba)	27-May-2014 11:04	Teance
000058/001:1-ALT13120141	ALT13120141	001	001	infodba (infodba)	30-May-2014 15:28	Teance

Properties	ALT13120133/001	ALT13120133/002
-Last Modifying User	infodba	infodba
-Date Created	27-May-2014 11:04	12-Jan-2015 10:50
-Type	ItemRevision Master	ItemRevision Master
-Object	000056/001	000056/002
-Is Modifiable	Y	Y

### ▼ H3D Export of Result

H3D files exported from HyperMesh can now optionally contain contour results plotted from the Post panel, the Comparison tool, and the Check Elms panel.

### ▼ Resolved Issues

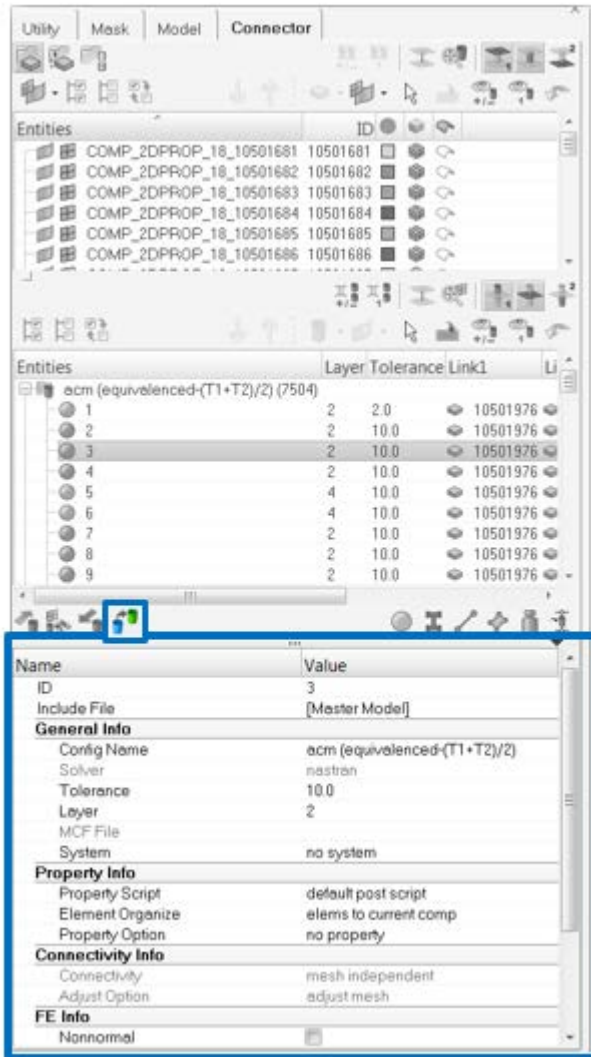
- Loads associated to geometry failed to get deleted after deleting the geometry.
- Display reserve in Model browser was not working correctly for assemblies.
- Performance issue associated with creating contact surfaces on solid elements.
- Active and Export state are not correctly saved in the HM binary file.
- Field loads are calculated differently when shells are on top of solid elements.
- Part replacement issue related to missing second layer of washer.
- Performance issue related to updating multiple composite properties.

## Connectors

### ▼ Connector Entity Editor (CEE)

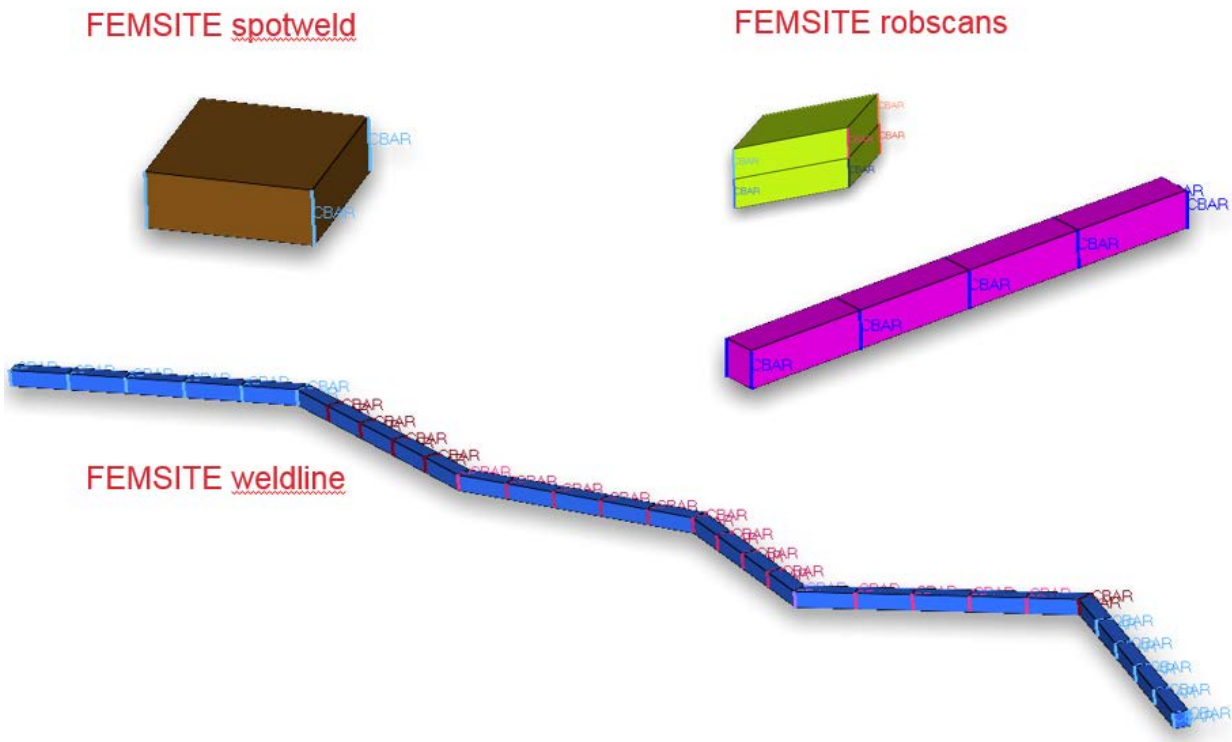
The CEE can be found in the lower part of the Connector browser and gives easy access to all important connector attributes of currently selected connectors. With the CEE it's now possible to precisely change single attributes and realize various connectors at once.

All attributes are described with tooltips.



### ▼ FEMSITE Interface Support

A new interface is enabled to support the FEMSITE Connector capabilities.



▼ **New "Modified" Connector State**

The connector color is changed from green (realized) to blue (modified) now, if their attributes have been changed in the Connector Entity Editor. This signifies that the information on the connector might be inconsistent with the current realization state. A following re-realization turns the connector to green or red and the state is then re-sync'd.

A new function is added which will re-realize all "modified" connectors with their updated attributes.

▼ **Automatic Differentiation Between lap-, t- and butt- Welds for Hexa and Penta Seams**

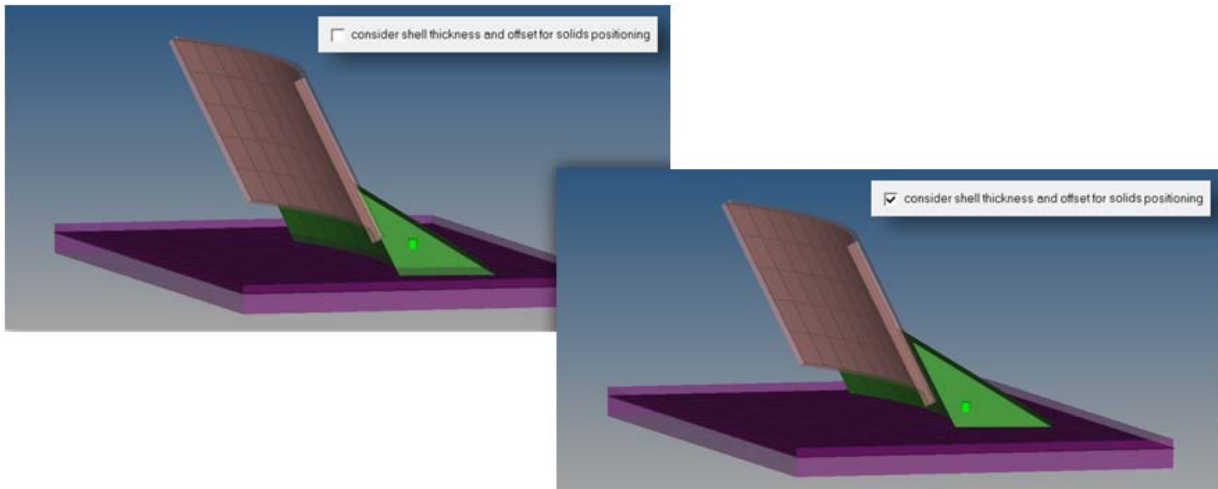
Hexa and penta seam realizations are slightly differently created dependent on the weld type (B, L, T). Before the realization is performed the angle between the two parts is measured. Dependent on that angle a lap-, t-or butt-weld is performed.

The connector angle control is located in the general connector options

The general penta seam realization type has been re-introduced.

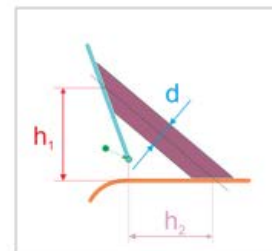
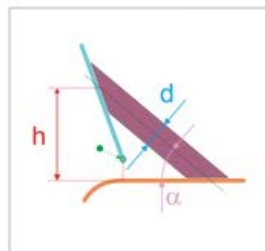
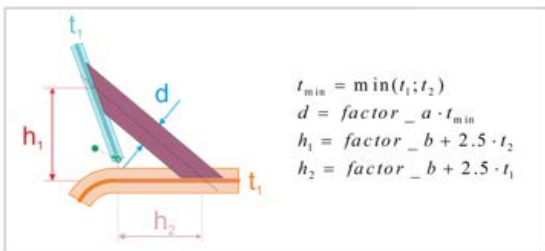
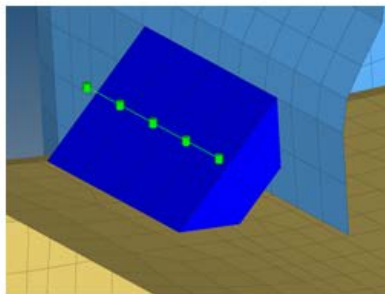
▼ **Consider Shell Thickness and Offset for Solid Positioning Option**

The connector option "consider shell thickness and offset for solid positioning" is now supported for penta seams.



▼ **Hexa Adhesive Seams in t- Cases**

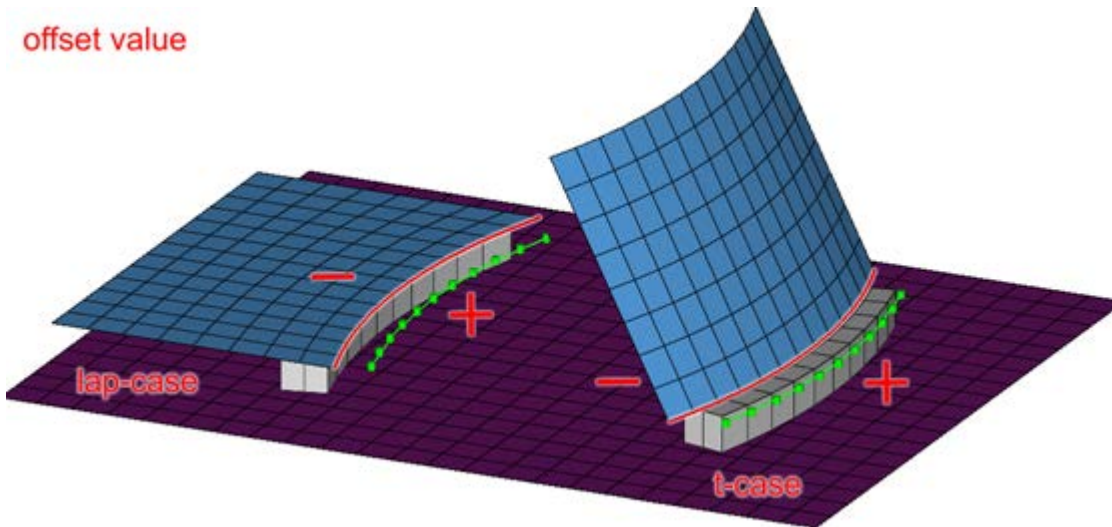
The hexa adhesive seam realization now has options like shell gap, constant thickness, mid thickness for T-cases. The hexa height is dependent on these settings.



▼ **Hexa Position to Edge**

For the hexa adhesive seam realization the position of the hexas related to the free edge can be user defined. The negative side is defined as the side of the smaller angle measured at the free edge between the two parts to be connected.





#### ▼ Hexa (Tapered T) Realization Type

A new seam realization type “hexa (tapered T)” has been implemented for various solvers. This realization type is intended to be used for t-cases. The size and exact position can be defined thickness dependent or the exact dimension and position parameters can be given.

#### ▼ Connector Style Conversion

Spot connectors can be converted to bolt connectors and vice versa. This can be done from the context menu in the lower part of the Connector browser.

#### ▼ New Link Type for Parts

A new entity type part has been introduced into HyperMesh. This entity type is also being supported for being used as link for the connector technology.

#### ▼ Connector Link Type Conversion

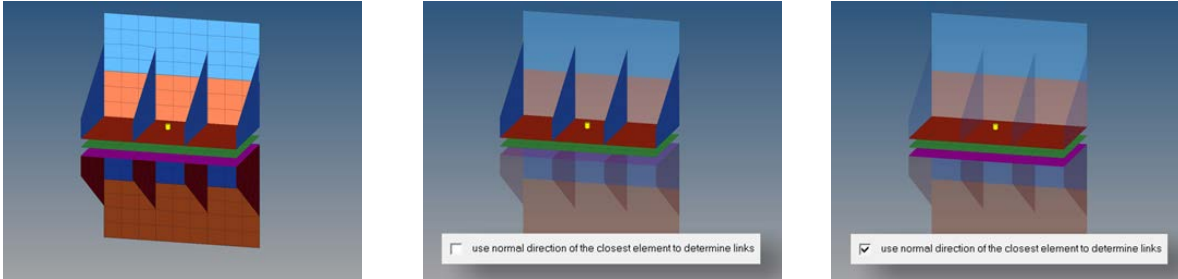
For connectors different link types are supported. It’s possible now to convert certain link types into another link type. Conversion is supported between part-, component- and property-links.

#### ▼ Updates for Seam Quad (angled + capped + L) and Seam Quad (angled + capped + T)

These realization types can be performed without the creation of the cap elements. Then the cap angle as well as the runoff angle are not considered anymore.

▼ **Use Normal Direction of the Closest Element to Determine Links Option**

This option can be found in the Spot Connector options. In certain situations, when many small element cluster of different link candidates are very close to the connector position it might happen, that unintended links are found. To prevent this behavior the option can be activated and the normal direction of the closest link is considered for further link detection.



▼ **Option to Define Imprint Size for Seams**

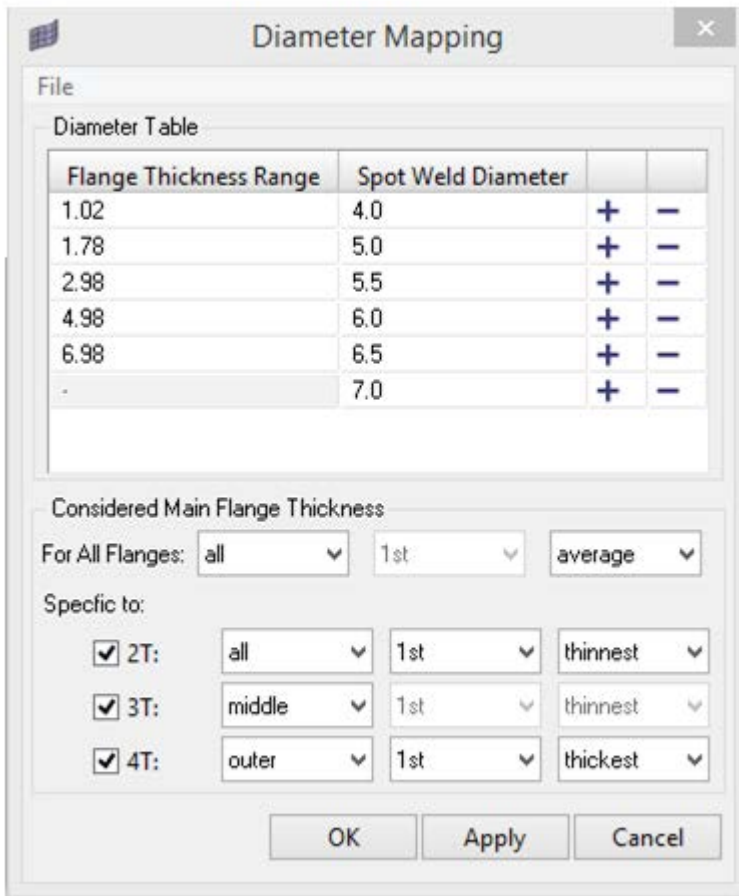
For seam imprint the imprint size can now be defined with a certain value, the average mesh size or as the pitch size.

▼ **RBE3 (Load Transfer) Update**

For the RBE3 (load transfer) realization the tolerance is used to detect the nodes, which should be considered for the connection. To be able to limit the number of attached node a limit per layer can be defined.

▼ **Diameter Mapping Table**

What has been formerly known as DvsT is the Diameter Mapping Table now. This table can be accessed from all ACM spot weld panels, when the diameter is not directly assigned. In the table the user can describe how the main flange thickness is recognized. The main flange thickness decides about the finally used ACM diameter. The thickness ranges can directly be modified in the table and a diameter mapping file can be saved from this window. When realizing ACMs the finally used diameter is written to each single connector.



#### ▼ Area Connector Realization for RADIOSS

A new area connector realization type for the RADIOSS user profile has been added. This realization performs a typical hexa adhesive connection with contact definition.



## Geometry

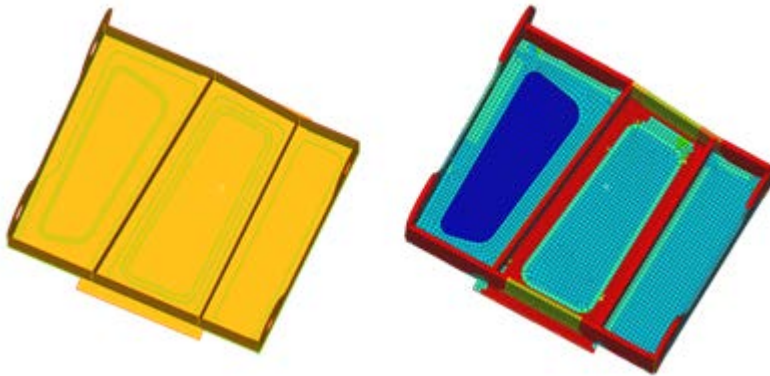
---

### ▼ Midsurface Skin Offset

A new "skin offset" midsurface extraction method has been added. This method generates a midsurface by duplicating and offsetting the inner skin surfaces and assigning a constant thickness. This is useful for parts where there is a constant thickness with no t-connections expected in the final result. This is significantly faster than other midsurface extraction methods, but is focused only on this specific model class.

### ▼ Midsurface Imprint

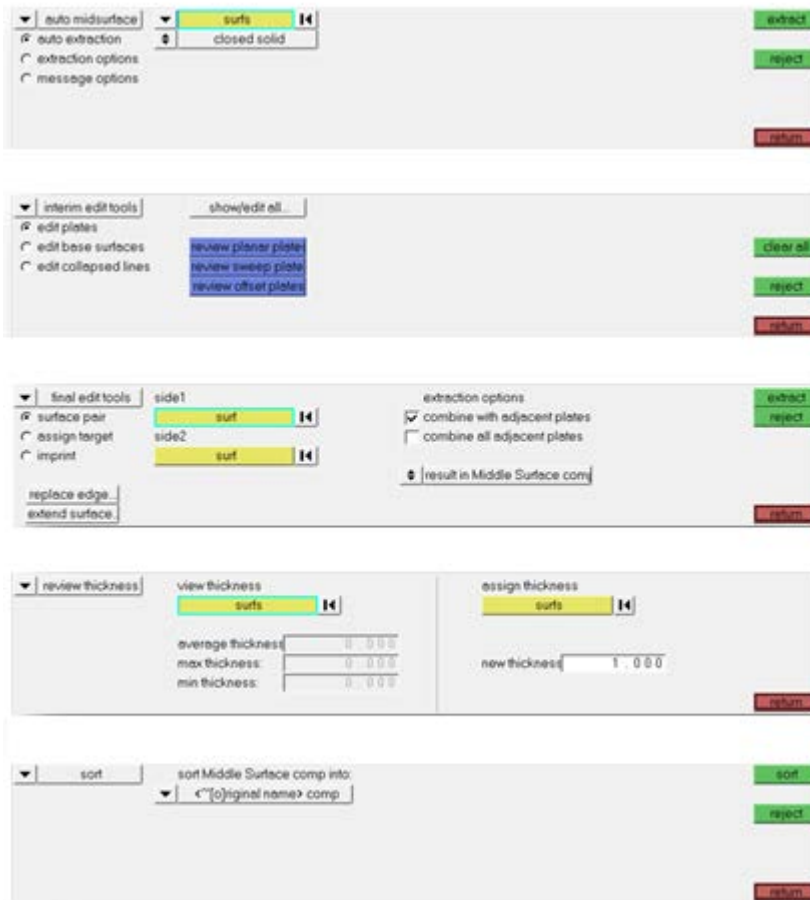
A new tool has been added to the Midsurface panel for imprinting solid features to the midsurface. This is useful when the midsurface does not properly represent solid features like pockets, thickness transitions, etc. Imprints can be made in multiple ways to accurately capture these features.



### ▼ Midsurface Panel Workflow

The Midsurface panel has been reorganized, with attention paid to the most common workflows. Tools have been separated logically into subpanels, with shortcuts to other relevant panels added. New options to control the behavior of warning messages have also been added.

Enhanced sorting behavior has also been adopted. Components created for sorting now automatically inherit their parent component's attributes (color, card image, material, property, etc.) and are organized into the same assembly/include/module structure. A new sort mode has been added to create incrementally named components, based on the parent component name. Finally, it is now possible to immediately sort the midsurfaces after extraction, instead of requiring a second manual step.



## ▼ Metadata Propagation

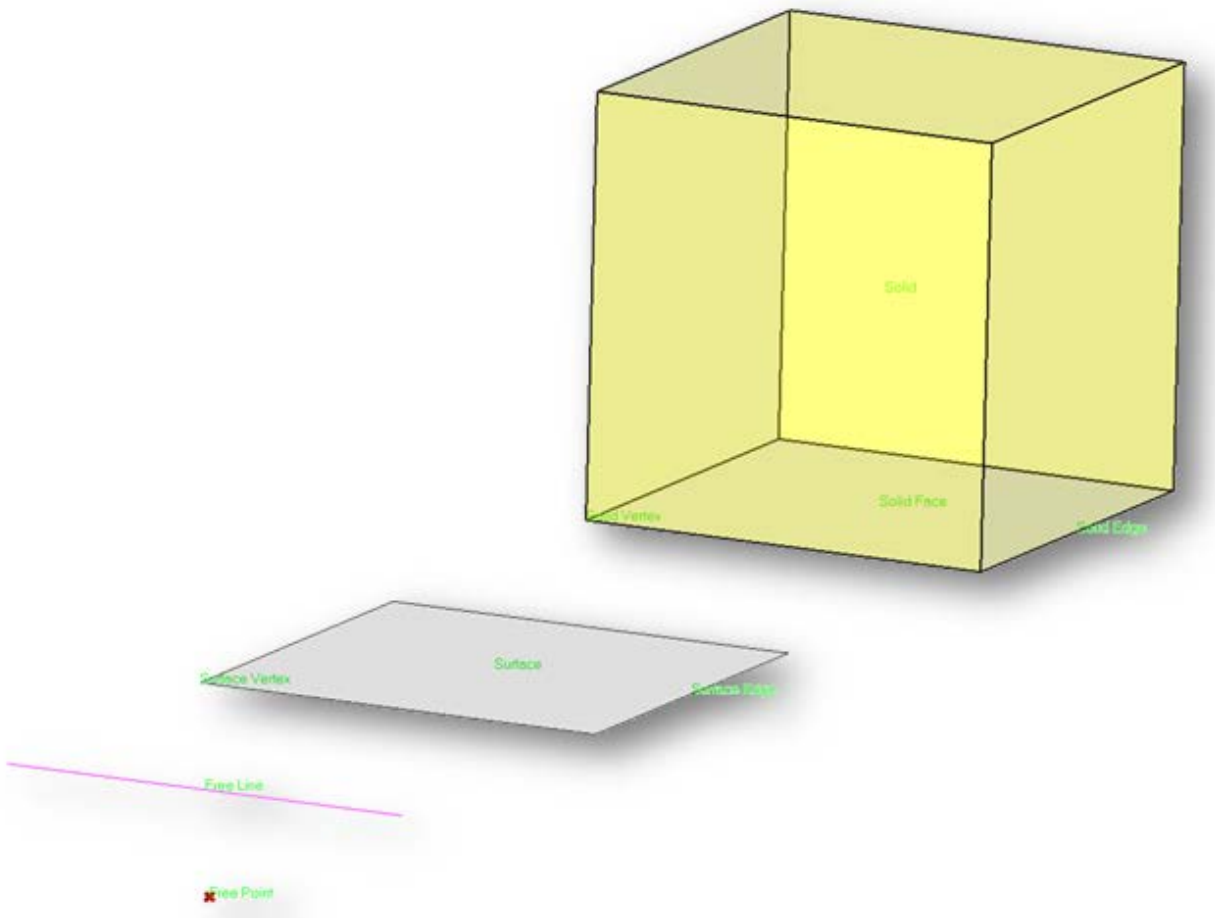
Metadata is now propagated when performing most geometry editing operations, and certain geometry creation operations (duplicate, spline/trimmed surfaces from free lines). This allows for metadata to be duplicated and combined during relevant operations, leading to improved automation and scripting capabilities.

## ▼ Spline Creation

- An option to keep the input line endpoints has been added.
- When creating a spline mesh, the nodes are now automatically equivalenced with the input mesh or geometry.
- The Spline from Bound Nodes tool has been updated to utilize all input nodes in the spline creation.

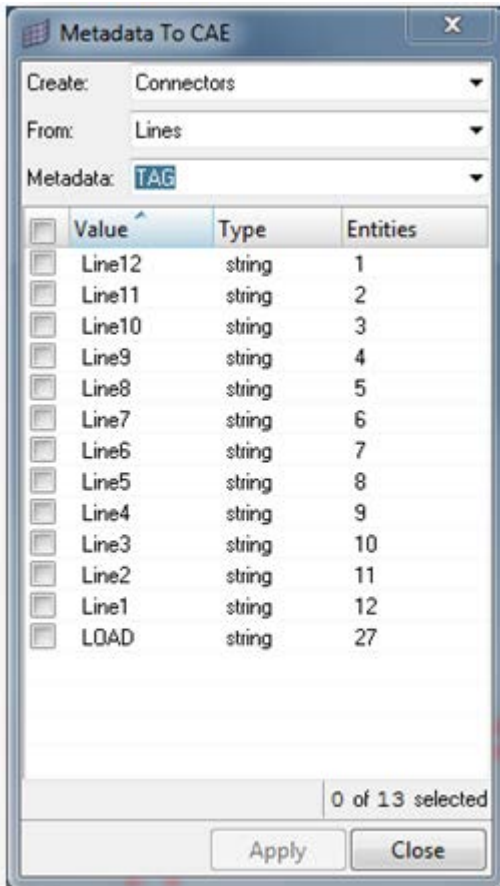
## ▼ Tags

All geometric entities are now supported for creation of HM tag entities.



▼ **Metadata to CAE**

This tool has been enhanced to support the creation of regions.



#### ▼ Resolved Issues

- Continued work to improve accuracy and robustness of geometric editing operations, particularly solid editing.
- Loads on geometry are now properly deleted when deleting geometry.
- Addressed model-specific midsurface creation and editing issues.

## Meshing

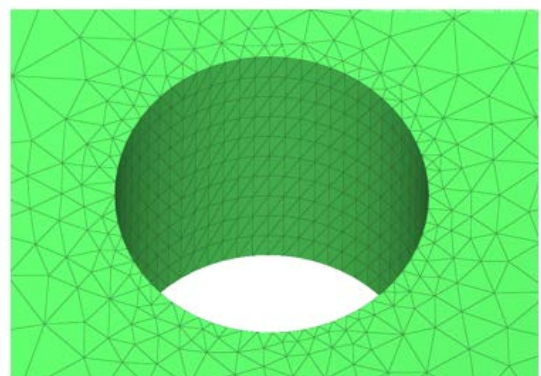
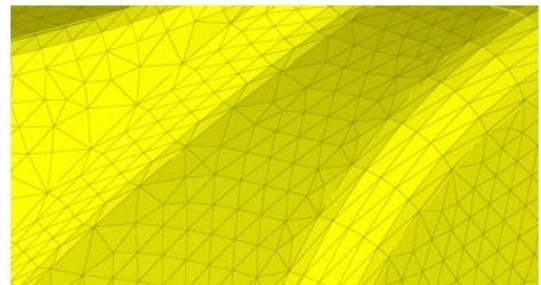
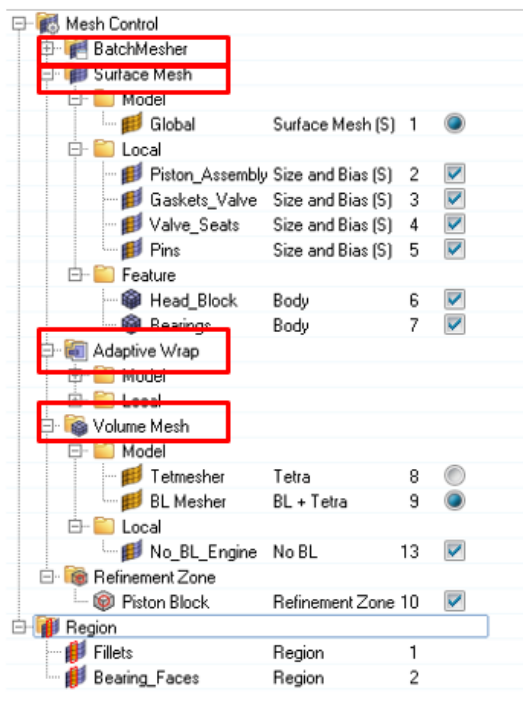
### ▼ Mesh Controls

A new browser-based workflow for meshing has been implemented. This workflow utilizes the new "mesh control" entity. This new entity:

- Allows for the definition of formal, centralized, reusable meshing scenarios.
- Is saved to the .hm file for data persistency.
- Has both model and local controls available.
- Allows for easy reapplication during design changes, via the use of regions.
- Utilizes the Entity Editor for setup and review.

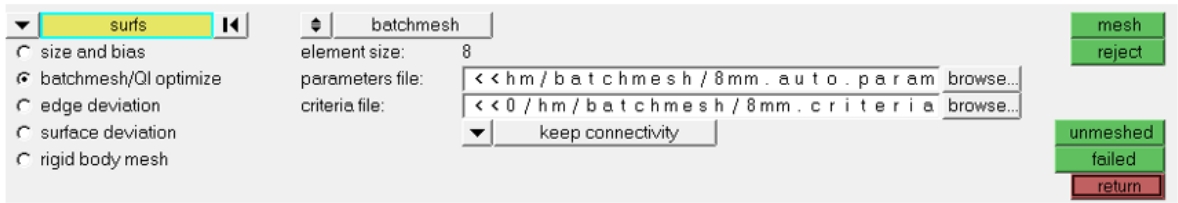
The following meshing algorithms are available:

- BatchMesher (single CPU).
- All surface and element-based automeshing algorithms, including proximity, angle-based and geometric refinement capabilities.
- Feature-based surface tria meshing.
- Volume meshing, including standard and CFD tetra meshing, and local BL control.
- Refinement zones for box, sphere, ellipsoid, cylinder, cone and frustum shapes, which apply to surface, wrapper and/or volume meshing.



▼ **BatchMesher**

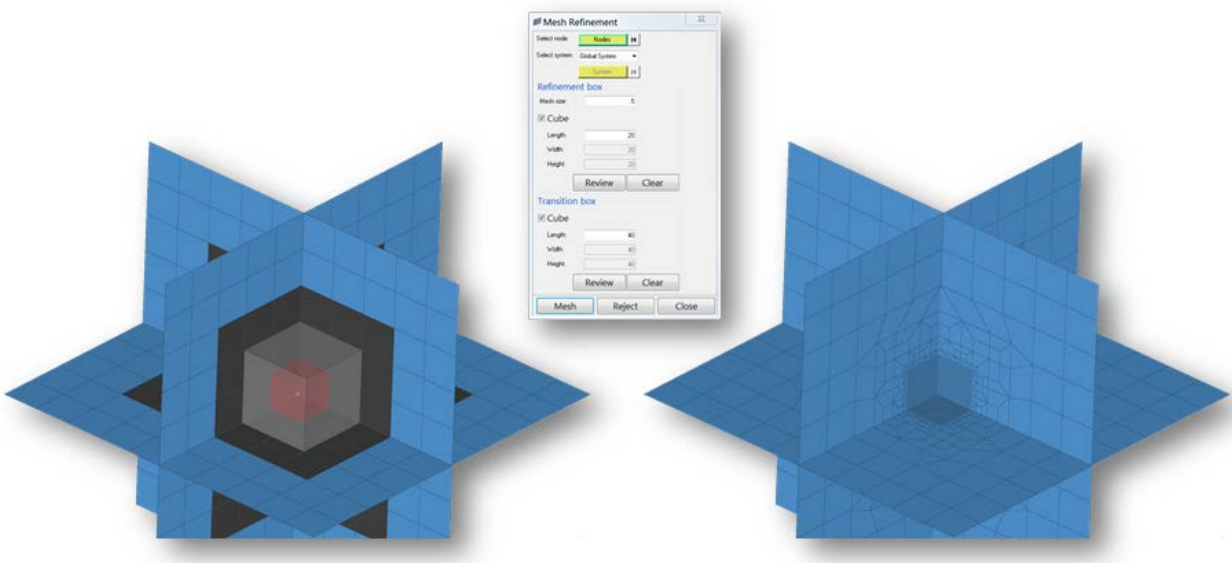
- Now accessible within the Automesh panel, via the renamed "batchmesh/QI optimize" surface meshing subpanel. This is useful for meshing individual components, or select surfaces of components using a single CPU. The standalone BatchMesher is still recommended for use for multi-part meshing, as it has the added benefit of multi-CPU processing.



- Continued improvement on feature recognition and capture, focused primarily on fillets, holes, beads and bird-beaks.
- Significant mesh flow improvements
- Added support for taper to the criteria file

▼ **Mesh Pattern Refinement**

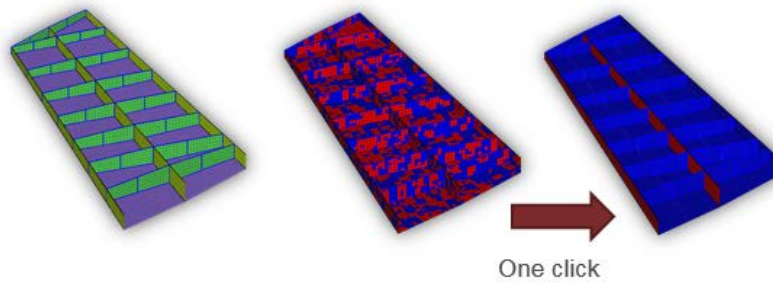
The ability to refine shell elements by pattern within a cube is now possible. A target mesh size, along with a refinement box and a transition box, are input.



▼ **Improved Normals Adjustment**

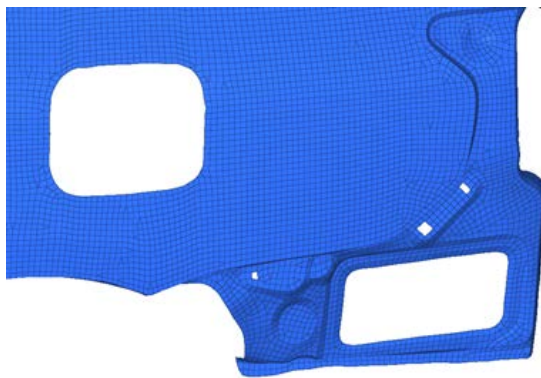
The workflow for adjusting element and surface normal has been significantly improved. The panel has been redesigned to aid in a more streamlined workflow, adding capabilities to adjust entities more efficiently and with better performance.



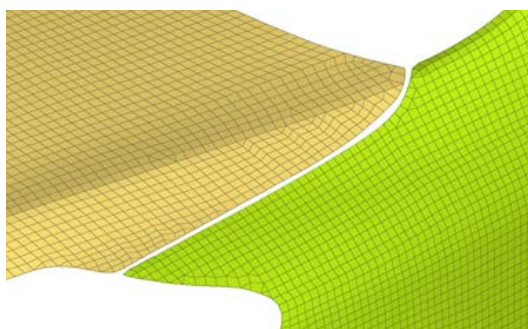
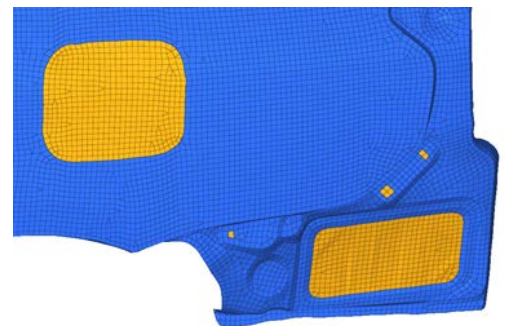


### ▼ Fill Hole/Gap Tool

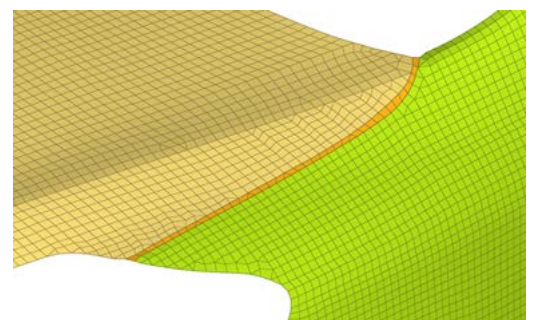
- Auto detect and fills the holes/gaps within selected components/elements
- Remesh the filled holes/gaps/patch with neighboring element type and size
- Ability to detect and fill the feature holes



Automated Hole Fill



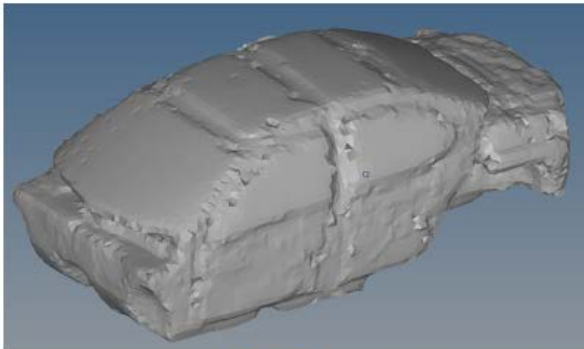
Automated Gap Fill



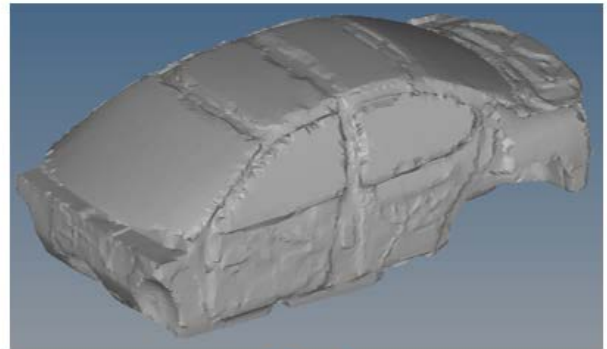
### ▼ Acoustic Cavity

Improvements have been made to the Acoustic Cavity mesher and workflow:

- Improvements have been made to better capture features and pockets.
- Creation of a variable size all tetra-mesh is now possible.
- The maximum element size can now be input directly, or calculated based on the maximum frequency and number of elements per wavelength.
- Enhanced UI with access to previously defined cavities, creation of ACMODL cards and tools for creating MPCs between cavities.



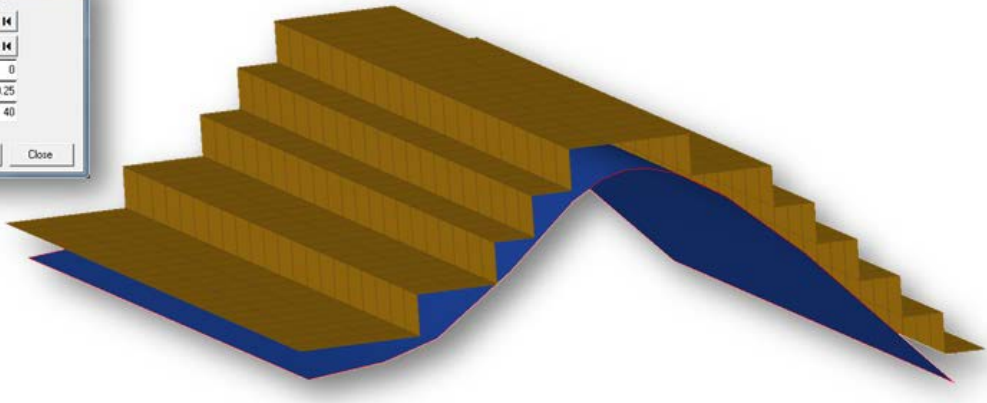
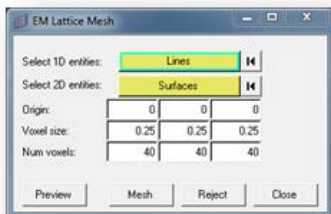
13.0



14.0

### ▼ Electromagnetic Lattice Mesh

A new tool has been added for generating EM lattice meshes.



### ▼ Resolved Issues

- R-body meshes now have improved connectivity with fewer gaps
- Addressed several issues and added a few enhancements for the Mesh Coarsening tool.
- Improvements for remeshing elements using the "surface edges" mode have been made.



- The performance of creating features using the "connected" mode has been significantly improved.

## APIs

---

### ▼ Command File

- `command.cmf` has been replaced by `command.tcl`. This allows for significantly improved ease of scripting and readability.
- The ability to block writing of all view commands to the command file has been implemented.
- New environment variables have been added to control the command file read/write location (`HM_COMMANDFILE_DIR`) and mode (`HM_COMMANDFILE_MODE`). The available modes allow for appending to a command file, overwriting an existing command file, or creating a unique command file.

### ▼ `hmmenu.set`

- New environment variables have been added to control the `hmmenu.set` file read/write location (`HM_MENUSET_DIR`) and overwrite behavior (`HM_MENUSET_OVERWRITE`).

### ▼ Hole Detection

A new, consolidated suite of APIs has been added for detecting and reporting information about holes in both geometry and FE.

### ▼ Resolved Issues

- Improved the performance of group entity `baseelement*` data names.
- Resolved multiple issues with `*setvalue` and `hm_getvalue`.

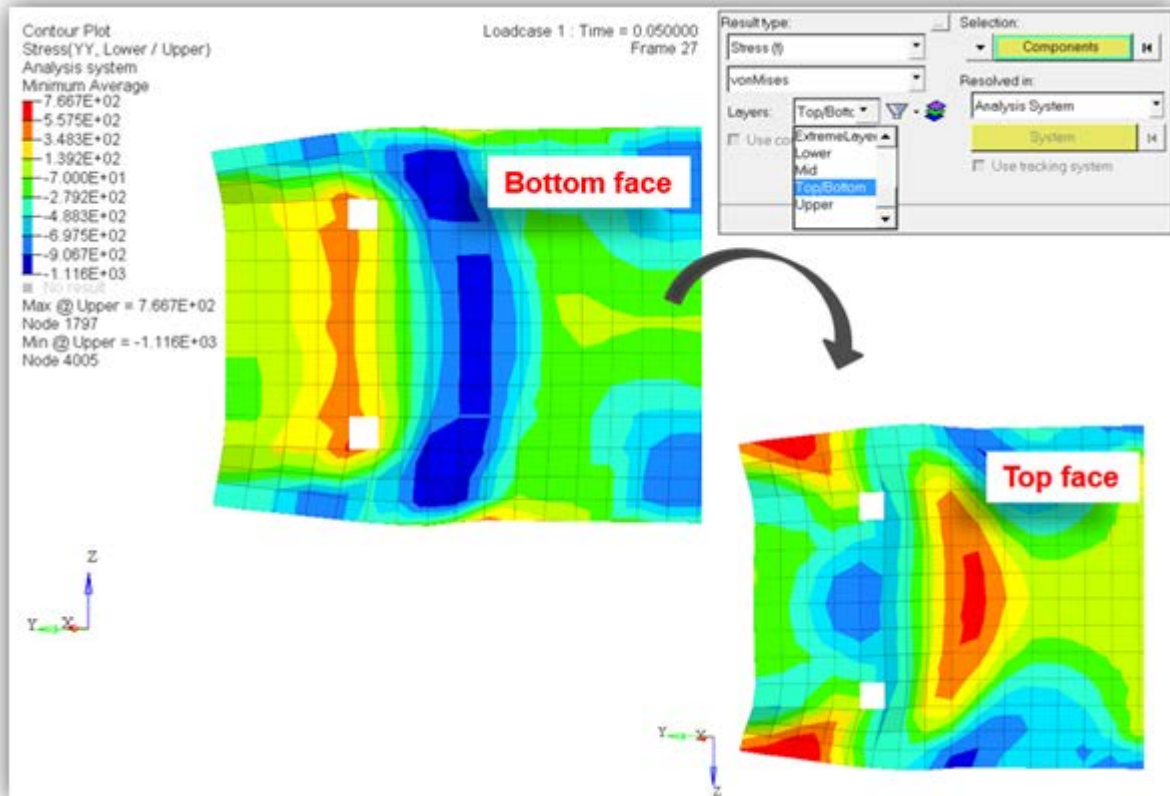
# HyperView 14.0 Release Notes

## Highlights

HyperView version 14.0 brings a number of new features, general enhancements and bug fixes. These include Top/Bottom Contouring, addition of Material/Property information in the HV Browser, result display for multiple overlaid models, a browser in HyperView Player, and video overlay enhancements. In addition, several issues have been resolved (For a complete list, see below).

### ▼ Top and Bottom Contour

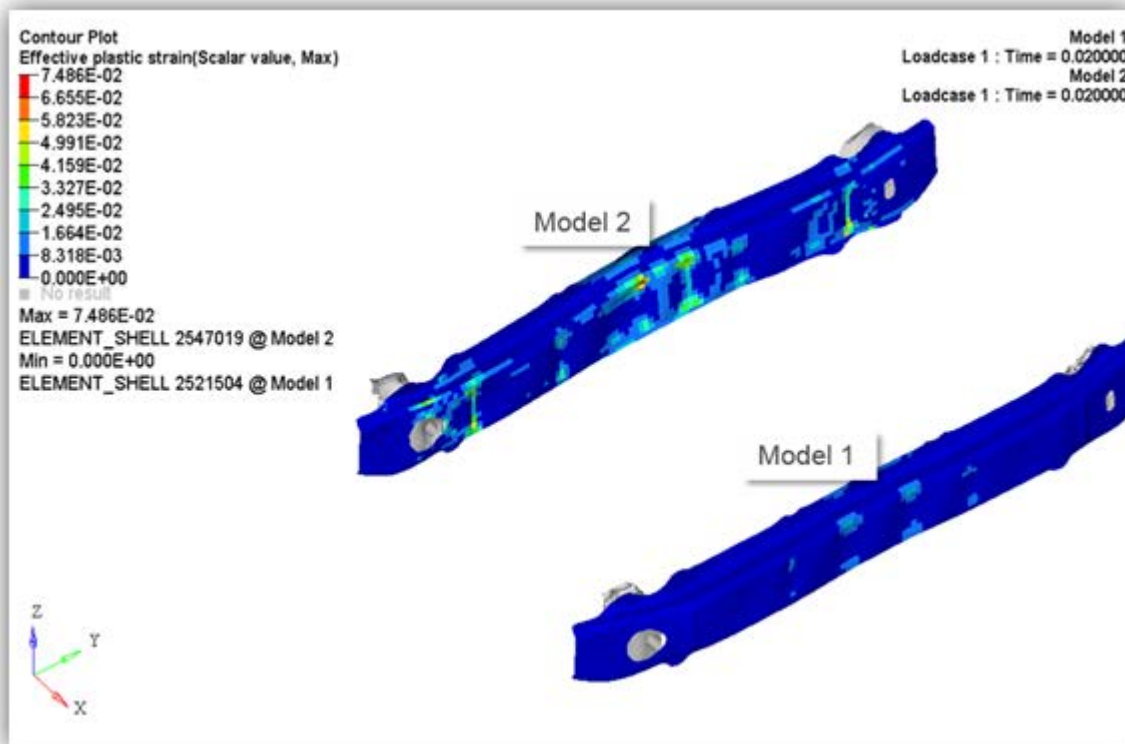
This new method of contouring can be used to read and contour results (for example, stress or strain) on both the top and bottom layers of a shell element simultaneously in a single plot. The user can choose any of the available layers as top and bottom. This can be used to review which side of an element/part is under tension or compression.



### ▼ Multiple Model Result Overlay

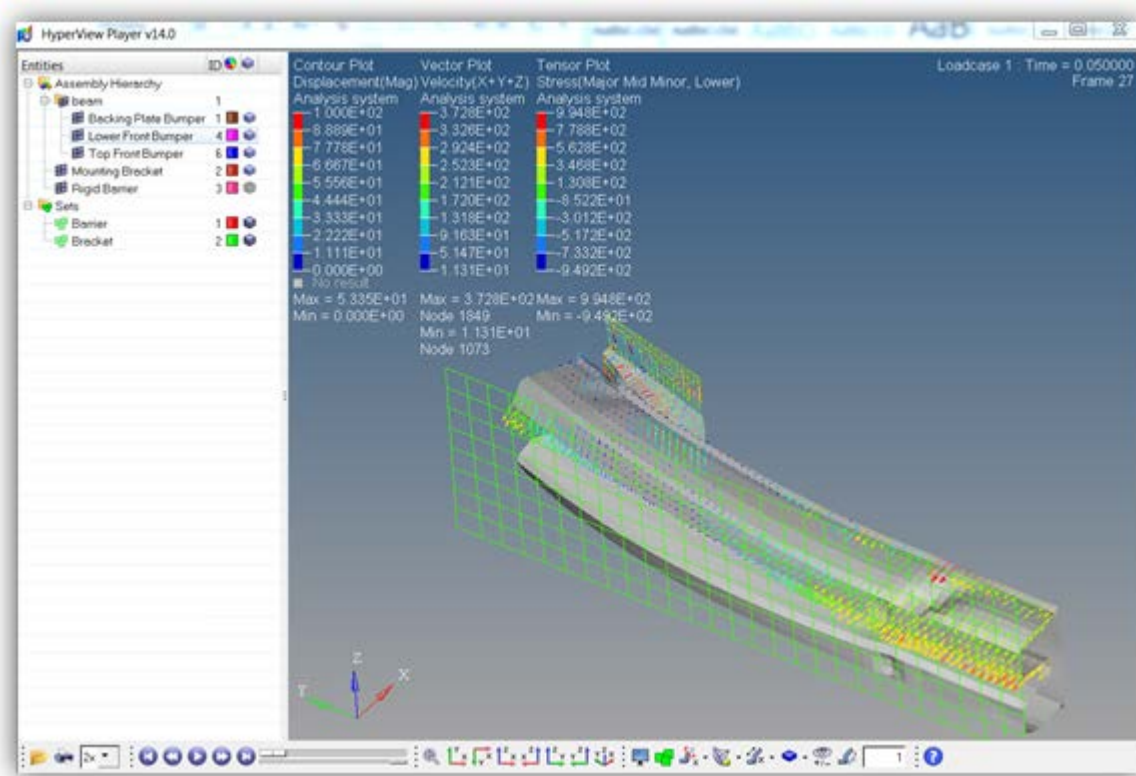
This enhancement allows users to plot and view results for multiple overlaid models in a single window without having to switch between the models (In previous releases, result plot can be viewed only on the current model). When multiple models are overlaid, users can select any of the displayed parts and apply a specific result. This provides a more convenient way to compare

multiple models in a single window layout. For a given plot type (contour, vector or tensor) only a single result-type can be plotted across all models at a given time and the list of result-types available are based on the current model. Specific rules governing copy/overlay of windows, import of legacy session files have been established. Please refer to HyperView User's Guide documentation.



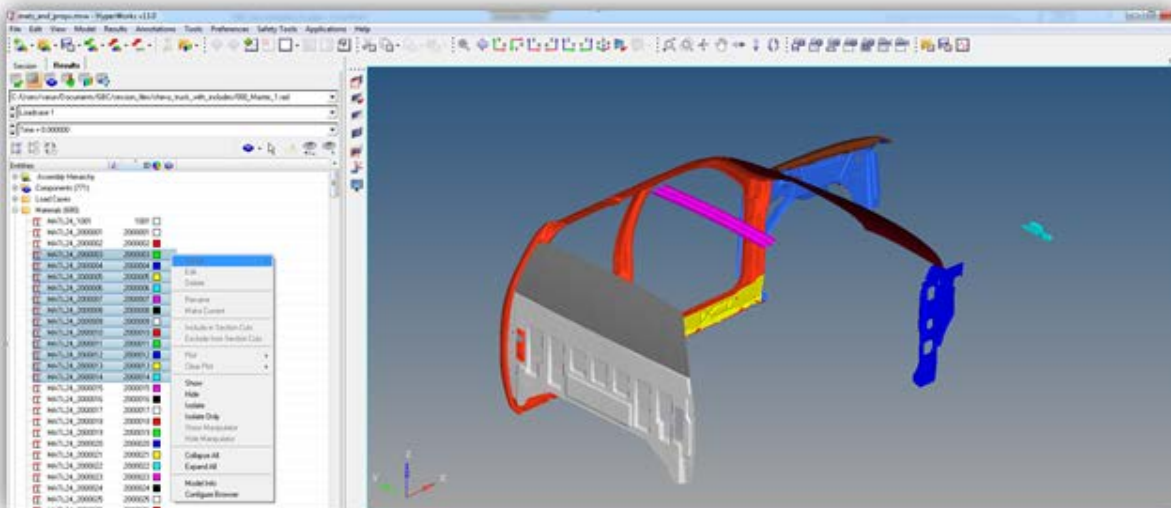
### ▼ HyperView Player Browser

A basic browser showing assembly, components and sets information has been added to HyperView Player. The components will be listed along with sub-assemblies under the assembly folder. Users can change the mesh style of components and the draw style of sets. They can also show, hide or isolate component(s), set(s) or assemblies via a context menu.



▼ **Material and Property Information in the Browser**

Limited support of material and property information is provided in the browser through the Advanced Result-Math template. Name and ID of the materials and properties will be displayed. Users can show/hide/isolate by material or property and use virtual collector to pick part in graphics area to see corresponding material or property.

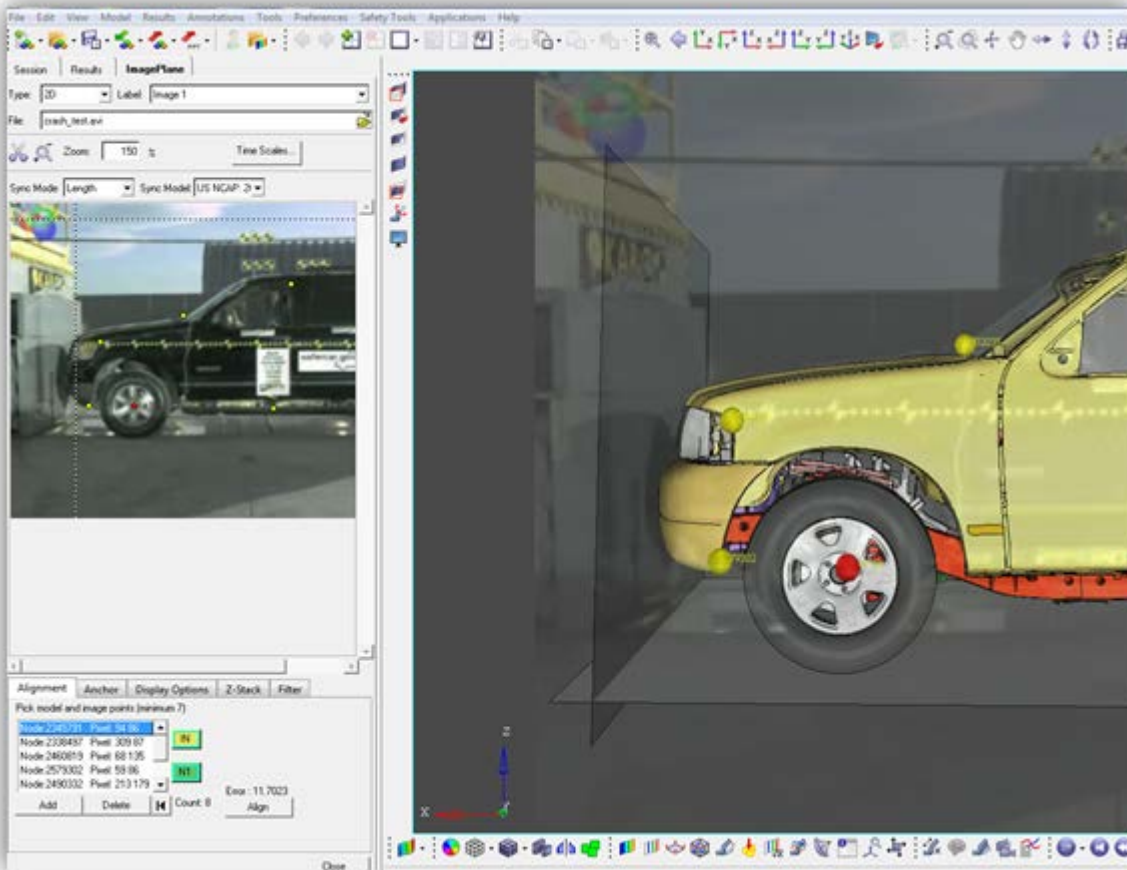


## ▼ Video Overlay

(Also available in HyperWorks Desktop 13.0.111hotfix)

Significant improvements have been made to the video overlay (Image Planes) functionality in this release of HyperView. It is now possible to achieve a good overlay of Model and Video/Image by providing the properties of the camera that was used to capture the image or by a multiple point alignment method. In the former, camera properties such as focal length, sensor height etc., are input through the views entity in the browser. This provides a more direct way of achieving alignment between model and image. A new projection type under views called Lens has been added for this purpose.

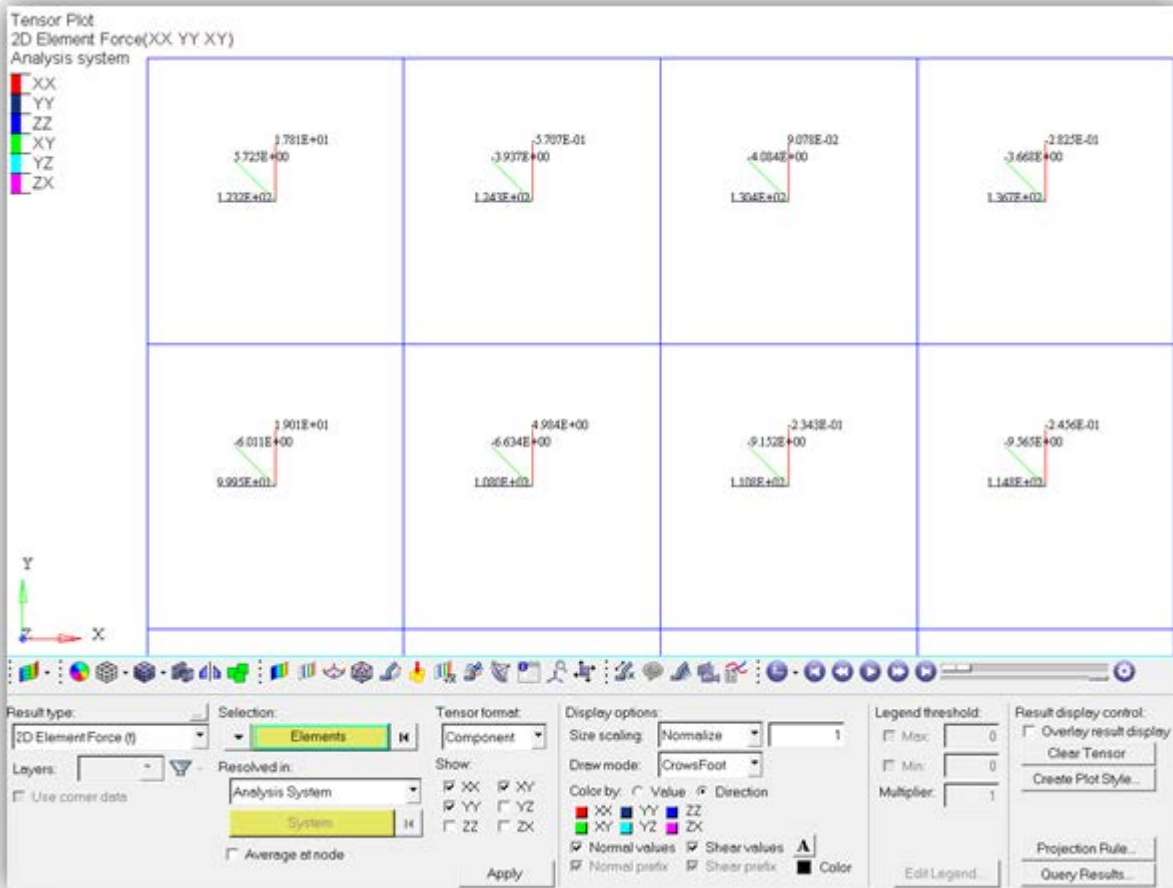
In the latter method, a new tab has been added under the 2D Image Plane section and this method works by allowing the user to pick points (pairs of pixels in the image and corresponding nodes in the model). A minimum of seven points are required as input. It is also recommended that points are picked at different depths and preferably towards the center of the image/video. The GUI will show a dynamically updating error value to indicate to the user the quality of the selected points. HyperView will then use the points to adjust the model size and orient it in the appropriate view so that a good alignment is achieved between the model and the image/video.





### ▼ Crow's Foot Draw Mode for Tensors

Crow's foot representation is provided as one of the draw modes in the tensor panel. Any 3 vectors can be chosen, but this is used widely in the aerospace industry to review in-plane forces on shell elements.



### ▼ Improved Visualization

Significant performance and usability improvements have been achieved by automatically controlling the amount of mesh line detail shown on the screen. The mesh lines are drawn appropriately dependent on the model zoom level. For example, if the model is viewed at a global level, the mesh lines will be suppressed; when the model is viewed more closely, then mesh lines will appear. The mesh lines are also drawn inheriting the component color as opposed to pure black mesh lines which was the case previously. Improvements have also been made to the visualization and performance by utilizing current GPU technology to increase large model handling. Added optimizing view controls to help increase dynamic model handling speeds by simplifying graphics.

### ▼ Updated 3Dconnexion Device Support

Updated support for 3Dconnexion devices, including better default settings and usability controls through the updated driver APIs within HyperView.

### ▼ Added Better Support for Integrated GPUs on Notebooks

HyperMesh will now try to utilize the dedicated GPU (graphics) in a laptop or notebook using a combination of Intel's integrated GPU and NVIDIA's or AMD's dedicated graphics cards.

### ▼ General Enhancements

- A checkbox called **use corner data** is added to the Vector panel and will be enabled if such data is available in the op2/xdm result file. Using this option will render corner bound forces and moments on 1D elements as vectors, provided that the Result-Math template is set to Advanced and the OptiStruct/Nastran input deck is used as model file.
- A new toolbar called **Visibility Controls** is added to HyperView for quick visibility manipulation – Isolate, Hide, Display All/None/Reverse and Unmask. This toolbar works on both component and element selections.
- Ply systems for orthotropic/composite shells in RADIOSS are supported. The systems for these individual plies (layers) can be reviewed through the Systems Review dialog. Stresses and Strains can be resolved in the ply system. To enable this, the datatype PHI must be written to the RADIOSS animation file.
- Some modifications have been made to the FLD panel: the existing compression line has been renamed shear line. A new line called **uniaxial compression line** has been added. The Edit Legend dialog has a new category called **High Wrinkle tendency**.
- Option to turn off text "Value =" when value option is selected in the Measures panel is provided.
- Option to show subcase along with value at same time for an envelope loadcase is provided in the Measures panel.
- HV Legend background color will also be made transparent when capturing the graphics area through image capture toolbar.
- The text 'CornerData' will be displayed in the legend title if the result plot is generated using such data.
- Option to select any set of simulations steps to trace is provided in the Tracing panel.
- "High Ascii" international characters are supported in the Notes panel.
- Options to turn ON/OFF, select font and pick color of prefix is added to the Vector panel.
- Options to turn ON/OFF, select font and pick color of "principal prefix", "Normal prefix", "Shear prefix", is added to the Tensor panel.
- Tresca is added to the list of computed invariants in the Contour panel.
- New option to quick query a contour result is provided. This will display contour value and the ID of the entity to which it is attached.
- Option to set the precision to zero is provided in the Measure panel.
- Triaxiality and Lode angle parameter is added to the list of computed results for stress tensor in the Contour panel.
- Time delay used in export of GIF files is saved to HW settings (hwsettings.xml) file.



## ▼ Resolved Issues

- By face selection method does not work when normal of elements are flipped.
- Result math operator BCNodeToElem shows N/A on mid-side nodes for 2nd order elements with corner bound values.
- Application crashes when animating an adaptive model with solid elements.
- Modifying averaging method drop down resets resolved-in system to Elemental system.
- Drop in performance when expanding all folders in the Include browser in Linux.
- Video tracking is not restored when session file is re-opened.
- Contour Legend appears vertically compressed on an Ultra-High Def display in Linux.
- Saved/reloaded views result in the model disappearing from the display.
- Last load case missing in H3D containing envelope of all load cases (also resolved in 13.0.111hotfix).
- Application crash while trying to display contour of corner scalar data.
- Mismatch in contour between source file and H3D file in the case of derived load case containing solid elements.
- Include view in Results browser does not show non-Cartesian systems.
- Application crash when accessing Include browser after loading H3D file created from Madymo output.
- Plotting corner data results and exporting H3D exported both corner and centroidal data causing performance issue.
- Contour plot on H3D does not match the one on the source result file for a variety of use cases.
- User profiles not written upon exiting HyperView on Linux OS.
- Unable to select nodes after deleting pages in the session.
- Perspective setting is not saved to the session file leading to a change in model view when the file is re-opened.
- Value filter setting not used when user applies contour settings to Iso panel.
- Difference in Spaceball mouse speed between HyperView and HyperMesh.
- User specified translation for exploded view fails to restore when session is re-opened.
- View synchronization has clipping issue for models with exploded views.
- Contour plotted through the browser shows N/A due to the presence of white space in the result type names.
- Unable to pick entities on non-current model when creating measures using Entity Editor.
- Tolerance settings not restored in FBD panel.
- Crash using GetReaderList API when a non-existent file is passed as an argument.
- Application crashes when using user-defined systems on measures involving overlaid models.
- Graphics issue when overlaying and animating FEMZIP result files.

- Incorrect stress results shown on axisymmetric elements in Abaqus ODB result files.

▼ **Known Issues**

As of September 1, 2015 (Version45), Google Chrome browser has disabled support for NPAPI based plugins citing security risks. This means HyperView Player embedded in HTML pages will not work with Chrome browsers. We are currently investigating other methods to support HVP plugins on Chrome. Until an alternative solution is found, users are recommended to use other browsers that still support NPAPI such as Firefox and Opera. Windows users can use Internet Explorer which uses ActiveX to support embedding in the browser.

## Solver Interface

---

### ▼ General Enhancements

- LS-Dyna FEMZIP (`d3plot.fz`) result files up to v8.08 can be read in HyperView.
- PAMCRASH 2G result files up to v2014 can be read in HyperView.
- Reading of more than 999 RADIOSS Animation files is supported.
- OP2 FEMZIP result files (`op2.fz`) can be read in HyperView.
- Assembly hierarchy comments written to the Nastran input file by ANSA/PATRAN can be read in HyperView.
- OP2 result files generated by PARAM, POST, 1 option in input deck is supported.
- Post-processing of axisymmetric elements (CTRIAX6) results in OP2 Result file is supported.
- Reading of LS-Dyna binary result file BLSTFOR is supported.
- SVAR (user defined state variable) result type in ANSYS result files is supported.
- Kinematic couplings will be represented as RBEs when ODB is loaded into HyperView.

### ▼ Resolved Issues

- HyperView reads older version of ODB result file from memory even after loading updated version.
- Issue reading multiple LS-Dyna `d3eigv` result files.
- Kinetic energy result plot not displayed on CONM2 elements in OptiStruct result file.
- Stress results on CTRIAR elements in OP2 result file not available for plotting.
- Memory issue when loading certain Ansys RST result files leading to application hang.
- Composite stress and strain results from Nastran and OptiStruct result files.

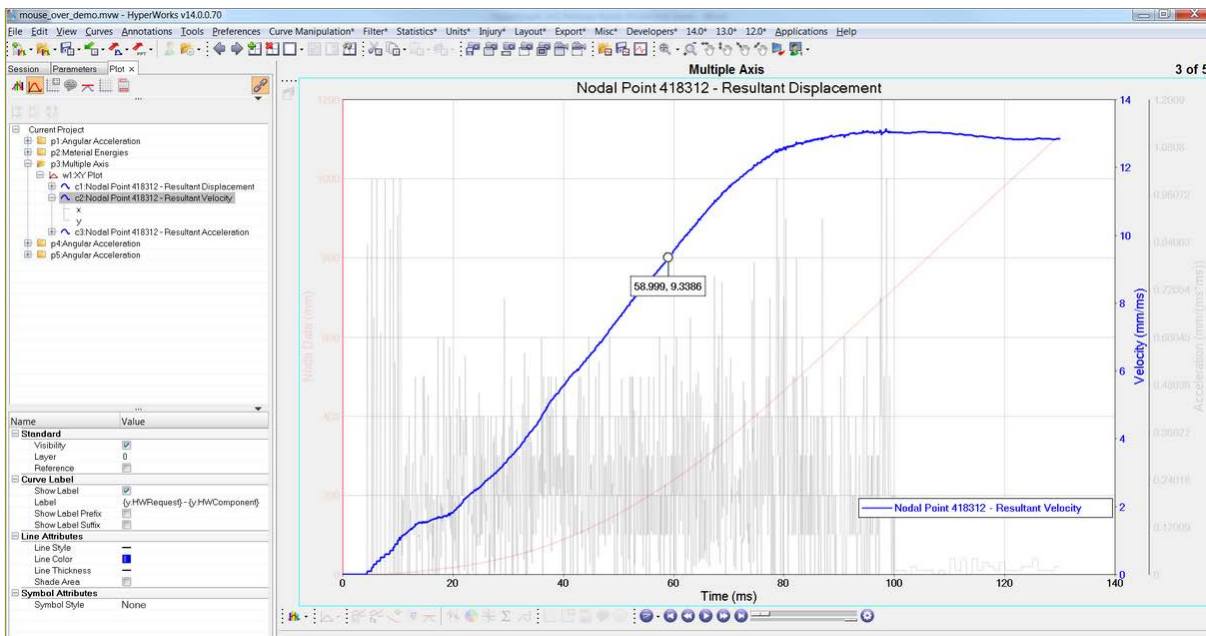
# HyperGraph 14.0 Release Notes

## Highlights

The 14.0 release allows users to import and analyze data more quickly than previous versions. The Inspect mode enables quick data mining for a huge number of curves, Plot Browser enhancements such as filter support for all entities will speed up the editing of multiple entities at once, and features like Summary Tables Using Report Overlay will make tedious hand-editing of summary tables obsolete.

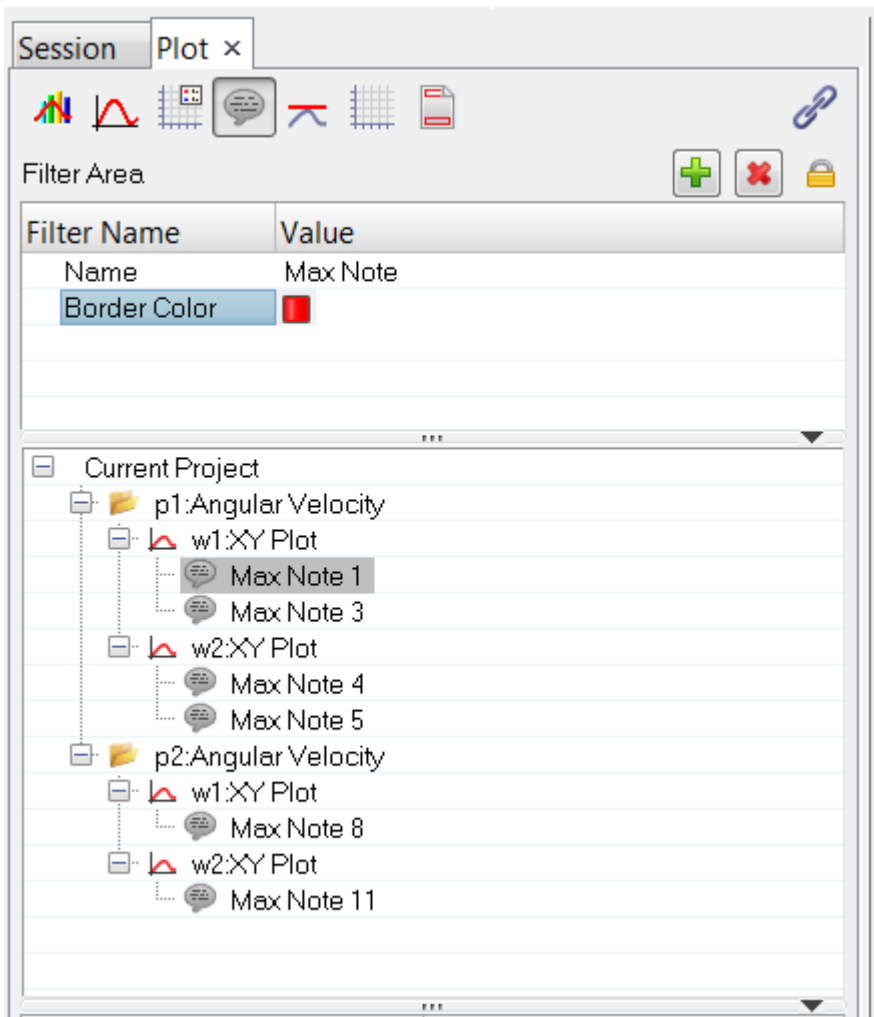
### ▼ Inspect Mode - Mouse-over Curve and Data Point Highlighting

HyperGraph contains enhanced functionality for displaying and analyzing even a large number of curves, thereby improving a curve's visibility and usability. Inspect mode, which increases the screen area of the plot, is triggered via double-click. In this mode, single curves can be highlighted by fading out all other curves both in the graphics area and in the legend, and data point markers can be shown on the pre-highlighted curves when the mouse pointer moves over them. Similarly, notes can be displayed as a symbol and pop-up at mouse-over, which allows for the visualization of many notes in a small area. Pre-highlighting of all plot entities is also supported, including the highlighting of entities in non-active windows. For plots with multiple axes, only the axes associated with the highlighted curves will be shown.



### ▼ Plot Browser - Filter Support for All Entities

The Plot Browser now supports filtering in all Plot Browser Views for all HyperGraph entities. In previous versions, filtering was only possible for curves and vectors in the **Curve View**; now filters for axes, notes, legends, headers, footers, and datum lines in the corresponding views can be added. Adding filters via the property tables using the right mouse button makes it fast and easy to filter for every entity property.



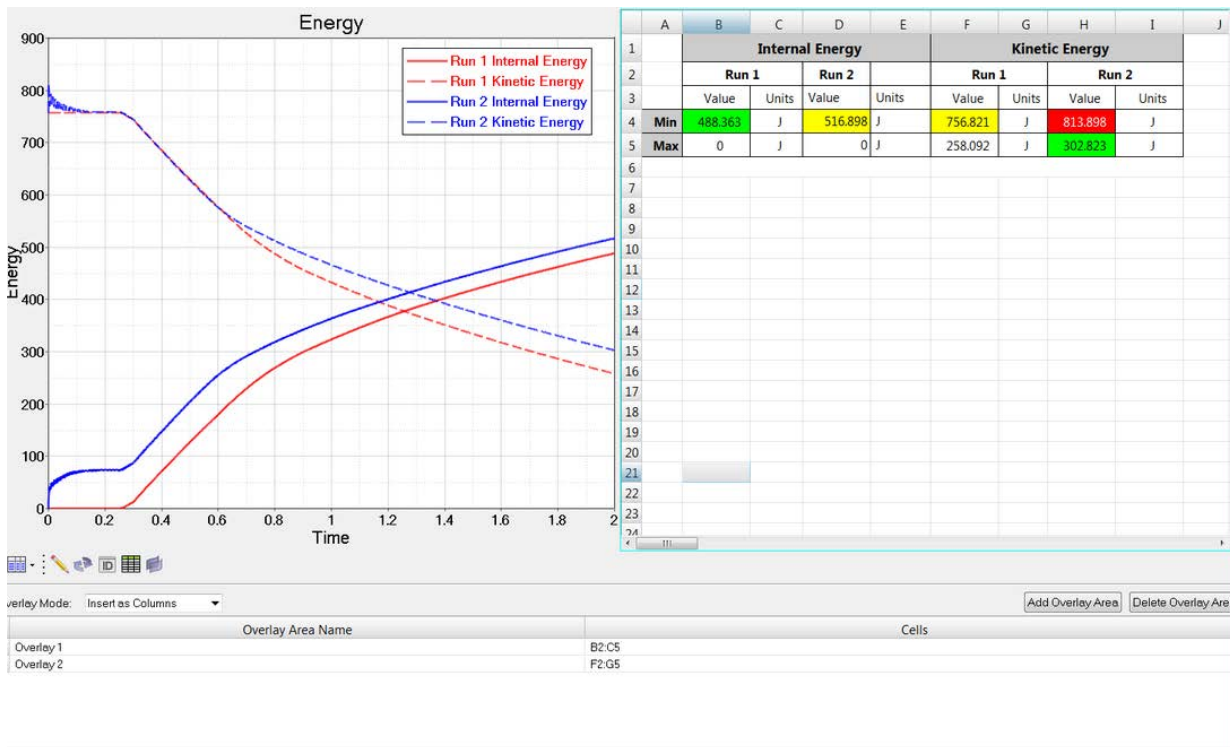
### ▼ TableView Enhancements

Multiple layout enhancements to TableView come with this build. The default table size is increased and cells can be merged and split over columns. Text alignment, both horizontal and vertical, is now possible, and the border thickness can be controlled. Cut, copy and paste from and to Excel is now supported, in addition to the export and import of Excel \*.csv files.

	A	B	C
1			
2		<b>Body Part</b>	<b>Maximum</b>
3		50th% Hybrid3 - LOWER TORSO - Res. a...	9.74753
4		50th% Hybrid3	22.1824
5		50th% Hybrid3	10.9405
6		50th% Hybrid3	10.1703
7		50th% Hybrid3	
8			
9			
10			
11			
12			

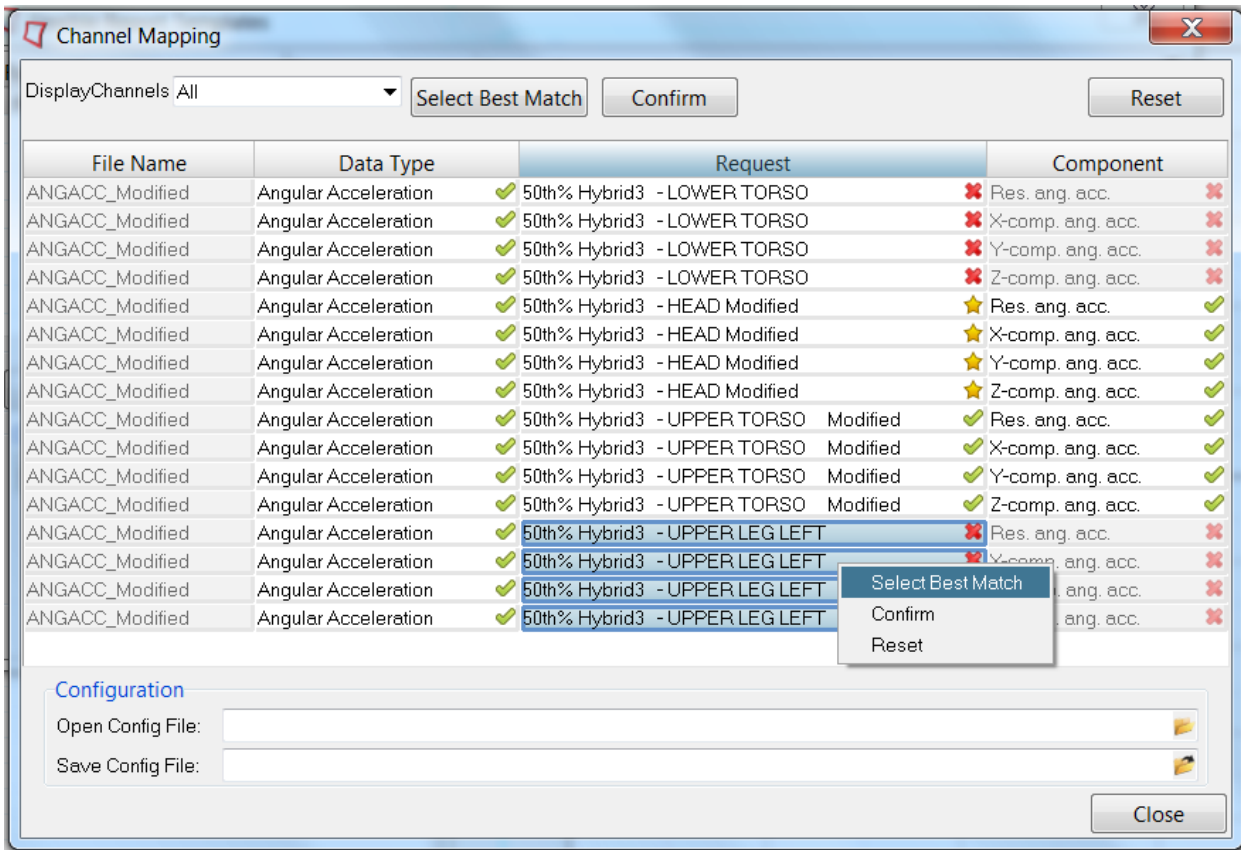
#### ▼ Summary Tables Using Report Overlay

This new feature can replace customized solutions which are designed to create summary tables for overlaid reports. Two methods can be used to overlay tables: appending rows/columns or inserting (interlacing/alternating) rows/columns. "Append" mode will append whole rows or columns to the table, as part of one single overlay area, every time a report is overlaid. "Interlace" mode allows the user to define multiple interlace areas which will be inserted in such a way that results of the same type can be seen side by side, e.g. the summary table will show first the head acceleration values side by side, then the chest intrusion side by side, for all overlaid runs in a crash evaluation. Header cells which are above these areas will be expanded.



### ▼ Flexible Reports – Best Match Algorithm

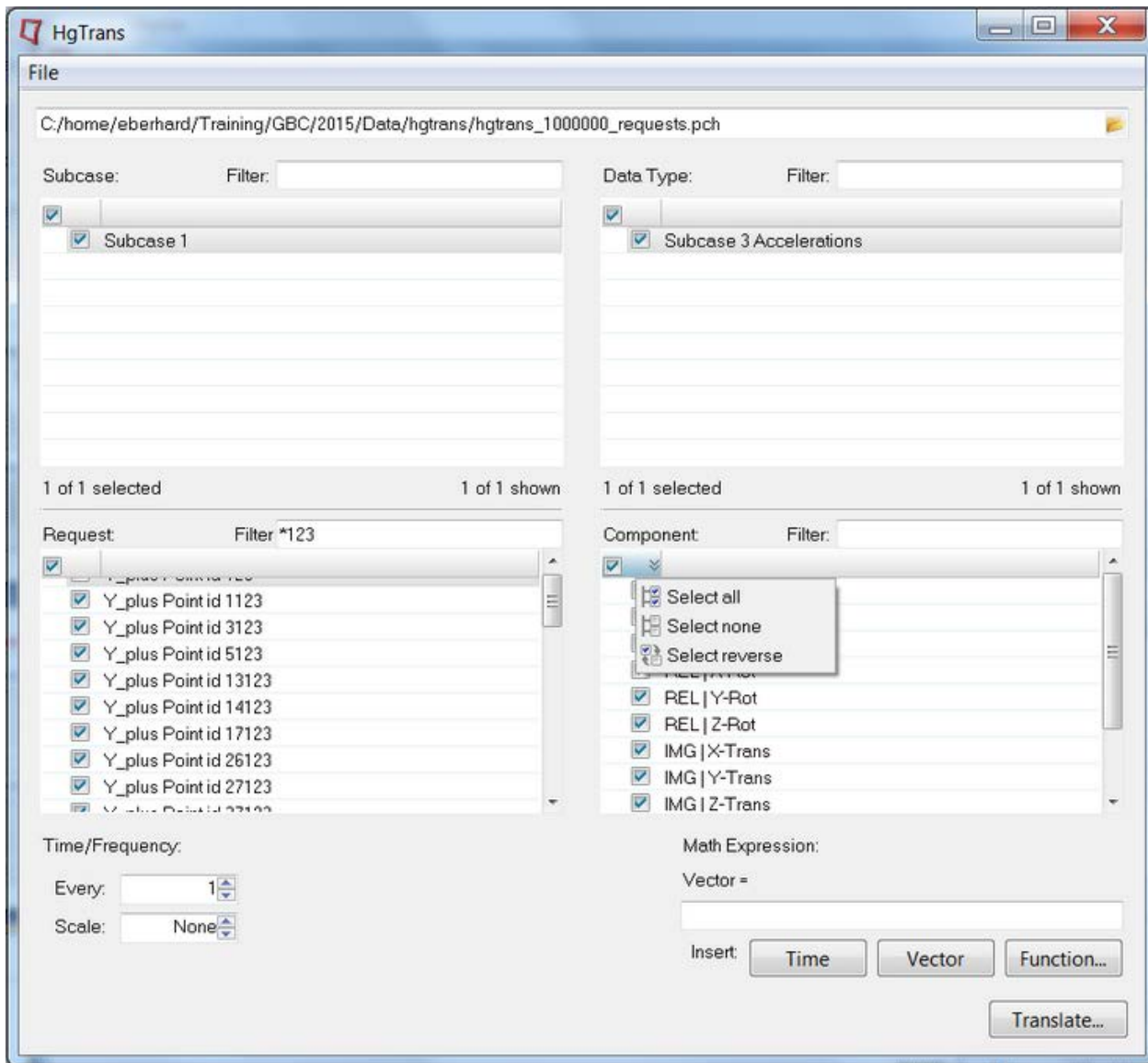
This feature allows the user to search for the best curve channel match based on the Type, Request and Component names when curve channels are not available in the selected result files. A typical use-case would be if the user wants to use a report template for only slightly different channel names, e.g. for driver and co-driver in crash test data. The results are sorted using a string match to the shortest distance, and the user can confirm and edit the search results before the report will be applied.



### ▼ HgTrans Enhancements

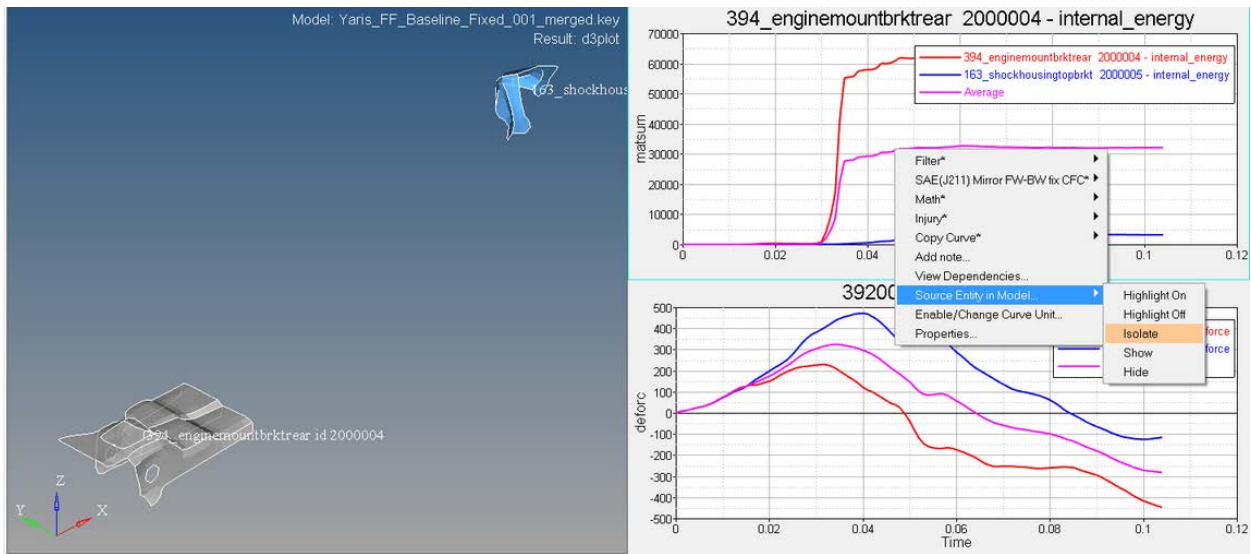
HgTrans now supports improved filtering which allows for the handling of large files with huge request lists via a GUI which has a modern look and feel. Filtering using wildcards allows much more detailed filter strings, and the user is warned before a filter string which would return such a huge list, therefore slowing down the interactive workflow, is applied.





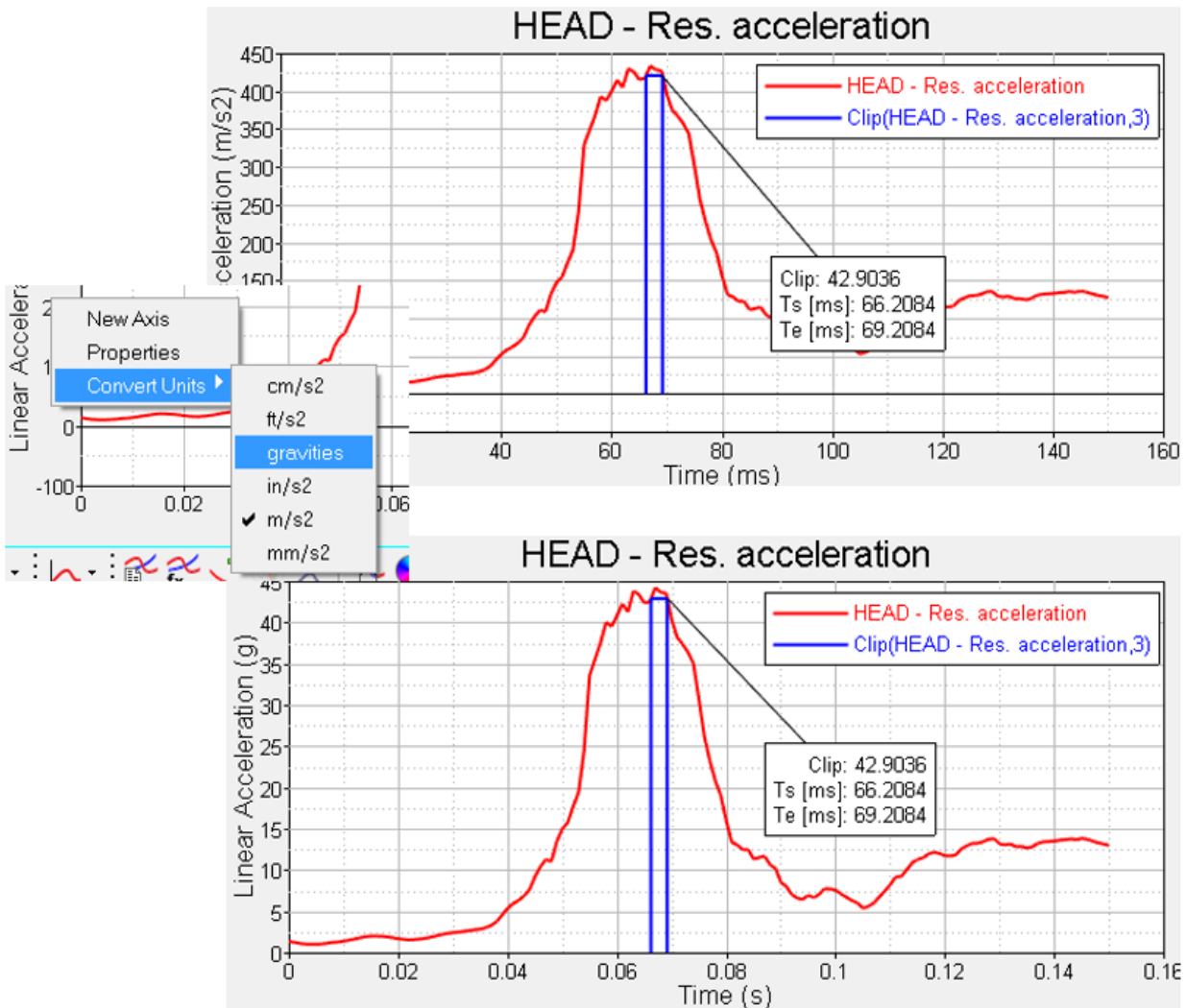
### ▼ Highlight Respective Entities in HyperView

If one or more HyperView windows are open on the same page as a HyperGraph 2D window and the user selects "Highlight Source Entity in Model" from the context-sensitive curve menu, the referenced node, element or component will be highlighted in HyperView. Using the "Isolate" option will show only the components which contains the respective entities. This will also work for math reference curves which refer to multiple result curves of multiple entities. This feature is supported for LS-Dyna only.



### ▼ Unit Scaling Support for Math Functions

The enabling of Unit Scaling for math functions allows to the user change the axis units without the need to change scale factors in vectors or math expressions in notes. The unit scaling converts the input as required by the function and synchronizes the output units with the input units. This is supported for injury and trig functions; the related Vehicle Safety Tools macros are adapted automatically if Unit Scaling is enabled. Preference file statements are available to support unit scaling for customized functions as well.



### ▼ Math Functions

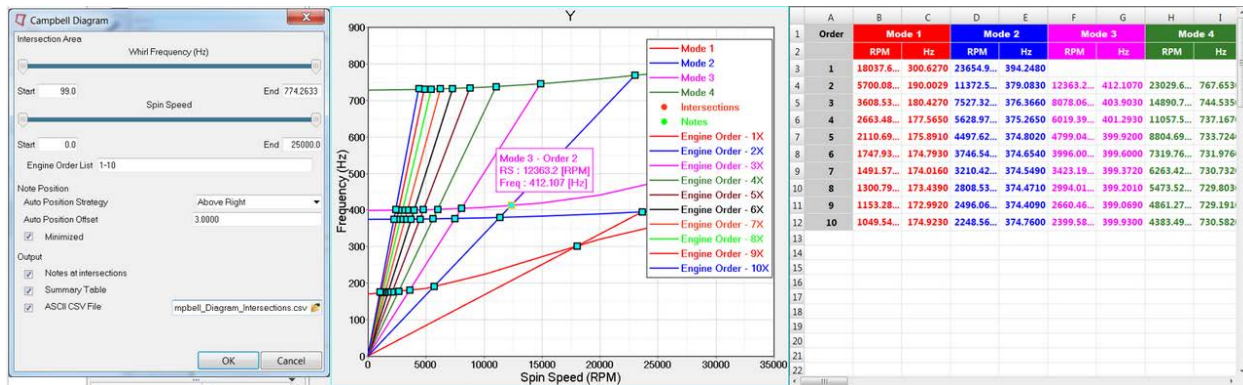
New math functions are added:

- OLC++ : Calculates both the Occupant Load Criteria (OLC) and OLC++
- Sliding Mean Integral: Smooth's the curve using the integral of a sliding window
- Relative Displacement: Removes rigid body motion from time history results. Supported Solvers: RADIOSS, LS-Dyna, PamCrash
- Logarithmic interpolation of curves where one or both axes have a logarithmic scale

### ▼ Campbell Diagrams

Campbell Diagrams illustrate the dependency of the Eigen frequencies on the rotation rates. The user can define, in a separate GUI, the intersection area, the engine order list for full or half orders and the layout of the required output. Summary tables can be displayed in the TableView client and

exported in CSV format. Notes at intersections can be displayed in minimized mode so that the information for each only appears when the mouse pointer hovers over the note's symbol.



### ▼ Solver Interfaces

- PCH Reader: Improved support of subcases for faster selection in the Build Plots panel.
- Nastran NX: New reader for NVH post-processing of f06 files.
- LS-Dyna: Request names will show only NAME and not NAME\_ID by default.
- RADIOSS: CLUSTER data support is enabled in HyperGraph readers.
- RADIOSS: Unit support has been added for T01 files.
- PAMCRASH: Support of ERFH, HDF5 and THP files for PAMCRASH-2014.

### ▼ General Enhancements

- All horizontal and vertical axes can be (un-)frozen at the window level, by using the new **Axis frozen** property.
- The Butterworth filter function has been updated with an optional argument to reduce the ringing on both ends of the filtered curve.
- Nij function now supports Out-Of-Position for 5% dummies.
- File type \*.hgdata has been added to the file browser in the Build Plots panel.
- The file browser list in **Multiple Files** has been made self-adjusting so that it can incorporate long path names.
- A new Templex function, `getcwd()`, returns the current working directory.
- A new Templex function, `setreaderversion()`, allows the user to set the HyperGraph reader version from within a Templex input file.
- The file filter in browsers (such as the **Build Plots** browser and the **Define Curves** browser) will now be saved on Linux.

## ▼ Resolved Issues

- Deleting pages with curves referenced in curves from type math on other pages no longer saved these curves with corrupt math references in a session file.
- The default limit for the number of lines supported by the `hgtxtcolumn.exe` reader that reads multiple formats of ASCII is set from 200,000 to 400,000 lines.
- Positioning notes in the plotting client via the Tcl command “notehandle SetPosition” works now as expected, the upper left note corner are positioned to the closest pixel defined by the axis coordinates given to the command.
- The performance problems overlaying reports with hundreds of pages and multiple curves per page from H3D files are fixed.
- The performance reading big LS-Dyna binout files is no longer affected if this binout files contain a large amount of meta-data.
- HyperGraph notes that contain text longer than the plot graphic area can now be positioned using the mouse pointer. The text alignment works as expected.
- Curves from legacy files that were trimmed using the start- and end-index feature during the import failed in 13.0.110. In 14.0 they will be trimmed as expected.
- The support of element type 33 was added to the pch-file reader. In 13.0.110 the import of files containing results from this element type failed.
- Pch-files with more than 10,000,000 lines can now be imported into HyperGraph 14.0. They failed in 13.0.110.
- The ISO-MME reader failed on Linux in previous versions if the channel names in the files and on the channel files where different considering case-sensitivity. This issue is fixed.
- Trimming curves during import using Advanced Options with Start- and End-Index failed for some result types, the first point was ignored. In 14.0 this will work as expected.
- When notes where attached to the maximum of a curve and these curves where trimmed using the Modify Curves panel, the notes could have vanished despite fitting the plot. In 14.0 the notes only disappear when the attachment point is not in the graphics area.
- Adding min- and max-notes via the context menu of the Plot browser no longer fails when the Vehicle Safety Tools are loaded.
- Every time a session was loaded and saved the y position of the note changed slightly. 14.0 no longer shows this behavior.
- Changing in polar plots the dB - scaling factor to a value different than one no longer changes the phase angle.
- In the Define Curves panel, large integer values of curves from type value are no longer converted to scientific format which caused the loss of valid digits saving and reloading the session.
- Support of result files that contain Swedish characters in the file path works in 14.0.
- The precision with which the curve data will be saved in a session file is now stored in the `hwsettings.xml` file for permanent usage.
- Notes attached to the magnitude of complex curves scaled with db20 reference values now show the y-axis values using expressions like {y} or {x} and not the vector source value as in previous versions.

- Curves in complex plots based on math references expressions no longer get scaled twice using db20 reference values unequal to one.
- Curve labels created by macros from VST (e.g. resultant) are now evaluated correctly and list the different source curves.
- The changes made in curve names created via the Measure panel of HyperView are now saved in report templates and no longer get lost when the report was loaded.
- The axis in complex plots can now be edited when the phase/magnitude positions are swapped which was not possible in previous versions.
- The bug introduced in 13.0.110 that using report template some notes could be missing in the graphics area is fixed in 14.0.
- HgTrans supports the selection of USERDEFINED types from Adams req files.
- Notes could be placed on top of the legends, the behavior of the versions before 12.0 is restored.
- The file list in the Build Plots panel shows now all the plot files used in a loaded session, independently if they were used in the panel or not.
- In 14.0 the axis scaling will be preserved if the axis min/max values were calculated by a fit with curves based on math expressions using customized Templex functions.
- HyperGraph 14.0 no longer crashes while exporting curves created by the HyperView Measure panel as CSV Blocks.
- The stability of the ISO-MME reader has been improved to handle data types which contain metadata but no requests or components.
- EXCEL csv-files created with Export Curves no longer contain empty lines at the end.
- When you load a report template via a customized menu button it will load and highlight the corresponding report definition. This was broken in previous versions.
- Most of the functions to vertically modify curves using the Modify Curves panel only work if the curves are strictly increasing both and x- and y-direction. The curves will now be checked and a warning will be given if the curves are not meet this requirements.
- In 14.0 complex plots labels with an increased font size no longer penetrate into the graphics area.
- The new subcase-based pch-file reader in 14.0 supports Panel Participation data, which fails in previous versions.
- Using high integer values for axis min/max values (e.g. 8 digits node IDs on the x-axis) are no longer converted into scientific format.
- HgTrans crashed in 13.0 loading op2-files. This issue is fixed.
- In previous versions external functions (IPC functions) used to round off to the 6th decimal point when returning real values. In 14.0 these values will have double precision.
- In 14.0 the performance of animating curves has been improved when big and/or colored cursors are selected in the Options panel.
- In the Build Plots panel "X Request" pop-up dialog a filter like in the pop-up dialog of "Y Request" has been added.

- In the Notes panel the field which shows the label of attached curves has been increased to show long curve labels.
- The scaling error of the PSD5 function is fixed in 14.0

▼ **Known Issues**

- Handling tables with more than a couple of hundred rows slows down the performance of TableView significantly. In 14.0 TableView is designed to support report summary tables and should not be used to handle tables with more than ~500 rows/columns.

# MotionView 14.0 Release Notes

## Highlights

---

The Altair multibody solution, MotionSolve, consists of the following capabilities:

- MotionView – the modeling environment
- MotionSolve – the solver
- Automotive Solutions – for suspension design, vehicle dynamics, ride comfort and durability
- HyperGraph/HyperView – the post-processing environment

The 14.0 release of MotionSolve contains three major capabilities that span the breadth of the multibody solution – modeling, simulation and post-processing.

- 3D Contact
- Nonlinear Finite Elements (NLFE) - Experimental
- Advanced Driver and Road for vehicle simulation

Refer to the Multibody Solutions Release notes for a consolidated release notes. This section describes release content specific to MotionView (includes Auto solutions) only.

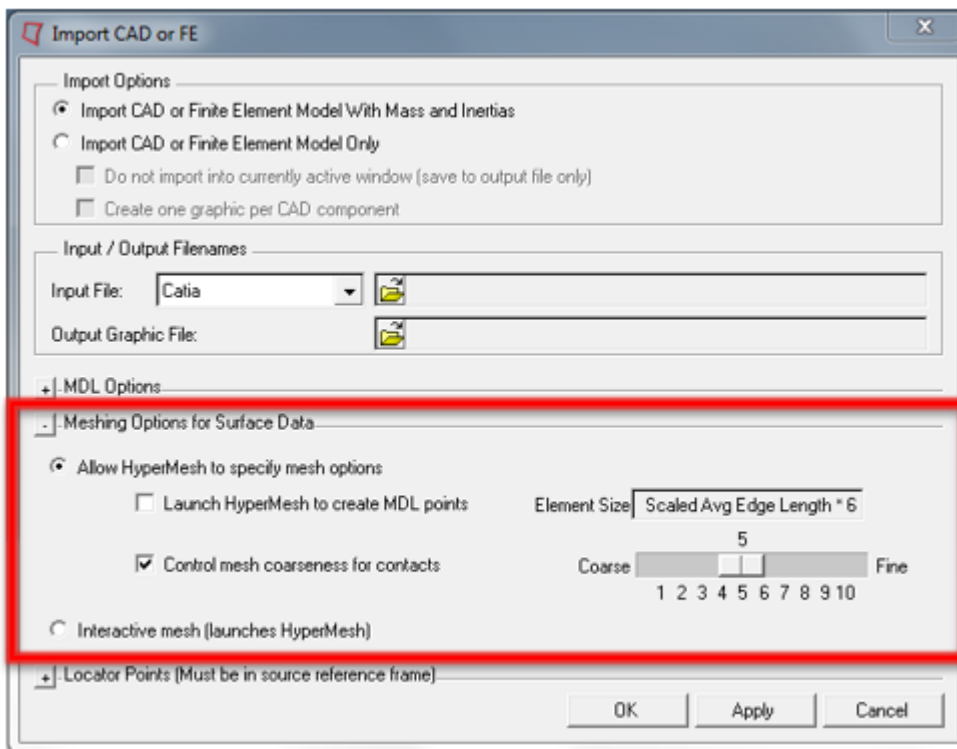


## 3D Contact

The 3D Contact capabilities in MotionSolve have been completely revamped. In line with these changes MotionView 14.0 has major improvements and features for easy modeling and obtain a robust contact simulation results.

### ▼ CAD Geometry can now be Imported Easily and Reliably to Provide the Geometry for 3D Contact

New meshing options are now available when importing CAD geometry. These enable you to control the mesh density in a very intuitive way.



Accurate contact force generation requires uniform meshes. New algorithms, more suited for uniform mesh generation are now used. This new meshing option is only enabled for input geometries and not for FE models.

### ▼ Contact Models are Defined with Minimum of User Input

The Properties tab in the Contact panel has been reorganized to have sub-tabs for normal force and friction force.

Normal Force		Friction Force	
<input checked="" type="radio"/> Impact	Stiffness: <input type="text" value="100.0000"/>	Damping: <input type="text" value="0.1000"/>	
<input type="radio"/> Poisson	Exponent: <input type="text" value="2.1000"/>	Penetration depth: <input type="text" value="0.0500"/>	
<input type="radio"/> Volume Model			
<input type="radio"/> User Defined			

Normal Force		Friction Force	
<input type="radio"/> Disabled	MU Static: <input type="text" value="0.2000"/>	Stiction transition velocity: <input type="text" value="1.0000"/>	
<input type="radio"/> Dynamic Only	MU Dynamic: <input type="text" value="0.1000"/>	Friction transition velocity: <input type="text" value="1.5000"/>	
<input checked="" type="radio"/> Static & Dynamic			
<input type="radio"/> User Defined			

Reasonable contact and friction parameters are provided as defaults. You do not have to change these unless your contacts have special properties.

Parameters are checked for validity as they are entered and the user is alerted to invalid values before the model is run.

Output Requests to measure contact forces are automatically created.

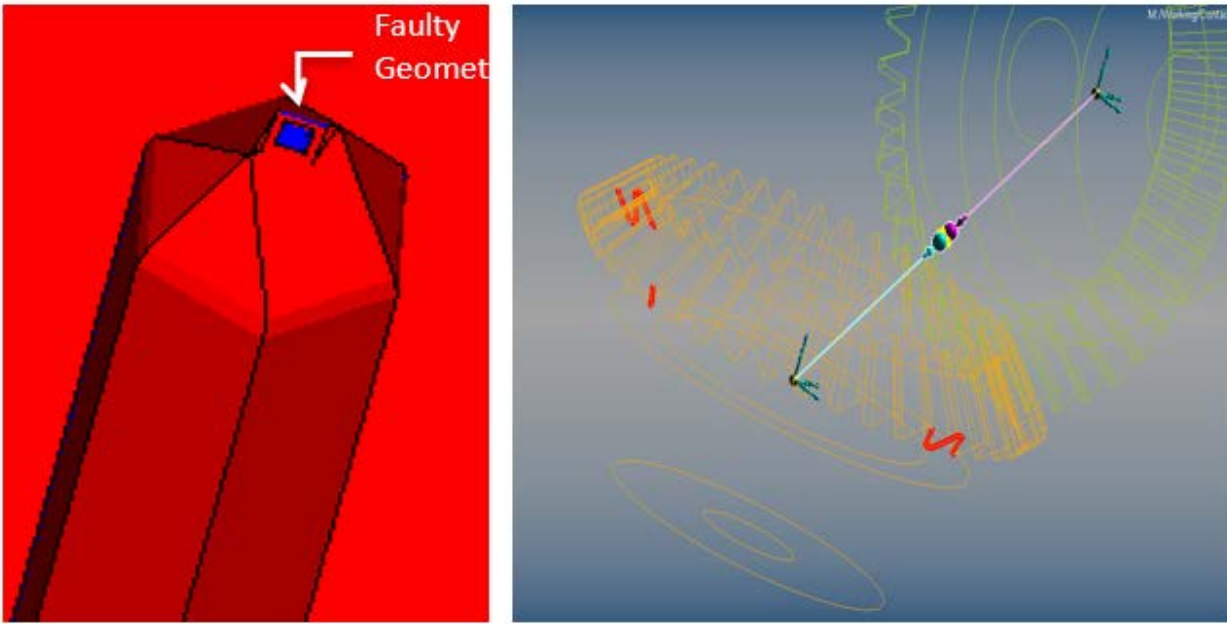
▼ **Extensive Geometry Checks are Performed Automatically to Avoid Potential Failures in the Solver**

All contact meshes examined and invalid or “dirty” geometry is clearly identified during modeling. Contact surfaces for all meshes can be visualized using “Highlight contact side” option in the Connectivity tab.

<input type="checkbox"/> Highlight contact side
<span style="color: red;">Contact side</span> <span style="color: blue;">Material side</span>

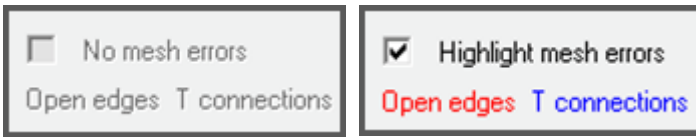
Surface normals can be inverted with just one operation. This is especially useful when contact is happening inside a selected closed geometry.

Red indicates outward normals and blue indicates the inward normals.



The normals for a mesh are automatically corrected when inconsistencies are detected. This is especially useful when tria-meshes are imported from STL files or Wavefront OBJ files.

Open edges and T connections, if any, can be highlighted as shown in the figure on the top right. If the mesh is clean (i.e. it forms a closed volume), the panel indicates "No mesh errors" else the "Highlight mesh errors" option becomes available.



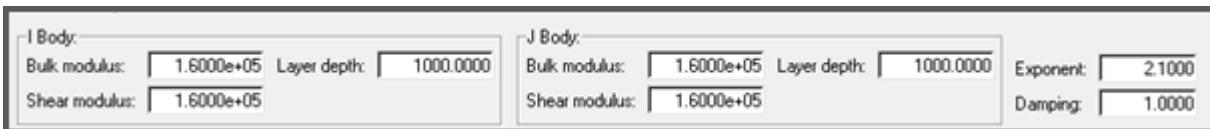
### ▼ Mesh Cleaning and Healing Tools

MotionView now includes a new mesh manipulation tool that allows you to clean previously unusable meshes for contact solution. The capability is accessed from the Import CAD or FE utility.

Import CAD or FE utility now provides additional cleaning options that enable you to fill holes, thicken surfaces, eliminate T-junctions and shrink-wrap unusable meshes. All of this is done with a minimum of user input.

### ▼ New Contact Models are Supported

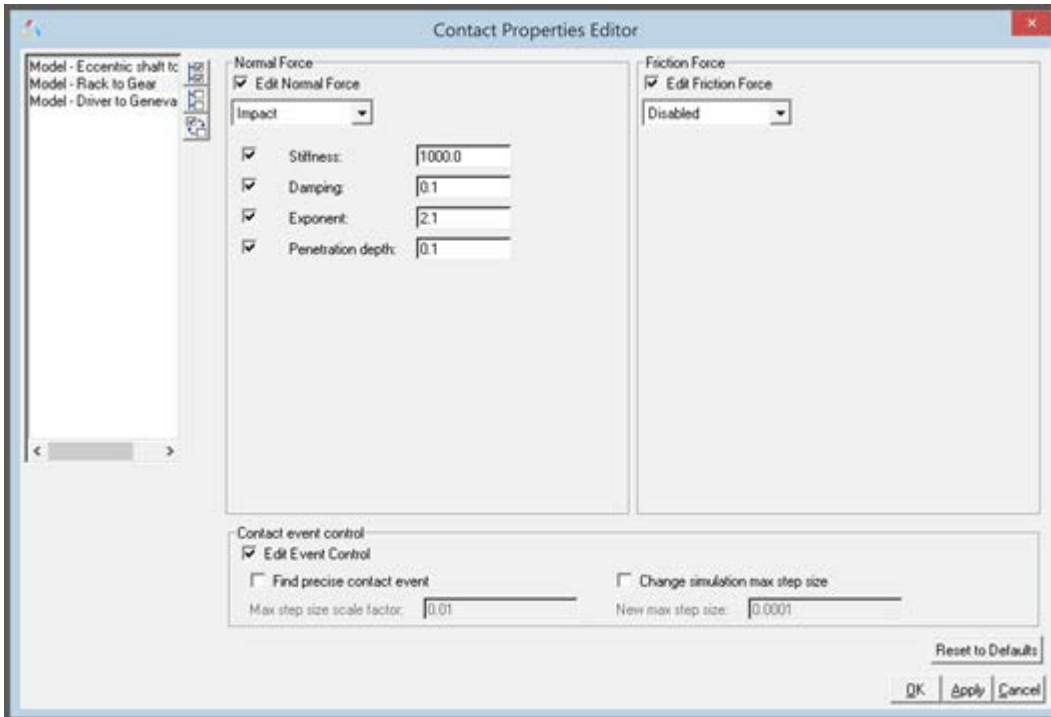
A new Volume Model for normal force calculations is now supported. User defined contact models for both normal and friction forces are also supported.



In addition, user-defined normal and friction force definitions are supported.

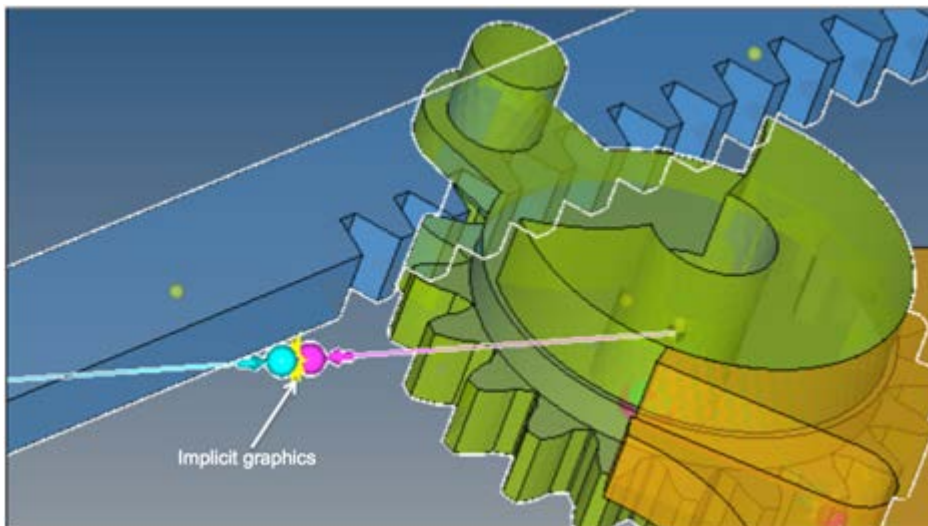
### ▼ Updated Contact Properties Editor Macro

This macro tool has been updated to take into account the new contact properties available in the solver and makes setting properties across multiple contacts quick and intuitive.



### ▼ Contacts are Visualized During Modeling

A new “implicit graphics” contact icon is now visible. You can visually see the geometries that have contact connections.



Associated graphics are highlighted when contact entity is selected.

▼ **Precise Contact Detection is Supported**

Advanced options are available to detect the precise event when contact occurs. An option is also available to change the maximum step size subsequent to contact detection. This option introduces a Sensor entity with a zero crossing attribute into the MotionSolve XML file. This Sensor will control integration step size so that the onset of contact is accurately detected.

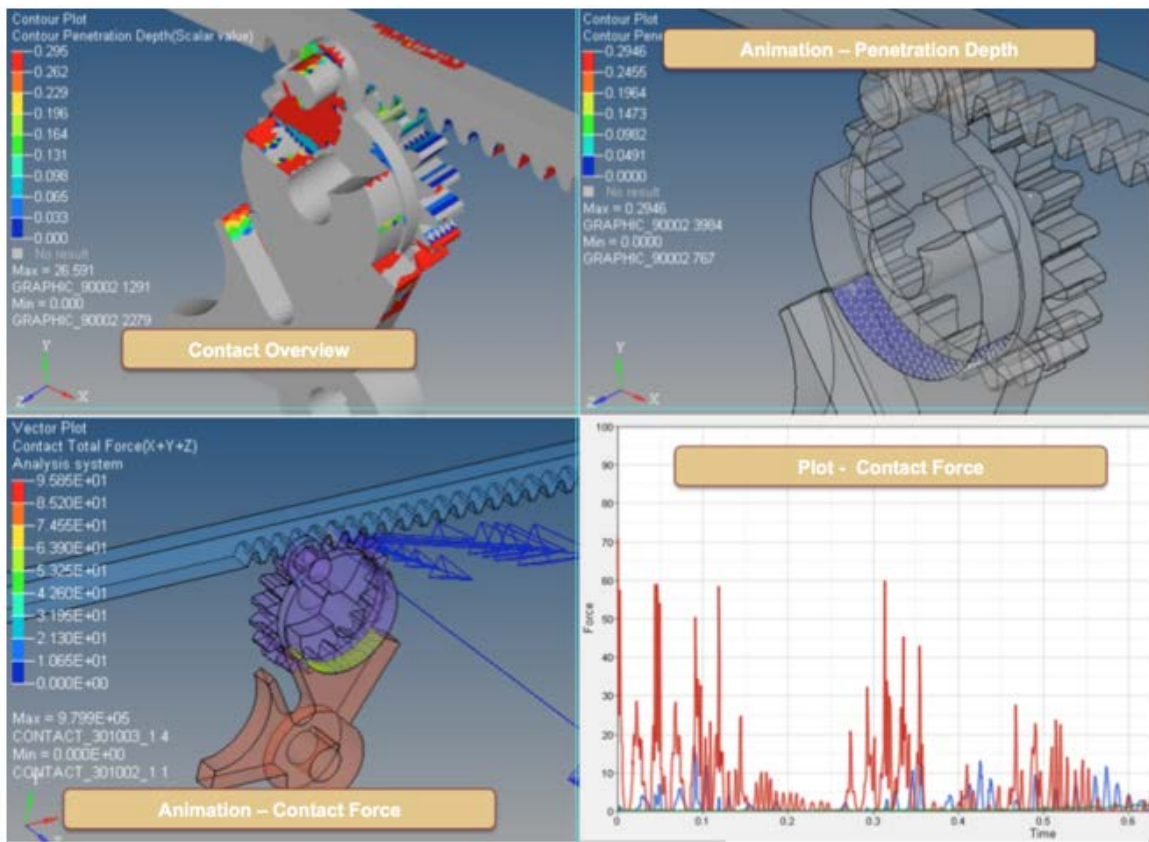
Refer to Rigid to Rigid Contact section in MotionView User Guide to learn more about modeling contacts.

▼ **Contact Report for Easy Post Processing**

MotionView now has capability to generate a Contact Report automatically to simplify the post processing of contacts.

For every model that involves contacts, the auto-generated report contains the necessary plots and animations that enable you to easily understand contact behavior in your model.

- A 'Contact Overview' is first generated so you can see where the contact has occurred in the model.
- Two animations of the model, one showing penetration depth and the second showing normal contact force are available for review
- Finally, the report also contains 'Overall Contact Force vs. Time' plot for every contact in the model.



## Nonlinear Finite Elements

---

MotionSolve 14.0 introduces the “NLFE Body” which allows you to model non-linearly flexible components in your multi-body system. This is an experimental feature in the current version. The NLFE Body is based on the Absolute Nodal Coordinate Formulation (ANCF). Nonlinearity can occur for two main reasons: (A) Geometric nonlinearity and (B) Material non-linearity. The NLFE body supports both.

MotionView has new capabilities for modeling NLFE systems. A generic NLFE Body can be created using either the CABLE or BEAM element.

### ▼ CABLE

The CABLE element does not resist shear or torsion forces. This implies that the cross section of this element does not change as the entity deforms.

This element can be used to model cables, wires, ropes, etc. Only the Linear Elastic material model is supported.

### ▼ BEAM

The BEAM element resists all forms of deformation, which implies that the cross section can also deform with load. This can be visualized in HyperView while using this element.

Multiple cross sections are supported for the BEAM element:

- Circular (solid and hollow)
- Rectangular or Box (solid and hollow)
- Channel
- Cross
- Hat
- H, I, L, T and Z sections

This element can be used to model beams, springs, belts, rubber components etc. You may choose from either a Linear-Elastic or a Hyper-Elastic material.

Three Hyper Elastic materials are supported – Neo-Hookean Compressible, Mooney-Rivlin and Yeoh models. MotionView provides a set of recommended values for the model parameters. You may however, create new materials and enter your own parameters.

### ▼ 3D Visualization

MotionView provides 3D visualization of the NLFE body based on the profile and cross section properties.

### ▼ Defining Preloaded/Relaxed Configurations

An NLFE body can now have a pre-loaded position in addition to the model position. The user can import these in a CSV format. Upon import, the model position will be shown in wireframe mode and the loaded position in solid.

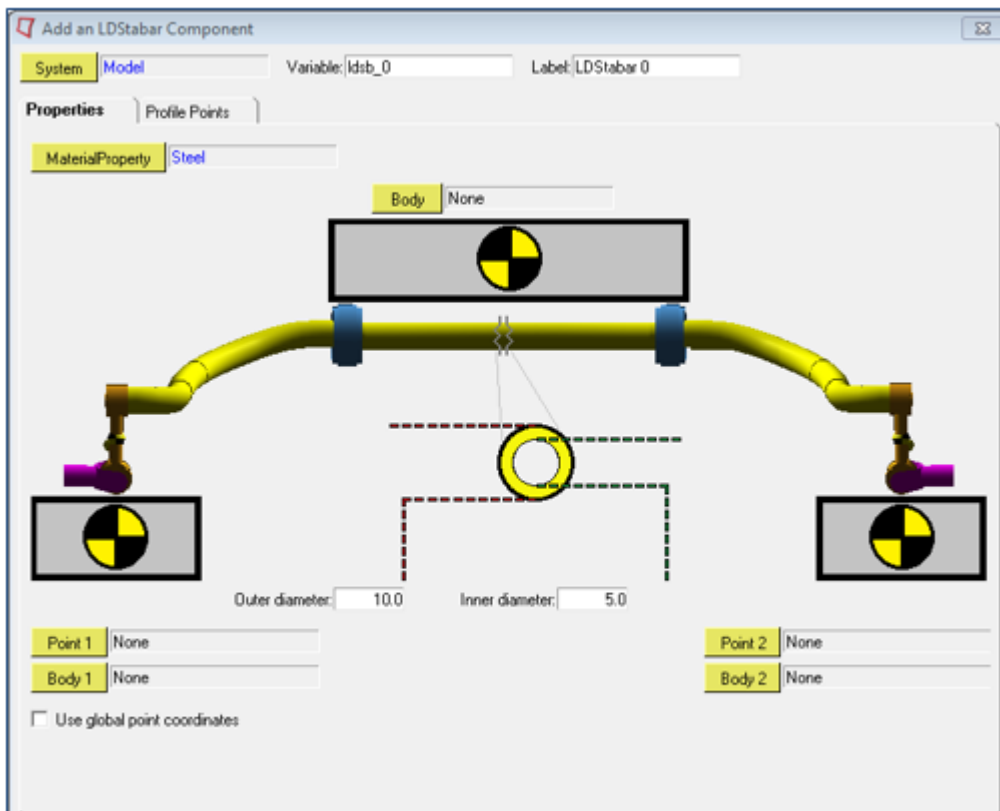
### ▼ Saving Pre-load Configurations

With a combination Save command in MotionSolve and a Python script, it is now possible to save the configuration of the NLFE at the end of simulation into a CSV file. This csv file can be read into MV as preloaded configuration. An option is also available to visualize the unloaded and pre-loaded profile together.

### ▼ NLFE Subsystems

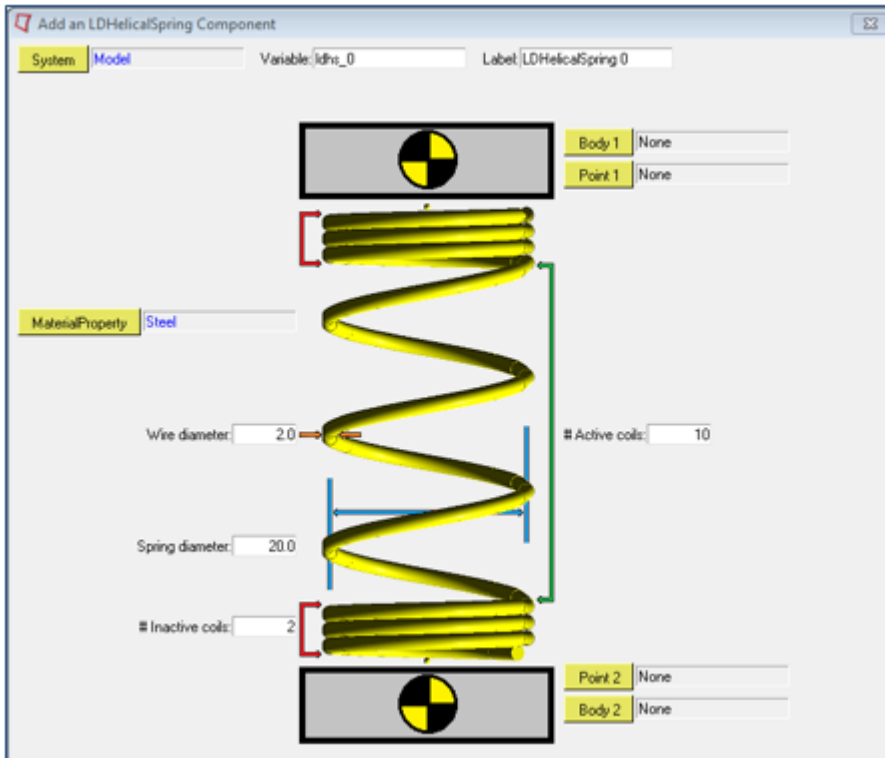
In addition to the generic Cable and Beam modeling elements, MotionView also supports a small set of NLFE subsystems that are specific to the Auto industry. These subsystems may be accessed from the vertical toolbar to the left of the graphics window. The following subsystems are supported.

#### Stabilizer bar





## Coil spring



Refer to NLFE Subsystems to learn more about using these subsystems in your model.

MotionSolve supports a much larger set of NLFE modeling elements. You can use these in a MotionView model by using a Templex template. Native MotionView support is not available at this time.

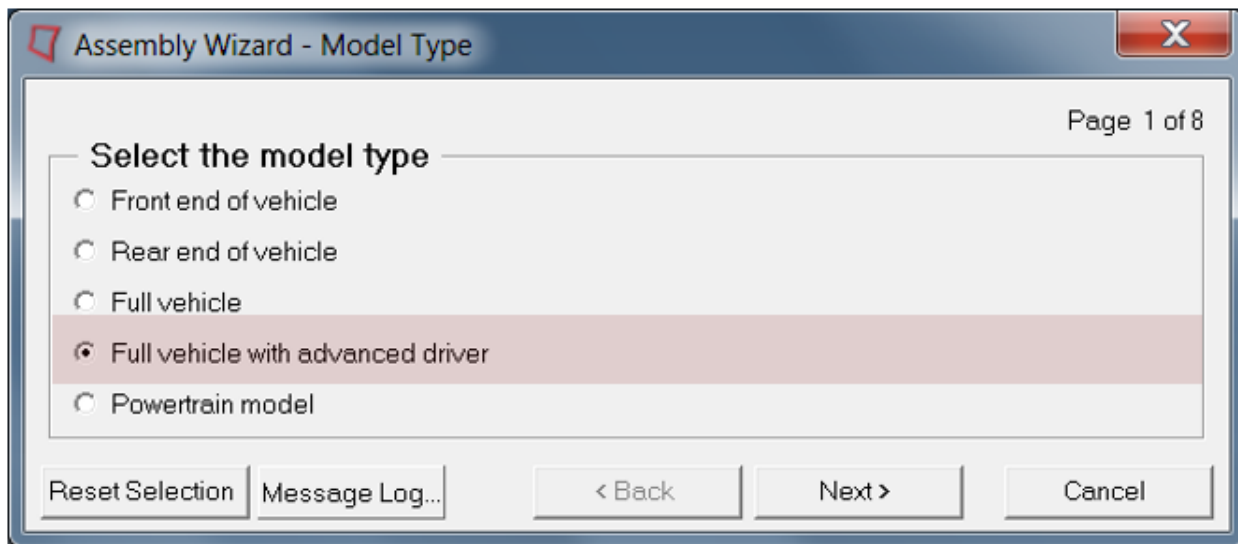
## Advanced Driver and Road Vehicle Simulation

---

### Driver

The Altair Driver is a set of MotionView models and libraries that drives virtual vehicles with closed loop and open loop inputs to perform a variety of events. Examples include driving on a rough, curving road while controlling speed for durability or ride comfort analysis, or driving a skid pad at increasing speed to predict understeer.

This is achieved by controlling the five vehicle inputs Steering, Throttle, Gear, Brake and Clutch so that the vehicle is able to accomplish the required maneuvers. The driver is completely integrated into MDLLIB. You can build a full vehicle for use with the Advanced Driver using MotionView's Assembly Wizard. You do this by selecting the **Full vehicle with advanced driver** option, as shown in the dialog below.



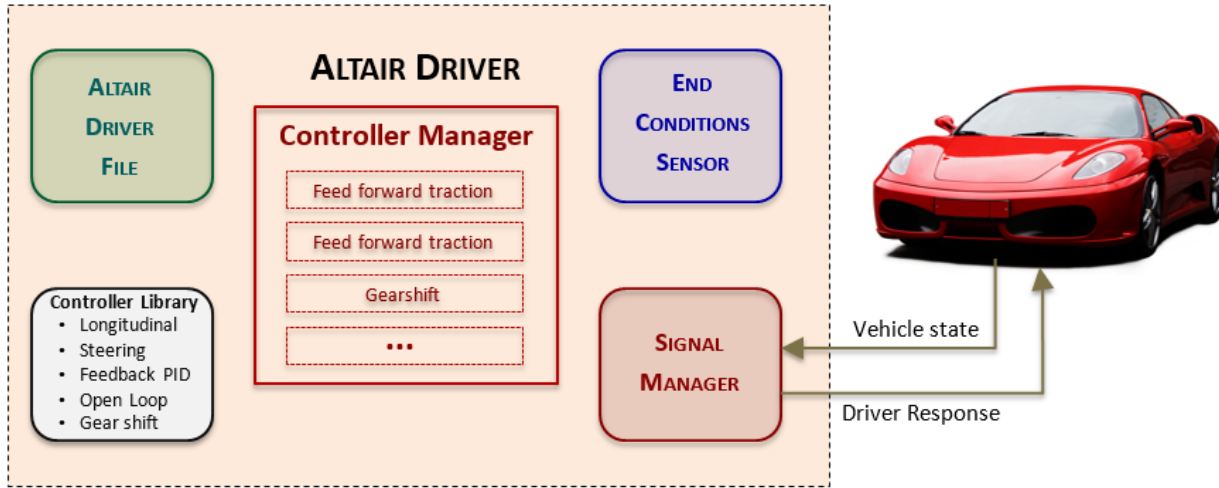
Once a vehicle + driver is built, the next step is to define what the combined system should do. You specify driver events to do this. In this release, the input to the driver model is an "Advanced Driver File". This is an ASCII TeimOrbit format file that scripts a simulation as a sequence of maneuvers. The Altair Driver and the Advanced Driver File support both open and closed loop events.

With Altair Driver File you can define any number of full vehicle events for the driver to execute.

- A complex simulation may be decomposed into many simpler "atomic maneuvers"
- For each maneuver
  - You can select the controllers for vehicle inputs
  - You can define termination conditions for each maneuver.
- You can define open loop, closed-loop or user-defined controllers to control:
  - Longitudinal speed or acceleration.
  - Vehicle path or lateral acceleration.
- Switching controllers during a simulation

- Defining path and speed profiles parametrically, in a table, or by referencing a data file.

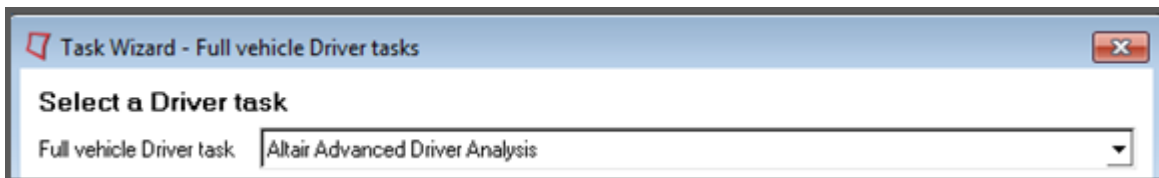
The overall relationship between the Altair Driver and the vehicle is shown in the figure below.



Full vehicle models built for use with the Advanced Driver include the following new systems:

- A four-wheel disc brake system for slowing the vehicle
- Either a linear torque map power train or an IC engine with clutch and manual gearbox for accelerating the vehicle.

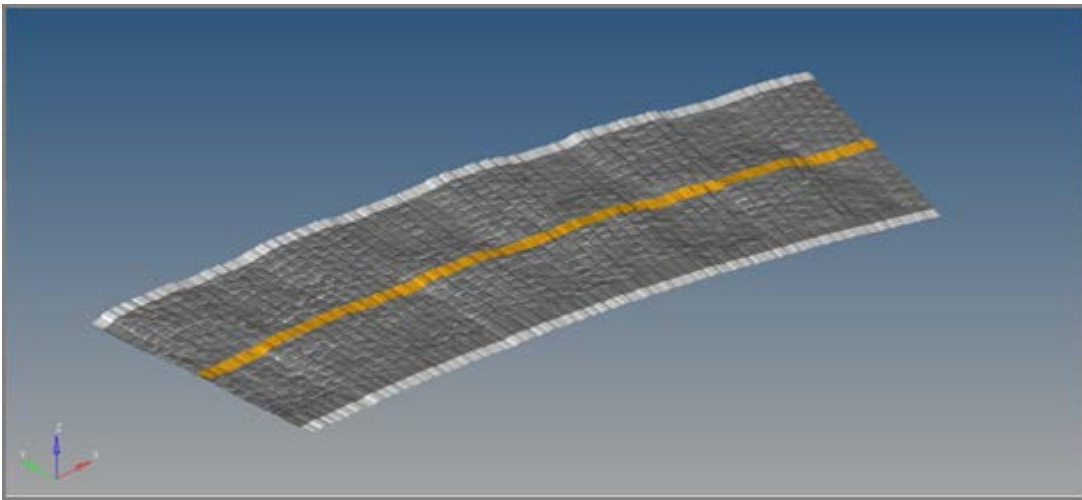
Once the full vehicle is assembled use the task wizard from the Analysis menu to add an Advanced Driver Analysis as shown below.



## Road

Use the Road Graphics Tools to build a MotionView graphic system to visualize the road or to extract a driver path from Curved Regular Grid (.crg), Regular Grid (.rgr) and 3D Spline Road files. Using the Road Graphics Tool you can resample the road to reduce or increase the number points in the driver path or the number of surfaces in the resulting graphics. You can also add lane divider lines and shoulders to the road, and set the colors of the road, the lane divider lines, and shoulders.

The road graphic system the tool creates is automatically imported into your model, but also saved to disk so you import it in any MotionView model.



## Other MotionView Enhancements

---

This section documents other important enhancements that have been made to MotionView.

### Geometric Stiffening for CMS Flexible Bodies

You can model geometric stiffening for CMS flexible bodies in MotionSolve. Geometric or stress stiffening is an effect most commonly seen in beams under tension where the frequency of the fundamental modes increases as the tension is increased. Such an effect can be observed in rotating beam structures like helicopter or wind turbine rotor-blades.

Include the stress stiffness data in your CMS flexible body by including the following in your FEM file:

`CMSMETH, 1, CBG, , 15`

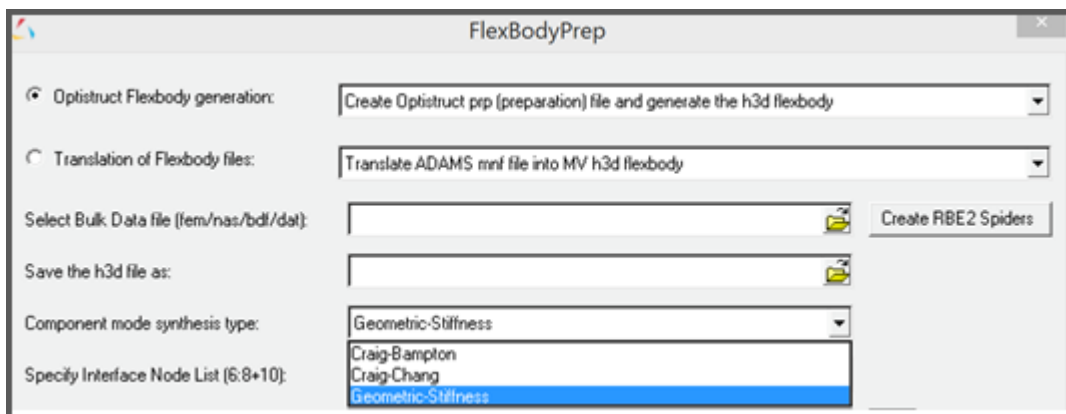
CBG is the keyword that instructs OptiStruct to create the CMS flexible body with additional geometric stiffness data.

You can then import this flexible body in your MotionView model as you would ordinarily. Note: Only the Craig-Bampton method is currently supported for calculating the geometric stiffness data.

This new MotionSolve capability is completely supported in MotionView also.

#### ▼ Flex-body Generation with Geometric Stiffness can be Done using the Flexprep Tool

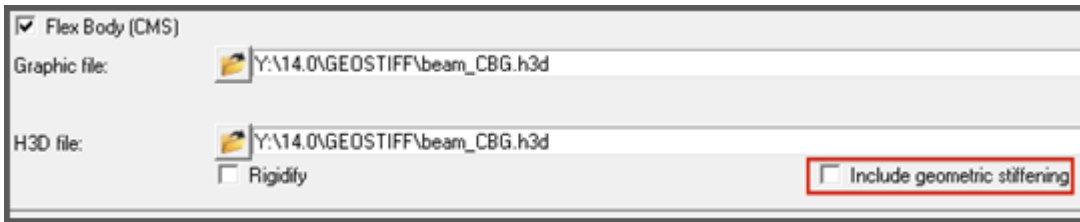
A GUI option is available to specify geometric stiffness.



Selecting the highlighted option would include the relevant CMSMETH statement of the form `CMSMETH, 1, CBG, , 15` into the FEM File.

#### ▼ Geometric Stiffness Option can be Specified in the Body (Flex body) Panel

“Include geometric stiffening” checkbox is now available in the Flex body panel. This option will only be activated when the flex h3d contains the geometric stiff (GEOSTIFF) data that is saved while generating the flexible body during the CMS process.

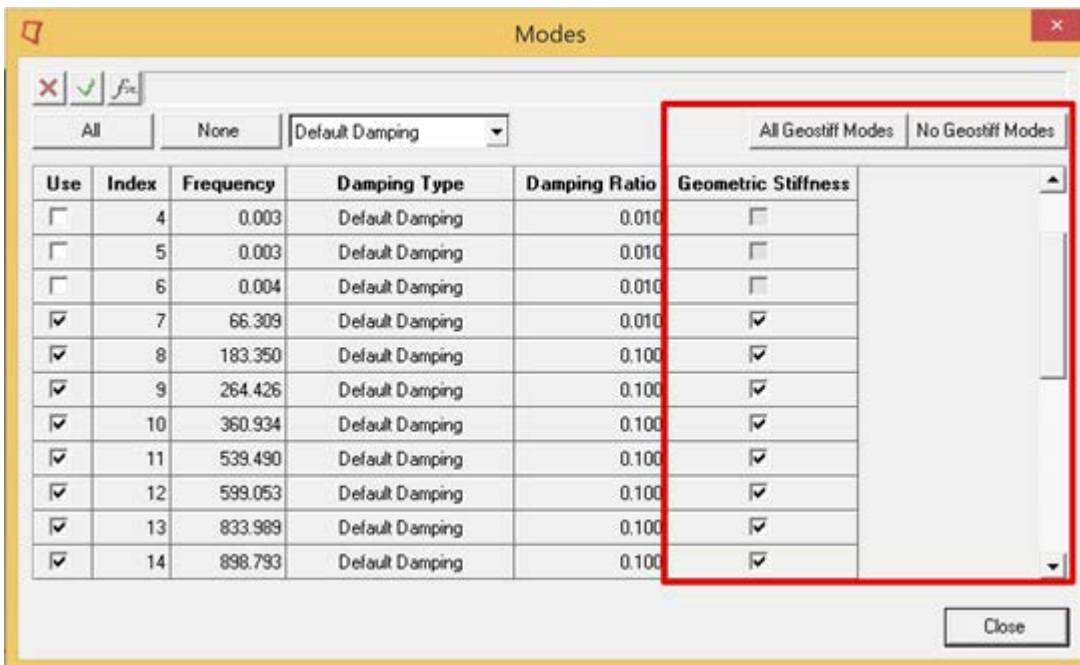


Checking on this option will enable export of the geometric stiffness data into MotionSolve XML file.

▼ **Geometric Stiffness Data can be Selectively Chosen by Mode for Export to MotionSolve**

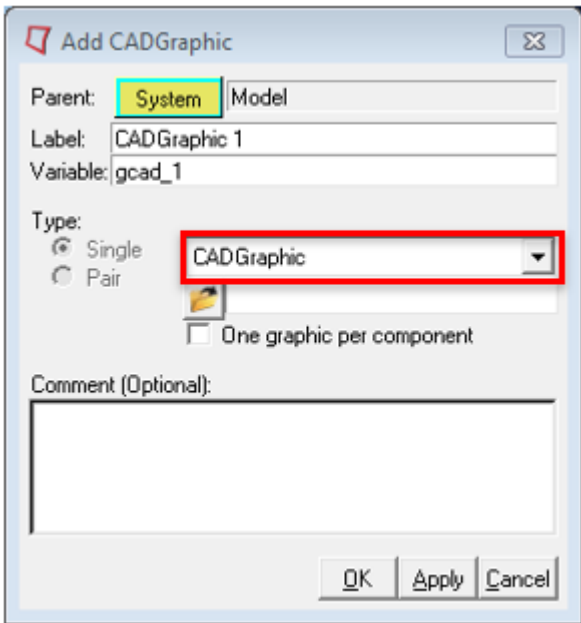
The “Modes” dialog in the flex-body panel now contains an additional column to control export of geometric stiffness data for each mode.

Turning off geometric stiffness data for certain modes can significantly improve the solution time. However, you should be sure that the modes that are turned off have a minor influence on geometric stiffening effects.



▼ **Direct Geometry Import**

In this release, MotionView introduces new functionalities based on the Parasolid geometry engine. A new ability to import geometry of type “CADGraphic” is now available. All common formats can be imported.

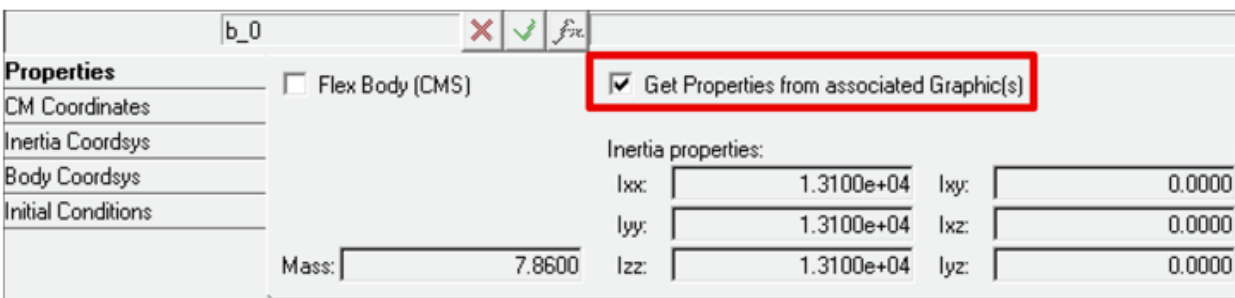


You will experience several benefits because of this capability.

- All common CAD file formats are supported and can be easily imported as graphic statements
- CAD Geometry imported in this way has mass and inertia attributes
- The Body to which CADGraphics are attached, can automatically inherit the mass and inertia properties based on the geometry properties
- User can pick specific components in geometry and export them to solver. A separate parasolid file will be generated with the component being selected. No mesh data written to XML file.
- User has the flexibility to change mesh refinement level of geometry by adjusting the fidelity factor before exporting to MotionSolve.

The underlying Parasolid geometry engine is also used to automatically compute mass and inertia properties for simple analytical shapes such as spheres, cylinders, boxes and frusta.

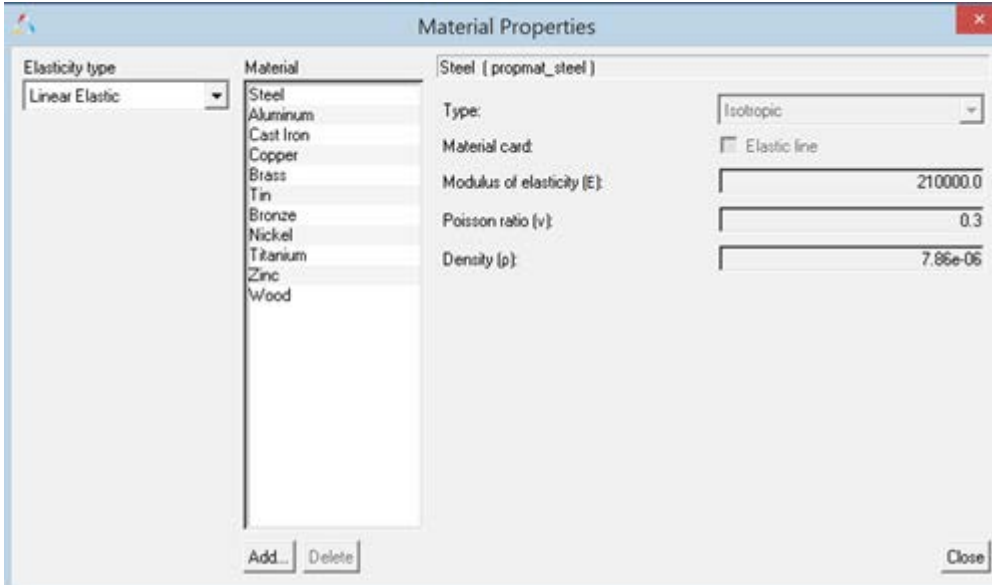
An option is provided in body panel using which user may choose to inherit mass and inertia properties from associated graphics.





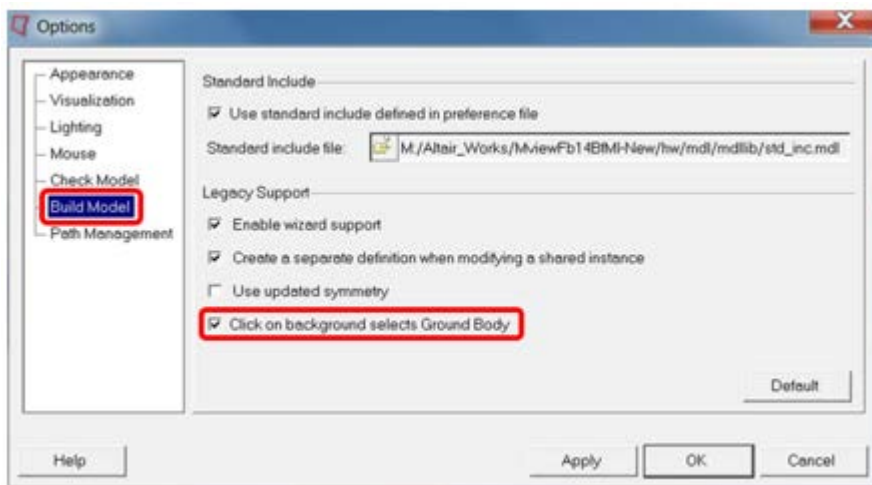
## ▼ Materials

A default set of materials with material properties is now available. These materials can be used either in an NLFE Body or CADGraphics. The Materials can be accessed using the menu **Model > Materials**.



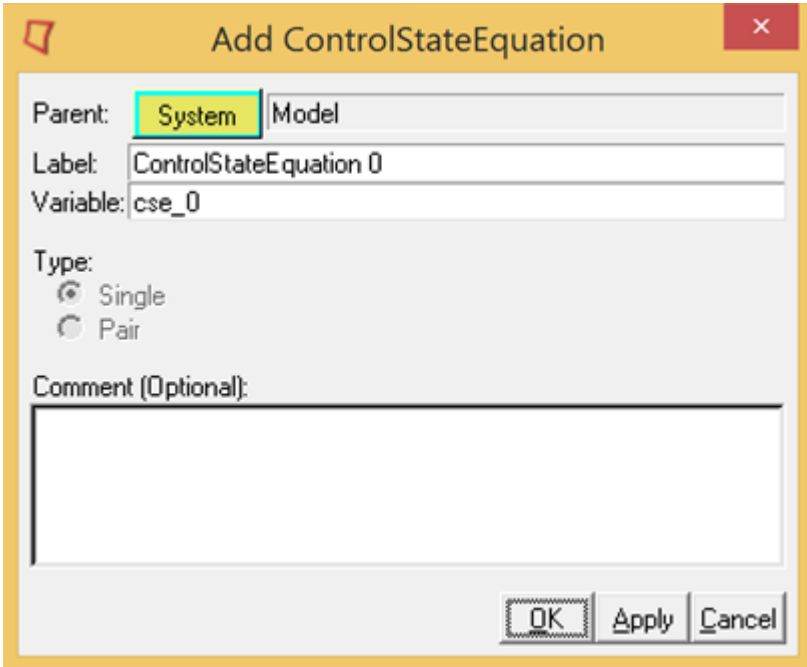
## ▼ Usability Improvements

- Multiple picking from graphics area to cut/delete entities. Hold Shift key to select many entities.
- Merging multiple bodies from the graphics area. User can select two or more bodies (using the Shift key), use the right click to merge the bodies to a new body or to Ground Body
- A new option is now available in MotionView to select a Ground Body. When Body collector is active (to pick a body as a reference into an entity panel), clicking on empty space in the Graphics area picks Ground Body with following option ON.

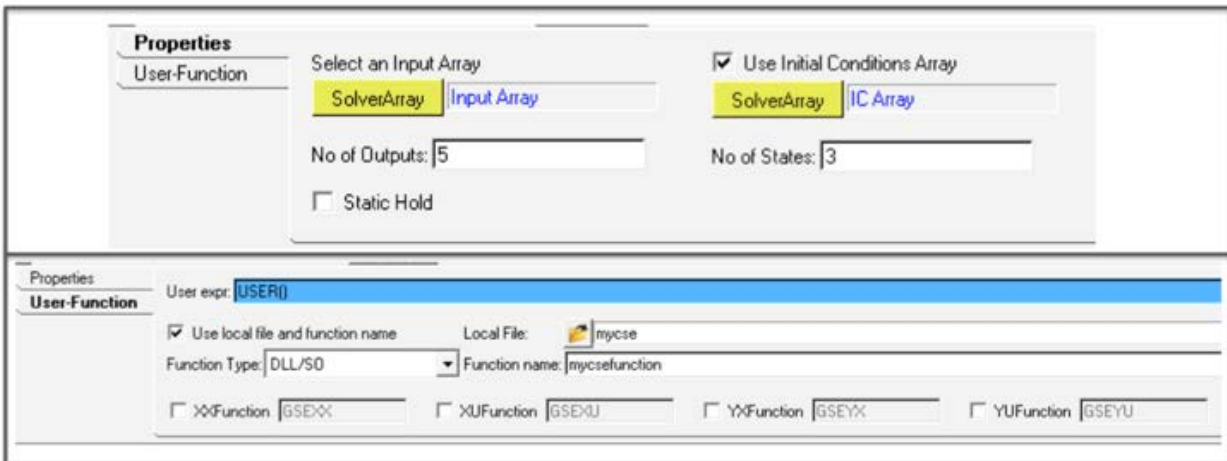


▼ **General State Element (GSE)**

The General State Element, also known as [CONTROL\\_STATEEQN](#) in MotionSolve, is now fully supported. This includes the user interface and complete MDL support. You no longer need a Templex template to use this capability. A Control\_StateEquation icon is available in the “Control Entity” toolbar to add the entity.



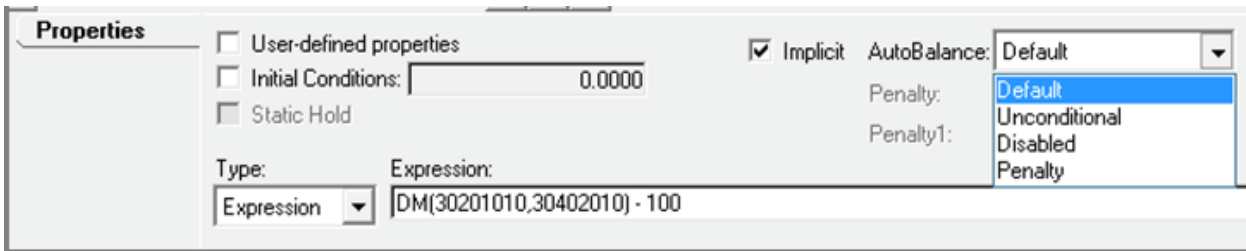
The two tabs in the panel can be used to define all the necessary inputs to the control state equation.



▼ **Implicit Variables**

This enhancement is applicable only in the MotionSolve mode. Traditionally, the [REFERENCE\\_VARIABLE](#) in MotionSolve has explicitly defined a new algebraic variable through an expression or a user-subroutine. Solver Variables are now extended so that they are *implicitly* defined. This means that [VARIABLES](#) can now be used to define constraints, and the [VARIABLE](#),

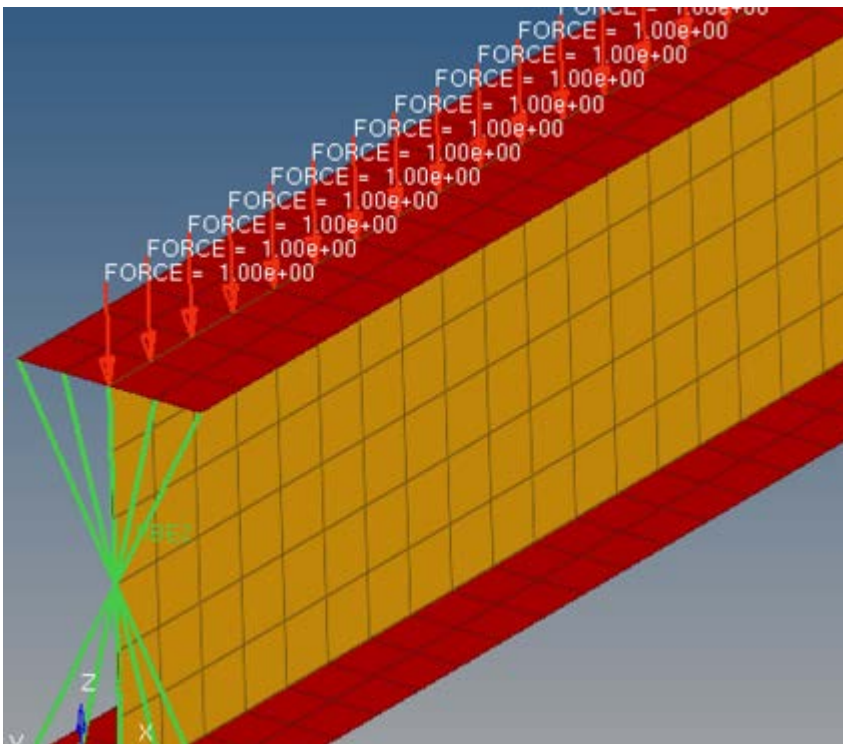
instead of assuming the value of the expression, now becomes the Lagrange Multiplier required to enforce the constraint.



The SolverVariable panel along with the implicit option has additional controls to tell MotionSolve how the Lagrange Multiplier has to be handled.

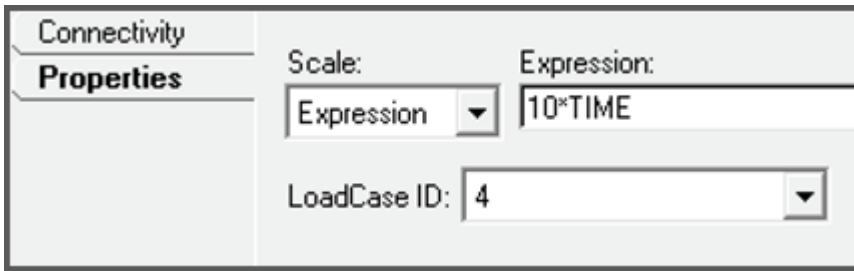
### ▼ Modal Force

MotionView now supports adding a modal force on a flexible body (Force\_FlexModal in MotionSolve). A modal force can be a distributed force such as fluid pressure or aerodynamic load that can vary with regards to time.



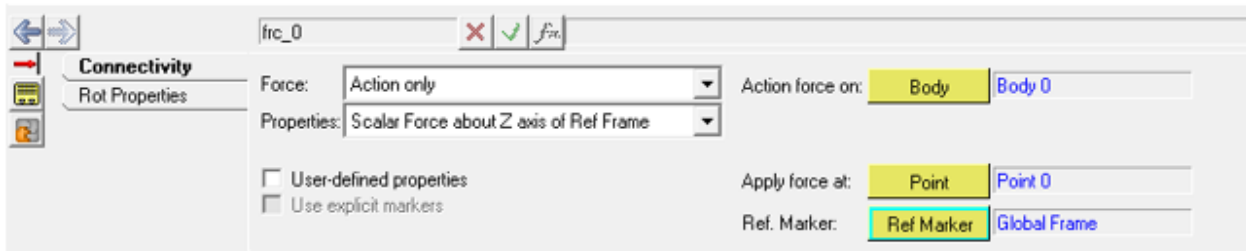
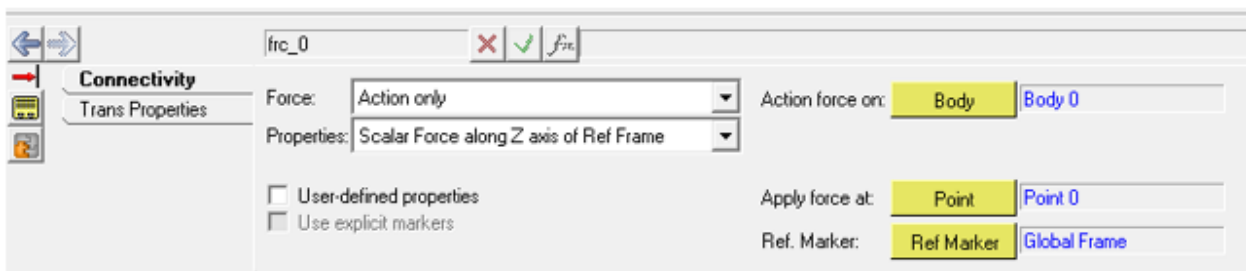
The distributed force should be applied on the FE model that would be used to generate the CMS flexible body. The force would be decomposed into modal form.

MotionView presents the modal force as load case ID as available in the flexible body H3D. The modal force can be scaled during simulation if needed.



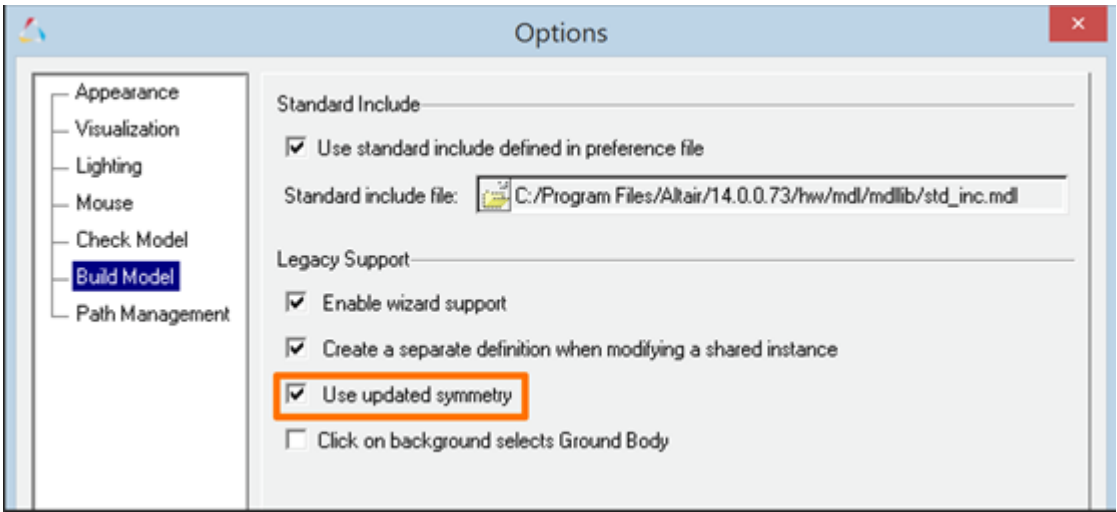
### ▼ Scalar Action only Force and Torque

MotionView now supports a Scalar action only force or torque. This means that Action Only forces or torques can be applied in a direction specified by the z-axis of a Marker.

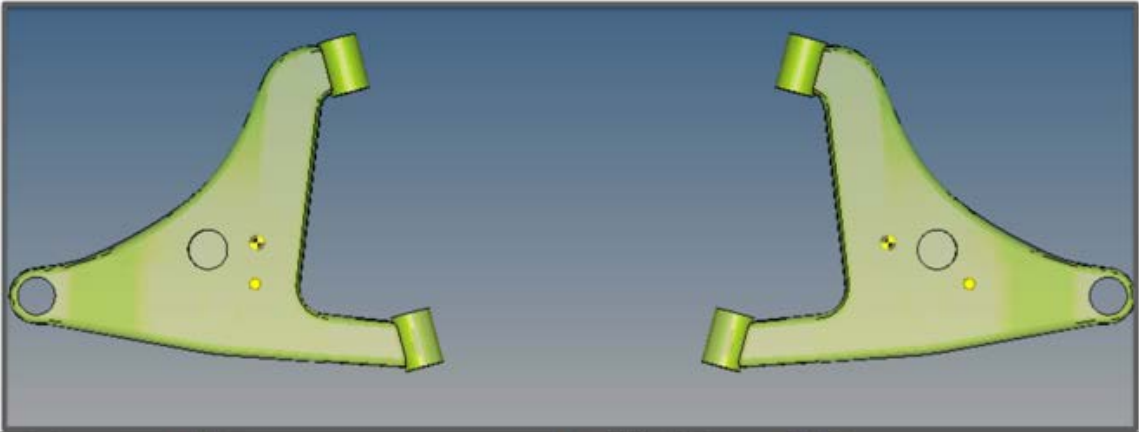


### ▼ Mass and Inertia Value Correction for Symmetric Pair Bodies

A new symmetry option is now available in MotionView to define symmetric body pairs. Cross products of inertias are properly negated (about the Global ZX plane) when a body is reflected. This new option can be turned on using the **Tools > Options > Build Model**. The symmetric pair's properties correspond to the bodies being represented as mirror reflections of each other.



The new calculation considers the offset between the CM marker and IM marker if any. Non-symmetry of the CM location is not considered. Also, the symmetry is not applied to initial conditions.



▼ **"Body Deformable" Changed to "Flex Body (CMS)"**

The option "Body Deformable" that is used to define a flexible body has been renamed to "Flex Body (CMS)" to distinguish it from the NLFE body. (The NLFE Body is added through the "Add Body" dialog)

The "Flex Body CMS" panel has additional options:

- Rigidify – to set the MotionSolve attribute `rigidified = "true"` in `<Body_Flexible`
- Include Geometric stiffening – To include geometric stiffening during solution of flexible body whose H3d has geometric stiffness data

#### ▼ Import CAD or FE

- Now opens with mesh options expanded
- Graphic H3d files are automatically populated while selecting an input file
- Multiple confirmation messages related to successful import are eliminated.

#### ▼ API for Calculating Aggregate Mass, CG and Inertia Properties of Systems

MotionView now provides a new TCL API, `GetAggregateCGAndInertia`, which allows you to calculate the aggregate mass, center of gravity, moment of inertia and product of inertias for multiple systems or multiple bodies in your model. Documentation for this function is included with this release.

#### ▼ Enhanced FatiguePrep Tool

MotionView now supports translation of `.plt` to RPC format.

#### ▼ Recent Files

The File menu now lists the four most recent MDL files used.

## Resolved Issues

---

### ▼ Graphic Selection/Picking

Linux/Mac Entity Selection: When a body is selected in a browser on Linux, all the mesh lines of the graphics of that body were highlighted. This is now resolved

Pick center choice with Ctrl+Middle mouse button was not being honored correctly. The graphics used to rotate about a different center than selected. This issue is fixed.

### ▼ ADM Import

ADM files with expressions having large blank spaces from the beginning of the line leads to expressions being truncated. This issue is fixed in this release.

Import of ADM that did not contain MATRIX information for flexible body resulted in crash of application. This issue has been fixed.

### ▼ Flexprep

Issues related to incorrect modes and interface nodes while converting .h3d files to .mnf files and .mnf to .h3d files are resolved within this release.

Erroneous GENSTIFF and SELNOD matrix was being written out while converting .h3d to .mtx when using kgf as the solver unit. This issue is fixed within this release.

An application error while using Create RBE2 spiders option in flex prep is fixed. The error appeared while closing HyperMesh button with the Close (x) button.

### ▼ Graphics Entity

A custom ID specified for a graphic entity was lost on Check Model. This issue is fixed.

MotionView would not report error when components specified in \*Graphic statement of File type were missing. This issue has been fixed.

An application error while changing the mesh scheme for File Graphics under a Graphic system is fixed.

### ▼ MDL

An issue with false referencing of entity due to use of same variable names in a system as well as at Model level is now fixed. When an entity (e.g. p\_0) that existed at Model level was referenced in a system that contained entity of same type and same variable name (p\_0) was being saved with a local address (p\_0 instead of MODEL.p\_0).

White spaces in the USER expression in Sensor entity evaluate subroutine resulted in the expression being truncated. This issue is resolved.

Setting of MAF export option was lost on model save. This issue has been resolved.



#### ▼ Miscellaneous

Statics simulation settings - Maximum # of iterations is now defaulted to 75

Write results data per time option is now set to LAST by default instead of ALL. This resolves issues in post-processing due to multiple results at certain time steps in a sequential simulation.

Outputs of type Entity sets on bodies used to result in many outputs with the same label. This issue has been resolved.

An application error in Curve panel while changing type from File to Values is fixed. The error was encountered when a file path was unresolved.

An application error encountered while using the Back/Forward button in any panel is resolved. The error used to occur while using the button after deleting an entity that was in the selection history.

## Auto Solutions Resolved Issues

#### ▼ Auto Spring Using Force as Install Method

Choosing force as the install method for AutoSpring resulted in an incorrect spring force at design position during the simulation. This was due to an incorrect spring displacement offset being calculated. This issue has been resolved.

#### ▼ Auto Bumpstop Pair Impact Formulation Incorrect

Choosing "Impact Length" had no effect and the distance entered was treated as the clearance in the stop. Choosing "Impact Length" now works properly.

#### ▼ Auto CDTire Pair Tire Graphics not Symmetric

The tire graphics are now centered about the plane passing through the wheel center and perpendicular to the wheel spin axis. Previously, the changing the tire aspect ratio shifted the tire graphics along the wheel spin axis.

#### ▼ Ideal Steer Angle Incorrect

The ideal steer angle in the Static Design Factors report for half vehicle analyses is now output correctly. Previously, a spurious value was output.

#### ▼ CDTire v4.1: CDT20 - Integrator Failures Occur with the Default CDT20 Property File

Testing using the CDTire model 20 with road surface model 2000 (RSM2000) showed simulation failures. Because CDTire model 20 uses a point-follower contact and road surface model 2000 is three (3) dimensional, they should not be used together. To prevent inexperienced users from encountering such failures, the default property file for CDTire in MotionView has been changed to a CDT30 property file, which is appropriate for all road surface models. However, no changes to CDTire or MotionSolve were made to resolve this issue. In MotionView the following system definitions for CDTire now reference a CDT30 property file by default:

- `.../hw/mdl/autoentities/definitions_mdl/sysDefCDTire.mdl`
- `.../hw/mdl/autoentities/definitions_mdl/sysDefCDTirePair.mdl`

#### ▼ CDTire v4.1: CDT20, CDT30, and CDT40 Messages Truncated

Multi-line information and error messages for CDTire models 20, 30, and 40 were truncated on output when used with legacy road models and CDTire v4.0. Fraunhofer ITWM corrected the problem and improved the messages for models 20, 30, and 40.

For example, when the tire property file could not be found CDTire v4.0 output the following message:

```
USRMES:USER [1]
INFO: CDTire:: Using CDTire version 4.0.0

USRMES:USER [1]
INFO: CD Tire - Time Discrete Interface.
cdt CDTire R 4.0.0 -> TIRE 1 INITIALIZING
cdt TIRE MODEL 30 SELECTED
cdt CTRL FILE SELECTED:
cdt ERROR:
```

CDTire v4.1 now reports:

```
USRMES:USER [1]
INFO: CDTire:: Using CDTire version 4.1

ERRMES:USER [1]
CDTire> CDTIRE_INIT_F: PRIME PAR FILE NOT FOUND AT:

ERRMES:USER [1]
CDTire> ./prop_files/tie_30.21
```

#### ▼ CDTire v4.1: REQ838 - Adams Models Fail in MotionSolve

The MotionSolve implementation of REQ838 incorrectly ordered the user subroutine input parameters causing Adams datasets for CDTire to fail when submitted to MotionSolve. The implementation of REQ838 in MotionSolve now orders parameters as documented by Fraunhofer ITWM for Adams. The MotionView definitions for CDTire now create output requests using REQ839 to avoid conflict with ADAMS.

▼ **CDTire v4.1: REQ838 – Models Without Output Requests Calling REQ838 Fail**

Customers reported that models without output requests calling REQ838 sometimes failed. However, Altair was unable to reproduce the issue. The problem occurred in the MotionSolve code that interfaces CDTire. The code was updated to correct the problem.

▼ **CDTire v4.1: RSM2000 - Does not work with CDT20, CDT30 and CDT40**

This release includes new CDTire new binaries from Fraunhofer ITWM that fix the issue.

▼ **CDTire v4.1: Models 20, 30, and 40 Fail with Property File Read Error**

When the initial character of the first two lines (as well as any other comment line) of a CDT20, 30 or 40 tire property file began with a numeral, CDTire reported errors reading the file. Fraunhofer revised CDTire to ignore any comment lines in property files for models 20, 30, 40 and in control files to correct the problem.

▼ **MF-SWIFT: Initial Velocity Reporting Issue in MF-SWIFT Tyre Model**

Previously MotionSolve incorrectly reported the initial velocities for tires using the MF-SWIFT model. MotionSolve now correctly reports the initial velocities for tires using the MF-SWIFT model.

▼ **Using FIESUB 810 Causes Failure in Quasi-Static Model**

- With the previous version, using FIESUB 810 caused a solver failure for a Quasi-Static model shared with Altair. This was determined to be due to a programming error in the code, where the row and column values for the FIELD Jacobian were interchanged. This has been fixed with this release.
- A side-effect of this change is that linear analysis of models containing FIESUB 810 may yield slightly different results. This is to be expected, since the bug-fix involves a correction to the Jacobian matrix of the system. The following changes may be seen for these models:
  - The set of independent coordinates may be slightly different
  - The eigenvalues and eigenvectors may be slightly altered
  - The ABCD matrices may be slightly different

▼ **Fixes to ACTCLC Routine**

The contact patch velocities, slip angle and inclination angle were incorrectly computed in the previous version of the ACTCLC routine for the user tire. These have been corrected to be consistent with the documentation that has been shared previously with Honda.

▼ **MDLLIB: Tire Envelope Analysis on Half Vehicle Models now Support Property File Based Shock Absorber**

This was non-functional in previous releases. This issue has been fixed.

▼ **MDLLIB: Support for Models using Legacy Steering Boost System**

This was non-functional in previous releases. This issue has been fixed.

▼ **MDLLIB: Full Vehicle Analyses - Incorrect Initial Rotation Velocities Using SETIC/SETWIC**

Previously when using the SETIC/SETWIC option for initial velocities, the initial rotational velocities of half-shafts, hubs and other bodies in the driveline were inconsistent with the initial rotational velocities of the driven wheels. The initial rotational velocities are now consistent.

▼ **MDLLIB: Full Vehicle KnC Analysis**

KnC analysis results for the same suspension match between Full Vehicle and Half Vehicle Analyses. Previously, there were small differences.

▼ **MDLLIB: Steering Column System**

The column support joint is moved to the lower column shaft center of mass. This makes the support joint easier to identify when, for example, attaching the steering column to a flexible body.

## Known Issues

---

### ▼ NLFE

- Providing a specific LPRF is not supported for NLFE Body
- Creating Helical Spring Subsystem with preload is not supported yet

### ▼ Material Properties

- MotionView does not scale material property values when units are changed
- In the Deformable Surface panel and the Deformable Curve panel, the expanded table does not allow you to pick an entity into a collector.

### ▼ Contact

- Switching between radio selections in Properties tab may be slower for large models
- Importing ADM – CONTACT statements are not converted into Contact entity in MotionView in some cases. The statement may end up in the Unsupported statements template.

### ▼ Modal Force

- Including Modal force during flexible body generation is not yet supported through Flex prep
- Modal Force does not have a graphic representation in MotionView

### ▼ Geometric Stiffening

- Unchecking and rechecking "Include Geometric stiffening does not retain the Geostiff modes checkbox

## Auto Solutions Known Issues

### Road Tools:

- The road tools dialog should offer a means to abort road graphics generation because of the large time require for some road file.

### Driver:

- Feedforward Controller doesn't follow acceleration profiles well
- When using the driver, the tire longitudinal slip shows oscillations at time=0
- Zero vehicle speed causes steering controller to fail.
- Driver event UI is experimental: To enable the event editor button on the driver panel in MotionView set the environment variable HW\_MV\_EXPERIMENTAL to ADVANCED\_DRIVER.

### MDLLIB

- Full vehicle events with a powertrain often fail to solve because the suspension spring preloads are too small causing the excessive suspension jounce. We recommend increasing the spring preloads to compensate for the mass of the powertrain, and as always you should update models.
- Steering systems contain dummy bodies to support analyses, but these have no meaning to users.
- Some combinations of wizard selections do not build because referenced system definitions use the same varname.
- To reduce the length of the Constant Radius event the length of the initial straight section should be based on the initial vehicle velocity

# HyperCrash 14.0 Release Notes

## Highlights

---

This release includes several major usability improvements for HyperCrash: multi-edit enables you to modify multiple entities of the same subtype quickly; orthotropic material orientation can be reviewed for shell and solids (RADIOSS only). Model checker is customizable and can be modified to follow company best practices/standard.

## General Enhancements

---

### ▼ Multi-Edit

This enhancement allows to edit and modify multiple entities of the same type very quickly. For example: change the element formulation for a number of properties, update a flag in a number of contacts.

This feature is available on the following pages:

- Rigid bodies
- Contact Interface
- Part (LS-DYNA only)
- Property
- Material
- Element Panel

### ▼ Material Orientation

The material direction of shell elements can be reviewed from:

- The part panel (for \*PART\_COMPOSITE)
- The property panel (for /PROP/TYPE10; /PROP/TYPE11; \*SECTION\_SHELL...)
- The tool bar (new icon allows to review fiber orientation on elements)
- The model checker (orientation check)

### ▼ Model Checker

A number of enhancements have been made to allow the user to customize the Model Checker:

- Checks can be organized in different model checker files
- Level of checks can be changed easily in the right-click context menu



- Modifications can be saved and maintained across sessions
- Filtering capabilities allows you to show only errors, warnings

#### ▼ Meshing Editing

- Mirror functionality can now be applied in any direction
- Nodes can be aligned between 2 nodes or points in the Move panel
- Connection are now automatically deleted when the structural elements they refer to are deleted
- Free nodes can be deleted via window selection mode

#### ▼ Tied Contact Status

A new projection method is implemented and enables user to accurately report the number of untied nodes in a given tied contact. The level of the check in the Model Checker has been adjusted to match the level that will be reported by the solver based on the contact definition.

#### ▼ Searching and Filtering

Search and filtering capabilities are available in generic panel; select in model and property/material file window. Entities can be searched by name, by ID, ID range, and so on.

Similar search capabilities are also available in the Model browser, but filtering is currently not available.

#### ▼ Pre-simulation

Parameters defined to run pre-simulation for dummy positioner and seat deformer can be customized and saved as a user profile. This profile is saved in the preference file and kept across session.

#### ▼ Seat Belt Routing

A slider bar allows to control the orientation of the end vector and the position of the middle node.

## RADIOSS Block User Profile

---

### ▼ New Monitored Volume Panel

The monitored volume panel now use the generic page. All entities related to monitored volume can be created and modified from this page. The porous surface are now supported for AIRBAG1 and COMMU1. It is also possible to convert old airbag definition (AIRBAG, COMMU, FMVBAG) to the new format (AIRBAG1, COMMU1, FMVBAG1).

### ▼ Support of XELEM (Multi-Strand Elements)

Multi-strand elements can now be created from the Seatbelt Generator page.

### ▼ Enhanced Extraction

A new extraction logic is available and can be activated by activating the environment variable `HC_ADVANCED_CONNECTION_EXTRACTION` in the `hypercrash.tcl` file. This new extraction logic does not change the model definition.

The connection are extracted based on part name, logical connection are created by type and organized into the corresponding panel. Automatically organized into the correct connection type.

If the environment variable is activated, HyperCrash will propose to automatically extract the connection from the imported file (if any detected).

### ▼ Keyword Support

Newly supported keywords:

- /SURF/BRIC/FREE
- /SURF/BRIC/EXT
- /MAT/LAW84
- /FAIL/EMC
- /MAT/GAS/CSTA

Updated keywords:

- Failure model: FLD, TBUCTHER
- Interface: Type2, Type6, Type7, Type11, Type16, Type19, Type24
- Material: Law2, Law32, Law43, Law52, Law58, Law59, Law70, Law73, Law77, Law83
- Monitored Volume: AIRBAG, AIRBAG1, COMMU1, FMVBAG1, PRESS
- Property: Type4, Type8, Type12, Type13, Type17, Type19, Type43
- /SENSOR/INTER
- /DAMP and /DAMP/INTER

## LS-DYNA User Profile

---

### ▼ Support of #SET\_TYPE\_GENERAL

Creation and modification of all \*SET\_TYPE\_GENERAL is now available from the Set panel.

### ▼ Keep Data After \*END

All information stored after \*END that HyperCrash does not recognize is now maintained in I/O. This allows users to keep comments or metadata written by other pre-processors.

Note that only I/O is ensured; the information that is not supported will not be updated during the session in case of renumbering, delete, remeshing, and so on.

### ▼ Keyword Support

The following materials are newly supported:

- \*MAT\_024\_HAZ
- \*MAT\_219
- \*MAT\_230
- \*MAT\_266
- \*MAT\_272

The following keywords were updated to LS-DYNA R7:

- \*CONTROL\_CONTACT
- \*MAT\_106
- \*MAT\_125
- \*MAT\_169
- \*MAT\_ADD\_EROSION

### ▼ Solver Conversion

The following enhancements were made on the conversion from LS-DYNA to RADIOSS:

- \*MAT163 is converted into LAW33
- \*MAT\_S05 is converted into spring TYPE13
- \*MAT\_S08 is converted into spring TYPE13
- \*MAT\_HILL\_FOAM is converted into LAW62
- \*MAT\_OGDEN\_RUBBER is converted into LAW42

The following enhancements were made on the conversion from OptiStruct to RADIOSS:

- CELAS1 and CELAS2 are converted into spring type8
- PBEAML and PBARL are converted into beam type3
- SET GRID and SET ELEM are converted into equivalent node and elements sets
- An HTML conversion report is generated in the directory in which the OptiStruct or Nastran model is read

#### ▼ Resolved Issues

- Conversion from OptiStruct to RADIOSS of second order tetra elements was incorrect.
- Converting BRICK20 to SHELL16 elements lead to a crash
- Direction of the isoparametric frame for brick element was incorrect in the RADIOSS Block user profile
- Mesh size was taken into account when checking for initial penetration of \*CONTACT\_SURFACE\_TO\_SURFACE; \*CONTACT\_NODE\_TO\_SURFACE in LS-DYNA user profile
- Resolved several issues on interface type24:
  - Visualization (slave/master)
  - Conversion from type7 to type24
  - Mandatory fields
  - Model checker

# BasicFEA 14.0 Release Notes

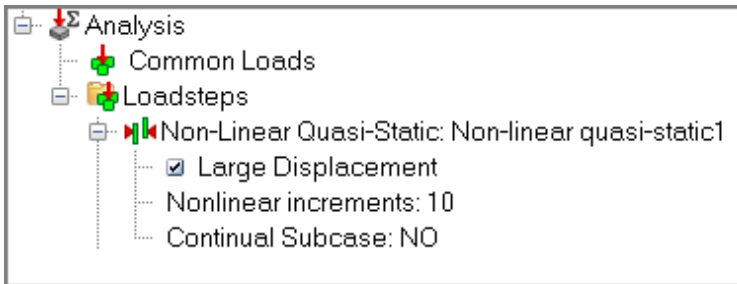
## Highlights

---

For v14.0, user experience was improved, so that you can approach additional analysis types, and expand on the ones already introduced. Specifically, new non-linear capabilities, new fatigue references, and load types were added. Additionally, there are several browser updates, which enhance the user experience by focusing on efficiency within the browser to cut down modeling and editing time.

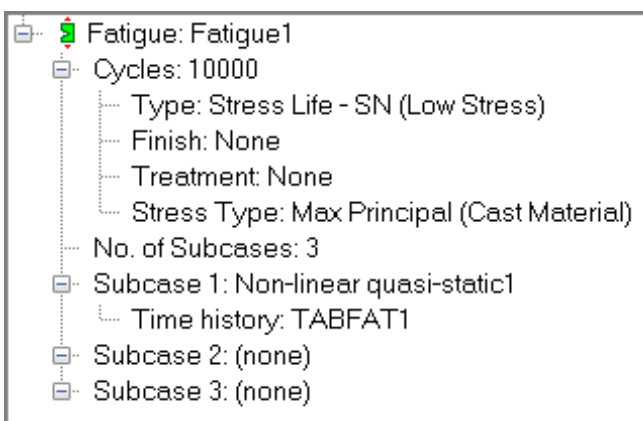
### ▼ Non-Linear Quasi-Static Large Displacement Subcase, LGDISP

Previous support on small displacement non-linear quasi static support has been expanded upon to include large displacement analysis. This includes incremental subcase control and the use of continual subcase so that you can reference and use previous solutions.



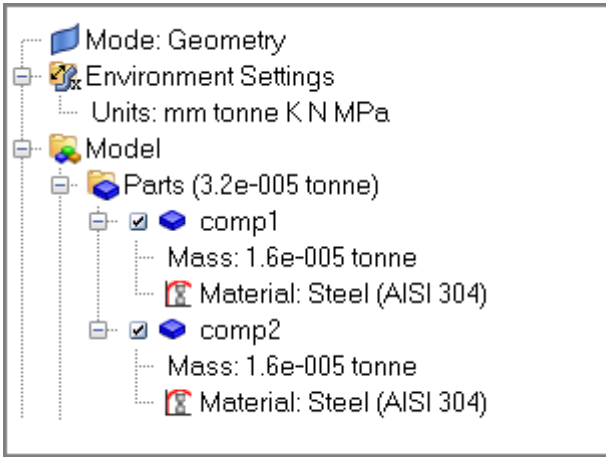
### ▼ Fatigue Subcase – Subcase Reference and Time History Support

Non-Linear subcase references are now available under the fatigue subcase. Additionally, you can now specify time history (TABFAT) and up to 10 linear and non-linear subcase references in the browser. There is also the ability to import your own time history information in supported formats.



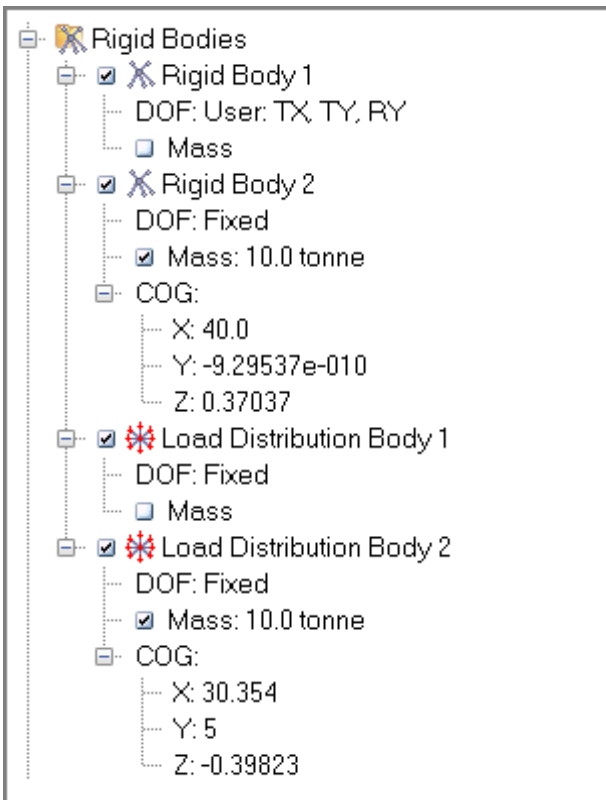
### ▼ Mass Summary for Parts

Unit specific mass values are now displayed for each part and cumulatively for the entire model.



### ▼ New Mass Option for Rigid Body and Load Distribution Body

You can now enable a mass (CONM2) and center of gravity (COG) for the independent node of Rigid Body and dependent node of Load Distribution Body.

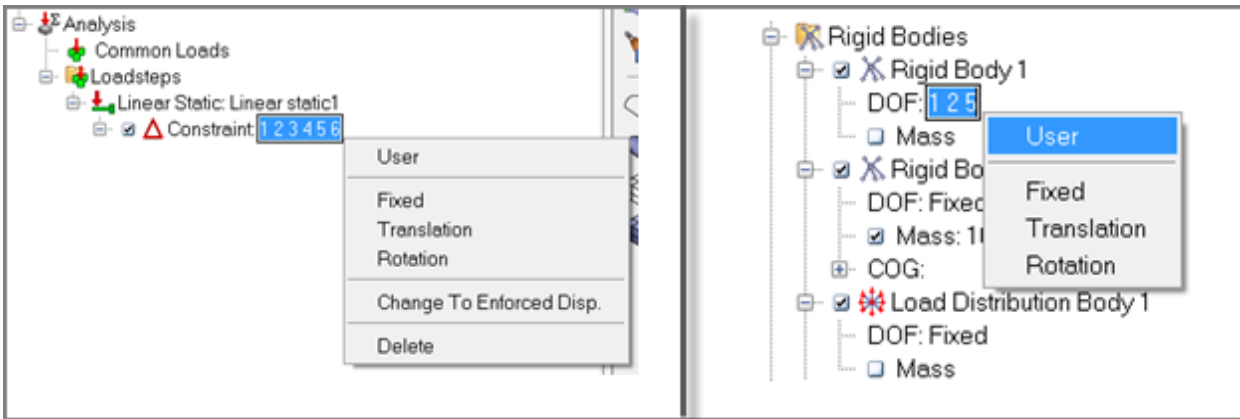


▼ **New Boundary Conditions**

- Force >> On Point
- Enforced Displacement

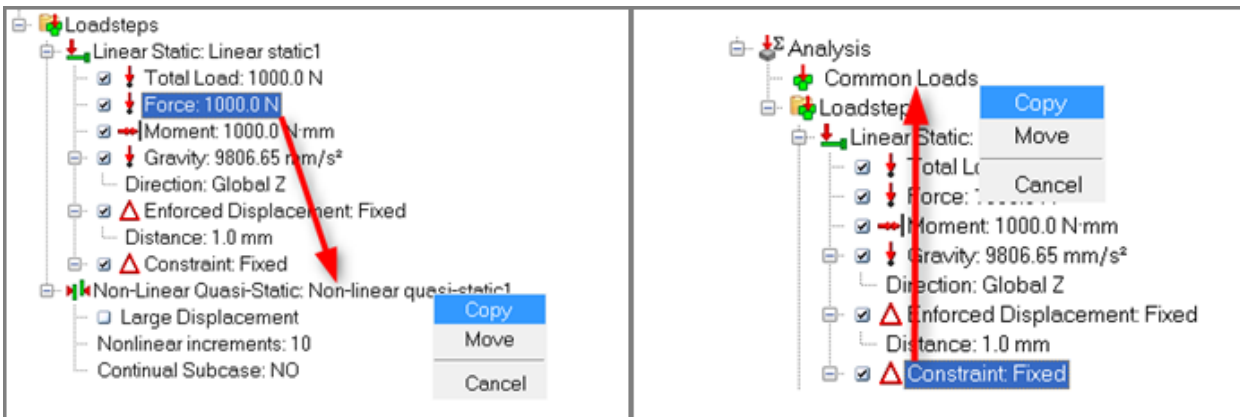
▼ **New Boundary Condition Editing Mechanism**

You can now edit rigid body, load distribution body, constraints, enforced displacement via a user input field for DOF 1-6.



▼ **Drag and Drop Boundary Conditions in Browser**

Boundary Conditions can now be dragged and dropped between different loadcases, if valid, and between loadsteps and common loads.



▼ **Resolved Issues**

- Thickness values with units are now a supported input.
- Fixed issue with Mesh settings for single parts. Independent settings are now kept for parts and global setting.



- Fixed contact direction issue with Sliding Pins to make more consistent.

# Automate 14.0 Release Notes

## Highlights

---

- Task development and debugging features in the Task Builder and Model Process browser are more intuitive and easier to use.
- Key functions are simplified including sharing and running processes.
- New options include: Batch Mode, Process as Wizard, and Session Retrieval.

### ▼ Core Enhancements

- **Save and Retrieve Session:** This feature lets you save and retrieve process states in a HyperWorks session.
- **Simplified Task Builder:** The new Task Builder helps you create tasks and datatypes and save them inside of task catalogs that are easily accessible from the Task browser.
- **Save to Process Catalog:** Seamlessly save new processes to a catalog for reuse.
- **Preview and Export with Process as Wizard:** Encapsulate and share complete processes inside of a Wizard. Recipients can execute custom processes without knowledge of Automate.
- **Process as Command:** Execute processes from third party applications such as Microsoft Excel. Run Automate processes in Batch Mode.

### ▼ Usability Enhancements

#### Better Look and Feel

- Meaningful command icons
- Highlight indication

#### Search and Help

- Search in all browsers to quickly retrieve items.
- Catalog Help provides information for you to understand task and process usage.

#### Easy Access to Properties and Easy Debugging

- The Model Process browser provides easy access to task properties and helps you isolate problems in a model and verify values for inputs, outputs, and parameters.

#### Simplified Task and Process Assembly

- Port selection is easier and helps you avoid mistakes when connecting tasks and processes.

▼ **Demos**

The following demos are part of the installation package and include helpful task development examples:

- Element Check Process: Learn how to perform 1D, 2D, and 3D element checks based on user-defined criteria.
- Excel Connect: Learn how to expose MotionView model building and analysis processes via Excel.

# HyperMath 14.0 Release Notes

## Highlights

---

HyperMath 14.0 is the first release of HyperMath delivered on Mac OS-X.

The use of the Intel MKL libraries contributes to improve the performance of the computations.

With HyperMath 14.0 some improvements are added to the editor and the support for the TCL scripts is enhanced.

### ▼ Mac Support

HyperMath 14.0 is now available on Mac platform (Mac OS-X).

### ▼ MKL Libraries

The Intel MKL libraries are used to improve the performance of the computations (such as matrix operations).

### ▼ Editor Improvements

The HyperMath editor new enhancements:

- The ability to execute the selected part of an HyperMath (.hml) script
- Better access to the "Find" with a "Find Again" shortcut
- A shortcut to switch tabs when multiple scripts are open
- On TCL command window, every TCL command executed will display the evaluated output on screen
- Added a command to clear the TCL console

### ▼ Recording Macros

HyperMath's Editor now has a Macro recording menu. You can start/stop recording edition actions and replay them (one to many times). Macros can be stored, recalled and removed.

### ▼ TCL Support Enhancements

The support of the TCL interpreter is improved in HyperMath 14.0. In particular, the editor offers a direct access to the help pages of the HyperMesh APIs. In case of cross execution or cross debugging between HyperMesh and HyperMath, the output of the TCL commands is now done in the console of HyperMesh.

▼ **User Interface Improvements**

Plots are enhanced. For instance, axis ticks can be defined now by entering a step size.

▼ **Resolved Issues**

- Improved plot axis values (round-off)
- Other HyperMath language (HML) issues
- Issues related to the script edition

# HyperStudy 14.0 Release Notes

## Highlights

---

HyperStudy 14.0 provides several exciting new features that enable users to explore designs more thoroughly and efficiently than ever before. From quickly picking your design parameters in HyperMesh to a new streamlined response definition experience, getting HyperStudy setup to run your investigation has never been easier. The new inclusion matrix allows optimizers to utilize existing run information, increasing the optimizer's efficiency and making the most out of your data. The automatic cross-validation check can save time by calculating checks on the Fit quality without a dedicated validation matrix.

### ▼ Inclusion Matrix

The inclusion matrix enables optimizers to optionally accept available run data as an input. This can save significant time during the optimization process, as the optimizer can utilize previously run data. This inclusion data typically comes from previously run approaches, for example, DOE or another optimization, meaning users are empowered to make sure no run data is left unused when optimizing.

### ▼ File Assistant Wizard to Build Responses

The File Assistant Wizard lets you select a target file, and a guided set of dialogs produces a completely defined response. This efficient process simplifies the response creation steps by moving the response construction syntax of the expression builder to the background.

### ▼ Parallel Coordinate Plots

Parallel Coordinate Plots, also known as Snake Plots, have a dedicated tab in each approach's post-processing. The plots feature controls to only show subsets of variables and responses as well as window filters to visualize areas of interest. These versatile plots are very effective in mining information from large data sets.

### ▼ Extended HyperMesh Parameterization

User created parameters in HyperMesh are expanded to include the Abaqus, Nastran, and RADIOSS user profiles. These parameters can be imported by HyperStudy from HyperMesh models as design variables.

### ▼ Cross-Validation Introduced in Fit Post-Processing Diagnostics

Cross-validation is a technique used to assess the Quality of a Fit without requiring a validation matrix. This feature is now automatic on all Fit types in HyperStudy. While a dedicated validation

matrix is still suggested for more complete checking, this time saving calculation can provide quality information, when a validation matrix is not available.

#### ▼ Filtering Out Data in Fit Approaches

Including data gaps, duplicates, or outliers when building a Fit can lead to poor quality results. New data filters automatically clean up data sets without manual intervention, helping ensure the Fit is not biased by bad data.

#### ▼ Other Enhancements

- New SimLab model type
- New FEKO model type
- Fit tradeoff tabs redesigned providing an intuitive and efficient results browsing experience
- Fit quality metric introduced to dynamically track extrapolation
- Histogram and Box Plots merged into Distribution tab
- Discrete variable support has been extended within the user interface
- A uniform distribution is now available for discrete variables
- Introduced the Editor option to visualize fixed field demarcation lines
- New database to handle larger data sets
- P-values available in least squares regression diagnostics and ANOVA tables
- Autobuild for Fit is removed as an option; the Fits will always be built.
- Table cells now support the display of string variables values

#### ▼ Resolved Issues

- The execution status of runs when multi-execute is enabled has been updated
- The tooltip on the Optima tab now presents updated information
- Precision loss of importing data into Fit is addressed
- Improved speed and performance of switching plots
- Safe opening of a running study with a second HyperStudy occurrence is supported
- Fixed issues with using relative paths and MotionSolve models
- Fixed a memory issue when starting HyperStudy from HyperMesh
- Corrected matrix size calculation when creating a Full Factorial DOE with linked variables

#### ▼ Known Issues

- An indexing change in the structure of stress results read by 14.0 products could impact responses defined in older studies. Responses built using expressions like `resvector`, which



use indexed values, will require a different set of indices when compared to previous versions. Newly defined responses will not be affected.



# HyperForm 14.0 Release Notes

## New Features

---

### ▼ Incremental RADIOSS

- Support for modeling of Pad entity that uses spring to apply the force during forming in the user process.
- Support for complete hot stamping setup including Blank transfer, tool closure, forming and quenching in a single setup using user process
- Trimming panel updated to add node selection for identifying the area to remove/keep for 3D trimming

### ▼ General

- Support of tube hydroforming process model setup in the User Process.
- Support of standard forming process: Single Action Draw, Single Action Draw with Pad, Double Action Draw, Double Action Draw with Pad, and Triple Action Draw in the User Process.
- Support for trim line optimization and blank shape optimization in the Process Optimization setup.
- Support for more than one tooling for the gravity setup in the Auto Process.
- Spring back distance calculation has been updated to plot the distance value with signs based on relative position of the part after spring back.
- Entity Editor has been added in the Model Browser to edit solver keywords.
- New option available in HyperForm Settings to include or exclude strain, strain history from gravity run in forming. Default is set as Exclude.

### ▼ Die Module

- New Die browser that tailors user in natural work flow to create die surface from the part geometry.
- New Die browser has following additional options:
  - Automatic hole recognition and identify the holes for future trimming operation
  - Automatic hole filling
  - Automatic and manual flange identification and remove flanges from the draw die design
  - Contour options for
    - Undercut negative draft in the part relative to draw direction
    - Draw depth based on flat binder along the draw direction
    - Draw depth based on created binder surface along the draw direction

- Part tipping updated to consider only the features formed in the part in the drawing stage.
  - Allows you to create multiple stage draw dies.
- New option to develop trim line based on the flanges and the addendum.
- Binder creation and editing improved to handle binder by sections and part on binder.
- Binder edit options has been redesigned to allow users to update the binder through sections with options of changing them in 2D and 3D plane and ability to cast them at other locations as needed.
- Part on Binder has been added as a new option to edit the binders created to match surface of the part.
- Prep Geom panel has been updated with new options to
  - Fill side cutouts
  - Remove the surface edge radius
  - Identify holes and fill them
  - Extend surface beyond the selected part boundary

#### ▼ One Step

- Automatic filling of holes in the part from the One Step browser.
- Blank nesting tool layouts are updated to support one-up, two-up and mirror layouts with options for each of them.
- Addendum definition added as part of the blank nesting to add additional material to flattened blank.
- Carrier option added to support nesting for progressive die use cases. Added carrier options includes side carrier, central carrier, nested carrier, double side and central carrier, double side and nested carrier, double side carrier.
- New Batch RIT and Nesting tool has been added that performs Results Initialization and Nesting in batch mode from input file that could be CAD geometry or HM file.

## Resolved Issues

---

### ▼ Incremental RADIOSS

- Blank penetration into the tooling at the end of the gravity run.
- Trimming functionality has been improved to avoid small elements and degenerated elements in the trimmed edge to improve spring back.
- Element connectivity not exported after trimming in the STA file for solid element trimming.

### ▼ General

- Auto position and return home position are fixed to work for local system for both sheet and tube.
- Multistage manager provides warning message if there are spaces in the path rather than an error during run.
- Material definition not carrying to all stages in the Multistage Manager.
- Draw bead creation improved to avoid skewed draw bead lines due to projection.
- Improve auto position speed to handle pin – sheet contact.
- Log file concept implemented to update user on the progress in the optimization.

### ▼ Die Module

- Addendum modify panel updated to highlight the ribs for selection.
- Addendum generation updated in Addendum Modify panel updated to create continuous surface with no gaps.

### ▼ One Step

- Warning message added if the material selected is not exported into the created deck.

# HyperXtrude 14.0 Release Notes

## HyperXtrude - Metal Extrusion

---

### ▼ Interface for Super Alloy Extrusion

The Extrusion Wizard is enhanced to support super alloy extrusion with glass pad lubrication.

### ▼ Improvements to the Extrusion Wizard Batch

Many major improvements are implemented in batch version of the Extrusion Wizard to create a complete HX data deck in the background. These improvements include automatic geometry cleanup, robust meshing methods and better reporting of the status and errors.

### ▼ Bearing Optimization Support in the Extrusion Wizard

Extrusion Wizard batch is enhanced to support bearing optimization model setup.

### ▼ Models with Choked Bearing Region

Extrusion Wizard batch is enhanced to support models with choked bearing regions.

### ▼ Bearing Regions with Non-uniform Starting Points

Extrusion Wizard batch is enhanced to support models that have bearing regions starting at varying z-coordinates.

### ▼ Batch Version of Tool Deflection Wizard

A batch version of the Tool Deflection Wizard is implemented to be used by the HX Job Manager and HyperXtrude 2015.

### ▼ Weld Strength Data

The interface is enhanced to automatically write the boundary and parameter data required to compute weld strength.

#### ▼ Grain Size Data

The interface is enhanced to automatically write the default parameters used for computing grain size in the \*.hxx file. This will enable users to modify these parameters as needed by editing this file.

#### ▼ Support for Coupled Analysis with OptiStruct

The Extrusion Wizard is enhanced to support setup of coupled analysis with OptiStruct. The interface will automatically write both HX and OptiStruct data decks. This feature is supported in batch mode as well.

#### ▼ Multi-cycle Analysis Support via Extrusion Wizard Batch

Extrusion wizard in batch mode now supports transient multi-cycle analysis. In addition, it will automatically setup time step data. Hence, from HyperXtrude 2015.1, a transient multi-cycle analysis with billet skin can be performed with just two additional mouse clicks.

#### ▼ Resolved Issues

- Re-enabled HX Job Manager to show partial/full run status of solver jobs on remote machines.
- Bug fixes to fit material module and adding new materials to user's location.
- Resolved data handling errors in Create Billet macro.
- Resolved an issue in computing Billet Diameter for models with symmetry.
- Bug fixes and improvements to efficiently update shell scripts on remote machines that are used by HX Job Manager.
- Resolved issues in writing metadata for HX reports
- Resolved few minor GUI bug fixes.

## HyperMold User Profile

---

### ▼ Automatic Geometry Cleanup

A tool to automatically perform geometry cleanup has been added. This will help the user to remove unwanted edges on surfaces and sliver surfaces.

## Moldflow User Profile

---

### ▼ Fusion Meshing Wizard

An easy to use fusion meshing wizard is added to the interface. This tool will estimate element size (which can be changed) and then mesh the model such that it has a good mesh match ratio.

### ▼ Resolved Issues

- Resolved issues in automatic 2D meshing algorithm used by different wizards
- Resolved issues in extracting and plotting thickness for Midplane models

## Moldex3D Solid and Shell

---

### ▼ Resolved Issues

- Resolved issues in automatic 2D meshing algorithm used by different wizards
- Resolved issues in extracting and plotting thickness for midsurface models



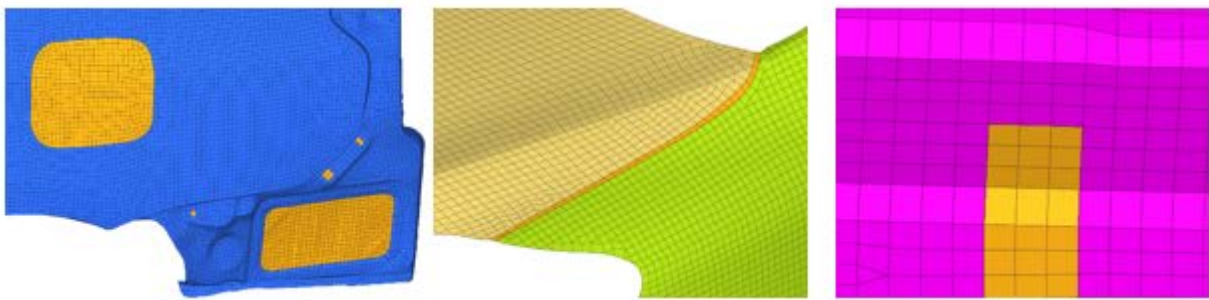
# CFD 14.0 Release Notes

## General

---

### ▼ Fill Hole/Gap Patch Utility

This tool is located in the mesh menu for all user profiles. It is useful utility to fill the holes and feature loops. It works for 1st order tria or quad elements. The selection makes it easier to fill the holes for selected components. It is an automated utility to fill the holes within selected components or elements. You can manually fill the holes using node/free edges selection.



### ▼ Merge – Disconnect Components based on Connectivity

The **Merge** and **Disconnect** tools are located in the Utility tab.

- **Merge:** Merges the selected components to one component. This is useful for combining/organizing components for meshing or analysis.
- **Disconnect:** Disconnect a component based on connectivity of mesh. This is very useful when you are working with `.stl` models.

### ▼ Mesh Control Browser

The Mesh Control browser is located in the Mesh menu for all user profiles.

Mesh controls provide the capability to automate and streamline the meshing process for **surface meshers**, **adaptive wrapper** and **volume mesher** in one browser. Using mesh controls enable you to define mesh settings for all of the entities that need to be meshed in one meshing job.

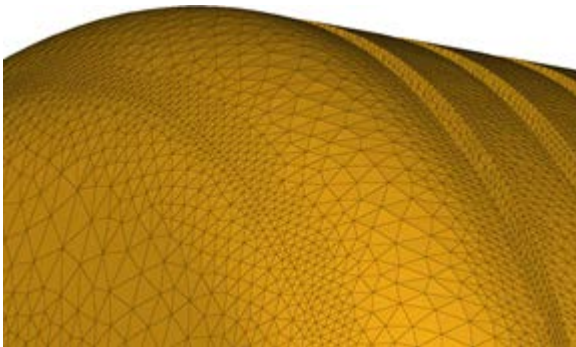
- **Local Settings:** Define local settings for surface mesher, adaptive wrapper and volume mesher.
- **Export/Import:** Mesh settings defined in mesh controls are stored in the database. Mesh controls can also be saved to an external `.xml` template file for reuse in other models.

## Surface Meshing Enhancements

---

### ▼ Surface Deviation Based Element Remesher

New multithreaded mesher, similar to surface deviation to remesh the elements is now available. It can be accessed via mesh controls. It captures curvature and provides smooth transition from smaller sized elements to larger sized elements. The new mesher internally identifies the elements features based on defined feature angle and captures and refines them. It provides the capability to refine the mesh based on proximity and angles specification. You can define any additional features to be captured and refined.

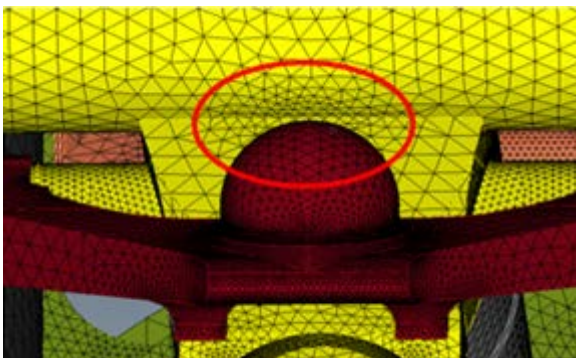


### ▼ 2D Mesh Refinement Controls

New options to refine 2d mesh based on the proximity, angle and shapes are added and can be accessed through mesh controls along with surface deviation meshers.

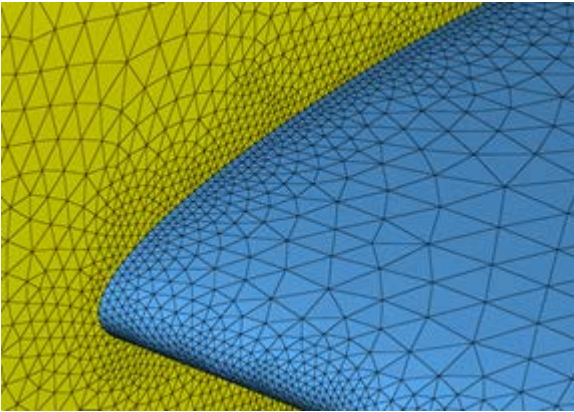
#### **Proximity based refinement:**

Provides the capability to refine surface or elements mesh based on proximity. It automatically identifies close proximity areas and refines elements based on user specification. You can define refinement based on refinement size, element size/gap ratio or through refinement curve.



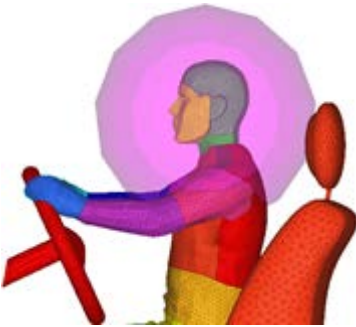
#### **Angle based refinement:**

Provides the capability to refine surface or elements mesh based on angle between surfaces or element faces. It automatically identifies refinement candidates and refines elements based on user specification. You have control over angle limits and refinement size.



### Shape based refinement

New capability to refine 2D mesh based on shapes with refinement zones. Refinement zones can be accessed within the Mesh Control browser. You can define box, sphere, cone, frustum, cylinder for refinement. Graphical objects provide better idea of shape definition and location. Refinement zones are used to refine mesh with a defined size. Refinement zones are also supported for surface meshing (surface deviation only), adaptive wrapper, and volume meshing.

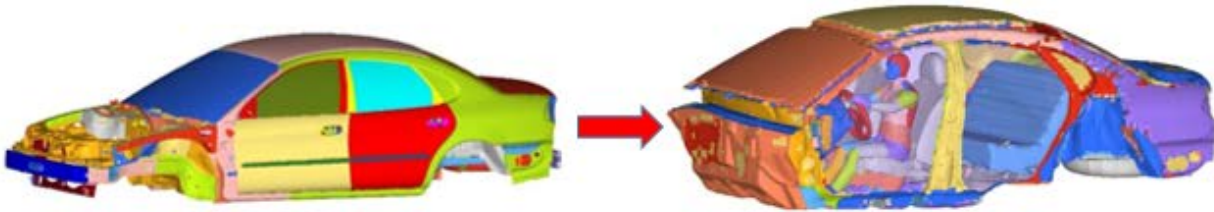


## Wrapper Enhancements

---

### ▼ Cavity Wrapper

New functionality has been added to the existing wrapper to wrap the cavity as well. With the same wrap settings, you can choose to perform exterior wrapping or cavity wrapping.



### ▼ Local Wrap Settings

New capability to define localized wrap settings via the Mesh Control browser. Local controls will take the precedent over global settings.

### ▼ Wrap Selection

New functionality to define which volumes to be wrapped is introduced. This functionality can be accessed in the Model Adaptive Wrapping Mesh Control. You have the option to wrap all the volumes, nth largest volumes, volume enclosed by seed node or exclude volume enclosed by a seed node.

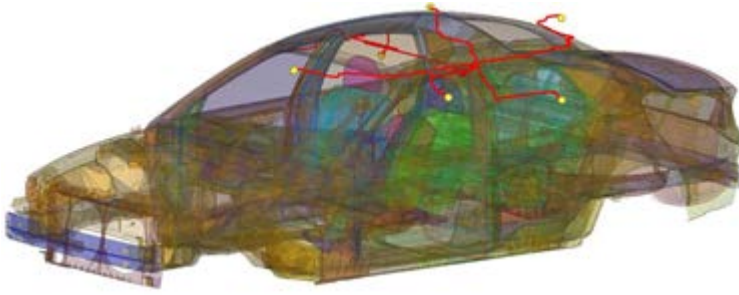
[-] <b>Wrap Selection</b>	
Wrap Type	Exterior
Wrap Selection	All
[+] <b>Wrap Mesh</b>	All
[+] <b>Refinement</b>	Wrap Enclosed
	Wrap Nth Largest
	Exclude Enclosed

### ▼ Improved Performance

Performance of wrapping is improved in this release. Feature capturing algorithm is much faster compared to 13.0.110. Few function is also multithreaded.

### ▼ Improved Leak Detection

New functionality is added to define source and target seed node for wrapping. Now you can define the enclosed node and target areas where leak checking is required. Multiple target points can be defined. It is a very useful visual utility which help to seal desired cavity. If the leak is identified, patching the gaps can be done by Fill-hole Patch tool or by increasing gap patch tolerance.



# Volume Meshing Enhancements

---

▼ **Performance Enhancements**

Performance of boundary layer meshing is significantly improved in this release. 55 mi BL Penta BL elements can be generated in one hour on a full scale external aero problem compared to 5 hours in 13-110. Boundary layer meshing is completely multithreaded.

▼ **Local BL Settings**

New capability to define localized boundary layer settings. User can define local BL settings for desired element set or components in mesh control browser. Local controls will take the precedent over global settings.

▼ **BL Aspect Ratio Based Growth Rate**

New capability to define growth rate based on desired aspect ratio. This option enables the growth rate definition for boundary layers to be based on the defined aspect ratio of the final layer. After the first few initial boundary layers, if this type of growth rate method is selected, the rest of the BL will grow to achieve the user-defined **Final layer height/base ratio**.

▼ **BL Acceleration Based Growth Rate**

New capability to define BL acceleration for boundary layers. You have the option to start the accelerating BL growth after few BL layers. This option will provide the capability to achieve faster BL- tetra transition.

▼ **1st Layer Height as Factor of Base 2D Elements**

This feature enables you to define 1<sup>st</sup> layer height as a factor of input 2d element size. This option is useful when the size of 2D elements varies significantly and a constant first layer height is not needed. With this factor, a smooth BL to tetramesh transition for large variation in 2d element size.

▼ **Volume Selector**

New functionality to define which volumes to mesh and option to define solid and fluid volumes before meshing has been introduced. This functionality can be accessed in the Mesh Control browser. You have the option to mesh all the volumes, largest volumes or volume touching reference elements. While selecting volume touching reference element, you can distinguish fluid and solid volume as well.





### ▼ Boundary Layer Contour for Visualization

This feature provides new functionality to output boundary layer contours after volume meshing. This new capability provides better options for reviewing boundary layer mesh rather than masking elements or creating cross sections. You can define the option via the Volume Selector in the Mesh Controls browser. After meshing, you can load the `.res` file to visualize the contours of **first layer BL height**, **number of layers generated** and **total boundary layer height** on entire model.



### ▼ AcuSolve Full Import Deck Support

New capability to import AcuSolve solver settings as well while importing `.inp` file has been introduced. While importing AcuSolve solver decks (`.inp` file) the defined parameters in the solver deck will also be imported in the model and can be accessed via the Solver browser. Any unsupported parameter will be collected in "Unsupported Entries" under each folder category of the Solver browser. You can manually edit those entries and the modified values can be exported.

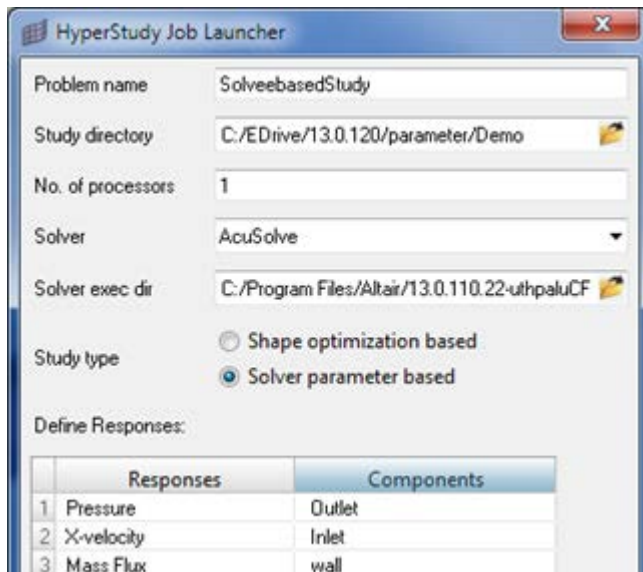
### ▼ AcuSolve Solver Parametric Studies

- **Parametric Definition:** You can now define discrete or continuous parameters and assign it to AcuSolve solver variables.

Name	Value
Name	component1
Type	INFLOW
<b>Simple Boundary Condition</b>	
Activate simple boundary condition	On
Precedence	1
Inflow type	Velocity
Reference frame	<Unspecified>
Inflow velocity type	Normal
Normal velocity ## (m/sec)	0.0
<b>Radiation Surface</b>	
Activate radiation surface	Select Parameter / Parameterize

Name	Value
Name	component1
Type	INFLOW
<b>Simple Boundary Condition</b>	
Activate simple boundary condition	On
Precedence	1
Inflow type	Velocity
Reference frame	<Unspecified>
Inflow velocity type	Normal
Normal velocity ## (m/sec)	parameter1 (0.1 0.2 0.3)
<b>Radiation Surface</b>	

- **Parametric Studies:** Once AcuSolve case is setup, parameters are defined and assigned to solver variables, you can use the HyperStudy Job Launcher to transfer all of the relevant data to HyperStudy. Afterwards, with a single click a design of experiment (DOE) study can be performed.



#### ▼ Resolved Issues

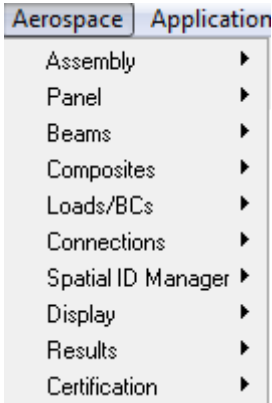
- Improved surface deviation mesher: Issues related to sliver elements has been resolved. Mesh flow has been improved. Enhanced smooth mesh across curved using better averaging of elements size
- Improved rigid body mesher: Fixed the issues of rigid body mesher creates free edges. Now free edges and t-connections has been reduced significantly
- Over refinement is avoided in the wrapper: Fixed the issues in wrapper to avoid over refinement where not necessary. This also results in performance improvement.



## Aerospace 14.0 Release Notes

### ▼ Aerospace Toolbar has been Rearranged

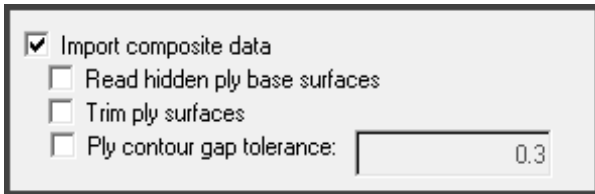
The Aerospace toolbar is rearranged into assembly, geometry, panel mesh, beam mesh, loads, composite, connectors, results and certification for ease of grouping of functionality.



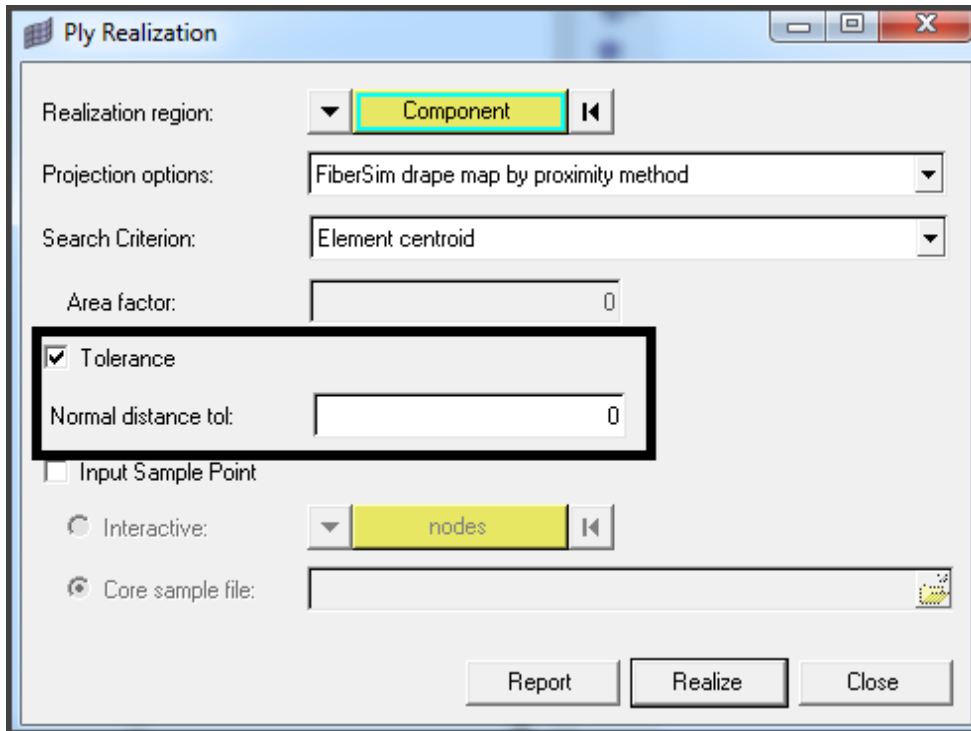
### ▼ Composite Enhancements

The Composite Import dialog has been enhanced to import ply base surfaces in addition to ply contour lines.

Each ply also now contains separate ply orientation system, and base surface.



A new ply realization method is added to Fibersim composite models.

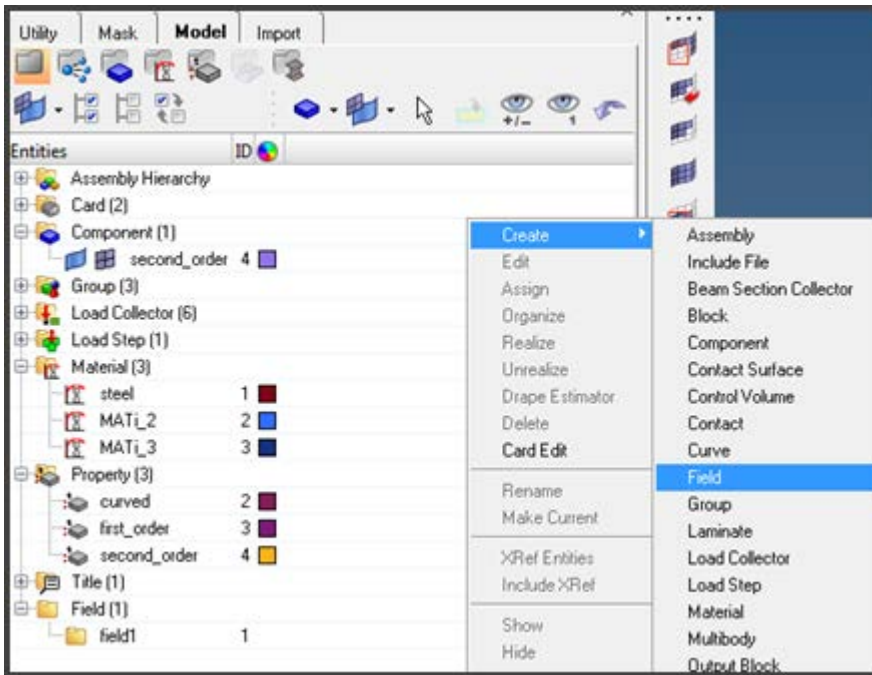


Composite realization for RADIOSS and LS-DYNA is now supported, including draping correction.

### ▼ Field Mapping

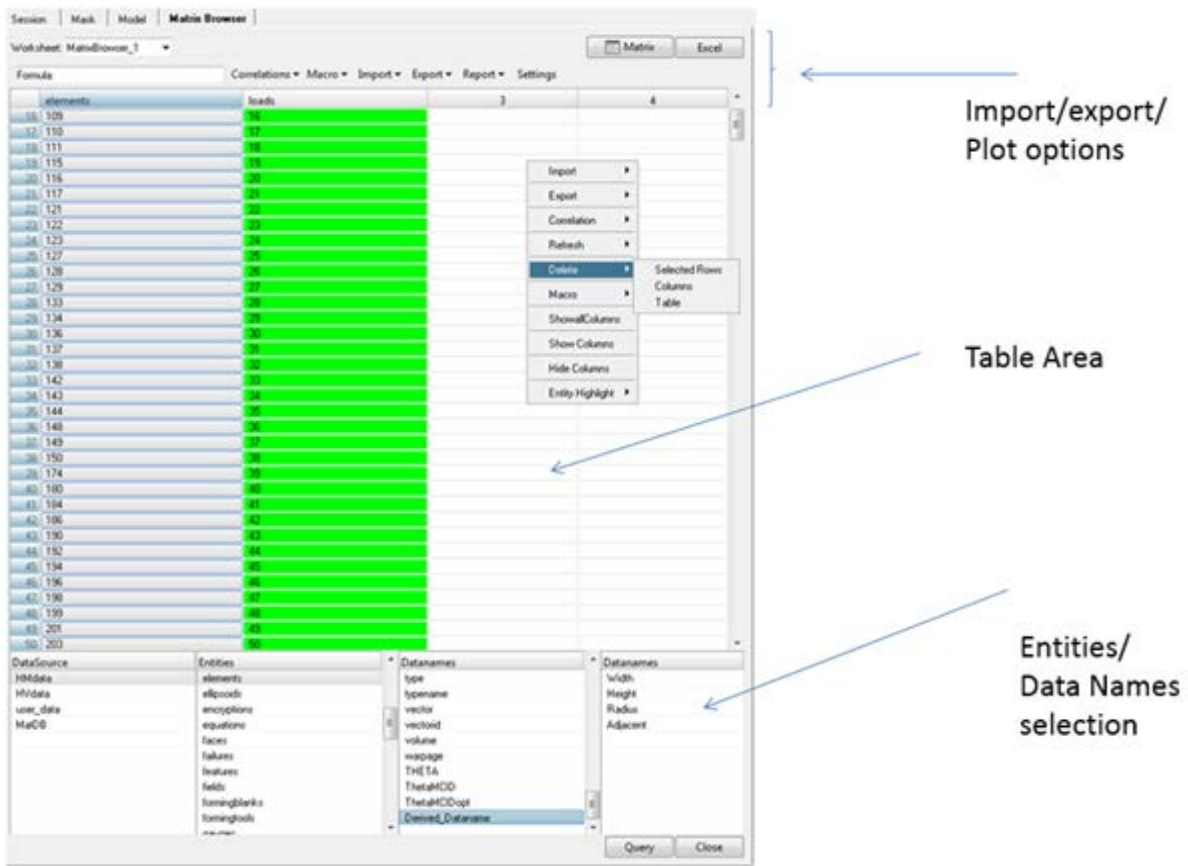
A new database entity called *Field* has been added. It is supported in the Entity Editor.

This entity helps map pressure, temperature, property (ID, thickness) and material orientation to new sets of elements and models from HyperView results files, CSV files, old solver decks and matrix tables.



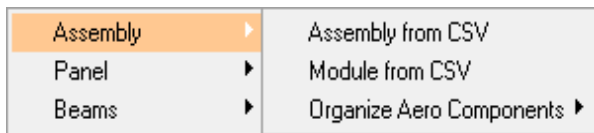
### ▼ New Matrix Browser

Version 2 of the Matrix Browser improves ease of use. This also supports multiple load case browsing from HyperView.

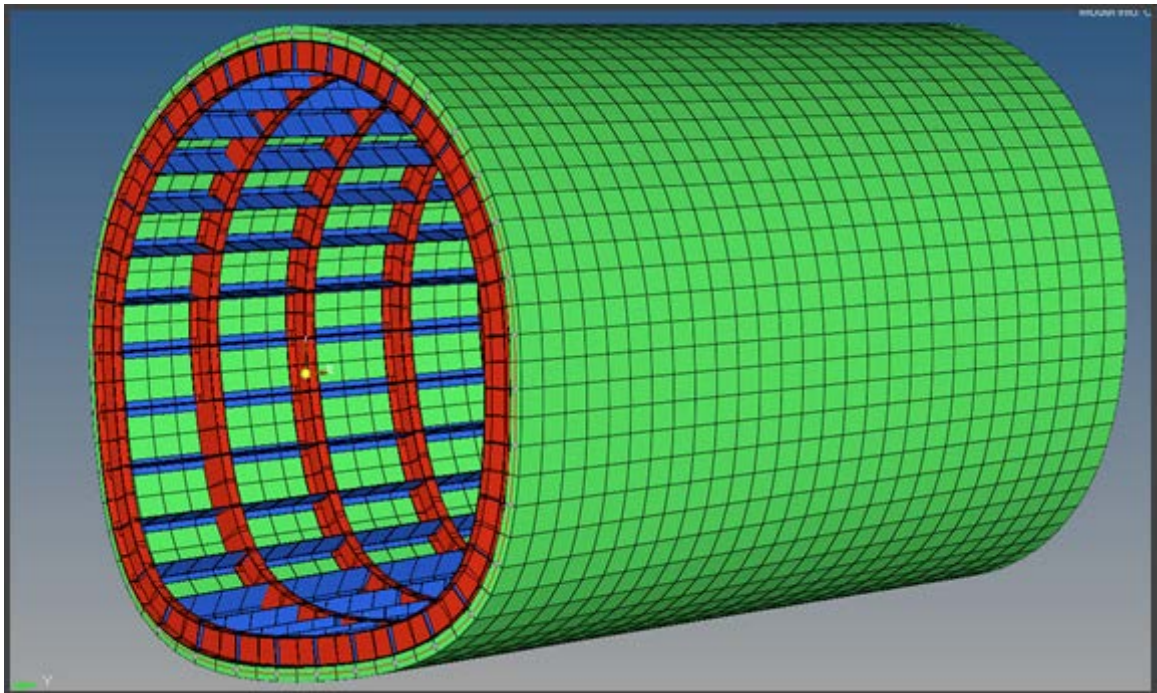
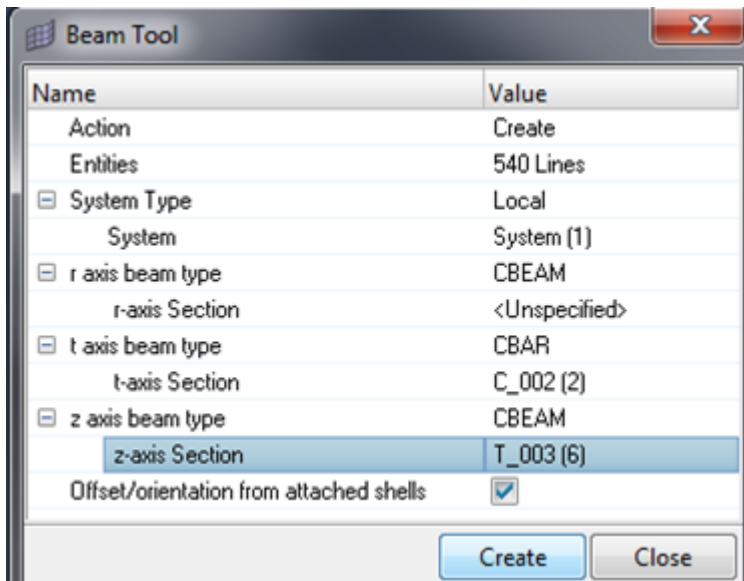


### ▼ Aerospace Tools

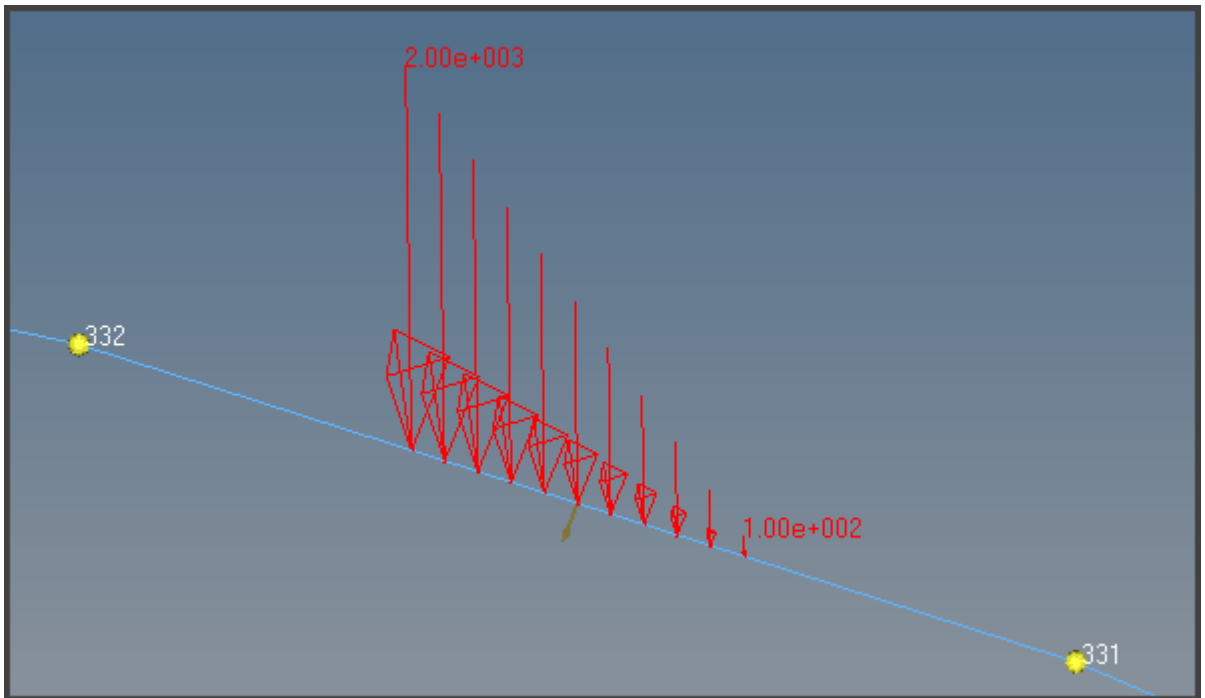
- **Assembly:** Organize models from Solver Input.



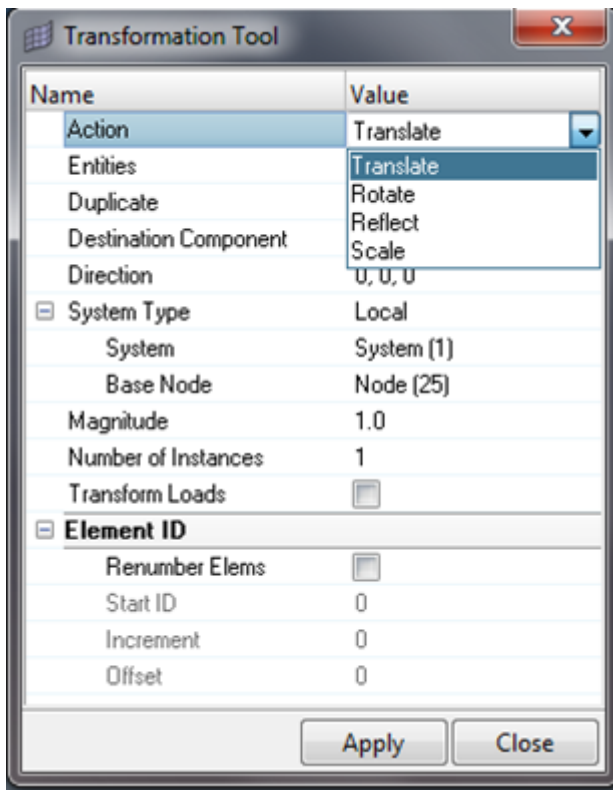
- **Beam from lines:** 1D element creation, update, orientation, offset and property calculation is enhanced for Aerospace users.



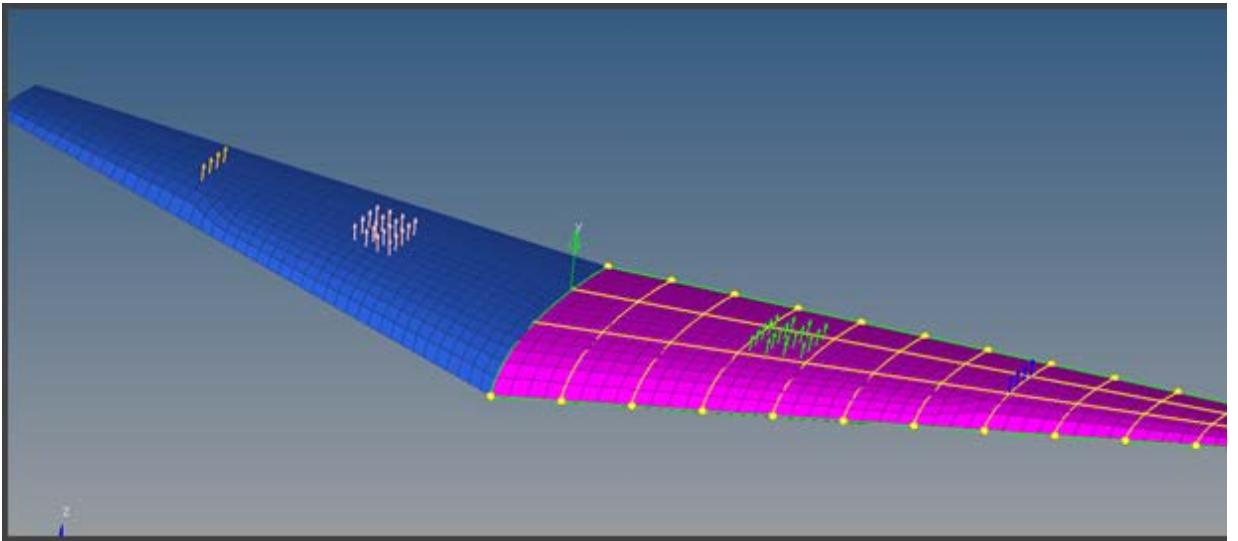
- **Loads:** New PLOAD1 creation/review, RSPLINE creation, review tools are added.



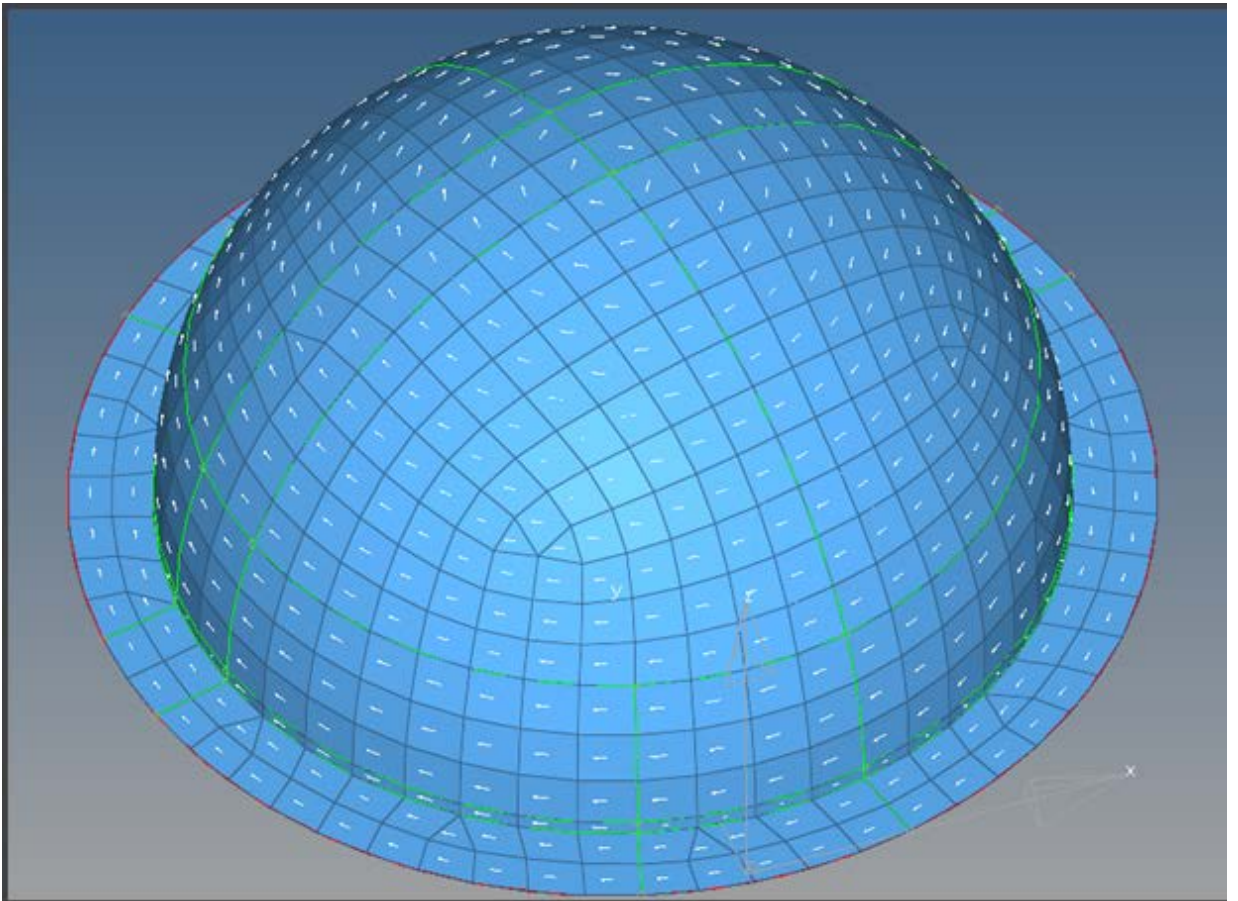
- **New Load transform tool:** Translate, rotate, and reflect models with copying loads and multiple instancing is enhanced.







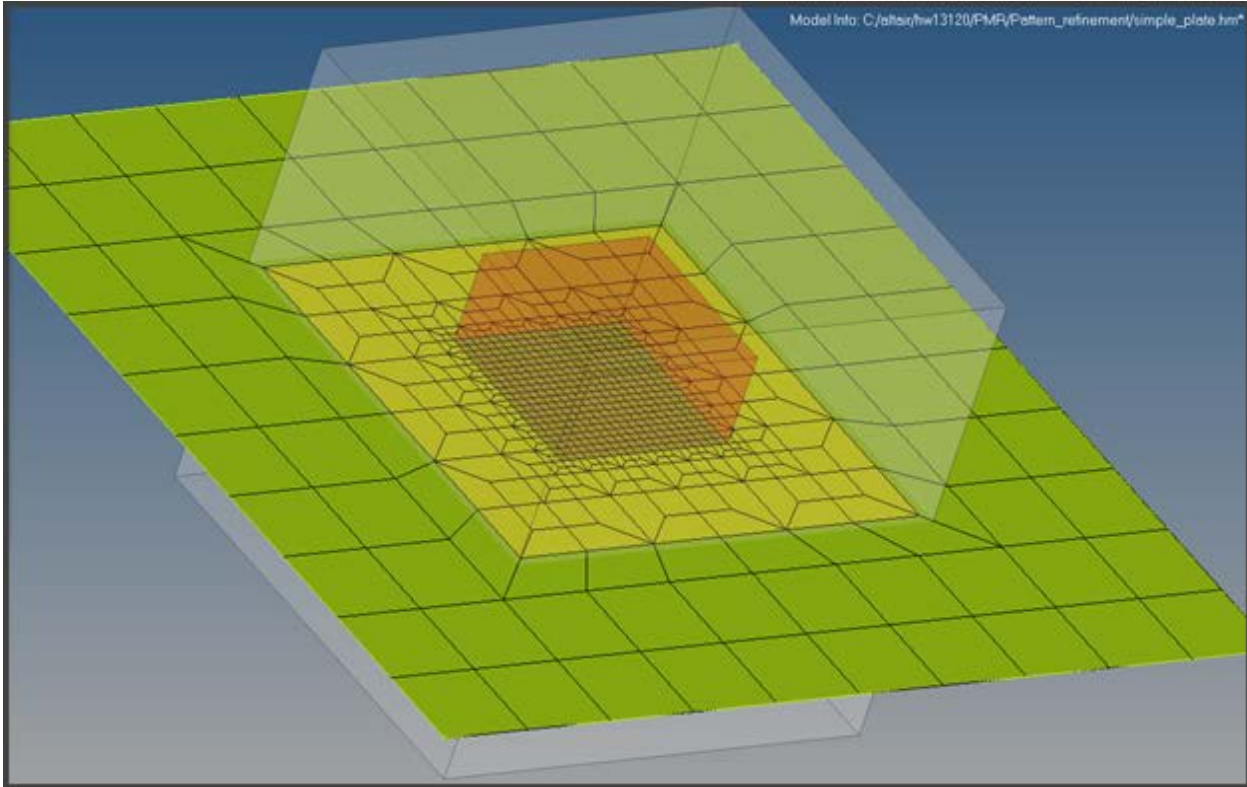
- **Composite:** New material orientation creation from curves has been added.



- **Results:** Min/Max, Envelop, ranking, bar end loads, and shear plots from HyperView results in HyperMesh has been added
- **Certification:** ESACOMP based composite certification added. This is a link to ESACOMP-APA product.

### ▼ Mesh Refinement

New element refinement tool based on quad mesh pattern is added.



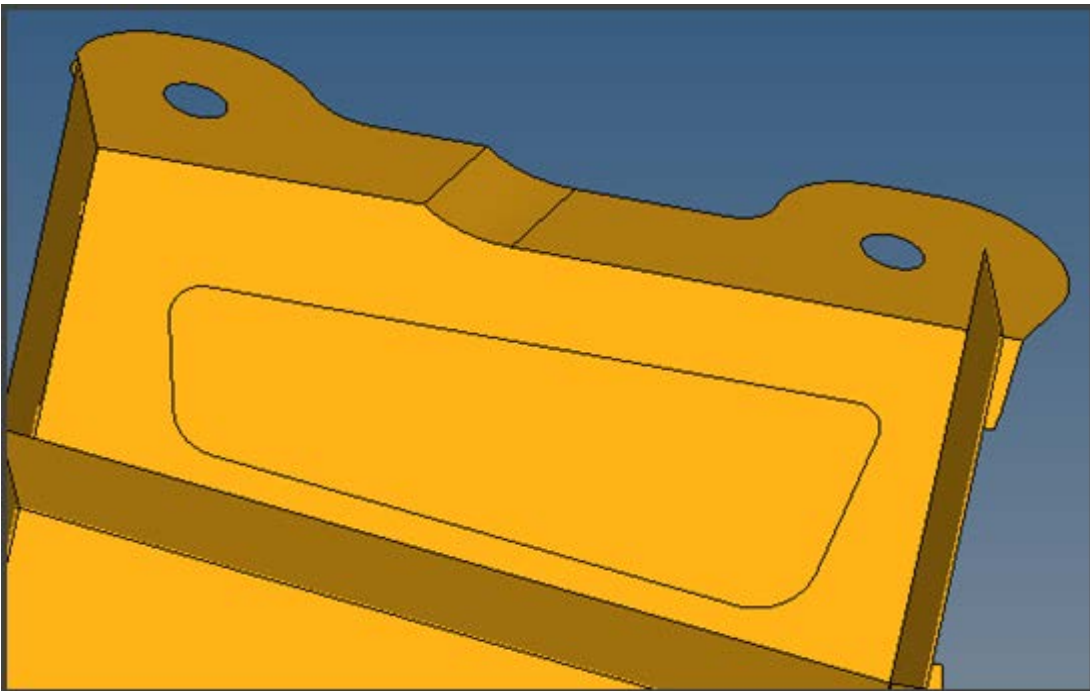
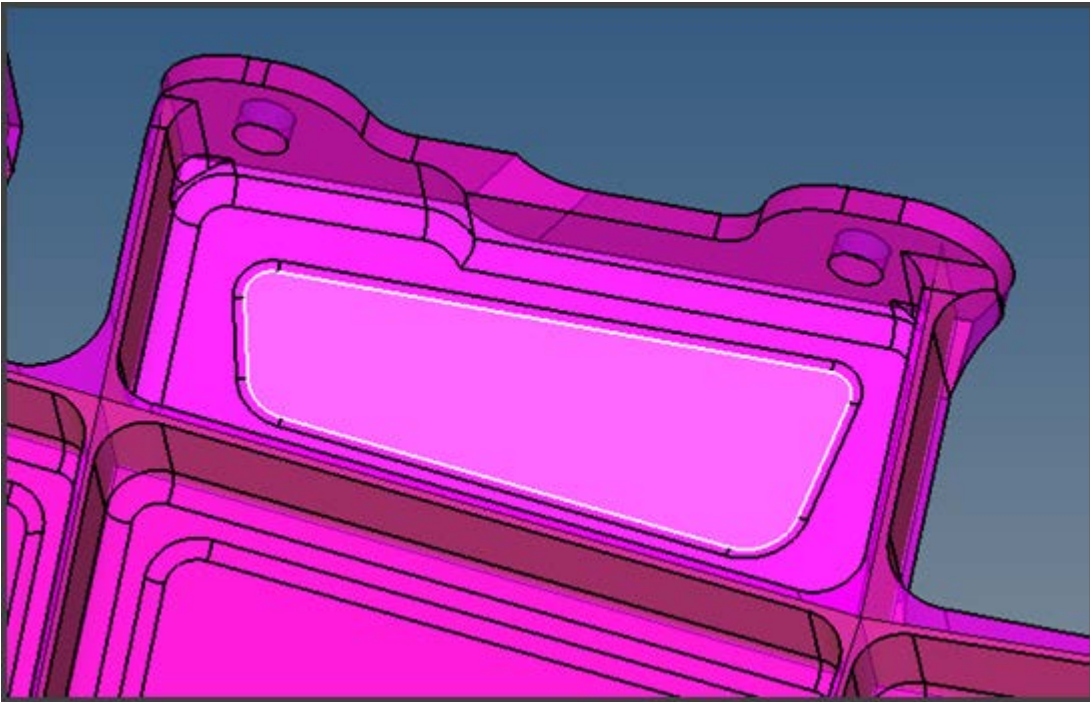
### ▼ HyperView Enhancements

- 1D element corner bound forces results are supported.
- 1D element forces are supported for nonlinear SOL 106 and 400 Nastran.
- 2D element forces based CROWS plots are supported.
- Free body diagram support in HyperView.
- New Aerospace profile is added to HyperView.

### ▼ Midsurface Mesh Imprint

Ability to imprint pockets from solids to midsurface models are added.





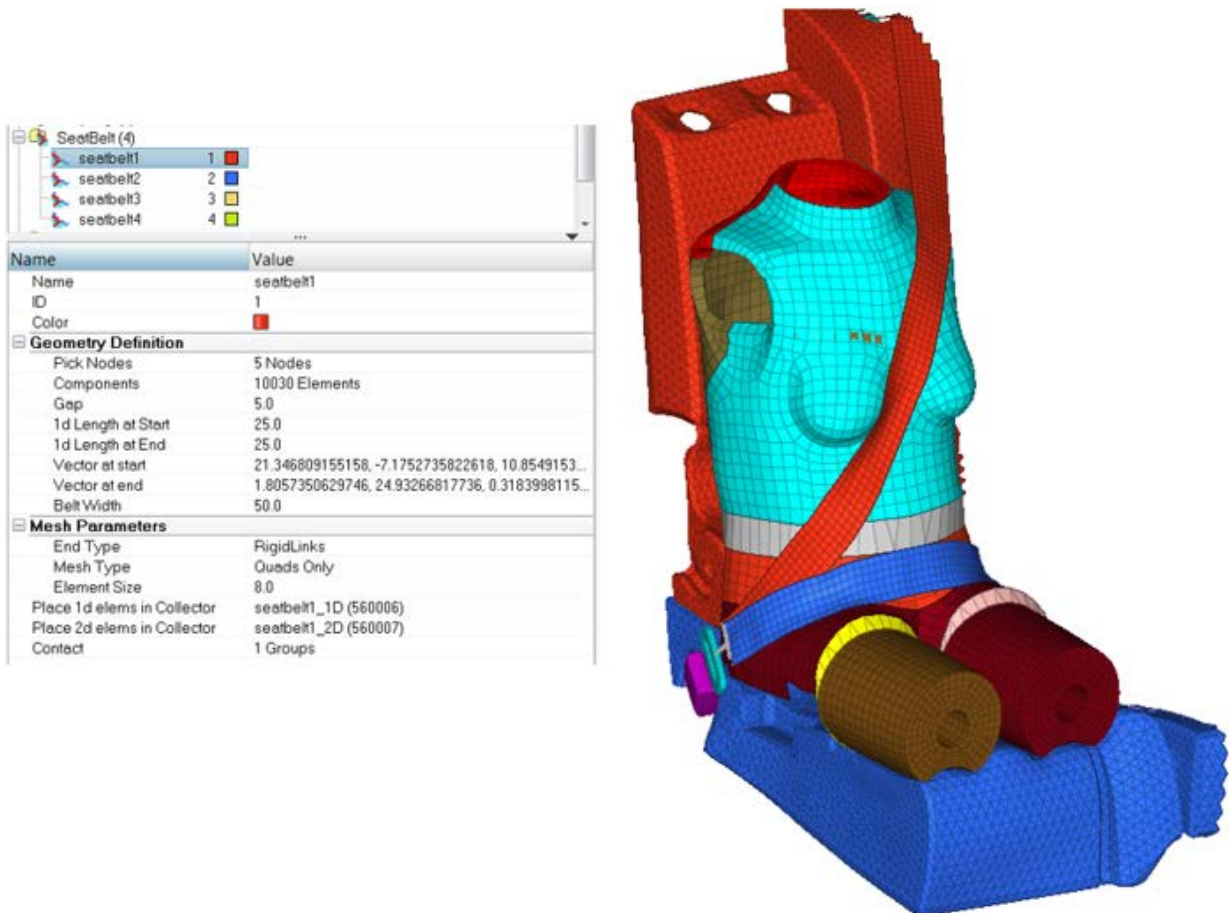
# Crash 14.0 Release Notes

## Highlights

This release includes several improvements for the Crash interfaces. A new seat belt entity is introduced that allows you to easily create and modify a seatbelt during the session. For the Pedestrian Impact Tool, EuroNcap protocol 7 and 8 are now supported.

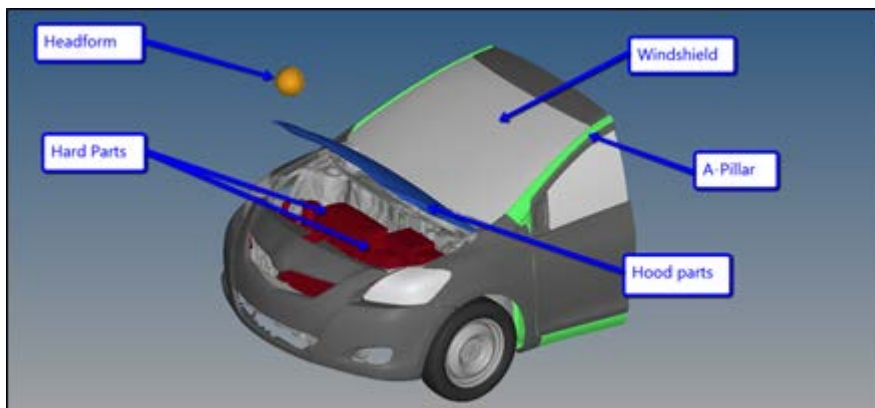
### ▼ Seatbelt Routing

- The seatbelt entity is introduced. This entity allows you to update the seatbelt generated during the HyperMesh session.
- A new algorithm enables you to define a belt path with more than 3 nodes.
- The position of the nodes that define the belt path can be changed modified interactively. This feature allows you to control the location and quality of the belt generated.
- Default material, property and contacts are automatically generated.



### ▼ Pedestrian Impact Tool

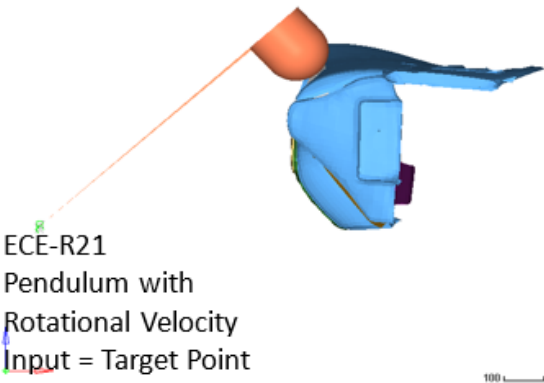
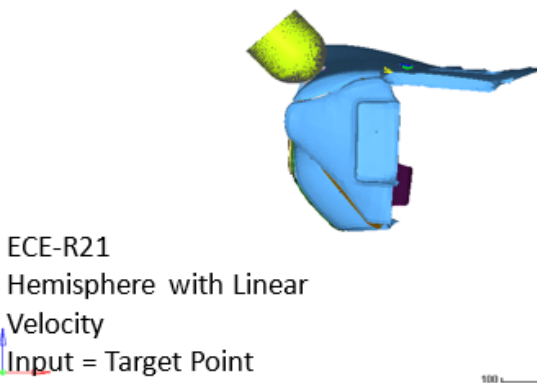
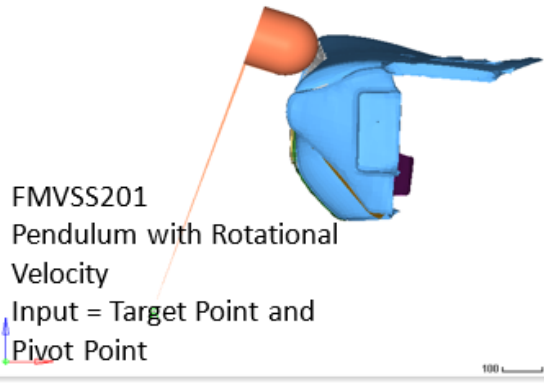
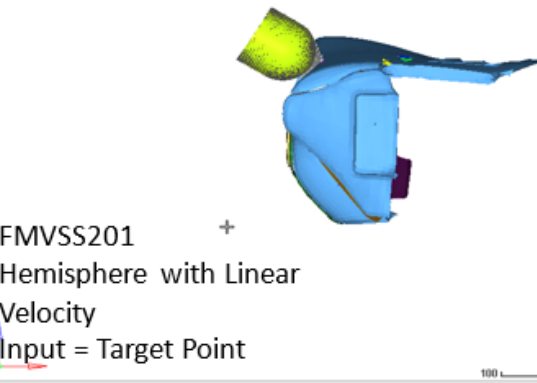
- EuroNcap protocol 7.0 and 8.0 are now supported.
- Upper and lower leg impact points are automatically generated
- Velocity of the upper leg impactor is computed automatically
  - Marking, positioning and export of adult and child points can be done at the same time
- In the “position impactor” step, the user can export transformation and velocity cards for selected target points (RADIOSS only)
- Reorganized the export to improve performance. The vehicle, contacts, transformations and impactor are exported in separate includes.
- A tutorial is now available to guide the user through the process (CRASH-2200: Pedestrian Head Impact Simulation)




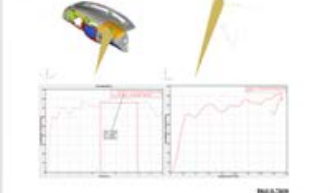
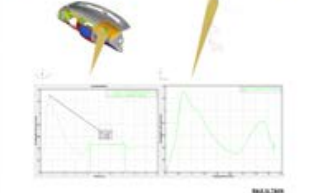
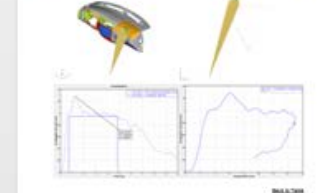
### ▼ FMVSS201/ECE-R21

#### Pre-Processing capabilities

- Added positioning for FMVSS201 and ECE-R21 supporting both hemisphere and Pendulum type headform with linear velocity and rotational velocity.
- Using SGrP, highlight zone marking
- Graphically pick impact target points or import via CSV file
- Visualize/review the positions
- Auto contact definition
- Velocity correction for ECE-R21
- Export files with Include structure
- CSV export/import

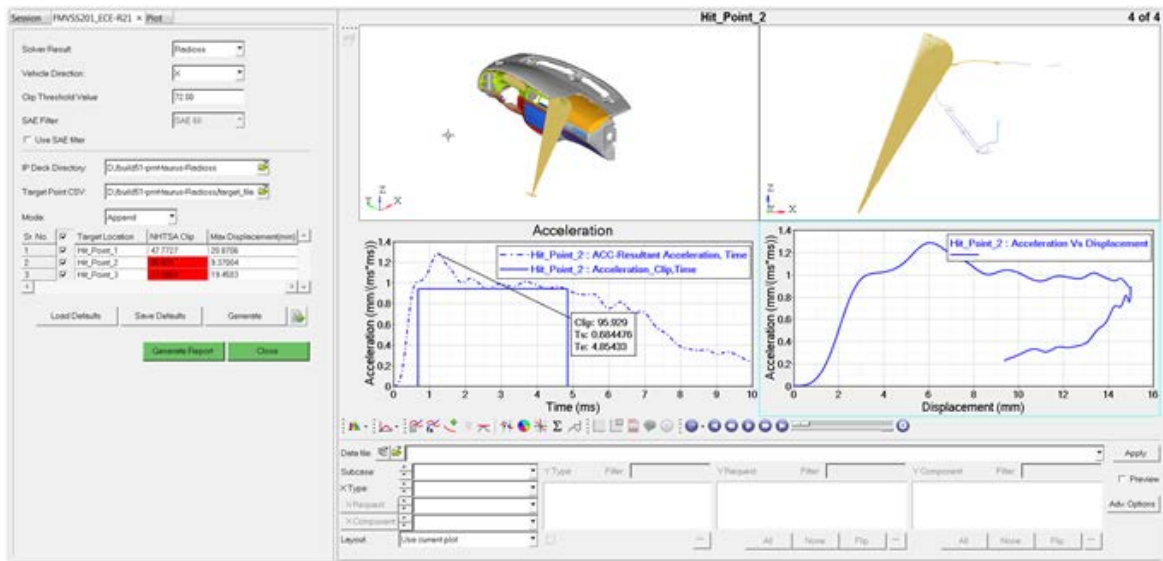
 <p>ECE-R21 Pendulum with Rotational Velocity Input = Target Point</p>	 <p>ECE-R21 Hemisphere with Linear Velocity Input = Target Point</p>
 <p>FMVSS201 Pendulum with Rotational Velocity Input = Target Point and Pivot Point</p>	 <p>FMVSS201 + Hemisphere with Linear Velocity Input = Target Point</p>

<p>Summary Report of FMVSS201/ECE-R21 Analysis</p>	<p>Project Details</p> <p>IP Model Image</p>  <p>Registration: FMVSS 201 Project: Vehicle Configuration: Work Request: Analysis Model: Number of Target Point: 3</p>	<p>IP Impact Results Table</p> <table border="1"> <thead> <tr> <th>Sl.No</th> <th>Target Location</th> <th>CLF Value</th> <th>Max. Displacement (mm)</th> </tr> </thead> <tbody> <tr> <td>1</td> <td>SH_Point_3</td> <td>71.0054</td> <td>10.4031</td> </tr> <tr> <td>2</td> <td>SH_Point_1</td> <td>41.7727</td> <td>20.6708</td> </tr> <tr> <td>3</td> <td>SH_Point_2</td> <td>69.4029</td> <td>6.3704</td> </tr> </tbody> </table>	Sl.No	Target Location	CLF Value	Max. Displacement (mm)	1	SH_Point_3	71.0054	10.4031	2	SH_Point_1	41.7727	20.6708	3	SH_Point_2	69.4029	6.3704
Sl.No	Target Location	CLF Value	Max. Displacement (mm)															
1	SH_Point_3	71.0054	10.4031															
2	SH_Point_1	41.7727	20.6708															
3	SH_Point_2	69.4029	6.3704															
<p>Summary at Hit_Point_3</p> 	<p>Summary at Hit_Point_1</p> 	<p>Summary at Hit_Point_2</p> 																

## Post-Processing capabilities

- Automatically post-process all hit points
- Table of clip values and Max displacement
- Color coding of clip values based on user defined threshold
- Sorting of results based on clip value or max displacement
- Easy navigation from table to plots
- Option to overlay different hit point
- Export to PowerPoint of standard report

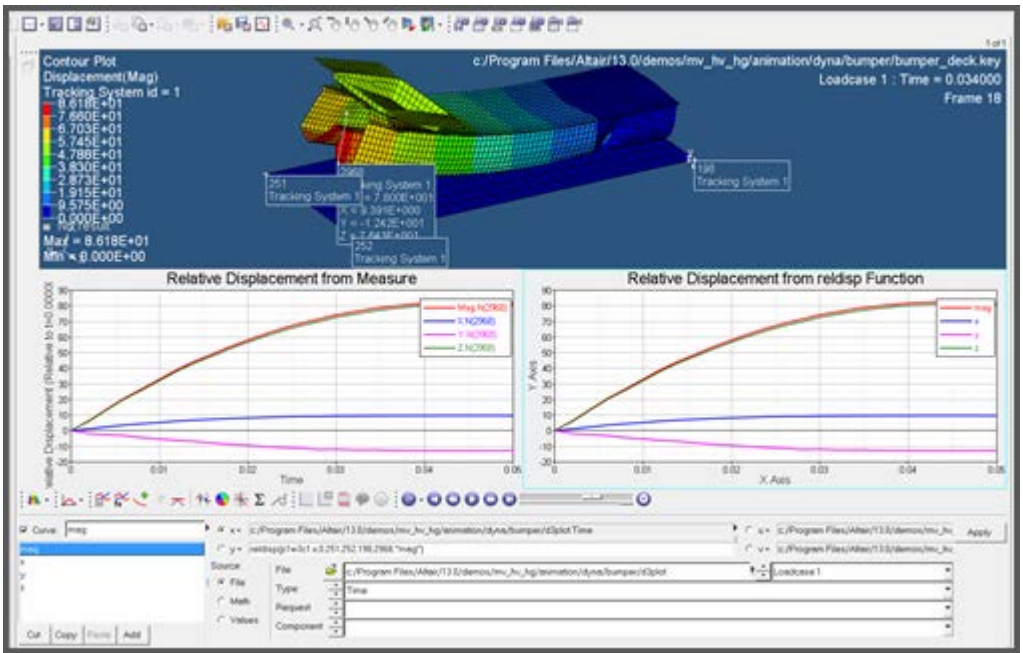


## Relative Displacement

A new function has been added to remove rigid body motion from time history results. Supported solvers include RADIOSS, LS-Dyna, and PamCrash. This tool is available under the Safety Tools menu when the Vehicle Safety Tools preference file is loaded in HyperGraph/HyperView. The function is very similar to plotting data from HyperView with tracking system on with Plane option.

Session	<b>Plot Relative Displacement</b>	Plot
File type:	Dyna - d3plot	
File name:	C:\Program Files\Altair	Load
Loaded files:	{C:\Program Files\Altair\13.0\demos\mv_h	
Ref coordinate system:		
Plane Type :	OXY	
Filter		
N1 :	251	N251
N2 :	252	N252
N3 :	198	N198
Result Request :		
Filter :	2968	
N2968		
Result component :		
Plot Window :		
New Plot		





▼ **Unit Support for Injury Functions**

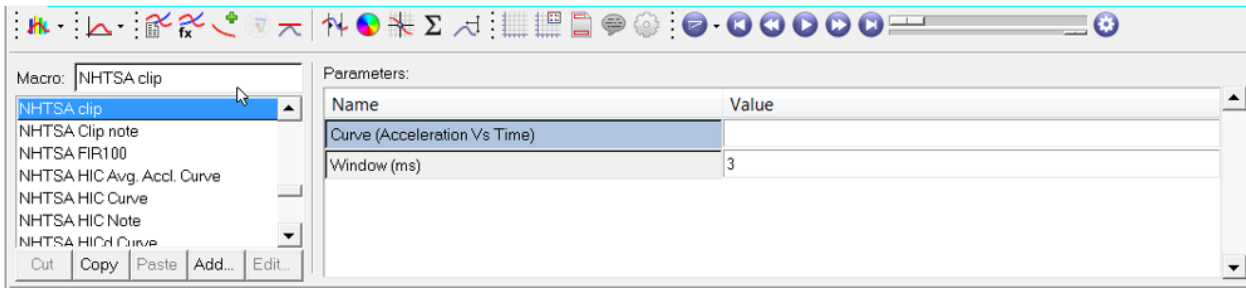
All the safety injury functions under the Injury drop down menu are now unit supported. There is no need for manually scaling the curves if units are enabled in HyperGraph.



Below is an example for Clip. Without units enabled, you need to manually define the scale factors for the curve since the expected units are g vs sec.

Name	Value
Curve (x=sec, y=g)	
Window (ms)	3
Time conv.	1.0
Accel conv.	1.0

With Units enabled for the session, HyperGraph will recognize the units for curve and appropriately scale it to give the proper injury result.







## **CAD and Solver Interfaces**

# Abaqus Interface 14.0 Release Notes

## Highlights

---

Transformation and Position entity is added to support \*NMAP. New elements related to fluid and temperature analysis have been added. Complete support for \*Parameter.

### ▼ \*NMAP keyword Support

- Transformation and Position entity is added to support \*NMAP keyword.
- Transformation entity can be used to define translation, rotation and scale operations.
- The position entity takes multiple transformations and applies to a NSET.
- Export option is toggle on by default and will remove transformation and export \*NMAP keyword to the solver deck.

### ▼ Support of \*PARAMETER

- Create, edit and reference of \*PARAMETER in other keywords using Entity Editor.
- \*Parameter can be imported/exported to solver input deck.
- The parameter defined in HyperMesh can be seamlessly imported into HyperStudy.
- Parameter can be a double, integer, string, double expression or integer expression.
- Parameter can be created from both the Model and Solver browser.

### ▼ Box Trim Utility

The Box Trim utility in the Abaqus user profile has undergone significant updates. It can now remove unused entities and automatically update entities which pass through the box trim – the objective is to ensure a run ready deck post box trim.

### ▼ Part Replacement

- Quickly replace a part with either an existing part in your model or a part from an external file.
- Property, material, contacts, loads, special elements and sets will be replaced to new component.

### ▼ New Element Type

- Fluid analysis elements FC3D4, FC3D5, FC3D6 and FC3D8 are supported.
- Thermal Analysis elements C3D4T and C3D6T are supported.

#### ▼ Table Entity

- COOR3D (Distribution\_Orientation) and Angle (Distribution\_Angle) card image is added to the \*Distribution keyword. The \*Orientation keyword is updated to select the table entity.
- \*Spectrum used in Response spectrum analysis. This keyword is mapped to table entity. \*response spectrum is updated to select the table entity.

#### ▼ Initial Condition for Temperature

Add new options **Gradient across shell Section** and **Define shell/Beam temperature points**.

- **Gradient across shell Section** can be used to define Initial temperature at multiple temperature points.
- **Define shell/Beam temperature points** can be used to define Initial condition as Temperature gradient across shell/beam section.

#### ▼ Fastener Improvement

\*Fastener can be define by creating element at the provided set of nodes location or solver can generate the elements. Prior to 14.0 only \*fastener with element was supported. Added support for mesh independent fastener.

#### ▼ Contact Comparison Tool

Instead of creating surface for \*Tie, solver can generate the master and slave based on the tolerance value. This tool can be used for visualization of these connection in HyperMesh. If the Nastran file is converted to Abaqus then you can retain the RBE and compered with the solver generated Tie.

#### ▼ New Keywords Supported

- \*Spectrum
- \*RESPONSE SPECTRUM
- \*SELECT EIGENMODES
- \*MODAL DAMPING
- \*FILTER option is supported for Field Output
- \*Parameter
- \*NMAP

### ▼ Resolved Issues

- Fixed a Tcl error in Abaqus Contact Manager Shell to Solid Coupling while selecting either master or slave entity with space in their name.
- When select an output block, its request types (ODB, fil, or dat) will be highlighted in step Manager. This avoid from users trying one by one to see to which option the output block belongs.
- Output Section Thickness “STH” and Plastic strain (PEEQ, PEEQT, PEMAG) is added for element output.
- Fixed the extra line that was added during export for \*EOS under \*Material.
- Support the import and export of \*ORIENTATION option defined in \*Kinematic coupling. This issue is exist if the \*kinematic coupling is defined on a set.
- Supported the option “Coupled slave nodes” for the keyword \*Contact Pair.
- Supported the option “COMBINE=DIFFERENCE” for the keyword \*Surface.
- Supported the option “ELEMENT DELETION=INSTANTANEOUS” for the keyword \*Section Controls.
- Supported the option “INITIAL GAP OPENING” for the keyword \*Section Controls.
- Supported the option “COHESIVE COMPRESSION DAMAGE” for the keyword \*Section Controls.

# ANSYS Interface 14.0 Release Notes

## Highlights

Enhancements have been made in the ANSYS interface to improve the modeling and visualization of models to solve.

- Mesh modeling enhancements
- Contact Manager enhancements which enable users to set ANSYS surface elements with ease and store Contact Manager settings for future use.

### ▼ Element Type Selection

When creating a mesh in previous versions of HyperMesh, you had to first select an element type of the appropriate element configuration using the Element Type panel, then you had to attach an ET Type (sensor) card to the component. This repetitive process verified that the ET Type matched that of the element type selected in the Element Type panel.

In HyperMesh 14.0, you will only be required to attach the correct ET Type (sensor) to the component. The generated mesh will have the same element type as the component as long as the mesh configuration matches the element type. You will no longer be required to duplicate this information in the Element Type panel, therefore making the Element Type panel obsolete. This panel can still be used to review the element type created for a mesh.

To change the element type, you will need to change the ET Type attached to the component. As long as the mesh configuration matches the element type, the elements will be updated. This enhancement has made the Update button obsolete, therefore it has been removed from the Element Type panel.

The status bar can be used to verify that the component is defined correctly. If a component has valid elements but does not have an ET Type attached, the status bar will read, "Attach HM\_COMP card image and ET Type" when the component is selected in the Model or Solver browsers. The status bar will also display an error if the element configuration and the ET Type do not match.

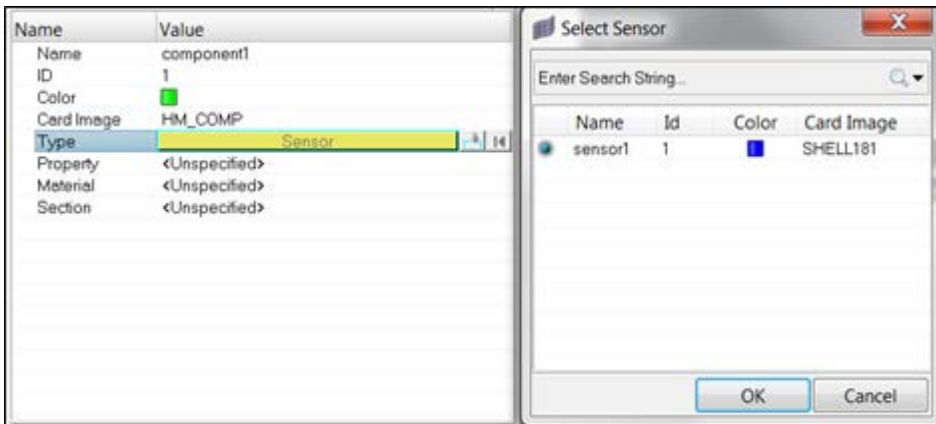
Enhancements have been made to make updating the mesh easier after any mesh updating operations have been performed, such as translating, copying, organizing, and so on. By making sure that the element type is the same as the ET Type attached to the component the mesh can be easily updated.

However, if a component does not have an ET Type attached or if the ET Type attached to the component is different from the configuration of the mesh being created, the mesh can still be created but the element type will be assigned the type set in the Element Type panel.

### New Element Type panel: Use this for review only



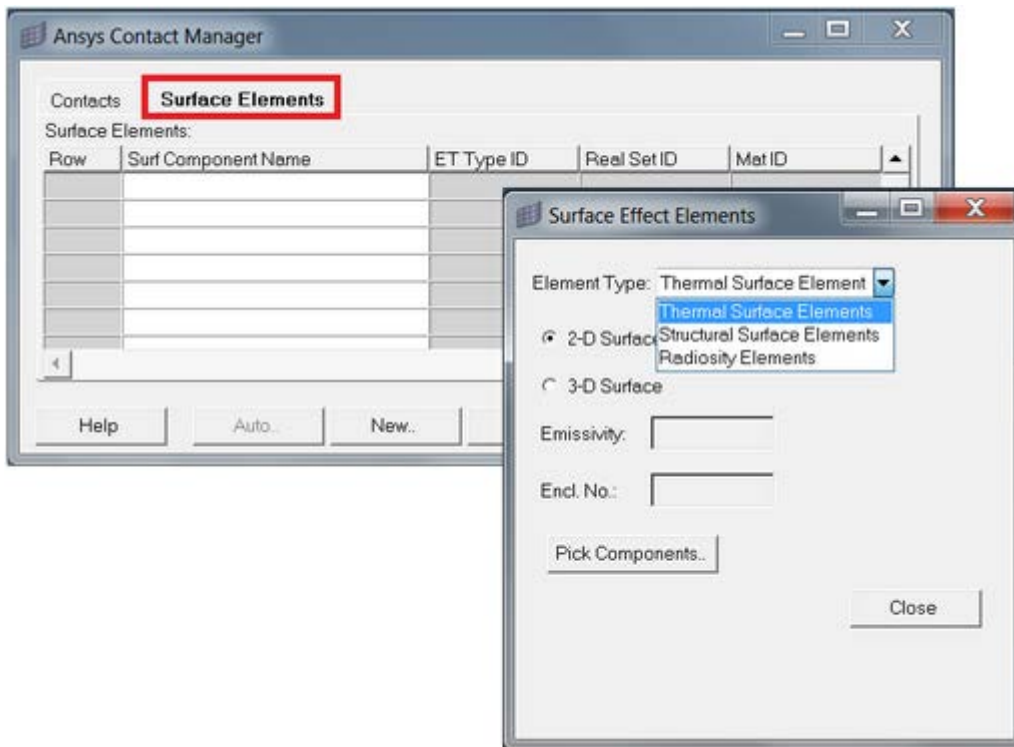
**Use Entity Editor > Component > Type to set the Element type**



**▼ ANSYS Contact Manager Enhancements**

Two new features have been added to the ANSYS Contact Manager.

1. The ANSYS Contact Manager can now be used to create and manage the ANSYS surface elements: SURF151, SURF152, SURF153, SURF154, SURF251, and SURF252.



The ANSYS Contact Manager has a new tab, Surface Elements, which enables users to review existing surface element components and create new surface element components of types: thermal surface elements, structural surface elements, and radiosity elements. 2D surface elements can be created on the edges of plane elements, and 3D surface elements can be created on the faces of solid elements. Extra nodes can also be attached if the surface element type selected supports the Attach Extra Node option.

This tool works similarly to the existing Contact Pairs tool in the Contact Manager. Users must select components and then select elements from the selected components to create surface elements on the faces or edges of the elements. Element key options, properties, and material properties can also be added in their respective tables that appear once the base elements have been selected.

In the ANSYS Contact Manager, an option has been added to enable users to save the Contact Element Key options, Real Sets (properties), and material settings they have defined for contact pairs. This enhancement will save time when creating new contact pairs with the same Key options, Real Sets (properties), and material settings.

Once the underlying elements of contact and target pairs have been selected, the Contact Manager prompts users to select applicable Key options, Real Sets (properties), and material settings. When finished, click Save Keyopts (Target Component Details dialog), Save Real Consts (Contact Property dialog), and Save Material (Contact Material dialog) to save the setting they have defined for future use. The next time a user creates a new contact pair, the settings they defined earlier are automatically applied. To restore the default settings, click Reset Defaults. All of the settings are saved in a configuration file located at:

```
<installation\hm\scripts\ansys>ContactManager\contactpair\SavingKeyoptionsAnsys.csv.
```

Users can edit this file directly if they choose.

#### ▼ Midsurface Mesh Tool in ANSYS Interface

The Midsurface extraction and meshing tool, located at Mesh > Create > Midsurf Mesh, is now fully supported for ANSYS interface with thickness assignment options. The thickness assignment options REAL SET and SECTYPE are only available for in the ANSYS user profile. Components, and properties on components are the available options as ANSYS interface only allows the indirect assignment of thickness.

The REAL SET option is available for legacy ANSYS elements such as SHELL63, where a thickness is assigned to elements.

For by REAL SET property and REAL SET, thickness from geometry is mapped to elements by creating a property card and assigning it a thickness. The property card will be assigned to the component of the element. The SECTYPE option can be used for the generation of new ANSYS shell elements such as SHELL181. Thickness from geometry is mapped to the Section card SECDATA with the type SHELL. The thickness value is mapped to the TK: thickness value in the SECDATA card, and creates only one layer. The section card is then assigned to the component of the element.

All of the other options perform the same as in the general HyperMesh profile.

#### ▼ Conversion of Beam Elements

If CBEAM is associated with PBEAM/PBEAML, then the properties within those cards are converted to ANSYS SECTION with the type BEAM. OptiStruct/Nastran beam cross sections of types: BAR, CHAN, HAT, I, T, ROD, and TUBE are converted to ANSYS equivalent beam sections.

CBEAMs are converted to ANSYS BEAM188 elements.

OptiStruct/Nastran Z and BOX sections are converted to ANSYS ASEC beam sections. CHAN, Z, and BOX sections are converted only if PBEAML has a HyperBeam section attached. PBEAM is converted to ANSYS ASEC Section if HyperBeam sections are not attached to the property card. OptiStruct NSM is converted to ANSYS ADDMAS.

Limitation: In this current version of the conversion tool, section offsets are not considered.

### ▼ Conversion of Shell Thickness: PSHELL and PSHEAR

In earlier releases, PSHELL and PSHEAR properties are converted to ANSYS SHELL63 (if elements are first order) and SHELL93 (if elements are second order). These elements are considered legacy elements, and are not used by most users. Therefore in version 14.0, PSHELL and PSHEAR properties are converted to ANSYS SECTIONS with section type SHELL. Thickness is now mapped to the TK value in the SECDATA cards of the SHELL SECTION. CTRIA3 and CQUAD4 elements are now converted to SHELL181 instead of SHELL63 and CTRIA6 and CQUAD8 elements are now converted to SHELL281 instead of SHELL93.

### ▼ Entity Editor Customization

In previous releases, the ANSYS Entity Editor contained customizations for the Et Type and Real Sets entities. In 14.0, the Entity Editor contains further customizations for the Material and SECTION entities. Tooltip help has been added for the material card MPDATA and BEAM SECTIONS. Each attribute in the material and MPDATA card has a tooltip help to indicate the type of material property and the type of input data. Tooltip help has been added to all types of sections supported under beam section collector. Tooltip help has also been added to other cards related to sections such as SECOFFSET, SECCONTROLS, SECLOCK.

### ▼ ANSYS Model Checker

The ANSYS model checker will now recognize contact element pairs that exist in the model. In previous versions, contact element pairs were reported under the duplicate elements error, which is incorrect.

### ▼ Support for Master Degrees of Freedom

The Master Degrees of Freedom used in reduced or Super element generation analysis are now read in HyperMesh as unsupported data, and will be written to the model file while exporting models from HyperMesh. Master Degrees of Freedom cannot be created in HyperMesh, currently they can only be imported and exported to prevent the loss of data.

### ▼ State Variable Result Type Support in HyperView

HyperView now supports the user defined state variable result type SVAR. These are special result types that are defined by the user using material subroutines. These results are now available in HyperView. If a result file already has these results, the users must select the read options Rad State variable Results – Yes before reading the result file. Once the result file is read in HyperView, users must select the Result type SVAR from the result type list and pick the variable number from the drop down menu below Result type. If elemental values are intersected, select the corner results option and set the Average option to none, or set it to advance if nodal average values are required. These results type are supported as scalar results in HyperView.



### ▼ Resolved Issues

- Issue with ANSYS Pretension Bolt utility where load orientation assignment was wrong when user uses 1D bolt option is fixed now.
- Order of entity sets is now maintained so that sets of sets (exported as CMGRP card) are exported at the end as these card point to the entity sets (exported as CMBLOCK) and these should be defined first in the model file. Also second or higher level of sets of sets are not supported in earlier release but in this release user can create, import and exports sets of sets up to 5 levels.
- All local system IDs were renumbered even though it was not required for the systems with IDs >10. Now renumbering occurs only on those IDs <10. The smallest available system ID will be assigned if it is created with ID <10.
- Issue with KEYOPTION definitions of BEAM44 element type where descriptions for KEYOPTS7 and KEYOPTS8 were wrong have been corrected.
- LSWRITE keyword issue where if removed from the model and reimported back to HyperMesh then the reader stops working is fixed.

# CAD Interfaces 14.0 Release Notes

## ▼ Updated CAD Version Support

The following CAD versions are newly supported

- CATIA v5-6R2015 (R25)
- Parasolid 27.0.172, for both import and export
- PTC Creo 2.0 M140 and 3.0 M040
- SolidWorks 2015 (2014 on MAC)
- UG NX 10.0. Support for NX 6.5, 7.0, 7.5 and 8.0 libraries has been removed from the native reader. However, older NX files can still be imported using any of the supported libraries (9.0, 9.5 or 10.0), or via the third-party reader.

## ▼ CATIA Publications

CATIA publications can now be imported as HyperMesh regions. The entities referenced by the publication are added as references to the region.

## ▼ Parasolid and NX Native Reader Robustness

Parasolid import and export, and NX import via the native reader, have been significantly improved via the use of native Parasolid libraries. The third-party Parasolid import/export available in previous releases is no longer in use.

## ▼ New Inspire Reader

HyperMesh now supports reading `.stmod` files from Inspire 2014 and earlier.

## ▼ Metadata Support

Additional metadata is now supported during import for ACIS, CATIA, JT, NX, Parasolid, ProE, SolidWorks and STEP formats. This metadata can be used for data transfer between CAD and CAE, as well as in process automation.

## ▼ CAD Export

CAD export has been enhanced for JT, Parasolid and STEP.

- The version of the exported file can now be chosen.
- Options for how the geometrical entities are named in the exported CAD file are now available.
- JT now supports assemblies both in "per part" and "monolithic" modes.

- The unit system is now more explicitly queried at export, to ensure proper selection by the user.

▼ **Resolved Issues**

- Addressed model specific defects and errors.

# EXODUS Interface 14.0 Release Notes

## Highlights

---

The EXODUS interface enables you to store a large amount of topological and result data in a binary format for finite element analysis.

In HyperMesh, the import and export of EXODUS files is supported in the EXODUS interface. An EXODUS file (\*.ex, \*.exo, \*.ex2) can be read using the Import tab, and the finished model can be written using the Export tab.

EXODUS supports the following types of entities, which can be created from the Solver browser:

- Nodes
- Element Blocks (Components)
- Node Sets (Sets)
- Side Sets (Contact Surfaces)
- Coordinate Frames (Systems)

# LS-DYNA Interface 14.0 Release Notes

## Highlights

---

This release includes several usability improvements for HyperMesh LS-DYNA interface: A new entity is introduced for Constrained Extra Nodes and SET of type COLLECT are now supported.

### ▼ New Entity for Constrained Extra Nodes

The constrained extra nodes entity is newly introduced in 14.0. The following keywords are mapped to this entity: \*CONSTRAINED\_EXTRA\_NODES\_NODE; \*CONSTRAINED\_EXTRA\_NODES\_SET.

These keywords were supported prior to 14.0 but were mapped to a different entity (group).

Upon import the following behavior is implemented:

- Solver deck: these keywords will be automatically mapped to the new entity without loss
- Legacy HyperMesh file: these keywords will be mapped from the old to the new entity without loss

### ▼ Model Checker Enhancement

A number of enhancement have been made to the Model Checker:

- Invalid definition of \*CONSTRAINED\_RIGID\_BODIES will be reported as an error
- Unresolved IDs reference in set and components are now reported as warnings
- Checking for untied nodes is faster and more accurate. In the right click contextual menu, added the possibility to create a set from the nodes found to be untied by HyperMesh

### ▼ Reader Enhancements

- Upon import of the keyword \*CONSTRAINED\_RIGID\_BODIES, HyperMesh will report a warning if the master and/or slave part ID are not defined.
- When a field is unsupported but the value in this field is null, HyperMesh will not report a warning anymore.

### ▼ Curve Editor

The Curve Editor now shows the curve ID and the X and Y scale factor applied on \*DEFINE\_CURVE. \*DEFINE\_TABLE is not listed in the Curve Editor anymore.

### ▼ Set Collect

Set of type collect are now supported. The concept of master and child set collect is introduced to allow multiple sets of type collect to share the same ID. The COLLECT option is currently supported for all variants of the following set type:

- \*SET\_NODE
- \*SET\_PART
- \*SET\_BEAM

### ▼ Support of ALIAS for LS-DYNA Materials

Supported ALIAS (three digits numerical designation) for material. Earlier these were listed as unsupported cards upon reading. Now HyperMesh can read these materials and map them to the corresponding descriptive designation. For examples:

- \*MAT\_001 is equivalent to \*MAT\_ELASTIC
- \*MAT\_009 is equivalent to \*MAT\_NULL
- \*MAT\_020 is equivalent to \*MAT\_RIGID
- \*MAT\_024 is equivalent to \*MAT\_PIECEWISE\_LINEAR\_PLASTICITY

### ▼ Newly Supported Keywords

- \*AIRBAG\_SHELL\_REFERENCE\_GEOMETRY (I/O only)
- \*CONTACT\_GUIDED\_CABLE
- \*DAMPING\_FREQUENCY\_RANGE\_DEFORM
- \*DATABASE\_MASSOUT
- \*DEFINE\_STOCHASTIC\_VARIATION
- \*ELEMENT\_SHELL\_COMPOSITE\_LONG
- \*EOS\_TABULATED\_COMPACTION (\*EOS)
- \*INITIAL\_VELOCITY\_GENERATION\_START\_TIME
- \*LOAD\_GRAVITY\_PART
- \*MAT\_ADD\_THERMAL\_EXPANSION
- \*PART\_COMPOSITE\_LONG
- \*MAT\_JOHNSON\_COOK\_STOCHASTIC
- \*MAT\_PLASTICITY\_WITH\_DAMAGE\_STOCHASTIC
- \*MAT\_TAILORED\_PROPERTIES (MAT\_251)
- \*MAT\_CONCRETE\_DAMAGE\_REL3 (\*MAT\_072R3)

## ▼ Updated Keywords

The following keywords were updated up to 971 R7:

- \*AIRBAG\_LINEAR\_FLUID
- \*AIRBAG\_PARTICLE
- \*AIRBAG\_WANG\_NEFSKE\_OPTIONS
- \*BOUNDARY\_FLUX\_SET
- \*CONSTRAINED\_GENERALIZED\_WELD
- \*CONSTRAINED\_RIGID\_BODIES
- \*CONTACT\_FORCE\_TRANSDUCER
- \*CONTACT (Optional card E)
- \*CONTACT\_SURFACE\_TO\_SURFACE
- \*CONTROL\_CONTACT
- \*CONTROL\_IMPLICIT\_DYNAMICS
- \*CONTROL\_THERMAL\_TIMESTEP
- \*DAMPING\_FREQUENCY\_RANGE
- \*DEFINE\_TRANSFORMATION
- \*INCLUDE\_STAMPED\_PART
- \*INITIAL\_STRESS\_SHELL
- \*LOAD\_BODY\_GENERALIZED
- \*MAT\_ELASTIC\_PLASTIC\_THERMAL (\*MAT\_004)
- \*MAT\_ELASTIC\_VISCOPLASTIC\_THERMAL (\*MAT\_106)
- \*MAT\_FABRIC (\*MAT\_34)
- \*MAT\_FORCE\_LIMITED (\*MAT\_029)
- \*MAT\_GENERAL\_NONLINEAR\_6DOF\_DISCRETE\_BEAM (\*MAT\_119)
- \*MAT\_GURSON (\*MAT\_120)
- \*MAT\_MODIFIED\_PIECEWISE\_LINEAR\_PLASTICITY (\*MAT\_123)
- \*MAT\_NONLINEAR\_ORTHOTROPIC (\*MAT\_40)
- \*MAT\_ORTHOTROPIC\_VISCOELASTIC (\*MAT\_086)
- \*MAT\_PIECEWISE\_LINEAR\_PLASTICITY (\*MAT\_24)
- \*MAT\_PLASTICITY\_COMPRESSION\_TENSION (\*MAT\_124)
- \*MAT\_RIGID (\*MAT\_20)
- \*MAT\_SAMP-1 (\*MAT\_187)
- \*MAT\_SCHWER\_MURRAY\_CAP\_MODEL (\*MAT\_145)
- \*MAT\_SIMPLIFIED\_RUBBER/FOAM (\*MAT\_181)

- \*MAT\_SPOTWELD (\*MAT\_100)

#### ▼ Resolved Issues

- The association between node/element ID and description in \*DATABASE\_HISTORY\_OPTION\_ID is maintained when some of these nodes/elements are deleted.
- HyperMesh was expecting the options in \*CONSTRAINED\_JOINT (FAILURE, LOCAL...) to be defined in a specific sequence. The keyword will now be imported irrespective of the input sequence.
- In the Export tab, an option allows you to control the export of the include file reference depending on the export status. This option does not apply to INCLUDE of type TRANSFORM.
- The local coordinate system defined in \*MAT\_RIGID was not imported correctly if the components of the vector (V1,V2,V3) were equal to 0 instead of being blank.
- The transformations were not applied in the correct sequence when both \*NODE\_TRANSFORM and \*INCLUDE\_TRANSFORM were used.
- Unresolved elements were not supported in \*DATABASE\_HISTORY leading to loss of data in I/O.
- Exporting a model with a large number of connectors was significantly slower than without.
- \*MECHANISM was exported before \*END.
- Resolved several I/O issues in the following keywords:
  - \*SENSOR\_CONTROL
  - \*SENSOR\_DEFINE\_CALC-MATH
  - \*MAT\_GENERAL\_SPRING\_DISCRETE\_BEAM
- The ID field was set to 8 instead of 10 for the following keywords:
  - \*LOAD\_SEGMENT
  - \*LOAD\_SHELL\_ELEMENT
- Curves referenced in loads (\*LOAD\_NODE\_POINT...) were not exported with the export "displayed" option.
- Legacy HyperMesh file:
  - Conversion of \*DEFINE\_VECTOR\_NODES was incorrect and lead to loss of data
  - Old HyperMesh file containing \*SET\_SHELL are converted to \*SET\_SHELL\_LIST
- Conversion from OptiStruct and Nastran to LS-DYNA:
  - Converting connectors linked to assemblies was causing an application error
  - SET ELEM are now converted to \*SET\_SHELL\_LIST instead of \*SET\_SHELL
  - Thickness visualization was not correct after conversion
  - Offset defined in QUAD4 elements was not considered
  - An application error occurred if there was empty components in the model



- When converting a model from LS-DYNA to RADIOSS, an application error occurred if there was “unresolved id” in the model.

# Nastran Interface 14.0 Release Notes

## ▼ Nastran User Profile Separation

Nastran user profile is now separated as MSC Nastran and NX Nastran.

## ▼ Symbolic Substitution

Symbolic substitution is supported as a parameter in MSC Nastran user profile for properties and materials.

## ▼ MSC Nastran 2013 Contact Cards

New MSC Nastran contacts cards and contact properties of version 2013 are now supported in the MSC Nastran user profile. The following cards have been added:

BCBODY1, BCONECT, BCTABL1, BCONPRG, BCONPRP and BCBDPRP

## ▼ MSC Nastran Sol 200 Cards

MSC Nastran Sol 200 cards are supported in MSC Nastran. The following cards have been added:

DVPREL1, DVPREL2, DVCREL1, DVCREL2, DVMREL1, DVMREL2, DCONADD, DCONSTR, DESSUB, DESGLB and DOPTPRM

## ▼ Newly Supported Keywords in MSC Nastran Profile

PACBAR

ERPPNL

ERP

RESTART

PARAM – SKINOUT, SS2GCR, VMOPT, MFLUID

SPCR

SPC1

BCPROP

BOLT

NONLINEAR

NOLIN1

NLSTRESS

PSHELL1

MONPNT

DAMPING

HYBADAMP

CTRLDEF

TABLES1 – TYPE 2

FFLUID in PSOLID card  
RROD  
INCL

#### ▼ General Enhancements

- The Part Replacement tool is now supported in both MSC and NX Nastran profiles.
- The Box Trim macro was enhanced to support solver specific changes in MSC and NX Nastran profiles.
- New options were added to the Solver option in the Import browser to read ANSA and PATRAN comments. The option is ON by default.
- A new check was added to the Model Checker to identify PSHELL & PSHLN1 referring to different MID's.
- A new Solver option in the Import browser to import models with ANSA and PATRAN comments; by default, option is ON.
- The Export template and Import reader were enhanced to support long format in entities.

#### ▼ HyperView Reader Enhancements

- Support for FEMZIP Nastran result files
- Nastran OP2 result file generated with option PARAM,POST,1 is supported
- Reader enhanced to support Composite Stress from NASTRAN 2012 generated OP2 files
- Support GPFORCE results in OP2 files for frequency response
- Support of 1D Element Forces (v) with Corner Data in XDB files
- Support of CSHEAR and CBEAM results as corner data in XDB files
- Support of ANSA and PATRAN comments Nastran Model reader options; by default, option is ON

#### ▼ Resolved Issues

- Nastran template to export CQUAD elements in sequential order.
- BSURF export and import issue.
- Performance issue while running 'unused sets' check.
- Mass was lost while importing CMASS4; mass is assigned to CMASS4 now.
- Blank line export issue with BCTABLE card.
- Nastran template to export IC card for SOL112.
- Abaqus > Nastran solver conversion resolved issues.
  - Performance improvement while converting models with beam elements.
  - Error in DOF mapping of SPRING1 to CELAS1 element type is resolved.
- Import reader issues while Nastran files with PATRAN comments.

- Performance issue related to updating materials to multiple PCOMP property.
- PCONVM is now listed in solver browser.
- BCTABLE is now read correctly.
- HyperView reader to display composite stress and strain from Nastran2012 OP2 file.

# PAM CRASH 2G Interface 14.0 Release Notes

## Highlights

---

This release includes several usability improvements: Entity Editor is now available, allowing users to create and edit options for keywords with filtered selection. Pamcrash2G2012 is now supported.

### ▼ Entity Editor

- Create and edit options are now available for all keywords mapped to named HyperMesh entities, such as component, property, material, contact, rigid wall, control volume, load collectors, system collectors, vector collectors, sets and sensors.
- New Entity Selection browser is introduced to select existing entities in the current session with options to filter based on name, ID, card image and so on.
- Entity Selection browser is auto filtered to show entities of valid solver keywords for selection to create error free solver keywords. This is on by default.
- Show/hide/isolate/Xref functions are available for each referenced entity in the Entity Editor for quick graphics review.
- Provided tooltips for the attributes in the Entity Editor.

### ▼ Support of Pamcrash2G2012

A new Pamcrash2G2012 user profile is now available. Pamcrash 2002-2008 templates are available in the installation and can be loaded; however they are no longer maintained.

### ▼ Support of SECURE

I/O support is provided for SECURE PART, MATER, PLY and FUNCTION.

### ▼ Transformation Enhancements

Transformation are now automatically applied during the import and disappplied at export. I/O support is provided for NPOS.

### ▼ Newly Supported Keywords

The following keywords were supported in Pamcrash 2012:

- Contact type 154
- ICTRL
- FPM turbulence model
- LCTRL

- NSMAS2

#### ▼ Updated Keywords

The following keywords were updated up to Pamcrash 2012:

Control Cards:

- EIGEN
- ECTRL
- OCTRL
- FPCTRL
- SPCTRL
- TCTRL

Others:

- BAGIN (FPM\_OPTION with VENT\_MODEL)
- Contact type: 33, 34, 36, 37, 46, 54
- Material type: 143 and 151
- MTOCO
- Ply model type 0 and 7
- RUPMO model type 7
- SENPT
- THLOC

#### ▼ Resolved Issues

- Importing a model with OTMCO without WEIGHT card defined caused the reader to crash.
- Reading of unsupported control cards caused the reader to crash. The unsupported control cards will now be skipped and a warning message will be reported.
- When a node is referred in an elements but not defined in the model, this node is skipped and an error is message is reported.
- Resolved I/O issues for the following keywords:
  - CONLO
  - CNTAC Type 36
  - MATER Type 24, 25, 62, 116, 161, 214, 222, 225, 302, 304,
  - MTOJNT
  - PART of type SHELL
- Added `.inc` as a valid extension in the Import and Export tab.

## RADIOSS Interface 14.0 Release Notes

### ▼ New User Profile

New user profile RADIOSS Block 140 is now available.

### ▼ Parameter Support

/PARAMETER is now supported for RADIOSS 12.0 and later.

- Create, edit and reference /PARAMETER in other keywords with the Entity Editor.
- Parameter type REAL, INTEGER, REAL\_EXPR, INT\_EXPR is supported. Parameter type TEXT is not supported.
- Parameter expressions are evaluated at import and are updated during the session.
- Parameters of type EXPR are created upon import of parameter references with a minus sign (-) before the prefix "&".
- All FE entities can be parametrized except loads, legal values, elements and node coordinates.
- Parameterization of entity ID and attribute of type entity is 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.

### ▼ Newly Supported Keywords

Supported up to RADIOSS Block 130, 140:

- /FAIL/NXT
- /UPWIND (Engine)

Supported as table entity in the Model browser in RADIOSS Block 110, 120, 130, 140

- /INIBRI/STRA\_F
- /INIBRI/STRESS
- /INIBRI/STRS\_F
- /PARAMETER/GLOBAL/REAL
- /PARAMETER/GLOBAL/INTEGER
- /PARAMETER/GLOBAL/REAL\_EXPR
- /PARAMETER/GLOBAL/INT\_EXPR

Supported in RADIOSS Block 130, 140

- /DT/THERM
- /FAIL/TAB1

- /MAT/LAW81
- /MAT/LAW83

Supported in RADIOSS Block 110, 120, 130, 140

- /ANIM/VECT/FREAC

#### ▼ Updated Keywords

The following keywords were updated up to RADIOSS Block 140:

- /FAIL/JOHNSON
- /IMPL/SOLVER
- /INISHE/STRS\_F/GLOB
- Interface type: 2,3,5,7,10,11,18,19,20 and 21
- Material law: 20, 51, 58, 77 and 80
- Property: SPH and type16

#### ▼ Resolved Issues

- PCOMPG are converted to /PROP/SH\_SANDW when converting from OptiStruct to RADIOSS.
- When converting a Nastran model to RADIOSS, an application error would occur if there were some unresolved IDs in the model to be converted.
- When converting from OptiStruct to RADIOSS, an application error would occur if there was a "\*" in the component name.
- GRNOD/GENE was not shown in Solver browser.
- Corrected the position of Einf and COE in /MAT/LAW78 under RADIOSS Block 130, 140.
- Solver conversion from OptiStruct to RADIOSS gives application error due to unresolved IDs.
- Added the blank line in /MAT/LAW51 for Iform=3 under RADIOSS Block 120, 130, 140.
- For the transformation of type translation, if the vector was defined with 2 nodes, the direction in which the displacement was applied in the opposite direction compared to the solver.
- Now Model browser's create submenu lists entities in alphabetical order.



## OptiStruct Interface 14.0 Release Notes

### ▼ Rotor Dynamics Cards

The cards required for Rotor Dynamics analysis in OptiStruct is now supported in the HyperMesh OptiStruct interface. The following cards have been added:

RGYRO, ROTORG, RSPINR, UNBALNC, PARAM\_WR3, PARAM\_WR4, PARAM\_GYOAVG and RSPEED

### ▼ Tooltip Help in OptiStruct Interface

Tooltip help is provided for OptiStruct cards in the Entity Editor to help you choose the right option. This avoids you from needing to open the full solver online help. View the Tooltip help by moving the cursor on the key options in element type or property options in property card.

### ▼ Lattice Optimization Card Support

A lattice optimization card is now supported in the Topology panel. Lattice control parameters are supported under a new subpanel with Lattice params. The following cards have been added:

LATTICE and all parameters in LATPRM.

### ▼ CONTACT Enhancements

- The following options are supported under CONTACT card: CLEARANCE, TRACK with SMALL and FINITE option, SMOOTH.
- PCONT card: Negative value (STIFF = Real < 0.0) to define stiffness scaling factor.

### ▼ SECTION Card Support

Defines a specific face of a cross-section of the model. The section is determined through a set of grid points and the particular face is specified by a set of elements.

The Resultant Section Force and Resultant Section Moment can be specified as responses via RTYPE=RESFORCE in DRESP1. The ATTA field specifies whether Resultant Section Force components (1,2,3) or Resultant Section Moment components (4,5,6) are selected. The ATTi fields should reference the Identification Numbers of the SECTION Bulk Data Entry that define the cross-section.

### ▼ Solver Conversion Enhancements

Abaqus > OptiStruct

- Enhanced conversion of TIE contacts
- Enhanced conversion of Contact Pair
- Enhanced conversion to map NLGEOM=YES to PARAM,LGDISP,1
- \*MATERIAL, ELASTIC = LAMINA to MAT8 conversion

- Multiple bug fixes

#### RADIOSS > OptiStruct

- Composite property and material conversion
- PROP/SH\_SANDW (TYPE11) is mapped to PCOMP for angle, thickness and material
- MAT/COMPISH (LAW25) is mapped to MAT9ORT

#### ▼ Axisymmetric Card Support

Axisymmetric elements, property and material are now supported in the OptiStruct interfaces. The following cards are supported:

CTAXI, CTRIAX6, PAXI and MAT3

#### ▼ Newly Supported Keywords

- PRETBOLT (OPTI) = YES/NO
- OUTPUT,H3DSENS
- CSEAM
- PSEAM
- CLEARANCE option in Contact card
- STIFF option in PCONT card
- FTYPE option in PCONV
- PARAM, EXPERTNL, AUTO
- NLHEAT
- POWERFLOW
- PARAM, EXCEXB
- PARAM, LFREQFL
- PARAM, HFREQFL
- PARAM, BUSHSTIF
- PARAM, ELASSTIF
- PARAM, BUSHTLMT
- PARAM, BUSHRLMT
- PARAM, ELASTLMT
- PARAM, ELASRLMT
- PARAM, MDK4OPT
- PARAM, GRDPNT
- PARAM, EXCEXB

- PARAM, FLLWER
- PARAM, EXTOUT, DMIGOP4, DMIGOP2
- PARAM, BUSHT
- PARAM, PELAST
- PARAM, HGHCMPK
- PARAM, XPOST
- PARAM, AMSE4CMS
- PARAM, CDITER
- PARAM, CDPRT
- PARAM, CDPCH
- PARAM, PRTRET
- PARAM, PRTDES
- SECTION
- RESFORCE option in DRESP1
- DGRIDB, DGRIDL options in DRESP2 and DRESP3
- CELLIN and CELLOUT options in DRESP3
- ADJUST=ALL option in Contact and TIE
- MFLUID
- CFAILURE
- ELIST
- PEAKOUT
- LATPRM
- LATTICE as continuation in DTPL
- SMOOTH and TRACK option in contact card

#### ▼ General Enhancements

- The Part Replacement tool is now supported.
- The Box Trim macro was enhanced to support solver specific changes in the OptiStruct interface.
- The Export template and Import reader have been enhanced to support long format in entities.
- OptiStruct compression run – New option Compress under run options in OptiStruct panel.
- Heat transfer SUBCASE reference in structural SUBCASE through TEMP card.
- Handling of OMIT card during import and export.

- New Solver option in the Import browser to import models with ANSA and PATRAN comments; by default, option is ON.
- LOCATION option support in STRAIN output.

#### ▼ **HyperView Reader Enhancements**

- Support of axisymmetric elements results from OptiStruct generated OP2 files.
- Reader enhanced to support Composite Stress from OptiStruct generated OP2 files.
- Support of ANSA and PATRAN comments OptiStruct model reader options; by default, option is ON.

#### ▼ **Resolved Issues**

- Export issue related to Export Displayed option not working correctly.
- Export issues with DRESP3.
- LS-DYNA to OptiStruct conversion errors.
- Abaqus to OptiStruct conversion errors.
- Correct BNDFIX1 format during export.
- Summary template can now calculate mass for PCOMP, PCOMPG and PCOMPP.
- OptiStruct template to export elements in sequential order.
- Export of DRESP1 for PSD and RMS stresses in conjunction with PBEAML.
- FATLOAD export to write as blank if TID is not unspecified.
- CBAR offset does not change now when importing the model.
- Op2 reader now reads stress results of CTRIAR elements in HyperView.
- Issues related to Contacts, LoadStep, Property and Material during conversion from Abaqus to OptiStruct are now fixed.
- Linear subcase is mapped correctly after conversion from Abaqus to OptiStruct.

# Samcef Interface 14.0 Release Notes

## ▼ .ZYG Macro Support

The macro command .ZYG is now supported as group (contact type) under the Interfaces panel. This command basically helps you to idealize a full scale model into a cyclic or periodic symmetry model. It glues a group of slave nodes of one end to a group of master facets at the other end.

In this release, you can also create .ZYG with an automatic mode. In this case no groups of slave nodes and of master facets are required.

## ▼ Pretension Manager Support

In this release, a Pretension Manager has been added which enables you to create and edit 1D and 3D bolt loads and bolt sections. Also, the Pretension Manager can be used to review the existing bolt loads and sections.

You can find it in the Tools menu.

## ▼ Reader Enhancements

Upon import of a Samcef deck, HyperMesh now recognizes keywords with both upper and lower case characters.

## ▼ Provision to Define Temperature Dependent Material Properties

In this release, the material card in HyperMesh allows you to attach a curve or plot to respective material attributes (For example, Young's modulus, Poisson's ratio etc.).

## ▼ Support of Parameter NEXCE in the Composite Definition

In this current release, you can specify distance as input for NEXCE for .ETA card as well. A similar parameter was supported under .PHP SHELL from the previous release. This allows you to define the offset distance of the composite element from its actual center.



## ▼ .HYP Macro Command is Written Out in a Compact Way

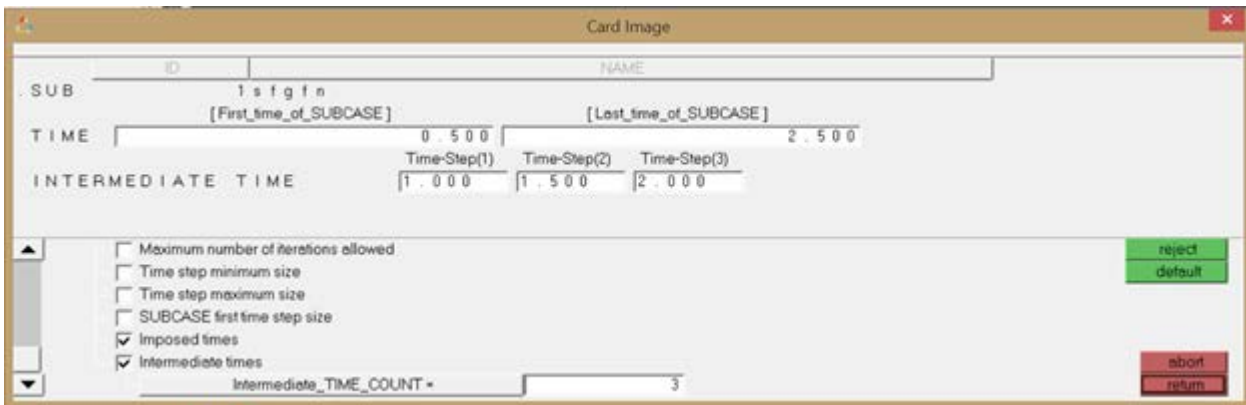
In the current release, both "I" and "J" parameters are used to write out the list of elements under the .HYP macro. These parameters write only the first and the last element number respectively.

This way of writing allows you to analyze and maintain the solver deck with less effort as it is very compact.

#### ▼ Enhancements in the TIME Parameter, Related to .SUB Macro Command

In the current release, while defining initial time and final time for a nonlinear analyses using parameter TIME under .SUB macro command, you can also specify real numbers along with the inter numbers. Previous releases used to take only integer numbers as input.

Also, a new parameter is added called INTERMEDIATE TIME, where you can specify time steps between the initial and the final time.



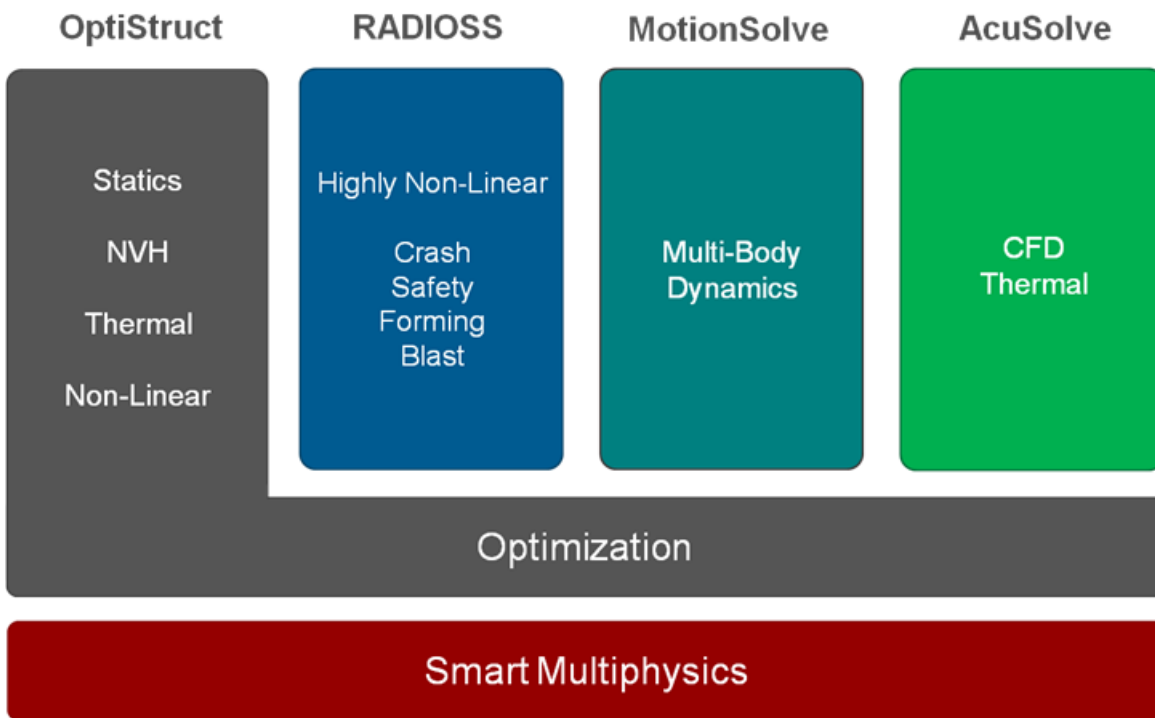
# HyperWorks Solvers 14.0 Release Notes

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.

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

With the 14.0 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 multibody dynamics with MotionSolve, and will also offer API to integrate additional solvers and solutions within a general multi-model framework.





# OptiStruct 14.0 Release Notes

## Highlights

---

Altair® OptiStruct® is a state of the art finite element solver for linear and nonlinear structural problems. It employs implicit integration schemes for static and dynamic problems. Besides mechanical loading, heat transfer coupled with structures is also available.

OptiStruct is designed with optimization at the core. The majority of solution sequences are available for optimization. A wide range of design problems can be solved addressing concept design and design fine tuning. In addition, RADIOSS and MotionSolve have been integrated to address multi-disciplinary optimization involving crash and impact, and multi-body systems, respectively. The optimization capabilities of OptiStruct are innovative and market-leading.

Analysis applications of OptiStruct include Automotive powertrain durability and vibrations, vehicle interior acoustics, vibrations of satellites, durability of heavy duty and off-road vehicles, component stress and vibrations analysis, detailed finite element analysis of airplane structures, random vibrations of ships and buildings, structural behavior of composite wings, buckling behavior, and many other advanced engineering applications.

Optimization applications of OptiStruct include material layout of structures and parts under static loads, sheet metal sizing under static and dynamic loads, improvement of acoustic behavior, design of parts for additive manufacturing, design of composite layups, and more.

The major items for the 14.0 release include:

- Finite Sliding Contact for Large Displacement Analysis
- Results at incremental load-steps for Nonlinear Analysis
- Fast Contact Analysis
- New Optimization Responses
- Enhanced Transfer Path Analysis
- Resolved Issues

## Stiffness, Strength and Stability

---

### ▼ **Nonlinear Analysis Results are now Available at Incremental Load-steps**

Small Displacement and Large Displacement Nonlinear Quasi-Static Analysis results are now available at incremental load-steps using the NLOUT bulk data entry and the NLOUT subcase information entry. The NINT and SVNONCNV parameters are available to control incremental output.

### ▼ **Finite Sliding Contact is now Available to Simulate Large Sliding at the Contact Interface**

Finite sliding contact is now available for Surface-to-Surface (S2S) contact to simulate large sliding at the contact interface. Finite sliding is activated by setting the TRACK field on the CONTACT entry to FINITE.

### ▼ **Large Displacement for 1D Elements is now Available**

Large displacement support for CROD, CBAR and CBEAM elements is now available. Note that Pin flags (PA and PB fields) for CBAR and CBEAM elements and Torsion for CROD element (J field on PROD entry) are not currently supported. 1D bolt pretensioning is now available for Large Displacement Analysis.

### ▼ **MPC-based Fast Contact is now Supported**

MPC-based Fast Contact is now supported for models where contact between parts is the only source of nonlinearity in the model. MPCs can now be used in conjunction with SUPORT, SPC entries and PARAM,CDITER to activate MPC-based fast contact.

### ▼ **CONTACT/CGAP(G)-based Fast Contact Analysis is now Available**

Fast Contact Analysis is now available for Node-to-Surface (N2S) contact and for CGAP/CGAPG elements. This is supported for models without material nonlinearity, or friction. Large displacement nonlinear analysis is not supported. PARAM,FASTCONT can be used to activate CONTACT/CGAP(G)-based fast-contact analysis.

### ▼ **Follower Loads are now Supported in Large Displacement Nonlinear Analysis**

Follower pressures and forces are now supported in Large Displacement Nonlinear Analysis. Pressures and forces are now dependent on the deformation with regard to both direction and area (pressure). The FLLWER bulk data and FLLWER subcase information entries are now available to activate and define parameters to control follower loading. PARAM, FLLWER is also available.

### ▼ **Nonlinear Convergence Parameters can now be Specified for Large Displacement Analysis**

Nonlinear adaptive convergence and time-stepping parameters can now be controlled for Large Displacement Analysis using the NLADAPT bulk data and NLADAPT subcase Entry. The NCUTS, DTMAX, DTMIN, NOPCL, NSTSL, and EXTRA parameters are currently available for nonlinear convergence control.

▼ **Time Dependent Loading Input for Nonlinear Quasi-Static Analysis**

Time dependent loading for Nonlinear Quasi-Static Analysis is available through the TLOAD1 and TLOAD2 bulk data entries with table data. The DLOAD subcase entry can be used in an NLSTAT subcase to reference TLOAD1/TLOAD2 bulk data entries.

▼ **Shell-to-Shell and Shell-to-Solid Element Support is now Available for Global-Local Modeling**

Shell-to-Shell and Shell-to-Solid Element support is now available for Global-Local modeling. Both the local and global models can contain shell elements. They can also be a combination of shell and solid elements.

▼ **Acceleration Loads**

Acceleration loads can now be applied to models using ACCEL, ACCEL1, and ACCEL2 bulk data entries. The ACCEL bulk data entry can be used to apply acceleration loads based on location. ACCEL1 bulk data entry can be used to apply acceleration loads on specific GRID points by specifying the particular GRID IDs. ACCEL2 can be used to apply acceleration loads to SETs of GRIDs in the model.

▼ **Rigid Bodies can now be Defined**

Rigid bodies can now be defined as element set, grid set, or surface using the RBODY bulk data entry. MASS and INERTIA continuation lines are also available for mass and inertia properties for the rigid bodies. The RBADD bulk data entry can be used to define a rigid body as a combination of rigid bodies defined by RBODY.

▼ **Contact Results in OP2 File**

Contact results, including Contact Force (v), Contact Pressure (s), and Contact Traction (v) are now available in the OP2 file using the CONTF bulk data entry.

▼ **Contact Area Output is now Available**

Contact area output is now available via the CONTF bulk data entry with the OPTI format option. This is supported only for surface to surface (S2S) contact. The contact area results are output to the .cntf file.

▼ **Beam and Bar Elements Results are now Supported in Random Response Analysis**

Power Spectral Density (PSDF) and Root Mean Square (RMS) element stress, strain, and force results are now available for Random Response Analysis in H3D and OP2 formats for CBEAM and CBAR elements.

▼ **STATSUB(BUCKLING/PRELOAD) can now Reference Small Displacement NLSTAT Subcases with S2S Contact**

STATSUB (BUCKLING/PRELOAD) can now reference small displacement NLSTAT subcases in models containing S2S contact. Previously, only N2S contact and linear static subcase references were supported.

▼ **Arruda-Boyce Model for Hyperelastic Material**

The Arruda-Boyce Model (Model=ABOYCE) is now supported for Hyperelastic materials on the MATHE bulk data entry. Additionally, reduced forms of the Generalized Mooney Rivlin material model (Model=MOONEY) are also separately available via Model= RPOLY/MOOR/NEOH/YEOH options.

▼ **Enhancements to Hyperelastic Materials**

The Poisson's Ratio (NU), Material Density (RHO), Coefficient of thermal expansion (TEXP), and Reference Temperature (TREF) fields have now been added to the MATHE bulk data entry.

▼ **Table Entries can now be Set to Have a Constant Y-Value Outside the Specified Range**

The FLAT field on each Table entry (TABLED1-3, TABLEM1-3, TABLES1, TABRND1, and TABDMP1 Bulk Data Entries) can be set to a non-zero value to activate the constant Y-value option.

▼ **Projection points GA and GB for CWELD and CGAPG are now Output in OP2 File**

The projection points GA and GB for CWELD and CGAPG are now available in the OP2 file.

▼ **New RIGID Entry to Select Rigid Element Processing Method**

The RIGID I/O Options entry is now available to select the rigid element processing method, LAGRAN (LaGrange multiplier method) or LGELIM (LaGrange multiplier method with constraint elimination). This is applicable to RBAR, RROD, RBE1, RBE2, RBE3, and MPC entries.

▼ **Stress and Strain Output for CYPRA Elements in Static Analysis**

Stress and Strain output is now available for CPYRA elements in Static Analysis in the OP2 file.

▼ **Control Output of .spcd Files**

OUTPUT, SPCD, YES/NO is now available to control the output of .spcd files. The default is YES.

▼ **Different Default for TFRAC on the CNTSTB Bulk Data Entry**

The default value of the tangential stabilization scaling factor with respect to the normal stabilization (TRFAC) on the CNTSTB bulk data entry has now been changed to 0.1.

▼ **Visualize S2S Contact Elements in HyperMesh**

When CONTPRM, CONTOUT, YES is specified, S2S contact elements (and large displacement N2S contact elements) will be output as PLOTEL+RBE3 into the <filename>\_contout.fem file for visualization. This filename\_contout.fem file can then be imported into HyperMesh on top of the base model to visualize the S2S contact.

▼ **SPC on RBE3 Dependent Node for Large Displacement Nonlinear Quasi-Static Analysis**

SPC is now allowed on the dependent nodes of RBE3's for Large Displacement Nonlinear Quasi-Static Analysis.

▼ **Dynamic Design Analysis Method (DDAM) is now Available**

Dynamic Design Analysis Method (DDAM) is now available.

▼ **Threshold-Based Output Options are now Available for Element Strain Energy (ESE), Stress (STRESS) and Temperature (THERMAL) Output**

Threshold-based output of results is now supported. The THRESH (absolute threshold), TOP (top "topn" results), and RTOP (top "rtop" percent of total results) options are now available for Element Strain Energy (ESE), Stress (STRESS) and Temperature (THERMAL) output.

▼ **Thermal Expansion of Rigid Elements is now Available**

Thermal expansion coefficient (ALPHA) on the RBE2, RBE3, RROD, and RBAR elements can now be specified.

▼ **Multiple OP2 Files can now be Requested for a Single Model**

The POST subcase information entry can be used to request subcase-specific output to separate files. The TOFILE, label option can be used to define a label to be appended to the model's filename in the generated subcase-specific output file. This is currently only supported for OP2 files. POST,TOFILE should always be used in conjunction with OUTPUT,OP2, FORMAT=OP2, or PARAM, POST,-1.

▼ **Temperature Load Output is now Available for Static Analysis**

Temperature load output is now available for Static Analysis via the OLOAD bulk data entry. Previously, only applied load vector output was supported.

▼ **FORCE Results are now Output for CBUSH and CELAS Elements for Transient Analysis to the H3D, OP2, and PCH Files**

FORCE results are now output for CBUSH, CELAS elements for Transient Analysis to the H3D, OP2, and PCH files.

▼ **RJOINT is now Available to Connect Coincident GRID Points**

RJOINT is now available to connect coincident GRID points by defining a rigid joint connection between them. Depending on the value of the CB field, the behavior of the rigid joint can be switched between hinges, universal joints, spherical joints and so on.

▼ **Corner Strains are now Available for Solid Elements**

Corner strains are now available for solid elements via the STRAIN(CORNER) entry. The "Corner Data" check box should be activated in HyperView to view corner results.

▼ **Grid Point Strains are now Available**

Grid point strains are now available for solid elements using the GPSTRAIN I/O Options entry. GPSTRAIN is currently only supported for Static Analysis.

▼ **Subcase-specific Random Response Analysis**

Subcase-Specific Random Response Analysis is now available. The results for Frequency Response Analysis and Random Response Analysis can now be decoupled. The ANALYSIS=RANDOM option can be used in a separate Random Response Analysis subcase and corresponding Random Response results can be requested in this separate subcase. The RANOM Subcase Information entry can be defined in this separate subcase pointing to RANDPS and RANDT1 entries.

▼ **Axial Deformation of GAP Elements are now Output**

Axial deformation of GAP elements are now output in the OP2 and PCH files.

▼ **Scaling Support for PBUSH Entry**

Scaling support for the values on the PBUSH entry is available via the PBUSHT entry. The KSCALE, BSCALE, GESCALE, and MSCALE flags are available on the TYPE field to scale the corresponding data (based on frequency) on the PBUSH entry.

▼ **MAT9 Entry is now Supported for Large Displacement Nonlinear Analysis**

MAT9 entry is now supported for Large Displacement Nonlinear Analysis.

▼ **The LOCATE Entry is now Available and RELOC is now Subcase Specific**

The LOCATE entry is now available and RELOC is now subcase specific. The LOCATE entry defines subcase-specific part location within the global-part. LOCATE bulk data entries are activated in each subcase by the LOCATE subcase entries. The LOCATE bulk data entries reference the identification number of a RELOC bulk data entry which defines the actual location of the part in the final model.

▼ **Linear Bushing Elements are now Supported in Large Displacement Nonlinear Analysis**

Linear bushing elements, via CBUSH and CBUSH1D, are now supported in Large Displacement Nonlinear Analysis. The SPRING continuation line on PBUSH1D is now not mandatory to use linear CBUSH1D elements in Large Displacement Nonlinear Analysis. Additionally, the TYPE=KN data on the PBUSHT entry is not mandatory to use linear CBUSH elements in Large Displacement Nonlinear Analysis.

▼ **Residual Energy Ratio (Epsilon) is now Always Output in the OUT File**

Residual Energy Ratio (Epsilon) is now always output in the OUT file. This now does not require the usage of GPFORCE. Epsilon is not available for a Large Displacement Analysis, Linear Gap and Fast Contact Analysis.

▼ **Contact Friendly Elements are now Automatically Switched ON When Applicable Within the Model**

Contact friendly elements (PARAM, CONFEL) are now automatically turned on when applicable within the model. The default is now PARAM, CONFEL, AUTO if the parameter is not specified in the model. This will activate contact friendly elements when 2nd order elements with N2S contact are present in the model.

▼ **Turn Off the Check for Negative and Large Compliance in Optimization**

PARAM, HGHCMPPOK is now available to turn off the check for negative and large compliance in optimization. This will bypass the ERROR in optimization runs even if the compliance is large or negative.

▼ **PBUSHT's and PELAST's in the Model can now be Turned ON/OFF**

Generalized spring and damper property (PBUSHT) and elastic spring property (PELAST) in the model can now be turned ON/OFF using PARAM, PBUSHT and PARAM, PELAST, respectively.

▼ **Large Displacement Nonlinear Quasi-Static Analysis can now be Subcase-Dependent**

Subcase-Dependent Large Displacement Quasi-Static Analysis can now be made subcase-dependent by using `NLPARM(LGDISP) = SID` in a specific subcase that selects a corresponding NLPARM bulk data entry.

#### ▼ **New Contact Stiffness Penalization Options are now Available for Nonlinear Contact Analysis**

The exponential and quadratic contact stiffness penalization options are now available on the PCONT entry for Nonlinear Contact Analysis. The STFEXP continuation line can be used to activate the exponential stiffness penalization and the STFQDR continuation line can be used to activate the quadratic stiffness penalization.

#### ▼ **Cylindrical Coordinate System is now Supported for Gravity Loading**

Cylindrical coordinate system is supported on the CID field of the GRAV bulk data entry.

#### ▼ **Multiaxial Fatigue Analysis is now Available**

Multiaxial Fatigue Analysis is now available. The MAXLFAT field on the FATPARAM bulk data entry can be set to YES to activate Multiaxial Fatigue Analysis. The MDMGMDL continuation line can be used to define multiaxial fatigue damage models on the DM1, DM2, and DM3 fields. The MATFAT bulk data entry has now been enhanced with parameters and coefficients for Multiaxial Fatigue Analysis.

#### ▼ **New and Enhanced Parameters**

- PARAM, CDITER: A new parameter, PARAM, CDITER can now be used to activate MPC-Based Fast Contact Analysis.
- PARAM, FASTCONT: A new parameter, PARAM, FASTCONT can now be used to activate CONTACT/CGAP(G) Based Fast Contact Analysis.
- PARAM, FLLWER: A new parameter, PARAM, FLLWER can be used to activate follower forces introduced by pressure loads (PLOAD4 entries) and concentrated forces (FORCE1/FORCE2 entries) in Large Displacement Nonlinear Analysis.
- CONTPRM, CONTOUT: A new parameter, CONTPRM, CONTOUT can be used to output S2S contact elements (and large displacement N2S) as PLOTEL+RBE3 elements to the `<filename>_contout.fem` file for visualization purposes.
- PARAM, CONFEL: The AUTO option is now available (and is the default) that turns on contact friendly elements automatically if 2nd order elements in N2S contact are present in the model.
- PARAM, HGHCMPOK: A new parameter, PARAM, HGHCMPOK can be used to turn off the check for negative or large compliance in optimization runs
- PARAM, PELAST: A new parameter, PARAM, PELAST can be used to turn on/off the PELAST entries in the model.
- PARAM, PBUSHT: A new parameter, PARAM, PBUSHT can be used to turn on/off the PBUSHT entries in the model.



### ▼ New and Enhanced I/O Options or Subcase Information Entries

- NLOUT: The new NLOUT entry can be used in the subcase information section to select incremental result output parameters used for Nonlinear Quasi-Static analysis.
- FLLWER: Used to activate follower loading in Large Displacement Nonlinear Analysis by referencing a FLLWER bulk data entry.
- NLADAPT: The NLADAPT Subcase Information entry can be used to reference an NLADAPT bulk data entry for nonlinear convergence control.
- DLOAD: The DLOAD Subcase Information entry can now be used to reference TLOAD1/TLOAD2 entries in NLSTAT subcases.
- RIGID: The RIGID I/O Options entry can now be used to select the rigid element processing method via the LAGRAN and LGELIM options.
- OUTPUT: The OUTPUT I/O Options entry has now been enhanced with the SPCD, YES/NO option to control the output of `.spcd` files.
- STRESS, ESE, and THERMAL: The RTHRESH, RTOP, and TOP fields have now been added to support threshold-based output.
- POST: The POST Subcase Information Entry has now been added to request subcase-specific output to separate files. The TOFILE option can be used to specify the label appended to the model name in the output files.
- OLOAD: The OLOAD I/O Options entry has now been enhanced with support for temperature load output.
- GPSTRAIN: The GPSTRAIN I/O Options entry has now been added with to output grid point strain results.
- ANALYSIS: The ANALYSIS Subcase Information entry has now been enhanced with subcase-specific random response analysis support. The RANDOM option is now available.
- NLPARM: The NLPARM Subcase Information entry has now been enhanced with the LGDISP option to allow subcase-specific Large Displacement Nonlinear Quasi-Static Analysis activation.

### ▼ New Bulk Data Entries

- NLOUT: The NLOUT bulk data entry defines incremental result output parameters for Nonlinear Quasi-Static Analysis.
- FLLWER: The FLLWER bulk data entry can be used to define parameters for follower loading.
- NLADAPT: The NLADAPT bulk data entry can be used to define parameters for nonlinear adaptive control.
- ACCEL, ACCEL1, and ACCEL2: The ACCEL, ACCEL1, and ACCEL2 bulk data entries can be used to define acceleration loads.
- RBODY and RBADD: The RBODY bulk data entry can be used to define a rigid body. The RBADD entry can be used to define a rigid body as a combination of rigid bodies specified using RBODYS.

- RJOINT: The RJOINT bulk data entry can be used to define a rigid connection between coincident grid points.
- LOCATE: The LOCATE bulk data entry can be used to define subcase-specific part locations within the global part.

#### ▼ Enhanced Bulk Data Entries

- CONTACT:
  - The TRACK field has been added to activate finite sliding.
- MATHE:
  - The Model=ABOYCE option can now be used to define an Arruda-Boyce material model for hyperelastic materials.
  - The Model= RPOLY/MOOR/NEOH/YEOH options are now available to directly define subsets of the Generalized Mooney-Rivlin material model.
  - The Poisson's Ratio (NU), Density (RHO), Coefficient of thermal expansion (TEXP), and the Reference temperature (TREF) fields have been added to the MATHE entry.
- TABLED1-3, TABLEM1-3, TABLES1, TABRND1, and TABDMP1:
  - The FLAT field can now be used to define zero Y-values for X-values outside the range of the table.
- TLOAD1 and TLOAD2:
  - The TLOAD1 and TLOAD2 entries can now be used to specify time-dependent loading in NLSTAT subcases via the DLOAD subcase entry.
- RBE2, RBE3, RBAR, and RROD:
  - The ALPHA field is now available to specify the thermal expansion coefficient for rigid elements.
- PBUSHT:
  - The KSCALE, BSCALE, GESCALE, and MSCALE flags are available on the TYPE field to scale the corresponding entries on the PBUSH entry.
- PCONT:
  - The STFEXP and STFQDR continuation lines and corresponding field parameters are now available to activate exponential and quadratic stiffness penalization options respectively.
- GRAV:
  - The CID field can now reference a cylindrical coordinate system.
- FATPARM:
  - The MAXLFAT field can now be set to YES to activate Multiaxial Fatigue Analysis.
  - The MDMGMDL continuation line can be used to specify multiaxial fatigue damage models.
- MATFAT:
  - The MATFAT bulk data entry has been enhanced with parameters and coefficients for Multiaxial Fatigue Analysis.

## Noise and Vibrations

---

### ▼ Dry Modes are now Output to the H3D File

Dry Modes are now output to the H3D file when MFLUID is used with PARAM, VMOPT,2.

### ▼ ERPREFDB and SPLREFDB Values are now Calculated Based on the UNITS Entry

The values of ERPREFDB and SPLREFDB are calculated based on the UNITS entry unless PARAM, ERPREFDB or PARAM, SPLREFDB are specified explicitly. When the UNITS entry and the PARAM's are not present then the values are set equal to 1.0.

### ▼ Panel-Wise Radiation Loss Factors

Radiation loss factors used in the equivalent radiated power (ERP) calculation can now be specified on the PANELG bulk data entry through the ERPRLF field. A real value can be specified to directly define the loss factor for all frequencies, or an integer value can be specified to reference a TABLEDi entry that defines the loss factor vs frequency table.

### ▼ Enhanced Automated Transfer Path Analysis

The Automated Transfer Path Analysis process is now enhanced with stability and accuracy improvements, SPCD support, and Direct Frequency Response Analysis support. The grids on the control volume are automatically output to the ASCII file.

### ▼ New Parameter PARAM, MDK4OPT is now Available

If MDK4OPT=1, the most common value of GE for structural elements may be internally set equal to PARAM, G to speed up the calculation of material damping. The traditional material damping calculation method is used, if MDK4OPT is set to 0.

### ▼ RVDOF and RVDOF1 Entries are now Available for Residual Vector Generation

RVDOF and RVDOF1 bulk data entries can now be used to specify degrees of freedom where the residual vectors are to be calculated to enhance the modal space.

### ▼ Preloading is now Supported for Modal Frequency Response Analysis and Direct/Modal Transient Response Analysis

Preloading is now supported and can be applied to Modal Frequency Response Analysis and Direct/Modal Transient Response Analysis subcases by specifying STATSUB(PRELOAD).

▼ **Modal Frequency Response Solution has been Enhanced**

Modal Frequency Response solution has been enhanced with the support of long integer pointer usage to utilize all available memory to run large models.

▼ **The Performance of the Craig Bampton (CB) Method has been Enhanced for CMS Analysis**

The performance of the Craig Bampton (CB) method has been enhanced for Component Mode Synthesis (CMS) Analysis.

▼ **Normal Displacement, Normal Velocity, and Normal Acceleration Outputs are Now Available**

Normal Displacement, Normal Velocity, and Normal Acceleration outputs are now available for Frequency Response Analysis. "Normal" indicates that the direction of output is normal to the surface on which a grid is present. DISPLACEMENT(NORMAL), VELOCITY(NORMAL), and ACCELERATION(NORMAL) entries can be used to request the corresponding normal results. Normal results are automatically only output for grids on the surface of the model or given grid set.

▼ **Grids from the Second Round of AUTOSPC are Written to a CMF File**

Grids from the 2nd Round of AUTOSPC is output to the `.HM.autospc.cmf` file. If it is an optimization run, this is executed only in the first iteration.

▼ **Non-Structural Damping is now Available**

Non-Structural Damping is now available via the NSGE, NSGE1, and NSGEADD bulk data entries which are referenced by an NSGE subcase information entry. The NSGE and NSGE1 bulk data entries define non-structural damping per unit area or per unit length for a list of elements, properties, or element sets. The NSGEADD entry defines non-structural damping as a sum of defined NSGE and/or NSGE1 entries.

▼ **Specified CBUSH Elements can be Included/Excluded from Residual Vector Calculations**

Specified CBUSH elements can be included or excluded from Residual Vector calculations using the VISC\_INC and VISC\_EXC options, respectively on the RESVEC subcase information entry. The VISC\_INC and VISC\_EXC options can reference CBUSH element set identification numbers for inclusion or exclusion.

▼ **RESVEC Calculation for Static Loading Using LANCZOS**

Residual vectors are now calculated when static loads are used in Lanczos run.

▼ **New Parameter Data**

- PARAM, MDK4OPT: This new parameter can be used to control the material damping calculation when different material damping values (GE on the MATx entry) are specified for different regions in a model.

#### ▼ New and Enhanced I/O Options or Subcase Information Entries

- DISPLACEMENT, VELOCITY, and ACCELERATION: The NORMAL option is now available to output normal results at surface grid points.
- NSGE: The new NSGE Subcase Information entry is now available to reference NSGE, NSGE1, or NSGEADD bulk data entries for the application of non-structural damping.
- RESVEC: The new VISC\_INC and VISC\_EXC options are now available to include or exclude CBUSH elements from residual vector calculations, respectively.

#### ▼ New and Enhanced Bulk Data Entries

- PANELG:
  - The ERPRLF field references the value of the radiation loss factor or the identification number of a TABLEDi entry to define radiation loss factor vs frequency.
- RVDOF and RVDOF1:
  - The new RVDOF and RVDOF1 bulk data entries can be used to specify the degrees of freedom at which residual vectors are to be calculated to improve the modal space.
- NSGE, NSGE1, and NSGEADD:
  - The new NSGE, NSGE1, and NSGEADD bulk data entries can be used to specify non-structural damping for elements, properties, and element sets.

## Kinematics and Dynamics

---

### ▼ Enhancements to EXB Output for AVL EXCITE

PARAM, EXCEXB has now been enhanced with selective invariant output options. The output of the all inertial invariants can now be turned off using PARAM,EXCEXB,SMOT. The output of only the second order inertial invariants is turned off using PARAM,EXCEXB,GMOTR. All invariants are included in the .exb file if PARAM,EXCEXB,GMOT is used. In addition, the performance is dramatically improved for the EXB generation run. GPSTRESS is now supported for solid elements in EXB generation and recovery runs.

### ▼ Enhanced Parameter Data

- PARAM, EXCEXB: The ability to control the output of inertial invariants is now available via the GMOT, SMOT, and GMOTR options.

## Thermal Analysis

---

### ▼ Transient Temperature at Multiple Time Steps can now be Applied in Thermal-Structure Coupling Analysis

Transient temperature at multiple time steps can now be applied in Thermal-Structure Coupling Analysis.

The TEMPERATURE subcase information entry has been enhanced by adding the HTIME = ALL (selects all time steps of a transient thermal subcase) field. To use this in Thermal-Structure Coupling Analysis, One Step Transient Thermal Stress Analysis can be performed. The TEMPERATURE(HTIME=ALL) entry should be specified in a Static Analysis subcase and point to a transient thermal subcase ID.

### ▼ Thermal Contact has now been Enhanced

The thermal conductivity drops to zero when the clearance value falls beyond the final value specified on the conductivity vs clearance table.

### ▼ Enhanced I/O Options and Subcase Information Data

- TEMPERATURE: A new HTIME argument is now available that can be set to ALL for One Step Transient Thermal Stress Analysis.

## General Updates

---

### ▼ Modal Stresses, Strains, and Displacements are now Output to the H3D File

Modal stresses, strains, and displacements are now output to the H3D file for Frequency Response and Transient Analysis with MODAL option on respective output request.

### ▼ The Path Character Limit on INCLUDE and ASSIGN Entries has been Increased

The filename and path character limit on INCLUDE and ASSIGN entries has now been increased to 600 characters.

### ▼ The Script Command Line Options are now Output to the STAT File

The command line options for the script are now output to the STAT file for a specific run.

### ▼ Full Length Subcase Label Support is now Available for PCH File Output

Full length subcase label support is now available for PCH file output.

### ▼ Distance Tolerance is now Available to Manage Duplicate GRIDs

Distance tolerance is now available to manage duplicate GRID points. The SYSSETTING, DUPGRTOL entry can be used to specify a distance value. If GRID points with the same ID are closer than the specified distance, they are considered to be equivalent. If GRID points with the same ID are farther than the specified distance, then OptiStruct errors out.

### ▼ The Number of Temporary Scratch Directories Specified using TMPDIR has been Increased

The maximum number of temporary scratch directories that can be specified using the TMPDIR I/O Options entry has been increased from 5 to 20.

### ▼ Repeated Subcases can now be Defined for Output Handling

The REPCASE Subcase entry can be used in the subcase section to indicate the start of a repeated subcase definition. REPCASE allows the user to define multiple output requests of the same type (with possibly different options) for the same subcase. A repeated subcase (REPCASE) should only contain output requests.

### ▼ New and Enhanced Subcase Information and I/O Option Entries

- SYSSETTING: A new setting, DUPGRTOL, is now available to control duplicate grids handling.



- REPCASE: A new subcase information entry, REPCASE, is now available to created repeated subcases for output handling.

## Optimization

---

### ▼ **Enhanced Stress Response for Topology and Free-Size Optimization is now Available via DRESP1**

Enhanced stress responses for Topology and Free-size Optimization is now available. Stress responses for Topology and Free-size Optimization can now be subcase-specific, component-based, and they are not only limited to von Mises stresses. The stress responses for Topology and Free-size Optimization can now be defined using the DRESP1 bulk data entry. Previously, stress responses for Topology and Free-size Optimization were only available via the STRESS continuation line on the DTPL bulk data entry. The new formulation significantly expands the functionality and performance of stress responses for Topology and Free-size Optimization.

### ▼ **Enhanced Free-Shape Optimization Results**

The “Grow or Shrink” result type is available. The shape is growing if the value is 1.0 and shrinking if the value is -1.0.

### ▼ **Output User Defined Response**

The DRESPONSE I/O options entry is now available, which outputs user-defined responses to the H3D file. This allows the post-processing of user defined responses (DRESP1, DRESP2, and DRESP3 bulk data entries) in HyperView and HyperGraph. All grid and element based responses (DRESP1) are supported, and additionally all such combinations (DRESP2 and DRESP3) are also available. The remaining responses are not supported since contouring them is not possible. Global responses are output in a separate global subcase (`_glob.h3d` file). DRESPONSE(EQUA) can be used to disable the output of DRESP1 responses.

### ▼ **Weighted Compliance for ESL-B2B Optimization**

The weighted compliance is now available for ESL-B2B Optimization. This is activated via DRESP1(WCOMP). Additionally, the optimization parameter WEIGHT is now supported for NLGEOM/IMPDYN/EXPDYN analysis.

### ▼ **3D Topology Optimization for RADIOSS-Optimization (RADOPT)**

- 3D topology optimization is now available for RADIOSS Optimization
- Composite manufacturing constraints are now available in Free-size Optimization for RADIOSS-Optimization (RADOPT)
- Size Optimization for composites is now available for RADIOSS-Optimization (RADOPT)

### ▼ **Free-size Optimization for Composites for ANALYSIS=NLGEOM/IMPDYN/EXPDYN Subcase**

Free-size Optimization for composites is now supported for NLGEOM, IMPDYN, and EXPDYN subcases.

#### ▼ **Control Entity Output in the OUT File for Optimization**

The output of retained responses to the `.out` file can be controlled using PARAM, PRTRET. The ALL option allows all retained responses to be printed, the NOSTR option excludes stress, strain, and force retained responses to the `.out` file, and NONE deactivates printing of retained responses. The output of design variables and design entities to the `.out` file is controlled by PARAM, PRTDES. The DVDP option prints both design variables and design entities to the `.out` file. The DV and DP options prints the design variables, and design entities (properties, materials, and elements), respectively to the `.out` file. The NONE option deactivates the printing of design variables and design entities to the `.out` file.

#### ▼ **Dang Van Criterion has been Enhanced**

The Dang Van Criterion has been enhanced on the FOS continuation line. If the value in Field 3 of FOS continuation line is an integer, it now references a TABLES1 entry. This is used to define the intersection points of the FOS curve, wherein, X-values are hydrostatic pressure and Y-values are microscopic shear. The units are defined on the MATFAT entry. The angle for the safe zone (5th field in FOS) is now available. If angle at a point in the domain is less than the defined angle, it is considered safe. The 6th field in FOS is the shear threshold for the safe zone. If microscopic shear stress is less than this value, it is considered safe.

#### ▼ **STACK Entries can now be Used in Topography Optimization**

STACK entries can now be referenced by DTPG bulk data entries in Topography Optimization.

#### ▼ **Smith, Watson, and Topper (SWT) Mean Stress Influence can be used for Optimization**

Smith, Watson, and Topper (SWT) mean stress influence can now be used in Fatigue Optimization.

#### ▼ **Soft Elements are Removed in OS Topology Optimization from the RADIOSS Nonlinear Run**

Soft elements are removed in the OS Topology Optimization run from the RADIOSS nonlinear run and reintroduced at the ESLM Linear Static Optimization run.

#### ▼ **Failed Elements from RADIOSS Nonlinear Optimization Run are now Skipped**

Failed elements from the RADIOSS Nonlinear Optimization run are now skipped when generating the equivalent static loads.

#### ▼ **A New Alternate Optimization Algorithm is now Available**

The DUAL2 Optimization algorithm is now available and can be activated using DOPTPRM, OPTMETH, DUAL2. It is recommended to try this algorithm if the DUAL algorithm fails (restarting the optimization job should suffice in such cases).

▼ **Frequency-Response Based Optimization Responses can now be Weighted**

A, B, C-Weighting are now available for optimization responses via the SCALE field on the DRESP1 entry.

▼ **Frequency-Response Based Optimization Responses can now be Calculated within Specified Ranges of Frequencies**

LOWFQ and HIGHFQ can be used to define the frequency range of the response calculation on the DRESP1 entry.

▼ **Resultant Section Force and Moment Response are now Available**

Resultant section force and moment responses are now available via RTYPE=RESFORCE on the DRESP1 entry.

▼ **Thermal Compliance can now be Used as an Optimization Response**

Thermal compliance can be used as an optimization response via RTYPE=TCOMP on the DRESP1 entry.

▼ **Enclosed Volume for 2D Elements can now be used as an Optimization Response**

RTYPE=VOLUME and ATTA=ENCLOSED can be used to create an enclosed volume response for Shape, Free-shape, and Topography Optimization. The ENCLOSED option indicates that this response is an enclosed volume defined by a closed 2D mesh (free-edges are not supported). For this enclosed volume response type, ATTI field(s) should be set to PID.

▼ **Vector-Based Input is now Available for Optimization Responses via DRESP2 and DRESP3**

Vector-based input is now available for optimization responses DRESP2 and DRESP3. The VTYPE#, RID#, and VOPT# fields can be used to identify the DRESP1 and DRESP2 (or 3) responses to be passed as vectors to the DRESP3 (or 2) entry. The VTYPEL#, RID#, LID#, and VOPT# fields can be used to identify the subcase-specific DRESP1 and DRESP2 (or 3) responses to be passed as vectors to the DRESP3 (or 2) entry.

▼ **Zero Crossing Frequency Response is now Available for Random Response Optimization**

Zero Crossing Frequency response is now available for Random Response Optimization. It can be activated by specifying ZCF in the ATTB field for RMS responses on the DRESP1 bulk data entry.

#### ▼ **Multiaxial Fatigue Optimization**

Multiaxial Fatigue Analysis is now supported for optimization.

#### ▼ **Thickness Gradient Constraints are now Supported for Free-Size Optimization**

Thickness gradient constraints are now supported for Free-size Optimization via the THICK continuation line. The TG field can be used to specify the maximum thickness gradient. The TGX, TGY, and TGZ fields can be used to define a vector that specifies thickness gradient direction.

#### ▼ **Linear and Planar Pattern Grouping for Free-Size Optimization**

Linear Pattern Grouping (TYP=20) and Planar Pattern Grouping (TYP=21) are now supported for Free-size Optimization on TYP field of the PATRN continuation line on the DSIZE entry. Linear pattern is typically designed to handle models with minimal or no curvature in the specified vector direction (which is typically orthogonal to the rolling direction in Taylor rolled blanks applications). Planar pattern grouping is designed to handle models with high curvature in the orthogonal planes of the defined vector, and with minimal or no curvature in the direction of the defined vector (the vector defined in planar pattern grouping should typically lie in the rolling direction in Taylor rolled blanks applications).

#### ▼ **OUTPUT,FSTOSZ is now Supported for PSHELL in Free-Sizing Optimization**

OUTPUT,FSTOSZ is now supported for PSHELL entries in Free-sizing Optimization.

#### ▼ **Enhanced Discreteness in Topology Optimization for Normal Mode and Modal Frequency Response Optimization**

The discreteness in Topology Optimization has now been improved for Normal Mode and Modal Frequency Response Optimization.

#### ▼ **Enhanced Discreteness in Topology Optimization**

The discreteness has now been improved for any analysis type in Topology Optimization. This can be activated by DOPTPRM,TOPDISC,YES. This parameter can be added to Topology Optimization in normal mode and Modal Frequency Response Analysis to enhance the discreteness even further.

#### ▼ **The Penalization on Mass is Removed in Topology Optimization when the Body Force Type of Loading is Involved**

Mass is not penalized anymore so that the force due to gravity, centrifugal is correct throughout the iterations in Topology Optimization. This also helps achieving more discrete results.

#### ▼ **Tapered Beam is now Available for Lattice Structure Optimization**

Tapered Beam is now available for Lattice Structure Optimization via PBEAML (TYPE=ROD) bulk data entry.

▼ **Lattice Smoothing and Remeshing Support is now Available for Lattice Structure Optimization**

Lattice Smoothing and Remeshing (via OSSmooth) support is now available for Lattice Structure Optimization using the LATPRM,OSSRMSH parameter. During Lattice Optimization, this parameter activates, via OSSmooth, the smoothing process and optionally the remeshing process if the mesh size value is specified.

▼ **Free-Shape Design Grids are now Divided by Geometry into Smaller Groups**

The FSSPLIT flag on DSHAPE allows splitting the design grids into smaller groups based on geometry. This allows capturing finer shape changes more effectively.

▼ **Large Shape Changes are now Supported for Optimization with Normal Mode and Buckling Analysis**

Large shape changes are now supported for Normal Modes and Buckling Analysis for a structure that contains N2S contact, CGAP, and CGAPG elements. Earlier only Linear/Nonlinear Static Analysis was supported for large shape changes in Shape Optimization.

▼ **The CSTRESS, CSTRAIN, CFAILURE Output from TOP, MID, and BOTTOM of Composites can now be Used as Responses for Optimization**

The CSTRESS, CSTRAIN, and CFAILURE output from TOP, MID, and BOTTOM of composites can now be used as responses for optimization. The T (TOP), B (BOTTOM), and O(MID) suffixes are now available for the regular corresponding composite stress, composite strain, and composite failure ATTA field options.

▼ **The ERROR and WARNING Messages from RADIOSS Optimization (RADOPT) are now Printed in the OUT File**

The ERROR and WARNING messages from RADIOSS Optimization (RADOPT) are now printed in the OUT file, instead of only on the screen.

▼ **The Euler Buckling Checks in the Phase 2 of Lattice Optimization are now Output**

The Euler Buckling checks in the Phase 2 of Lattice Optimization are now output to the H3D file.

▼ **The Default Constraint Violation Tolerance can now be Changed**

The DOPTPRM, CONTOL parameter can be used to change the default constraint violation tolerance from the current default value of 1%.

#### ▼ New and Enhanced Parameter Data

- PARAM, PRTRET: Controls the printing of retained responses to the .out file.
- PARAM, PRTDES: Controls the printing of design variables and design entities to the .out file.
- DOPTPRM, OPTMETH: A new DUAL2 optimization algorithm is now available.
- DOPTPRM, TOPDISC: A new parameter is now available to improve discreteness in Topology Optimization.
- DOPTPRM, CONTOL: A new parameter CONTOL is now available to control the percentage value of the constraint violation tolerance.
- LATPRM, OSSRMSH: A new parameter OSSRMSH is now available to activate smoothing and remeshing in Lattice Structure Optimization.

#### ▼ Enhanced I/O Options and Subcases Information Data

- DENSITY: A new type option, TOPO is now available on the DENSITY I/O Options entry to output density results for topology design elements. The SID option has also been added to specify a SET of elements.
- DRESPONSE: The DRESPONSE I/O Options entry is now available to output DRESP1, DRESP2, and DRESP3 responses to the H3D file. The EQUA option can be used to exclude DRESP1 output.

#### ▼ New and Enhanced Bulk Data Entries

- DRESP1:
  - DRESP1 stress constraints are now supported in Topology and Free-size Optimization.
  - The ZCF option is now available on the ATTB field for RMS responses.
  - The SCALE field on the EXTN continuation line on the DRESP1 entry is now available for scaling and weighting responses. The LIN, LOG, DBA, DBB, and DBC options are available.
  - The OCTAVE field on the EXTN continuation line on the DRESP1 entry is now available to calculate responses in octave bands. An integer value can be specified to define the octave band.
  - The LOWFQ and HIGHFQ fields can be specified to define the frequency range for the calculation of frequency response based responses.
  - The RTYPE= RESFORCE setting can be used to specify resultant section force and moment responses on the DRESP1 entry.
  - The RTYPE=TCOMP setting can be used to specify the thermal compliance response on the DRESP1 entry.
  - The RTYPE=VOLUME and ATTA=ENCLOSED settings can be used to specify the enclosed volume response on the DRESP1 entry.

- The T, B, and O suffixes are appended to the existing composite stress, shear, and failure item codes for the ATTA field options.
- MATFAT: The MATFAT bulk data entry is now enhanced with the STHETA and SSHEAR fields for safe zone definition. The Tfl field can now reference a TABLES1 entry that defines the intersection points of the FOS space.
- DTPG: The TYPE field on the DTPG entry can now reference STACK entries for Composite Topography Optimization.
- DRESP2 and DRESP3: The VTYPE#, RID#, and VOPT# fields and The VTYPEL#, RID#, LID#, and VOPT# fields are now available for vector-based optimization response input.
- DSIZE:
  - The TG, TGX, TGY, and TGZ fields are now available to apply thickness gradient constraints in Sizing Optimization.
  - The TYP field on the PATRN continuation line can now be set to 20 and 21 to activate linear pattern grouping and planar pattern grouping, respectively.
- DSHAPE:
  - The FSSPLIT continuation line is now available to activate the splitting of design geometry into multiple DSHAPE's based on their normal orientation. The SPLIT field controls the splitting of the design grids for this DSHAPE entry.



## Solvers

---

▼ **Automatically Switch to 64-bit Sparse Direct Solver**

The 64-bit sparse direct solver is automatically activated if `-len` is set greater than 16 GB.

▼ **Memory Estimate for MUMPS Solver**

The estimated memory for the MUMPS solver is now available in the OUT file.

▼ **AMSES has been Upgraded to 64-bit**

AMSES has been upgraded to 64-bit to be able to solve large models.

## Nonlinear Dynamic Analysis

---

### ▼ New MATX38 Bulk Data Entry

A new MATX38 bulk data entry material has been added as a MAT1 extension material type for Nonlinear Dynamics Analysis. It maps to the RADIOSS /MAT/LAW38 material type.

### ▼ New Bulk Data Entries

- MATX38: A new MAT1 material property extension for viscoelastic foam tabulated material (tabulated form) for Nonlinear Dynamics Analysis is now available.

## Resolved Issues

---

- Inertia Relief Analysis with INREL = -2 has now been enhanced to output the same displacement results for Inertia Relief subcases in models with static subcases and in models with only Inertia Relief.
- Internally created constraints for Draw Direction are now ignored for the output of Constraint Violation % in the .out file.
- The accuracy of results for Lanczos runs with SPCD and other types of residual vectors (loading, damping, USET), has been improved.
- An issue wherein the CBUSH force results did not take into account the varying GEi bush structural damping values correctly (via PBUSHT) has now been fixed.
- Runtime for models containing NSM bulk entries referenced by a large number of element and property data has now been improved.
- Models with local coordinate systems assigned on CBUSH grids with PBUSHT in nonlinear analysis are now correctly handled.
- Previously, grounded CBUSH in Large Displacement Analysis and grounded CBUSH (with nonlinear PBUSHT) in Small Displacement Analysis were incorrectly handled. This issue has now been fixed.
- A programming error that was printed with RBE3 in Large Displacement Analysis with dependent dof as 123 only has now been fixed.
- Previously, results may be incorrect when a local coordinate system is assigned to the dependent node of an RBE3 in a Large Displacement Analysis. This has now been fixed.
- The accuracy of sensitivity calculations for Shape Optimization with nonlinear contact has now been improved.
- Previously a Negative Jacobian error occurred when shape design variables are defined on pretension surfaces. This has now been fixed.
- An issue wherein viscous damping was not handled properly when bush offset was defined has now been fixed.
- An issue wherein results were incorrect if PBUSHT and SPCD were used in conjunction in a fluid-structure coupling model has now been fixed.
- No corner stress was output when linear material (other than MATHE) was referenced by PLSOLID. This has now been fixed and an appropriate error message is printed if any material type other than MATHE is referenced by PLSOLID.
- A performance issue with Topology Optimization using Multi Model Optimization has now been fixed.
- An issue with STRAIN results for models containing Hyperelastic materials has now been fixed.
- Surface to Surface contact now correctly recognizes the TOP/BOTTOM type on the ZOFFS field.
- H3D file is now output for Complex Eigenvalue Analysis in Rotor dynamics.
- Performance issue with ESE output request for Modal Frequency Response is now fixed.

- Results were incorrect for models with 3D bolt pretensioning with non-flat surfaces. This has now been fixed.
- An issue wherein results could be incorrect if a local coordinate system is assigned on the dependent node of RBE3's in large displacement analysis has now been fixed.
- ERP grid contribution of second order elements has now been improved.
- ELEMQUAL in `optistruct.cfg` is now correctly recognized.
- The CHACAB element formulation has now been enhanced.



## **RADIOSS 14.0 Release Notes**

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 industries worldwide to improve the crashworthiness, safety, and manufacturability of structural designs. For over 25 years, RADIOSS has established itself as a leader and an industry standard for automotive crash and impact analysis.

## Crash and Safety

---

Major improvements for airbags, in particular Starter capability to generate automatic Finite Volume Mesh for folded complex airbags to account for internal airbag topology: inter-chamber surfaces, retainers etc. Pre-simulation for airbag folding is becoming a standard, but having the folded bag respecting its package constraints requires additional morphing operations that are now implemented inside the solver.

Tied contact has an extended penalty formulation, to overcome issues with multiple incompatible kinematic conditions; main advantage versus previous formulation is the conservation of both forces and moments. There are no over-costs compared either to kinematic formulation or former penalty one.

Composites is another domain of major improvements. Ply based input, equivalent to OptiStruct input, is not only more intuitive, but allows seamless combined crash/impact and strength simulations, or multi-objective optimization. The coupling with MDS (Multiscale Design Systems) opens the door for material law identification as well design to answer specific dedicated behaviors.

Globally material laws and rupture criteria are fundamental issues for all the industries in the virtual testing approaches. A new material law for high strength steel with strain rate and temperature dependencies developed by MIT provides an accurate prediction of the local strain fields with a consistent damage indicator.

Damage is an important indicator for failure of components; stress and strain only are often not sufficient to estimate components failure, because failure in tension/compression/shear occurs at different stress and strain levels. Damage visualization, available for all failure criteria and both solids and shells, allows to have a more precise estimation about how close to failure are elements. It is also possible to initialize damage, for example to take into account damage due to forming process of a given component.

### ▼ Airbags

- **Destination Reference (Fitting):** Technology to morph folded airbags into given its destination. Folded airbag mesh usually does not fit into its destination in packaging space; the airbag is gradually morphed into its destination by defining both the envelop surface and the destination surface.
- **Automatic FVM mesh for multi-chambered airbags:** FVM mesh taking into account internal inter-chamber surfaces, retainer etc. is generated automatically by the Starter.
- **FVM solution speed:** Improvement of the FVM method parallelization, offering nice scalability in SMP.
- **Airbag tethers:** Advanced option in /PROP/TYPE4 ( $H_1 = 8$ : elastic total length function) allowing to define the force as a function of total actual length, as opposite to the regular force as a function of elongation ( $H_1 = 0$ ).
- **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 transverse shear modulus Gsh between layer.
- **Pyramid element available for gas mesh:** In case of manual mesh, element compatibility has been extended to pyramid elements.

- **Pyramid element** available for internal FVM mesh.
- **FVM compatibility with /LEAK/MAT:** Possibility to model leakage through porous fabric surfaces.
- **Reference state:** /EREF compatibility with TETRA4 elements. /EREF is compatibility with /REFSTA and /XREF in case there are no common nodes between these 3 options to define a reference state.
- **//SUBMODEL:** Compatibility extended to /MONVOL/COMMU1, /MONVOL/FVMBAG1, /LEAK, and /EREF.
- **Local parameters implemented for //SUBMODEL. This allows to parametrize airbag, seatbelt and other RADIOSS models.**

#### ▼ Contact TYPE6

- New friction formulation with static force and dynamic coefficient input; it allows to better control relative motion. For example, tangential motion starts only if the tangential force exceeds the static friction force.
- Damping coefficients are available for both normal and friction force as a function of relative velocity and of normal contact force.
- New option Icor to adjust the contact force due to initial intersection.

#### ▼ Tied Contact (TYPE2)

- New penalty formulation (Spotflag=26) designed to overcome limitations of the existing one (Spotflag=25): not only forces but also moments are fully transmitted from master segment to slave node, respecting global balance. It is less sensitive to errors due projections outside the master segment; performance are similar to Spotflag=25.
- Stiffness control for penalty formulation (Spotflag=25, 26): user can tune the penalty stiffness to prevent time-step drops, that might occur especially when the ratio slave/master stiffness is big.
- Slave node projection control: to prevent negative mass errors, in case the slave node projection is close, but outside the master segment, the user can force slave projection on the edge of the master segment (Iproj=0, default in v14.0).
- Error/Warning messages are compacted into the Starter output file, making it more readable and lighter in terms of file size. For example, instead of repeating the same error message for each node, a single error message is output with a list of the nodes.

#### ▼ Contact Default Values

- Possibility to define default values of contact flags, by contact type. For example, stiffness flag, etc.

#### ▼ Composites

- Ply-based input: new input type for multi-layer composites based on plies. Plies are identified with groups of elements, which is closer to designers experience and manufacturing process. RADIOSS and OptiStruct ply-based input are now fully consistent, allowing easy conversion from one format into the other both ways, making combined crash/impact and strength simulations seamless, including optimization.
- QEPH shells compatibility with orthotropic properties/materials: QEPH is an under-integrated shell element with physical hourglass control; with a very small computational over-cost gives results with high accuracy, close to fully integrated elements which are traditionally used for composites. QEPH v. Batoz elements computation time can be estimated as a factor of 3.
- Sandwich shell (PID11): Global material properties are now automatically calculated based on the properties of the layup in membrane and bending  
A material for part definition is still needed (for pre- and post-treatment purposes). The stability condition is recalculated and offset layup stability is improved; this may change results within the magnitude of the model numerical sensitivity. Global material properties are also used to compute time-step and contact stiffness.

#### ▼ Materials and Failure Models

**MDS** (Multiscale Design Systems): complete environment for analysis and design of heterogeneous material.

- Direct coupling with RADIOSS through dll.
- MDS can be used to identify the parameters of the RADIOSS laws for example law 25 for composite.

#### **Metals:**

- /MAT/LAW84: Swift-Voce elastic-plastic law with Johnson-Cook strain rate hardening and temperature softening.
- /MAT/LAW52 (GURSON): User-defined yield stress; input is yield stress table (yield stress v. plastic strain) activated with the new flag lyield. Strain rate hardening is also possible. The regular Johnson-Cook parameters are used if table\_ID=0.
- /MAT/LAW2: Possible to input yield-stress, Ultimate engineering tensile stress or engineering strain at UTS with new flag "lflag" in /MAT/LAW2.

#### **Foams:**

- /MAT/LAW77: porous foam material law. This porous material law is compatible with moving structures.

#### **Spotwelds:**

- /MAT/LAW59 and /MAT/LAW83: new flag lmass for mass calculation: spotweld mass can be computed either using the volume or the area (upper and lower average).
- Physical thickness could be taken into account for moments computation (True thickness in /PROP/CONNECT).
- /MAT/LAW59: The Starter checks and errors out if the slope of the input curve is higher than the Young's modulus.



- Spotwelds lines and clusters: /CLUSTER allows to group of elements with LAW59 or LAW83 material into a cluster of spotweld elements. Cluster allows to output summed force and moments acting to all elements of the cluster. Cluster failure criterion can be defined based in these forces and moments. Cluster can be used also with spring elements.

#### Polymers:

- Elastic-plastic material with a non-linear elastic section for solid (LAW65 compatibility extended to all solid elements: hexa, tetra and thick-shells).

#### XFEM

- Possibility to separate the crack initiation and advancement criteria extended to other compatible failure models ( $D_{adv}$  in /FAIL/TBUTCH and new function  $fct\_ID_{adv}$  in /FAIL/FLD).
- No more number of layer limitation with X-FEM in /PROP/SH\_SANDW (/PROP/TYPE11).
- By making /MAT/HILL (LAW32) and /MAT/HILL\_TAB (LAW43) compatible with /PROP/SH\_SANDW (/PROP/TYPE11) X-FEM can be applied to orthotropic materials.

#### Thermal stress and expansion:

- Generalized to all material laws (for both shell and solid elements).

#### Damage and Failure index:

- Possibility to output damage and failure index for all failure criteria using /ANIM/SHELL/DAMA or /ANIM/BRICK/DAMA.

#### ▼ Properties

- Hill's law for muscles: /PROP/SPR\_MUSCL is a spring property for modeling active/passive muscle behavior in crash events; it is an elastic-viscous plastic law with user-defined functions reflecting time dependent muscle behavior in tension (force v. deflection or force v. time).
- Possibility to define edge to edge contact for XELEM: a part of XELEM can be set in /LINE/PART to define contacts with /INTER/TYPE11.
- Improved hourglass control in HEPH element: Hourglass tangent modulus control (IHKT) extended to orthotropic properties (/PROP/TYPE6).
- Improved energy computation for pre-tensioner spring: initial/added internal energy for /PROP/TYPE32 (SPR\_PRE) to avoid negative energy.

#### ▼ Resolved Issues

##### Airbags:

- Non-consistent temperature/fluid velocity output when automatic mesh is used (FVMBAG1).
- Porosity curve shifted not respected if controlled by Ittf (=0, 1, or 2).
- LAW58 nonlinear curve input not read correctly.

- Starter issues Warning 941 (surface not closed) even when the airbag surface is closed.
- FVM: general robustness improvements (better control on the evolution of the number of finite volumes).
- /FVMBAG1 with tetra mesh and internal surfaces. Result of pressure in time history and animation does not match.
- Error message is issued, if vent a hole in /FVMBAG1 is not determined through a physical components.
- Added a warning for FVMBAG when a node belongs to several inflators.
- Starter will issue a Warning (instead of an error), if a vent hole area is not defined for UP airbags.
- Possible Starter segmentation violation error while try to write finite volumes in animation file (Ifvani=1 in /MONVOL/FVMBAG1).
- /IMPDISP does not does not imposed nodes, if /IMPDISP/FGEO used.
- Airbag internal surfaces is ignored by FVM.

#### **Contact TYPE6:**

- Formulation flag not detected (Iform, nonlinear behavior with constant unloading stiffness).

#### **Tied Contact TYPE2:**

- Possible segmentation violation when using interface TYPE2 in 2D analysis.
- With ignore=2, Starter does not report a problem of non-projected node as expected.
- With IGNORE >= 1, if slave nodes are not tied, warnings are compacted to improve readability.

#### **Contacts:**

- TYPE7: Possible Engine failure if Irem\_gap=2 with SPMD version is used.
- TYPE7: The gap calculated by RADIOSS for Igap=3 does not match the theoretical value.
- TYPE11: When contact is defined inside a submodel with different units, dtmin value is not scaled.
- TYPE17: Hertz contact may not work if several interfaces TYPE17 are defined.
- TYPE17: Contact might not be detected if the top face of SHELL16 is impacting the master SHELL16.
- TYPE21: Possible segmentation error due to restart writing is wrong if using Interface type 21 + INACTI=6.

#### **Composites:**

- /MAT/LAW25.
  - Wrong RADIOSS units for WPLAREF and WPLAMX: if Wp\_ref=0, Wp\_ref=1 will be considered whatever units system used which is not correct.
  - Possible engine segmentation violation error with solids and thick-shells.
  - Wrong strain tensor output in animation file for solids and thick-shells.

- No element (with LAW25 CRASURV Formulation) deletion despite number of failed layer in animation.
- Elements possible deletion during initialization of orthotropy directions with the /INISHEL\_ORTHO\_LOC card.
- Failure appears too early with Puck criterion; failure might occur even if the failure is switched off (all strengths to 1e30).
- Extended incompatibility checks with orthotropic law + isotropic property and print the warning message.
- /PROP/SH\_SANDW: Add error message if /PROP/SH\_SANDW is used with incompatible material.
- Unstable solution by use AMS + Ipos=1 in sandwich shell property and all the ply define on one side of the middle plane shell.
- Starter error out and info incompatibility for orthotropic material laws(LAW43,73) with sandwich property.
- Model with Idrill=1 + Batoz + Composites fails due to energy error limit reached.

#### **Material and Failure Models:**

- LAW35: Issue due to wrong unit convert for LAW35 in /BEGIN.
- LAW59: Possible wrong behavior of /FAIL/CONNECT on win64 (energy criterion is ignored).
- LAW83: Plastic strain output is wrong (in both ANIM and TH).
- LAW50: Strain rate not available and wrong Starter printout.
- LAW66: Wrong elastic-plastic behavior for solid elements.
- LAW70: Incorrect default Qa and Qb values in the Starter printout file.
- LAW70: Wrong results in animation, if using LAW 70 and Ismstr=11/Iframe=2.
- LAW42 and LAW65: Incorrect time-step estimation in the Starter.
- /MAT/LAW44 (COWPER): Calculates the strain-rate effect even with C=0 (no strain rate effect).
- /FAIL/LAD\_DAMA input: Wrong order of Ifail\_sh (shell) and Ifail\_so (solid).
- Possible segmentation violation Error when none plastic law (e.g. LAW42) with tensstrain failure criteria.
- INICRACK (X-FEM) might delete instead of splitting elements.
- LAW24: Dilatancy (s0/fc) default value, leads to NaN Error.
- LAW24 is compatible with tetra10.
- Different results between two unit systems used in a model with HEPH+IHKT=2+Hyperelastic laws (42, 62, 69, 82).
- Use LAW42 with Prony coefficients showing unexpected zero reaction force at displacement=0 in the unloading cycle.
- XFEM does not work correctly with multiple /FAIL/TAB1 cards (each of them with Ixfem = 1).

- Corrected failure criterion (/FAIL/SNCONNECT) to match the theoretical results and also correct ANIM damage output in (/ANIM/BRICK/VDAM).
- Possible segmentation violation error in engine for XFEM in win64 SMP.
- No difference in results even with different input curves in /FAIL/ENERGY.

**Properties:**

- When using 10 IPs over thickness, the structure shows strange stress output (waves) in animation files and in Time History file, as well. It does not happen, if using less than 10 IPs.
- When solved using RADIOSS SPMD, NSTRAND/XELEM parts are missing from animation files and elements are incorrectly grouped.
- Corrected curve description for PROP/SPRING in Starter output.
- /PROP/KJOINT (TYPE33) does not work if both ends of the spring are without /RBODY.
- In /PROP/KJOINT2 (TYPE45) friction force is incorrect in case SD+ and SD- are set to zero.
- Deformation of solid connect element will not be stable when connected shell elements are deleted.
- In /PROP/TYPE4 (SPRING) with H=7, the output force does not follow the prescribed force/displacement curve. And also fix the behavior for non-homogenous spring groups.
- Correct negative internal energy when using Isolid=17+LAW34.
- /PROP/TYPE22+LAW25: incorrect strain output for thick shell elements with Isolid=14.
- Possible engine error(NaN in IE) due to Starter change Ismstr=1 is changed to Ismstr=11 with REFSTA + LAW36.
- Correct shell stress results through the thickness when more than 1 IP defined through thickness.

## Forming

---

Extended thermal capabilities represent certainly a big step forward for hot-forming simulation capabilities, as well as general applications. Thermal expansion and heat transfer capabilities have been generalized to all materials and element types (solids and shells).

Controlling manufacturing process is sometimes crucial for the performance of a component. Damage evaluation, together with thinning and residual stress and strain is critical for an accurate description of the initial state of a component that after being formed, should undergo a set of tests. Damage initiation is now extended to shells. Damage can also be output into animation files, and might be helpful for failure risk prediction.

### ▼ Thermics

- Generalization of thermal expansion to:
  - all material laws
  - all small strain formulations
  - all element types (solids and shells)
- Improved results accuracy for thermal expansion.
- Radiation (/INTER/TYPE7): possibility to compute the radiative exchange between two parts which are not in contact.
- Thermal conductivity (/INTER/TYPE7): depending on contact pressure.
- Thermal time-step: usually thermal time step is several orders of magnitude larger than mechanical time-step. For example, for quenching simulations, thermal time-step might speed-up the solution by a factor 1000x about.
- Heat output in time history: new option QFRIC in /TH/INTER outputs thermal energy generated from contact friction.

### ▼ Damage Initialization and Visualization

- Manufacturing process may severely affect material properties; evaluation of damage due to manufacturing is critical for the performance of the components in real tests.
- Damage now can be output (\*.sta file) and initialized both for shells and solids.
- Damage can also be visualized in \*.anim file (/ANIM/BRICK/DAMA and /ANIM/SHELL/DAMA).

### ▼ Materials and Failure

- Metals: Input yield stress curve ( $I_{yield0}=1$ ) in orthotropic direction is possible (option  $I_{yield}$  stress flag in /MAT/LAW32, /MAT/LAW43 and /MAT/LAW73).
- Failure: NXT failure criterion is an extension of /FAIL/FLD, where the failure is checked against the stress (in FLD the failure is checked in the strain domain). NXT main

advantage v. FLD is that failure is stress and strain history path independent. Output at each integration point is available.

#### ▼ Drawbeads Contact TYPE8

Force on tools can be predicted with higher accuracy by taking into account the blank bending force due to drawbeads.

#### ▼ Titles in Animation Files

In forming simulations the time in animation file is not relevant, while some key information (like the tool displacement or distance from the blank) are not plotted. Now animation file titles can be defined in Engine file (1 title per animation file).

#### ▼ Resolved Issues

##### Thermics

- LAW80: possible segmentation violation error:
  - when using adaptivity
  - when using table input + single precision
- LAW80: possible wrong plastic strain output.
- Nodal temperature may depend on small strain formulation used.
- Different nodal temperature results between fine mesh and adaptive mesh.
- Possible incorrect energy balance (99% Energy Error).
- Possible incorrect volumetric expansion.
- Thermal conduction might happen in all /INTER/TYPE7 contact interfaces defined in a model, even if requested only in a single one.
- Incorrect thermal expansion results for solid model if full geometric non-linearities formulation (Ismstr=2, 4) is used.

##### General

- Possible NaN energy output per integration point (NPT > 5 in shell property).
- Possible Starter Error if STATE/ADMESH and SH3N are used simultaneously.
- Using STATE/THICK + Adaptive mesh, elements are written several times in .sta file.
- Starter errors out in Adaptive Meshing if Node ID and shell ID are > 900000000.
- Possible ERROR 760 during spring-back phase.
- For /FAIL/NXT criterion, results sigma1/h and sigma2/h could be output in all layers, in upper/lower/membrane layer or in N layers.
- Starter issues a warning – instead of error - message if \*.sta file includes elements which might have been removed from the Starter (due to trimming).

## Blast Simulation

---

Thru the Euler and ALE capabilities RADIOSS is brightly used for fluid structure interactions. In Version 14.0 improvements have been brought in the multi-material regarding the compatibility with the far field boundary conditions. A significant added value lies on the compatibility of the Tetra4 elements with ALE.

### ▼ TETRA4 Element Compatible with ALE

New TETRA4 element (Itet=3) compatible with ALE, reducing the volumetric locking.

### ▼ Multi-Material (LAW51)

- Drucker-Prager yield criteria introduced. (/MAT/LAW51 – Iform=11).
- Possibility to define relative pressure for inlet material boundary:  $P_{EXT}$  (Iform = 4 or 5).
- /INIVOL card is now compatible with boundary elements (Iform = 2 or 3).

### ▼ New Cards for Detonics Modeling

/MAT/LAW5 and /MAT/LAW51: Burn fraction method for JWL EOS can be defined using IBFRAC flag. Usually two functions are evaluated: one is based on time, the second one is based on volumetric compression. The maximum of the two functions is used by default to set burn fraction value. Now the user can deactivate one of them. This especially enables to keep only function based on time control to get rid of erroneous ignition in some specific cases.

### ▼ Resolved Issues

- Degenerated pentahedra now compatible with /SUPG.
- /MAT/LAW51
  - Possible time-step drop (Iform = 2, 3) due to Strain Rate incorrect value.
  - Possible Engine failure with negative density ( $\rho=NaN$ ) for unused submaterial in inlet/outlet formulation.
  - Possible Engine failure at cycle 0 on Windows. Issue occurring when using plasticity models with MM-ALE (Multi-Material-ALE) LAW51.
  - No strain rate dependency with JCOOK criteria.
  - Deviatoric stress is now taken into account for outlet frontier (Iform=3).
  - Fixed strain rate & temperature dependency in Johnson-Cook criteria when used with MM-ALE.
- /INIVOL not working correctly when surface is defined with mixed 3N and 4N shells.
- Multiple /INIVOL cards not working (last one overwrites previous ones).
- /INIVOL: possible failure on Windows platform.
- Drucker-Prager material LAW10 and LAW21.

- Artificial Viscosity was not taken into account while computing energy integration from EOS numerical solving. Might affect numerical result unless user sets artificial viscosity to 0:  $qa=qb=1e-20$ .
- Wrong Drucker-Prager behavior in tension:
  - 1- Yield surface was symmetrized around fracture point, while there shouldn't be any yield criteria below this point. This might affect numerical results since Yield surface was fixed
  - 2- If Yield surface is mathematically not well defined user is warned
- Introduction of  $P_{EXT}$  parameter. User can now input EOS written in relative pressure formulation: new  $P_{EXT}$  parameter enables to get correct energy balance.
- /MAT/LAW20:
  - Detonation wave velocity faster than expected results. ALE rezoning issue was fixed. It was computed only if second submaterial in input card required it.
  - Possible engine segmentation violation error if /ANIM/ELEM/VFRAC is used.
- Rotations can now be imposed on Lagrangian nodes merged to an ALE element.
- Error not returned when LAW3 format is the old v10 one: Using material card format 10.0 leads to error message in input files 110 120 130.
- Warning message is issued if old format of LAW3, LAW4 and LAW49 is used.
- /MAT/LAW37: Starter checks for massic liquid proportion out of range [0,1] and initial densities  $< 0$ .
- Reinforced concrete material LAW24: Check the warning "increment too big on element ###" for LAW24 material elements in engine. Fix of Possible high occurrence of Warning message about too big increment on load path.
- Possible read error with /CAA option, if /IOFLAG or /ANALY card was not present.

### User comfort (Input/Output/Manual)

- Manual for material law 11 (/MAT/BOUND) was rewritten to provide better understanding of different options (non-reflecting frontier(NRF), general outlet, inlet from stagnation condition, etc.)
- Improved user readability in output files for ALE keywords: listing outputs and documentation are now consistent.
- FSI interface 18 (/INTER/TYPE18) has no longer Istf flag. It is now removed from input card and the manual since it is now automatically set to 1.0. Other values led to error message about NEGATIVE STIFFNESS NODE.
- Starter is checking if a Lagrangian element is connected to an Eulerian one. Warning message is printed in this case.
- Surface defining elementary boundary condition (/EBCS) is now checked by Starter Warning messages are printed, if necessary.
- Initial pressures for law 51 submaterial are printed in Starter listing file.



## General Updates

---

Main enhancements relate to TYPE24 contact, SPH and Multi-Domain.

General contact TYPE24 has been enhanced concerning the treatment of initial penetrations; formerly with solids coated with shells, the contact defined both on shells and solids, with penetration larger than half the shell thickness, was causing nodes anchorages on the wrong side. This is no longer the case in 14.0.

SPH: major improvements regarding scalability on high number of cores; performance measured on several tests shows a solution acceleration factor of 10x about.

Multi-Domain compatibility has been extended to SPH and ALE; for FSI problems, due to their time-step and amount of elements differences between the fluid and lagrangian domains, the solution acceleration factor is much higher than in regular crash applications, measurable in something between 6x and 15x, depending on the model. Solution theoretical acceleration factor is also estimated at the Starter level and printed in output file.

### ▼ Contact 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.
- Initial penetrations treatment (INACTI=5), with possible release of nodes with initial penetrations, and INACTI=1000 to handle tiny initial penetrations.
- Automatic treatment of Solids coated with shells. The contact stiffness is the highest among the shell and the solid; the contact segment position is always determined by the shell.
- Scalability improved in H-MPP mode with high number of cores.

### ▼ Multi-Domain

- Single input file setup extended to FSI involving ALE and/or SPH: significant speed-up compared to regular Mono-Domain. Speed-up factors from 5x to 15 x with industrial models.
- Single Time History file output.
- /GAUGE card made compatible with ALE and SPH.
- Starter printout of a theoretical speed-up estimate when Multi-Domain is used.
- Starter now checks and errors out if more than 50% nodes are coupled with cross domain contacts (was just a warning in earlier versions).

### ▼ SPH

- Failed elements converted into SPH: instead of being deleted, solid elements can be converted into SPH particles. All failure and deletion strategies can be used, including minimum time-step.

- Parameter to reduce tensile instability.
- Heat transfer is enabled between SPH particles and finite elements.
- Thermal Expansion is also available for SPH.
- H-MPP performance improvement: an optimized domain decomposition improves greatly the scalability on high number of cores.

#### ▼ Optimization

- Design Response via Equations for Design Optimization (/DRESP2) and Equation Definition (/DEQATN) entries are supported in RAD-OPT Format.
- Support Free-size Manufacturing Constraints/Size Optimization for composites in RADIOSS-Optimization (RADOPT).
- Support Free-size Optimization for Composites in ANALYSIS=NLGEOM/IMPDPYD/EXPDPYD subcase.
- Support of 3D Topology Optimization in RAD-OPT Format.
- Support of Velocity response in RAD-OPT and ESL-B2B.
- Handling of "Soft" (Low-density) elements from OS ESLM Topology Optimization in RAD-OPT and ESL-B2B. "Soft" (Low-density) elements from OS Topology Optimization are removed from the next RADIOSS nonlinear analysis. They are assigned to a /DEL entry in the RADIOSS Engine file. These elements are reintroduced at the next ESLM linear static optimization run.
- Handling of failed elements from RADIOSS Nonlinear Analysis in RAD-OPT and ESL-B2B. Stiffness of Failed elements from RADIOSS nonlinear analysis run, is not considered when generating the Equivalent Static Loads for the subsequent linear static optimization problem.
- Support more animation options (/ANIM/VECT, /ANIM/BRIC/TENS/STRAIN, /ANIM/SHELL/TENS/STRAIN, /ANIM/EItyp/EPSP, /ANIM/EItyp/ENER, /ANIM/EItyp/HOURG and /ANIM/EItyp/THIC).

#### ▼ Output

- Output parameter value used in which card which field in output file.
- Weld beads, glue lines: /CLUSTER defines a cluster of elements for modeling spotweld/connection lines; failure can be applied to the cluster and/or to the single elements There are also specific cluster outputs for post-processing /ANIM/VECT/CLUST, /TH/CLUST.
- New keyword /INISHE/FAIL and /STATE/SHELL/FAIL: To improve predictivity of metal forming simulation, it is necessary to consider damage, together with stress, strain and thinning. Need damage output for shell in .sta file. (already have damage output for brick).
- New keyword /ANIM/ELEM/SSP (ALE only): To output sound speed.
- New keyword /ANIM/ELEM/SCHLIEREN: To output schlieren image (optical method widely used in CFD field).

- New keyword /ANIM/SHELL/TDEL and /ANIM/BRICK/TDEL: Time of deletion of solid and shell element.
- Outputs global plastic strain (equivalent to total mean plastic strain in all integration points) for beam (with /PROP/TYPE18).
- Animation file is generated when Engine fails with ERROR 166.

#### ▼ AMS

- AMS compatibility extended to RBE2 and RBE3.
- Added warning if master node of a rigid body is part of the mesh now errors out if running /AMS.
- Compatibility checks extended to /MAT/LAW13: if it is used with AMS, Starter errors out.

#### ▼ Cut-Section Method

- Read/write multiple sections into multiple SC01 files to prevent global SC01 file huge sizes resulting from the amount of stored nodal information. Now can be split into multiple independent SC01 files, enhancing usability, range of applications, as well as disk space and data transfer reductions. However, backward compatibility with a global SC01 option gathering multiple sections is maintained.

#### ▼ Submodels

- Compatibility extended to /SENSOR/TYPE11, and 12.
- Local parameters: parameters can be restricted to a SUBMODEL (do not apply to any other part of the model).

#### ▼ User Subroutines

- Improved scalability on high number of cores for User Material laws.
- Possibility to use free Fortran compilers for Windows (MinGW) and Linux (GCC); warning: performance might not be at the same level as Intel compiler.
- User material laws compatible with solid spotweld property.
- Add User Animation Outputs /FAIL/SNCONNECT.
- Possibility to specify in Starter (or execution) a specific dll. Users may need to use different dll or even no dll at all.
- Add deformation gradient tensor in Users' laws when Ismstr=10.

#### ▼ Implicit

- Implicit nonlinear compatibility extended to /RBE2, /RBE3 and /PROP/SPR\_PRE.

- Improve the convergence for implicit using /IMPL/SINIT.

#### ▼ Messages

- Starter messages have been reformatted to include option titles.
- Warning and Error messages have been compacted to improve readability and avoid huge out files. For example, Incompatible kinematic conditions checks for rigid walls and tied interfaces.
- Warning message is issued in case subset, surface or group are empty (instead of Error).
- Mass and inertia of each part are printed anytime (from the output level Ipri=0).
- Added DOF output for RBE3.
- Starter errors out if:
  - submodel version is not supported ( $\leq 44$ ).
  - only one node is defined for rotation axis in /TRANSFORM/ROT.
  - a wrong value (besides 0 or 1) is used for /BCS.
  - no defined node in input deck, which leads to NUMNOD=0 in .sta file.
  - Engine errors out if invalid engine keywords are used (for example: /DT/NODE/CST is not valid option but difficult to debug: was leading to deletion of all elements).
- Output complete error message in case of  $RHO0 \leq 0.0$  in LAW6 for user to better understand the error message.
- Improved the warning message in case a 4-node degenerated triangle shell element.

#### ▼ Resolved Issues

##### Contact TYPE24:

- Large sliding for SH3N and at corners.
- In case of T-connections and X-connections, possible wrong direction of contact forces.
- GapMax for shells.
- Single precision: possible loss of contact.
- Solids coated with shells: possible node anchorage in case of initial intersection. Improve coating shells treatment in /INTER/TYPER24 which may leads to too small stiffness or gap in contact.
- Wrong forces output if shell sliding in Contact TYPER24.
- TYPER24 + INACTI=5 Starter infinite loop.
- No contact treated in type 24 when master surface (shell) is defined by /SURF/SEG.
- Lost contact in int24 due to master surface using triangular shell elements.
- Possible Starter segmentation violation error with int24 in win64 and messages repeated too many times.

- Improvement of computational efficiency of interface type24 if using gravity in implicit simulation.
- Possible lose contact in interface type 24 if run with single precision.

#### **Multi-Domain:**

- Contact across domains: Contact segments were not deleted upon element deletion (flags Idel and Idel2 information was not passed between domains).
- Possible Engine segmentation violation, if /ATFILE in 1.rad file exists without /ATH in 0.rad.
- Segmentation violation in Engine phase when /TH/SUBS is present in the Starter file.
- Use of TYPE2 with penalty formulation may lead to very low time-step in the domain with small time-step.
- SPH with Multi-Domain TH combined T01/T02 output plot interruption between restart phases. May affect curve processing (for example a derivative operation).
- /GAUGE for ALE and /GAUGE/SPH compatibility with Multi-Domain.
- An infinite loop in the reading of .sta file at Starter level was fixed.
- SPH/INOUT issues were fixed.
- Starter issue with large ALE models.
- /GAUGE/SPH failing /PARITH/ON issue was fixed.

#### **SPH:**

- Possible Starter failure in H-MPP mode with large number of SPH particles.
- Damage tensor (LAW24) output to animation files does not work for SPH.
- Possible segmentation violation if mass is not defined; error message (ID=138) is introduced for an easy localization of the issue.
- Possible Starter segmentation violation if MAT\_ID=0 is assigned to SPH. Starter will issue an error if MAT\_ID=0 is used.
- Filtering in /GAUGE/SPH/ not working due to bad initialization of gauges which cause F\_cut frequency to be equal to 0 in all cases.

#### **Optimization:**

- When a /ESLPART entry was defined in RAD-OPT, the RADIOSS Contact definitions (/INTER/ entries) were not considered in the linear static optimization phase.
- Some of the generated equivalent static load subcases had zero compliance. The ESLM Time Step Screening Process in RAD-OPT and ESL-B2B now filters out zero compliance (equivalent static) subcases.

#### **Output:**

- Possible engine segmentation violation error with ANIM/BRICK/USRi.
- Output for USER4 and USER5 in LAW58 was engineer strain not true strain.
- Output Null (with Isolid=17) or wrong output (with Isolid=24) in /ANIM/BRICK/TENS/DAMA for /MAT/LAW24.
- T02 file generated after implicit run followed by explicit run might be empty.

- Possible Engine segmentation violation error when /OUTP/SHEL/USER5 or /OUTP/ELEM/USER5 used.
- Animation files might be corrupted when /ANIM/SHELL/EPSP/ii and /ANIM/SHELL/EPSP/ALL are used in same Engine input.
- TShell property reports incorrect plastic strain (extremely large values).
- Incorrect damage initialization with TETRA4.
- The equivalent plastic strain and damage values for LAW72+Ishell=12 is wrong in TH file.
- TH output does not take into account the contact viscous energy from /INTER/TYPE24.
- No Plastic Strain show in contour for solid element using /MAT/LAW44 Cowper Symonds.
- KE and TE from /TH/PART did not give the right results.
- HyperGraph can not read results (from /THPART) in the T01 file.
- Different results from abf output and T01 output.
- Wrong stress results in composite layers.
- Engine error out if /MAT/LAW01 with /STATE/SHELL/AUX/FULL.
- Possible Engine segmentation violation error when using /TH/QUAD.
- Wrong Output values in the abf file, if compare to the T01 Output.
- /TH/BRIC/EPSXX output the incorrect value for part with LAW4.
- Correct Isolid value of /INIBRI in .sta file in case Isolid=24 + TETRA4 used in model.
- .sta file is not compatible with TETRA10.
- Use /INIBRI/ST...\_F the initial stress and strain are not correctly mapped on the model.
- Strain tensor output in animation file for LAW25, solids and thick shells is not correct.
- /PROP/TYPE11+LAW25: no strain tensor output in animation file.
- /FVMBAG1 with tetra mesh and internal surfaces. Result of pressure in time history and animation does not match.
- Improve of the spring element output in Starter listing file \*\_0000.out (aligned the comment and value).
- Possible segmentation error in Engine, if using /OUTP/VECT/VROT for solid.
- Starter error when using /TH/FRAME.
- Wrong Internal Energy in case restart the job after it previously stopped by "STOP" in control file.
- Wrong animation results (DAMA variables and Von-Mises stress) if uses Sol2SPH +/ANIM/BRICK/DAMA.
- /STATE/BRICK/STRESS/FULL is not working for thick shell 16 node and for Brick 20 node.
- Possible segmentation error in Starter due to Initialize deformation with NPT=0 in /INISHE.
- Wrong strain rates (always Zero) in Animation files (using /ANIM/ELEM/EPSP) for material LAW2.

- Inertia info in warning message #542 (rigid body inertia) is wrong (mixed each other).
- Starter blocked, if using /INIBRI/AUX.

**AMS:**

- Possible instability due to large time-step and solids with full integration (Isolid=12).
- Suppress all Warnings relating to TYPE7 v. TYPE2 interfaces slave nodes incompatibilities when switching /INTER/TYPE2 from kinematic to penalty formulation (Spotflag 25 or 26).
- AMS divergence with m=8 in /MAT/BARLAT3.
- Time step higher than DTMIN for /DT/AMS and elements may cause instability in interface elements.

**Cut-section Method:**

- Cut Model cannot find SC01 file.
- Restarts were not functioning.
- Checks and auto-correction if some sections refer to a specific root-name and others do not.
- Wrong results (zero section force) if cut plane(defined with /SECT/CIRCLE, /SECT/PARAL or /SECT with frame) is just on a row of nodes.

**Submodels:**

- Units of the /RWALL parameters are not transformed in a submodel.
- /UNIT is not applied on the submodel.
- /FAIL/CONNECT is not supported in submodel.
- Model crashed due to SUBMODEL, in case of more than two times transformations are requested.
- /MAT/LAW65 is not supported inside submodel.
- Using submodel, M (Maxwell order flag) in LAW42 is ignored.
- Starter errors out if more than 101 //SUBMODEL and /TRANSFORM are defined (out of bound).

**User Subroutines:**

- Advanced user laws are not working in multithread.
- Wrong user values in animation output if user law has high number of uvar (like number 99).
- Results repeatability (/PARITH/ON) is not ensured for solid with user defined property (igtyp > 29).
- Possible segmentation error in Engine, if interface type 2 with failure + using user dynamic library.

**Implicit:**

- Possible Engine crash (ISTOP=4) of BCS solver on Windows (temporary files management on Windows).
- Implicit simulation does not end properly on several CPUs.

- /IMPACC and /IMPVEL provide incorrect displacement for both implicit static and implicit dynamic.
- Possible Run freeze for implicit nonlinear with RBE3.
- Possible Engine segmentation violation error due to Linear implicit + /INTER/TYPE24.
- Inconsistent results between multi-proc and mono-proc by implicit nonlinear using quasi-newton.
- Possible segmentation violation error due to big memory usage in case large slave node number of RBE2 used in implicit.
- Possible segmentation violation error in implicit nonlinear due to wrong hierarchy RBE2/RBE3 definition.
- In implicit linear+ explicit in SPMD, computed external work for implicit part incorrectly (doubled).
- Node displacement do not respect imposed displacement value in local skew for nonlinear implicit.
- Wrong displacement result in implicit analysis when /IMPDISP and /BSC (using local skew) used on the same node.
- No convergence in implicit if solid element with orthotropic laws (like LAW28).
- Run in SMP but failed in SPMD shortly after start due to masters of RBE3 are slaves in Rigid body.

#### Messages:

- Starter does not printout Material ID and material title in \*.out message or does not print info for number of airbag element in case automatic meshing of airbags failed.
- Starter reports incompatibility for orthotropic material laws with sandwich property.
- Starter prints out wrong default values for shells numerical damping (dn).
- Starter writes wrong pressure values to 0000.out file.
- Starter does not printout all hardening options for Springs.
- Possible Starter segmentation violation error if user sets incompatible value for Ish3n (Ish3n=24) in /PROP/SHELL.
- Possible Engine segmentation violet error if use /ANIM/BRICK/TENS/STRAIN/010 or ANIM/BRICK/TENS/STRESS/010, which those integration points are not existing.
- Message for brick activation is not correct in case of big identifier number (more than 8 digits).
- Correct error message which with ASCII characters.
- Possible segment violet Starter error due to use "TEMP" in /TH/NODA in non-thermal analysis.
- Remove Engine warning message, if using /UPWM/SUPG + /MAT/LAW51 for /MAT/LAW51 now is compatible with SUPG.
- Get unreadable error message, if define non exist material ID in part.
- Rupture message repeated in Engine output file many times, if use /LAW58 + /FAIL/TENSSTRAIN.





## Resolved Issues

---

- Wrong Energy balance with RBE2 (initial mass is counted twice in kinetic energy computation).
- Results not /PARITH/ON with RBE2 in SPMD (-np 8 -nt 1 ).
- /DEL is not working if use Idrill in property.
- Run hanged in MPI due to issue in ANIM/LSSENSOR with SPMD.

# MotionSolve 14.0 Release Notes

## Highlights

---

MotionSolve is a state-of-the-art multi-body 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.

The major highlights of the 14.0 release are listed below.

- Revamped 3D rigid body contact solution that improves robustness, accuracy and performance
- Non-linear finite element bodies are introduced in MotionSolve (beta release)
- Geometric stiffening support is added for linear flexible bodies
- A new Python Lexicon is available for model building
- Several enhancements to Linear Analysis

### ▼ Improved 3D Rigid Body Contact Solution

3D contact in MotionSolve has been revamped. Described below are the pertinent changes to the solver:

Change	Benefit
A new mesh-mesh collision engine has been implemented	The robustness of models making use of mesh-mesh contact improves significantly. The accuracy and performance have also been seen to be superior to the collision engine used in v13.0.
Contact simulations are parallelized by default in MotionSolve	The run time of simulations containing many contact forces has been seen to decrease significantly, with speedups of about 5x seen in some cases.
Special handling of primitive spheres as colliding geometries	When your model contains contact elements that make use of primitive spherical geometry (a sphere that is not meshed), MotionSolve automatically switches to a more efficient, analytical or semi-analytical collision engine.  The advantage of doing this is that the simulation time can be reduced by as much as 7x.
Initial penetrations are monitored to get the exact point of first contact	A zero crossing sensor that monitors the distance between contacting geometries is now available. The

	<p>zero crossing sensor controls integrator step-size to precisely capture the contact event.</p> <p>By doing this, large initial penetrations leading to unrealistic system behavior are avoided.</p> <p>Note: This sensor is only meaningful for impulsive contacts. If a contact is persistent, then a zero crossing sensor is not needed</p>
<p>New contact force model</p>	<p>A new "Volume" force model that relies on understandable physical parameters such as bulk modulus and shear modulus is now available. This means that contact parameters have a physical basis and can be easily defined.</p>
<p>Contact outputs are written to the output files at times when the penetration depth is maximum</p>	<p>An option to output the system state at the maximum penetration depth in-between two regular output steps is now available. When you run a simulation from MotionView, this is done automatically. You can filter out these "extra" outputs in HyperView.</p>
<p>Enhanced <code>CONTACT()</code> function</p>	<p>The <code>CONTACT()</code> function has been enhanced to provide many more details about the contact such as maximum penetration depth, number of contact patches, total number of contacts, area of contact, etc.</p>
<p>Contact overview table is printed at the end of the simulation</p>	<p>Contact metrics tables are now printed in the MotionSolve log file. You can examine the metrics table to determine what the maximum penetration(s) and normal force(s) is/are in your model and when it occurred. Two tables are printed – one ordered by maximum penetration depth and another ordered by maximum contact force:</p> <pre> Contact Overview TOP 5 (sorted by MAX_PD) ----- CONTACT    I_GRAPHIC  J_GRAPHIC  MAX_PD (TIME)  MAX_FORCE (TIME) ----- 301014     90022      90001      7.436E-01 (0.000E+00)  4.567E+02 (0.000E+00) 301015     90042      90001      7.436E-01 (1.000E-05)  4.575E+02 (0.000E+00) 301018     90102      90001      7.436E-01 (1.000E-05)  4.575E+02 (0.000E+00) 301017     90082      90001      7.436E-01 (0.000E+00)  4.567E+02 (0.000E+00) 301011     90082      90000      7.232E-01 (1.198E+00)  4.058E+02 (1.199E+00)  Contact Overview TOP 5 (sorted by MAX_FORCE) ----- CONTACT    I_GRAPHIC  J_GRAPHIC  MAX_PD (TIME)  MAX_FORCE (TIME) ----- 301018     90102      90001      7.436E-01 (1.000E-05)  4.575E+02 (0.000E+00) 301015     90042      90001      7.436E-01 (1.000E-05)  4.575E+02 (0.000E+00) 301014     90022      90001      7.436E-01 (0.000E+00)  4.567E+02 (0.000E+00) 301017     90082      90001      7.436E-01 (0.000E+00)  4.567E+02 (0.000E+00) 301011     90082      90000      7.232E-01 (1.198E+00)  4.058E+02 (1.199E+00) </pre>

▼ **Non-linear Finite Element Bodies in MotionSolve (beta release)**

MotionSolve 14.0 includes a beta release of the “NLFE Body” which allows you to model non-linearly flexible components in your multibody system. The NLFE Body is based on the Absolute Nodal Coordinate Formulation (ANCF). Nonlinearity can occur for two main reasons: (A) Geometric nonlinearity and (B) Material non-linearity. The NLFE body supports both.

A non-linear modeling component is typically required in the following scenarios:

- A large deformation is expected in the flexible component, or an unusually large deformation is observed in the flexible component
- Stresses in your flexible components approach yield point of the material
- The flexible component is composed of non-linear hyper elastic materials like rubber or polyurethane. The material exhibits a nonlinear stress vs. strain behavior or is hyper elastic
- A linearly flexible modelling component does not provide the fidelity that is desired

NLFE subsystems are included in MotionSolve as a flexible body that is specified through an ANCF XML file. The MotionSolve XML file fragment below illustrates this.

```
<Body_Flexible
  id           = "30102"
  label       = "NLFE Body 0"
  full_label  = "Model-NLFE Body 0"
  ref_marker_id = "30101010"
  ancf_file   = "simo-spinning-beam4_30102.xml"
  rayleigh_damping = "0.5"
  num_i_marker = "3">
  30102021 30102101 30102102
</Body_Flexible>
```

The NLFE subsystem can consist of any combination of atomic MotionSolve NLFE elements. For the example shown above, the NLFE subsystem is specified in the file `simo-spinning-beam4_30102.xml`. The contents of this file are shown in the table below.

```

<ANCFModel>
  <UNIT force="NEWTON" mass="KILOGRAM" length="METER" time="SECOND"/>
  <GRID id="301002" x="0.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0
1.0"/>
  <GRID id="301003" x="0.5" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0
1.0"/>
  <GRID id="301004" x="1.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0
1.0"/>
  <GRID id="301005" x="1.5" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0
1.0"/>
  <GRID id="301006" x="2.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0
1.0"/>
  <GRID id="301007" x="2.5" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0
1.0"/>
  <GRID id="301008" x="3.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0
1.0"/>
  <GRID id="301009" x="3.5" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0
1.0"/>
  <GRID id="301010" x="4.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0
1.0"/>
  <BEAMC id="20000" pid="10000" g1="301002" g2="301003"/>
  <BEAMC id="20001" pid="10000" g1="301003" g2="301004"/>
  <BEAMC id="20002" pid="10000" g1="301004" g2="301005"/>
  <BEAMC id="20003" pid="10000" g1="301005" g2="301006"/>
  <BEAMC id="20004" pid="10000" g1="301006" g2="301007"/>
  <BEAMC id="20005" pid="10000" g1="301007" g2="301008"/>
  <BEAMC id="20006" pid="10000" g1="301008" g2="301009"/>
  <BEAMC id="20007" pid="10000" g1="301009" g2="301010"/>
  <PBEAMC id="10000" mid="3000006" ri1="0" ro1="0.020000" ri2="0" ro2="0.020000" nf="1" nx="5"
nr="4" nt="10" ngx="10" ngr="1" ngt="1"/>
  <MAT1 id="3000006" e="2e+011" nu="0.3" rho="8000"/>
</ANCFModel>

```

MotionSolve supports a comprehensive set of NLFE modeling elements. The table below summarizes the current implemented atomic NLFE entities in MotionSolve. **Note, that MotionView supports only a small subset of these. You can use these in a MotionView model by using a Templex template.**

Element Type	Supported Element Set
Line Elements	BEAM9 (Gradient Deficient ANCF Beam Element) BEAMC (Beam Element with Circular Cross Section) BEAM12 (Fully Parameterized ANCF Beam) CABLE (Cable Element)
Surface Elements	QUAD12 (Rectangular Thick Plate Element) QUAD9 (Rectangular Thin Plate Element) TRIA12 (Triangular Thick Plate Element) TRIA9 (Triangular Thin Plate Element)
Solid Elements	HEXA12 (C1 Brick Solid Element)

	HEXA3 PENTA12 PENTA3 TETRA12 TETRA3	(C0 Brick Solid Element) (C1 Penta Solid Element) (C0 Penta Solid Element) (C1 Tetra Solid Element) (C0 Tetra Solid Element)
Scalar and Bushing Elements	ABUSH LINE2 LINE3 LINE4	(Bushing Element) (Two nodes line spring element) (Three nodes line spring element) (Four nodes line spring element)
Mass Elements	CONGM CONPM	(Concentrated mass element connection) (Concentrated point mass)
Connector Elements	CONN0 CONN1 CONN2 CONN3	(Connector element) (Connector element) (Connector element) (Connector element)
Linear Elastic Material Models	MAT1 MAT1LS MAT6	(Continuum Mechanics Approach ) (Linear Strain Material Model) (Elastic Line Approach)
Hyper-Elastic Material Models	MAT2 MAT3 MAT4 MAT5	(Incompressible Neo-Hookean Material Model) (Compressible Neo-Hookean Material Model) (Mooney-Rivlin material model) (Yeoh material model)
Anisotropic Material Model	MAT7 MAT7ORT MAT9 MAT9ORT MAT9TRA	(Thin Plate Element Anisotropic Material Model) (Thin Plate Element Orthotropic Material Model) (Anisotropic Material Model) (Orthotropic Material Model) (Transverse Isotropic Material Model)
Geometric Properties	PABUSH	(Bushing element properties)

	PBEAM9 (Arbitrary Cross Section Beam element properties) PBEAMA (Arbitrary Cross Section Beam element properties) PBEAMC (Circular beam element properties) PBEAML (Property card of the ANCF beam elements) PCABLE (Cable element properties) PLINE (Line spring property) PSHELL (Plate elements properties) PSOLID (Solid elements properties)
Other Inputs	GRID UNITS

For more information about these elements, you may refer to the NLFE Reference Manual.

MotionSolve also writes out a 3D representation of the BEAM and CABLE elements that can be animated in HyperView. Displacement, stress and strain contours can be visualized on the NLFE body. Other operations such as section cuts can also be applied.

For more details, please refer to the MBS solution release notes. Also, please note that this capability is being offered as a beta or experimental feature and is not expected to be used in a production environment.



## Solver Enhancements

---

### ▼ New Python Based Lexicon

A new Python based lexicon for MotionSolve is provided in this release. Using this lexicon, one can now create Python applications that can:

- Create a MotionSolve model
- Execute the model
- Use any other Python utilities that are useful in this context.

For instance, you could: Perform a MotionSolve simulation and display live plots during the simulation with Matplotlib (a Python based plotting tool).

Along with the Python based lexicon, a new Python lexicon Reference Manual is also available.

### ▼ Support for Multi-Iterative Coupling (MIC) in Co-Simulation with AcuSolve

With this release, MotionSolve supports multi-iterative coupling (MIC) while co-simulating with AcuSolve. This enhancement allows MotionSolve to re-compute its system states on the fly when needed during co-simulation. This is helpful while trying to solve “tightly” coupled problems more accurately. With this enhancement, your co-simulation results are more accurate and the co-simulation is more robust.

### ▼ MS\_USERDLL\_DIR Supports Python Scripts

You can use the MS\_USERDLL\_DIR environment variable to specify the location of your Python scripts (\*.py, \*.pyc) as well as your C++/FORTRAN user subroutines.

The environment variable MS\_USERDLL\_DIR can be set to a list of folder locations on your machine (separated by ;) where you want MotionSolve to search for your user subroutines (\*.dll, \*.so). This scenario is common when you don't specify the absolute path of the user subroutine in your model or if the user subroutine is not in the same location as the solver input XML file.

### ▼ Geometric Stiffening for CMS Flexible Bodies

You can model geometric stiffening for CMS flexible bodies in MotionSolve. Geometric or stress stiffening is an effect most commonly seen in beams under tension where the frequency of the fundamental modes increases as the tension is increased. Such an effect can be observed in rotating beam structures like helicopter or wind turbine rotor-blades.

Include the stress stiffness data in your CMS flexible body by including the following in your FEM file:

- `CMSMETH, 1, CBG, , 15`

CBG is the keyword that instructs OptiStruct to create the CMS flexible body with additional geometric stiffness data

You can then import this flexible body in your MotionView model as you would ordinarily.

**Note:** Only the Craig-Brampton method is currently supported for calculating the geometric stiffness data.

▼ **Linking with msautoutils via the User Subroutine Build Tool**

The user subroutine build tool shipped with this release of MotionSolve now links with the msautoutils library on both Windows and Linux. You can now use access functions exported by the MotionAuto Utilities as well as the MotionSolve UserSub API. This substantially simplifies the effort on your part to create your own road or other Auto functionality based on the published set of capabilities.

▼ **Updated ISTAT Codes for the GTCMAT Utility Subroutine**

The ISTAT return codes for the GTCMAT utility have been updated in MotionSolve.

ISTAT= 0	Normal return. Compliance matrix was calculated.
ISTAT= -1	Error return. $NM < 0$ .
ISTAT= -2	Error return. $NM > 100$ .
ISTAT= -3	Error return. $NC < 6 * NM$ .
ISTAT= -6	Error return. MKID refers to a non-existent REFERENCE_MARKER.
ISTAT= -7	Error return. The system has zero degrees of freedom.
ISTAT= -9	Error return. A singular Jacobian matrix was encountered during the calculations.
ISTAT=-11	Error return. A REFERENCE_MARKER is on GROUND. It cannot move.
ISTAT=-12	Error return. An initial static equilibrium was not performed.
ISTAT=-99999	Error return.

▼ **Linear Analysis Enhancements**

Many enhancements have been added to Linear Analysis. These include:

- A new option for “balancing” prior to eigenvalue analysis
- Support for strain/dissipative energy calculations
- Mode selection for linear analysis output

The updated statement is shown below with the 14.0 enhancements in red.

<Param\_Linear

```

anim_scale      = "real"
balancing       = {"TRUE" | "FALSE" | "AUTO"}
disable_damping = {"TRUE" | "FALSE"}
pinput_id      = "integer"
poutput_id     = "integer"
write_eig_info  = {"YES" | "NO"}
write_energy_dist = {"YES" | "NO"}
write_matlabfiles = {"YES" | "NO"}
write_simulinkmdl = {"YES" | "NO"}

{ke_modes_exclude = options}
{ke_modes_include = options}

{se_modes_exclude = options}
{se_modes_include = options}

{de_modes_exclude = options}
{de_modes_include = options}

```

/>

Options:

- Select a single mode
- Select multiple modes
- Select a range of modes
- Select all modes

### A new capability for “balancing” prior to eigenvalue analysis

For certain models, the eigenvalues calculated are sensitive to very minor changes in the state matrix used as input for the solution. This occurs when the eigenvector matrix is ill conditioned. MotionSolve detects an ill conditioned eigenvector matrix and “balances” the state matrix for a more robust eigenvalue solution.

Balancing refers to diagonally scaling the state matrix such that the row and column norms are numerically close to each other. You can control this behavior using the attribute “balancing” in the <Param\_Linear> model or command statement.

Set “balancing” to “TRUE” to force MotionSolve to perform balancing on the state matrix and “FALSE” to disable balancing. The default for “balancing” is “AUTO” which lets the solver decide when balancing is required based on the condition number of the eigenvector matrix. Please note, that balancing is a very inexpensive operation.

### Support for strain/dissipative energy calculations

For linear analyses, in addition to Kinetic Energy distribution, MotionSolve also calculates strain and dissipative energy for the following modeling elements:

- Force\_SpringDamper (TSPDP/RSPDP)
- Force\_Bushing (BUSH)
- Force\_Beam (BEAM)

- Force\_Field (FIELD)
- Force\_VectorOneBody/Force\_VectorTwoBody (VFORCE, VTORQUE, GFORCE)
- Force\_Scalar\_TwoBody (SFORCE)

The KE, strain and dissipative energy distribution is written to the log file, on the screen and the \*\_linz.mrf output file. To request MotionSolve to calculate these distributions, set the write\_energy\_dist attribute to "TRUE" in the <Param\_Linear> model or command statement.

### Mode selection for linear analysis output

You can now specify the modes for which MotionSolve writes the energy distribution on screen and to the log file. This is useful when the number of eigenmodes is large and you are interested only in a subset.

You may specify mode numbers to include or exclude, for each of the three distributions. Please see the documentation on <Param\_Linear> for more details on how this is done.

### ▼ Header Format Change for PLT Output Files

With this release, the header for the PLT output files has been changed to allow the PLT file to be imported into FEMFAT for fatigue analysis. The old and new PLT format are described below:

Old format: The old format contains 4 columns for each request and the version number is 8.1.

```
MotionSolve v8.1 Plot File
Generated on 12-AUG-2015 12:27:29, Altair Engineering, Inc.
  19      MMKGS_N_RACA 1.0000000000000000e+003
  70000005          4      30102080      30101080
rg1(Output Body1)
  70000006          4      30102020      30103020
r12(Output Body1)
  70000007          4      30103020      30102020
```

New format: The new format contains 5 columns for each request and the version number is 9.1. The fifth column is added as per the specifications from FEMFAT. If this value is non-zero, then the next line is treated as a comment line.

```
MotionSolve v9.1 Plot File
Generated on 12-AUG-2015 12:09:55, Altair Engineering, Inc.
  19      MMKGS_N_RACA 1.0000000000000000e+003
  70000005          4      30102080      30101080          5
rg1(Output Body1)
  70000006          4      30102020      30103020          5
r12(Output Body1)
  70000007          4      30103020      30102020          5
```

### Kinematic analysis for "structure" models

Previously, MotionSolve did not solve models that have 0 DOF and time dependent loading. To solve such models you needed a workaround with "dummy" differential equations etc. With this release, MotionSolve has been enhanced to recognize 0 DOF models that have time dependent loading such

that the simulation can continue. Such a scenario is useful when trying to determine joint reaction forces for models with 0 DOFs.

## ADAMS Compatibility

---

### ▼ FX function compatibility

The definitions of the functions FX, FY, FZ, TX, TY and TZ have been made consistent with what is used in ADAMS. The table below lists the changes, using the FX function as an example:

Function	Arguments	Behavior
FX (I, J, K)	I, J and K markers	Computes the X component of all the forces acting between marker I and J resolved in the coordinate system of marker K
FX (I, J)	I and J markers	Computes the X component of all the forces acting between marker I and J resolved in the ground coordinate system
FX (I)	I marker	Computes the X component of all the resultant forces at marker I. To calculate the X component of action-only forces at the I marker, specify <code>frc_expr_action_only = "TRUE"</code> in the <code>&lt;ResOutput&gt;</code> command element in the XML input file

### ▼ GRAPHIC Statement Compatibility

MotionSolve now supports the adm statement GRAPHICS/... for displaying force vectors in the animation H3D. A sample statement is

```
GRAPHICS/10215001, FORCE, ETYPE=ALL, EM=10215001
```

The following ETYPE are now supported:

ETYPE	ALL, BEAM, BUSH, FIELD, GFORCE, JOINT, JPRIM, NFORCE, SFORCE, SPDP, VFORCE, VTORQUE
-------	---

### ▼ Correct Conversion of Output Steps to XML for Multiple Analyses

Previously, a model that had multiple SAVE/RELOAD statements and analyses with different output steps was not translated correctly to MotionSolve. An example of such a scenario is shown below:

```
RELOAD/SYSTEM, FILE=try_model  
SIM/STA, END=5.0, STEP=20  
SIM/STA, END=6.0, STEP=5  
SIM/STA, END=6.1, STEP=5  
SIM/STA, END=6.5, STEP=4  
SIM/STA, END=6.9, STEP=4  
SIM/STA, END=7.0, STEP=5
```

This has been resolved within this release.

## Resolved Issues

---

### ▼ Solver Behavior Updated for Vehicle Models Containing Tires

With this release, the solver behavior for any model that contains tires has been updated to produce more accurate velocities and accelerations. The solver now recognizes force elements that model the tire and monitors the rotating wheel bodies so that the velocities and accelerations are computed more accurately.

For models that make use of the <[Force\\_VectorTwoBody](#)> modeling element to represent tires, you can expect to see smoother results slip ratio and tire rotational velocity.

### ▼ Using Python User-Subroutine Scripts on a Different Network Location than MotionSolve

Previously, MotionSolve was unable to access a Python script that was not located on the same machine as the MotionSolve installation. This issue has been fixed, now, you can run a simulation in a scenario where MotionSolve is installed on one machine and the scripts are located on another network location.

**Note:** You need to have read access to the network location where your scripts are located.

### ▼ Flexbody Visibility of MotionSolve H3D in HyperView

Previously, if your model consisted of multiple flexible bodies generated using a combination of PLOTTEL and non-PLOTTEL elements, the flexible bodies would not be displayed properly in HyperView. This issue has been fixed within this release.

### ▼ Resolved Issues in Linear Analysis

Several issues have been fixed for the linear analysis within this release:

- If you request Simulink or MATLAB output files after a linear analysis, MotionSolve requires a valid <[Control\\_PlantInput](#)> model element that defines the input and a <[Control\\_PlantOutput](#)> model element that defines the output for the linearized system. If these IDs are not specified or specified incorrectly, MotionSolve will issue an error message and stop the simulation. These IDs can be specified via the [pinput\\_id](#) and [poutput\\_id](#) attributes in the <[Param\\_Linear](#)> model or command statement. See the documentation for <[Param\\_Linear](#)> for more details. For some models, MotionSolve wrote incorrect B and C matrices after a linear analysis when “[write\\_matlabfiles](#)” is set to [TRUE](#). This was determined to be an issue with numerical tolerance and has been fixed for this release.
  - For a model that contained only flexible bodies, MotionSolve crashed while writing the Kinetic Energy distribution. This issue has been fixed within this release.
  - While requesting a linear analysis on a model with no force elements (or zero total force), MotionSolve crashed during the linear analysis simulation. This was determined to be an error in the solver code and has been resolved starting with this release.
- When the ADM/ACF deck contained more than one linear command statements, the translation to XML resulted in a <[Param\\_Linear](#)> statement which was not equivalent. This has been resolved within this release.



#### ▼ Building User Subroutines by using the GCC Compiler on Linux

In the previous release, you were unable to link your user subroutine code with the MotionSolve user subroutine API library ([ms\\_usersubapi](#)) using the GCC compiler. Starting with this release, this restriction has been removed. Now you can use both the GCC and Intel compilers to compile and link your user subroutine code.

#### ▼ Incorrect Gear Ratio Calculated When Using the Constraint\_Gear Modeling Element

Previously, a gear modeled using the [<Constraint\\_Gear>](#) statement resulted in an incorrect gear ratio calculation in the solver. This has been fixed within this release.

#### ▼ The SENVAL Function Returns an Incorrect Value When the Sensor is not Activated

The [SENVAL](#) function should return a value of 0 when the sensor it refers to is not active during the simulation; this behavior was not seen in the previous release. This has been fixed starting with this release.

#### ▼ SENSOR Causes a Crash if Deactivated Before the First SIMULATE Command

If your model contains a [<Sensor\\_Event>](#) modeling element that is deactivated before the first [<Simulate>](#) command is executed, the simulation would crash with the previous release.

This was determined to be a bug and has been fixed with this release.

#### ▼ Post-processing Module (MS-Post) Crashes with '0' Specified as a Marker ID

Previously, the post processing module in MotionSolve crashed if your model contained an output request that referred to a '0' marker ID. This scenario may occur with older MotionSolve models. This has been fixed such that the post module simply ignores such a marker request.

#### ▼ PROXIMITY Function Referring to a Deactivated Sensor Crashed the Simulation

Previously, if your model contained a PROXIMITY function that referred to a deactivated sensor in your model, the simulation would crash in such a scenario. This has been fixed such that the use of such a PROXIMITY function does not halt the simulation.

**Note:** It is important to keep track of which sensors are deactivated in your model since the PROXIMITY function referring to a deactivated sensor will return a 0 value.

#### ▼ Rigid Body Contact for Box Graphic Defined with a Negative Length

Previously, if you modeled 3D rigid body contact with at least one body being a box graphic defined by a negative length, the contact force was not generated as expected. This has been fixed with this release.

#### ▼ IF Function Doesn't Return User Defined Value

Previously, if you defined a solver variable using multiple IF statements and queried the solver for this variable's value in a [Post\\_Request](#) using [VARVAL\(..\)](#), MotionSolve did not return the exact values for the variable as defined by the user. Specifically, the values were incorrect at the points where the gradient of the expression for the solver variable is discontinuous.

This issue can be resolved by specifying [dae\\_interpolation = "FALSE"](#) in the [<Param\\_Transient>](#) model or command statement in the MotionSolve XML input file.

#### ▼ Missing "XP" and "ZP" of Marker in XML

Markers with direction cosine matrix very close to the identity matrix were not translated correctly to the MotionSolve XML input file. This was caused by a loose numerical tolerance in MotionSolve.

#### ▼ Sign of Velocity at Discontinuous Operating Point

Previously, for a kinematically constrained model, if you specified a motion on a joint using multiple IF statements, the velocity of the marker associated with the joint showed a difference if the model was solved using the [VSTIFF](#) or [DSTIFF](#) integrator.

This has been fixed within this release – the choice of the integrator does not matter for such a kinematic model.

**Note:** Use of the [IF](#) function to define motions is not recommended in MotionSolve, since the [IF](#) function tends to produce discontinuities which make it hard for the solver to converge and can produce erroneous results in certain cases. Please refer to the documentation on [<Motion\\_Joint>](#) for more tips on defining smooth motion inputs.

#### ▼ Incorrect Gear Ratio When Using "Constraint\_Gear" Model Element

Previously, a gear modeled using the [<Constraint\\_Gear>](#) statement returned an incorrect gear ratio in MotionSolve. This has been fixed within this release.

#### ▼ State Variable Doesn't Return User Defined Value Correctly, Especially in Transient Analysis

Previously, a Motion defined as a saw-tooth function using a series of IF statements returned results inconsistent with the user input. This has been fixed within this release.

#### ▼ Multiple Simulates with Time Dependent Motion Result in Incorrect Answers

Previously, if your model contained multiple `Simulate` commands and a `<Motion_Joint>` modeling element was modified in between the analyses, MotionSolve did not update the `<Motion_Joint>` value correctly thus yielding incorrect results or simulation failure. Such a scenario is shown below:

```
<Motion_Joint
  id          = "1"
  expr        = "STEP(TIME,0,0D,1,360D)"
/>
<Simulate
  analysis_type = "Static"
  end_time      = "1."
/>
<Motion_Joint
  id          = "1"
  expr        = "360D+0.0*(TIME-1)"
/>
<Simulate
  analysis_type = "Static" `
  end_time      = "2."
/>
```

This has been resolved with this release.

#### ▼ **Suspension Design Factor (SDF) Outputs Caused Certain Models to Fail During GTCMAT Calls**

Previously, certain models that had SDF outputs defined in them failed the GTCMAT calls. This was determined to be an issue in the solver code and has been fixed with this release.

#### ▼ **Save and Load Issue with Models Containing Joint Friction**

Previously, if you save and load your model that contains joint friction modelling elements, the reloaded model will not solve correctly. This has been identified as an error in writing the model upon saving.

## Known Issues

---

### ▼ Using Non-linear Finite Element Bodies

Certain models containing NLFE components may not solve with `max_order` > 2. Try to set the `max_order` to 1 or 2 and re-run your model

The damping parameter for NLFE components is a coefficient for a Rayleigh damping model. Since this is a numerical damping, it is advised to start from a value of 0 in your model, since critical damping can be achieved with a small value and a larger value may yield unrealistic results.

### ▼ Wrong Results with Custom Elements for a Specific Scenario

When using a custom element definition (including <Force\_JointFriction>) that uses GFORCE with the ground being defined as the I body, the results may be incorrect. As a workaround, insert a dummy body between the ground and the J body.

# HyperXtrude Solver 14.0 Release Notes

## HyperXtrude for Metal Extrusion

---

### ▼ Automatic Ram Speed Correction Based on Press Tonnage

The solver can automatically adjust the ram speed based on the limits imposed by the press data. This will help to determine the maximum ram speed possible on the press for the given process data.

### ▼ Improvements to Modeling Choked Bearing

An improved algorithm is implemented to compute friction at choked bearing regions based on test data and the computed results correlate well with extrusion field data.

### ▼ Dead Cycle Time

The solver is enhanced to include dead cycle time in a full cycle transient analysis. This helps in computing more accurate temperature distribution before the next cycle begins.

### ▼ Thermal Analysis of Tool Heating Systems

The solver has a new feature to do a thermal analysis of the tool heating systems and control tool assembly temperature. HyperXtrude can now perform a transient analysis to monitor temperature at the pre-defined thermocouple locations and control the heating elements based on the feedback from thermocouples. Since temperature plays a critical role in extrusion, this analysis is useful in understanding and controlling the thermal systems that heat components such as, container, liner, and die assembly.

### ▼ IGES File Export of Optimized Bearing Curves

When doing bearing optimization, HyperXtrude will now write CAD files in IGES format for the optimized/corrected bearing curves. This will enable easy import of these optimized bearing curves into HyperMesh or HyperXtrude 2015.

### ▼ Tool Deflection Analysis Using OptiStruct

The solver is enhanced to perform coupled simulation using OptiStruct for tool stress analysis. In this approach, after performing extrusion analysis on the workpiece, HyperXtrude will automatically invoke OptiStruct with the computed loads to perform tool stress analysis. Extrusion analysis is then continued by taking computed tool displacements into consideration.

▼ **Resolved Issues**

- Error in writing displacements in H3D file for bearing optimization runs has been resolved.
- Improvements to bearing friction module has been made to increase the accuracy of the solver.
- Resolved errors in gap contact load computations in coupled analysis using HyperXtrude.
- Resolved limitations in particle traces module. This data is used with a standalone code for visualizing particle traces with history in HyperView.
- Resolved issues when skin tracking is enabled in models that use lookup table for material data.
- Enabled function/table specification of heat transfer on free surface BCs.
- Resolved many minor issues related to transient analysis as part of dead cycle time implementation.

## HyperXtrude for Polymer Extrusion

---

### ▼ Improvements to Calibration Analysis

Major enhancements are made to calibration analysis and these include improvements to computational time and extracting additional useful results specific to cooling of the extruded profile. New results include solidified layers, solidification time, degree of cooling, and cooling time.

### ▼ Resolved Issues

- Issues related to viscosity data in lookup table format (as function of strain rate and temperature) have been resolved.

## HyperXtrude for Resin Transfer Molding

---

### ▼ Resolved Issues

- Issues related to VARTM resin infusion has been fixed.
- Improved robustness of filling even for complex models with drastically varying data



# AcuSolve 14.0 Release Notes

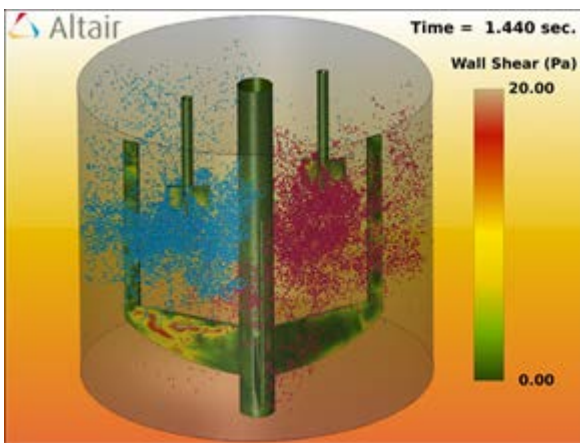
## Highlights

The 14.0 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 14.0 release provides improvements to AcuSolve's moving mesh technology, new multiphysics coupling in the form of finite mass particle tracing, revised turbulence models, newly developed tutorials, an expanded validation manual, and many other enhancements. 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.

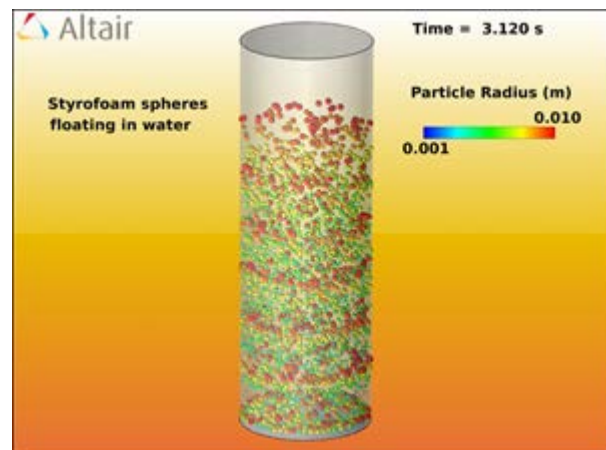
### ▼ Finite Mass Particle Tracing Using AcuSolve and AcuTrace

AcuTrace has been enhanced to support tracing of spherical particles having finite mass. When exercising this option, the particle velocity is no longer governed solely by the local flow velocity. The motion of the particle is computed by solving a conservation equation that incorporates the mass of the particle, the pressure force, the drag force, the viscous stresses, and the virtual mass force. The user is given control over which terms in the conservation equation should be included at run time and various drag laws are available to choose from. Wall/particle interactions are handled with user-specified coefficients of restitution to define the energy loss as a result of the collision.

In addition to supporting finite mass particle tracing, the bi-directional coupling between AcuSolve and AcuTrace also provides support for momentum exchange. For the case of bi-directionally coupled finite mass particle tracing, the effect of the particle's presence appears in the AcuSolve flow field. The momentum of each particle is computed, and the corresponding reaction force is added to AcuSolve as a source term in each element that the particle passes through.



Finite mass particles in suspension within a mixing tank



Buoyant particles rising in a tube

Note that particles are limited to idealized, spherical shapes in the current release, and the particles are assumed to constitute a dilute and disperse phase within the fluid. In addition to this, particle-particle collisions are not yet supported. As a general guideline for use, it is suggested that the

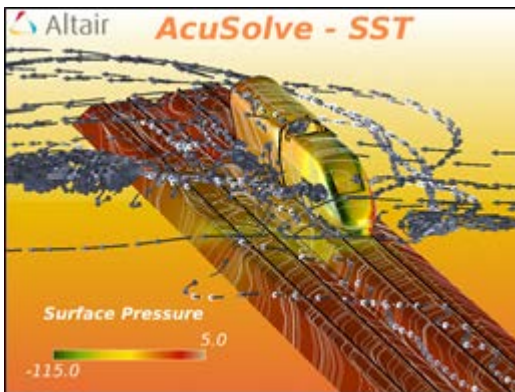
ratio of particle spacing to particle diameter should be on the order of 10 or greater for massless particles and on the order of 100 or greater for finite mass particles.

Common applications for finite mass particle tracing include fluidized bed simulations and other applications involving the transport of small particulates within a flow field.

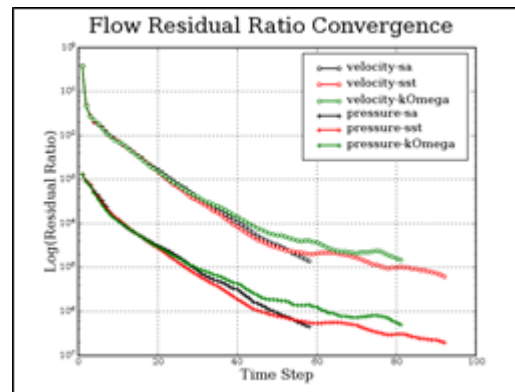
### ▼ Revised Implementation of SST and k-omega Turbulence Models

The 14.0 release contains major improvements to the SST and k-omega turbulence models within AcuSolve. The new formulation improves the accuracy and robustness of the models by incorporating a change of variables for the omega equation. Instead of solving for the standard k

and omega equations, the new formulation solves for k and  $\frac{1}{\sqrt{\omega}}$ . The change of variables does not alter the results produced by the underlying turbulence model, but does have significant benefits with respect to numerical robustness. The new formulation has vastly improved near wall behavior; having a much more agreeable wall boundary condition of 0.0 for the omega transport equation, as well as the ability to be managed easier through wall functions. Internal testing and validation has shown that this change of variables produces far superior behavior of the models in comparison to past releases.



SST results showing flow over a rail car



A comparison of convergence rate between turbulence models for the rail car

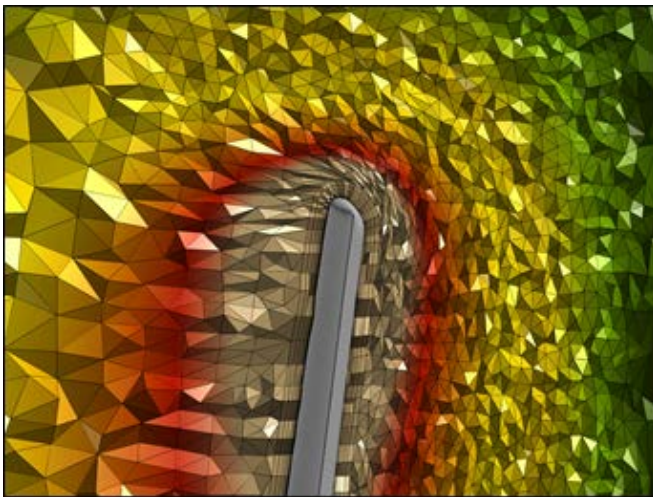
The new models are fully compatible with input files created in older versions of the code. The nodal initial conditions and boundary condition values are still specified in terms of the `eddy_frequency` variable. All conversions to the newly introduced transport quantity ( $\frac{1}{\sqrt{\omega}}$ ) are handled automatically. The new transport quantity (`sqrt_eddy_period`) will appear in nodal field outputs along with the `kinetic_energy` and `eddy_frequency` variables.

The revised implementations of SST and k-omega are complete with support for wall functions and wall roughness. Internal testing has shown that a stretch ratio of 1.3 provides optimal results when using wall functions.

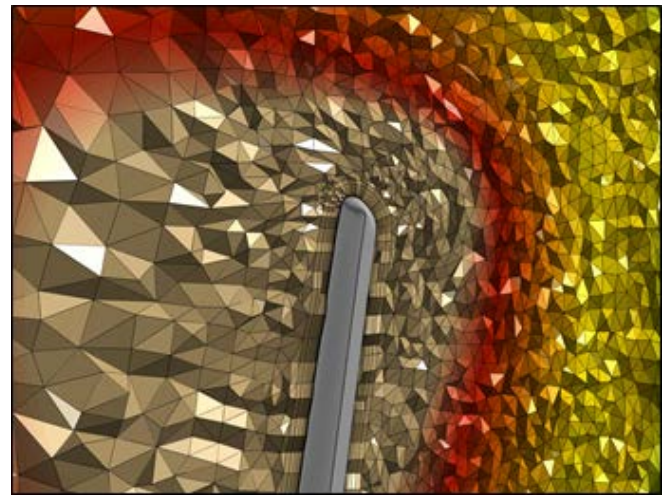
It should be noted that the most efficient solution strategy for these models involves iterating multiple times on the turbulence variables for every iteration of the flow equations. AcuSolve's `AUTO_SOLUTION_STRATEGY` command has been updated to reflect this solution strategy.

### ▼ New Mesh Quality Metric (MQM) Technology Added to ALE Mesh Motion

AcuSolve's Arbitrary Lagrange-Eulerian (ALE) mesh motion technology has historically been based on a hyperelasticity model to control the deformation of the mesh. Using this approach, the stiffness of a given element is 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 formulation leads to undesirable results. A new method of computing the motion of the mesh has been introduced in AcuSolve V14.0. Using the new formulation (i.e. Mesh Quality Metric or MQM), the mesh deforms in such a manner as to maintain the original shape of each element. Each time the boundaries of the simulation are deformed, the mesh is updated using an algorithm that attempts to optimize the quality of the mesh. The original shape of the elements is used as the reference for which the mesh was assumed to be of optimal quality. In practice, this method has a number of benefits over the hyperelastic model. This technique implicitly retains the shape of elements within the boundary layer, which reduces user burden to constrain near wall nodes to move with deforming surfaces. Additionally, this method inherently moves large deformations to the larger elements in the domain. Even with ~5000:1 aspect ratio elements near a deforming wall, the MQM approach retains the structure of the boundary layer elements while absorbing the deformation in the far field elements. See the images below for an example of the behavior of MQM. Note that the hyperelastic model required constraints on the boundary layer elements to deform to the level shown, whereas the MQM approach handled the boundary layer elements without additional constraint.



The mesh deformation predicted by hyperelastic mode



Mesh quality metric

### ▼ Support for Mixed Topology Surfaces and Volumes

AcuSolve V14.0 now supports the definition of surfaces and volumes using mixed element topology. In previous releases of the solver, it was necessary to specify all input commands such that only a single parent element set and element type were referenced by each command. This restriction has been eliminated with the 14.0 release and all solver input commands are functional using the mixed topology input.

Support for the mixed topology input is achieved through the introduction of the `SURFACE_SET` and `VOLUME_SET` commands. With these commands in place, it is now possible to read in the individual surface and volume definitions for later reference in the input file. Surface sets or volume sets of different topology may then be referenced in a single solver command.

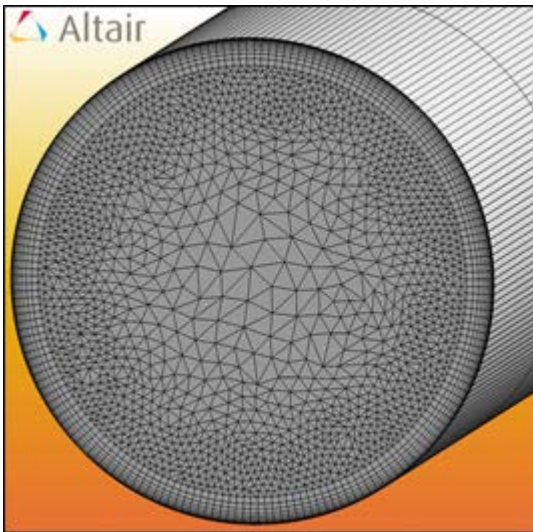


Illustration of mixed topology surface input in AcuSolve.

For example:

```
# +-----+
# | Surface sets |
# +-----+
SURFACE_SET( "inflow tri3 Fluid wedge6" ) {
    surfaces          = Read( "MESH.DIR/inflow.tri3.ebc.B" )
    shape             = three_node_triangle
    volume_set        = "Fluid wedge6"
}
SURFACE_SET( "inflow quad4 Fluid brick8" ) {
    surfaces          = Read( "MESH.DIR/inflow.quad4.ebc.B" )
    shape             = four_node_quad
    volume_set        = "Fluid brick8"
}

# +-----+
# | Simple Boundary Condition |
# +-----+
SIMPLE_BOUNDARY_CONDITION( "inflow" ) {
    surface_sets      = {"inflow tri3 Fluid wedge6","inflow quad4 Fluid
brick8"}
    type               = inflow
    inflow_type       = average_velocity
}
```



```

average_velocity      = 1.0
}
SURFACE_OUTPUT( "inflow" ) {
    surface_sets      = {"inflow tri3 Fluid wedge6","inflow quad4 Fluid
brick8"}
    integrated_output_frequency = 1
}

```

This new capability represents a significant change to the architecture of AcuSolve and is being released in phases. This first release includes support for all solver commands in the AcuSolve input file, and full post-processing support in AcuTrans, AcuFieldView, AcuProbe, and HyperView. The new input file format is currently not supported by AcuConsole, HyperMesh, AcuTrace, or AcuFwh. Support for the new format will be added to the remaining programs in a future release. Note that AcuSolve continues to support the older, single topology format for all commands as well. As such, full backward compatibility is retained.

▼ **Introduction of the AcuSolve Tutorials**

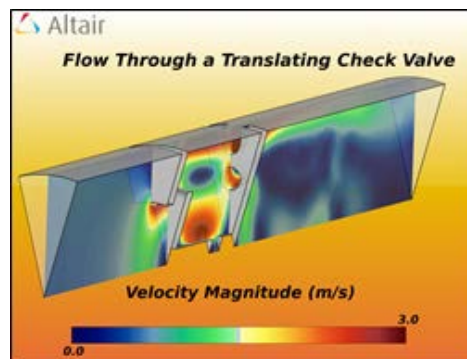
The AcuSolve installation now comes equipped with tutorials that provide detailed instructions for getting started with the code and working with some of the more commonly used features. The purpose of the document is to provide self contained sets of exercises and instructions for both new and experienced users of the software. The tutorials provide a brief overview of each simulation that is covered, along with the instructions on how to properly set the model up. All prerequisite files necessary to perform the simulations are included. The instructions within each tutorial include details on the meshing, solver settings, as well as post-processing of the model.

As with other AcuSolve documentation, the tutorials are available from the AcuSolve Help Welcome page. All files necessary to run the simulations that are described in the document are available in the distribution within the following directory:

<installation\_directory>/model\_files/tutorials/AcuSolve>. The current version includes 5 tutorials, ranging in complexity from steady, isothermal flow to externally coupled multi-physics applications. Additional tutorials will be added in future releases of AcuSolve.



AcuSolve Help Welcome Page showing links to the AcuSolve tutorials



Results from the translating check valve tutorial

▼ **Addition of Smoothing to Interpolated Mesh Motion**

A newly developed smoothing algorithm has been added to interpolated mesh motion (MMI). The impact of smoothing is to improve the behavior of MMI when the background mesh introduces local oscillations in the displacement in the mesh. This option reduces the likelihood of elements collapsing and also smooths the overall displacements that are enforced by MMI. It should be noted that this feature does have some additional implications. Under these conditions, the interpolated mesh motion algorithm is allowed to modify the position of any node that has MMI applied to it. For degrees of freedom that are not entirely constrained by the MMI, the smoothing algorithm can modify the position. For example, the behavior of `mesh_displacement_type = slip` is now different for surfaces that are part of an element set that contains an interpolated mesh motion. The unconstrained degrees of freedom (in the wall parallel direction) are now controlled by the smoothing algorithm and can have non-zero displacement.

### ▼ Separation of Mesh Motion Constraints from Simple Boundary Condition Commands

AcuSolve's `SIMPLE_BOUNDARY_CONDITION` command was developed as a mechanism to automate the application of suitable constraints to represent common types of physics. Inlets, walls, and outlets are examples of the common physics that are addressed by `SIMPLE_BOUNDARY_CONDITION`. As part of this command, the constraints that dictate the mesh motion were also added. Although this does provide a convenient way of assigning constraints, it can introduce some restrictions in flexibility. To alleviate this, a new command has been created that separates the mesh motion constraints from the `SIMPLE_BOUNDARY_CONDITION` command. The new command, `MESH_BOUNDARY_CONDITION`, provides a centralized mechanism for assigning mesh motion constraints that were previously associated with the `SIMPLE_BOUNDARY_CONDITION`, `FREE_SURFACE`, and `EXTERNAL_CODE_SURFACE` commands. To accomplish this, AcuPrep internally creates a table of constraints for each element face/node on the boundaries and identifies conflicts within the mesh motion boundary conditions. These conflicts may arise due to combinations of `MESH_BOUNDARY_CONDITION` and the legacy commands that provide similar functionality. In the case of conflicting constraints that have equal precedence values, the `MESH_BOUNDARY_CONDITION` is given priority. If the precedence values are not equal, then the higher precedence constraint is enforced. AcuPrep also checks for compatibility of the mesh motion boundary conditions on each node with the boundary conditions applied for the physics through the `SIMPLE_BOUNDARY_CONDITION` command.

An example showing how to use the newly introduced command to define a `planar_slip` mesh motion constraint is as follows:

```
MESH_BOUNDARY_CONDITION ( "slip wall" ) {  
    surface_sets          = {"tri faces","quad faces"}  
    type                  = planar_slip  
}
```

The introduction of the `MESH_BOUNDARY_CONDITION` command deprecates the `FREE_SURFACE` and `EXTERNAL_CODE_SURFACE` commands. In addition to consolidating commands, the `MESH_BOUNDARY_CONDITION` implementation contains more complete error checking. Surfaces that are defined as `planar_slip` (equivalent to `mesh_displacement_type=slip` in `SIMPLE_BOUNDARY_CONDITION`) are now checked to ensure that a consistent normal direction exists for the entire surface set. Full details of the command, along with all supported options, are included in the AcuSolve Command Reference Manual.

The `MESH_BOUNDARY_CONDITION` command is currently supported within the solver and post-processing programs. Full GUI support will be added in a future release. Note that AcuSolve

continues to support the legacy commands that provided control over mesh motion boundary conditions. As such, full backward compatibility is retained.

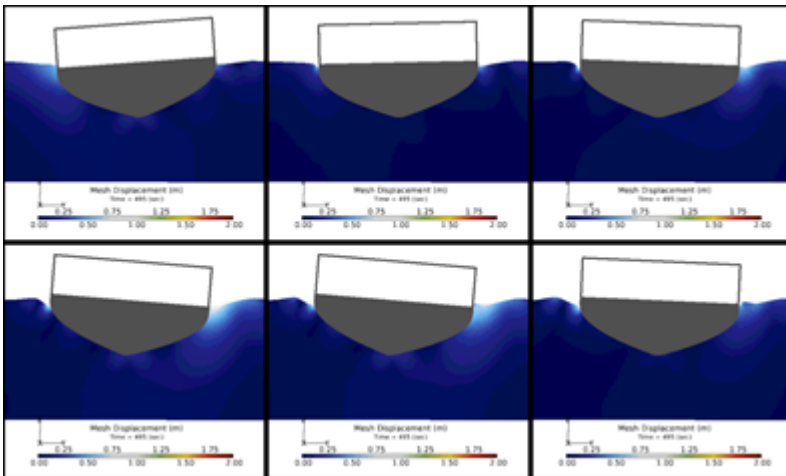
▼ **Added Support for Distributed Memory Parallel Runs Across Windows Machines**

AcuSolve now supports running in distributed memory parallel mode across multiple Windows hosts. In the past, this could only be achieved when running on a Windows HPC Server cluster. This restriction has been lifted, and it is now possible to run in parallel on multiple Windows hosts using Intel MPI, Platform MPI, MS-MPI, and MPICH2. It should be noted that some set-up work is required to install services and create shared directories. Please see the HyperWorks Installation manual for further instructions.

As part of this effort, the process of setting up parallel simulations for single host applications has been simplified as well. It is no longer necessary to install the SMPD daemon for Intel MPI when running in parallel on a single Windows host.

▼ **Improved CFD Post-processing With AcuFieldView**

AcuSolve 14.0 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 improvements when working with AcuSolve data. The AcuSolve direct reader has been updated to provide full support for mixed element topology surfaces. In addition to this enhancement, the performance of the direct reader has been improved dramatically. The performance improvements have been accomplished through a combination of code changes as well as newly added options that enable the user to skip compute intensive steps that are not necessary for all simulations. Some of the changes that have been made include an ability to avoid reading of AcuSolve's extended output variables. Disabling this option alone can improve read times by as much as 2x. Users have the option to remove duplicate surfaces as well, which brings consistency with AcuTrans. Please see the AcuFieldView User Guide for full details.



Multiple images displaying custom color maps and a multi-window layout.

▼ **Other Notable Changes**

AcuSolve 14.0 contains a number of other notable changes that are worthy of mention. A brief description of each is shown below:

- Uniformity index has been added as an output quantity. This quantity is part of surface statistics output.
- User defined functions on Windows now automatically recompile at run time if the library corresponding to the requested message passing protocol is not present.
- `moment_center` has been added to AcuTrans to enable user control of the location about which moments are computed. In previous releases, the moment was always computed about the global origin.
- Added ability to specify viscosity as a function of pressure.
- Added support for reading binary mesh files in all supported programs (AcuProj, AcuSif, etc.)
- Added `center_of_mass` output quantity to `ELEMENT_OUTPUT` command.
- Improved the performance and scalability of search algorithm used for `INTERFACE_SURFACE` command.
- Added ability to convert flexible body forces and displacements to table and statistics format using acuTrans.
- Added support for slip wall mesh displacement when coupling to MotionSolve.
- Added surface names into Patran files created by AcuTherm.
- Added ability to define external forces for rigid bodies using user defined functions.
- Added support for MS-MPI message passing protocol when running outside of Windows HPC Server environment.
- Added `-mpiopt` command line argument to AcuRun to enable specification of command line arguments to the `mpirun` command. The default value is `_auto`, which reverts to previous behavior.
- Speed improvements to AcuTrans and other utilities that access AcuSolve's nodes and elements through the adb API.
- Added ability to assign nodal initial conditions based on user defined functions and curve fits. The `NODAL_INITIAL_CONDITION` command is now able to have multiple instances for a given variable as well. This allows assigning initial conditions on different collections of nodes using different approaches.
- Added the `allow_stagger_reordering` option to the `TIME_SEQUENCE` command to allow AcuSolve to change the stagger order if it determines that changing the sequence of equations will lead to better stability. Note that this option will only have an effect for problems containing `FREE_SURFACE` commands at the current time.
- Changed the behavior of `mesh_displacement_type=slip` parameter in the `SIMPLE_BOUNDARY_CONDITION` command such that only planar surfaces are accepted.
- Added the ability to read lists of strings from file for all commands that accept a list argument. For example, it is possible to read a list of surface sets into the `SIMPLE_BOUNDARY_CONDITION` command by placing them into a file.



### ▼ Notable Resolved Issues

- Resolved a problem with the ALE formulation that could induce significant parasitic velocities on interior nodes.
- Updated domain decomposition methods to enforce consistent behavior across Windows and Linux platforms.
- Fixed a crash when integrated inflow conditions (`inflow_type=mass_flux|average_velocity|flow_rate`) were used on a surface with no no-slip walls adjacent to it.
- Fixed a bug that produced asynchronous time marching in AcuSolve/MotionSolve co-simulations.
- Fixed a bug that caused an assertion when computing turbulent diffusivity when the species stagger is not active.
- Fixed a bug that prevented `wall_velocity_type=normal` from being enforced.
- Fixed an overallocation of memory in AcuTherm when using `fct=direct`.
- Fixed a bug that caused spurious results when using the `INTERFACE_SURFACE` command on prism elements.
- Fixed a bug that caused thermal undershoot when viscous heating was active and played a dominant role in the heat transfer.

### ▼ Changes to Supported Platforms and Third Party Packages

- Updated Linux kernel to 2.6.18-348
- Removed mvapich entirely from installation packages and discontinued support
- Updated Intel MPI to version V5.1
- Updated Platform MPI to V9.1.2 community edition
- Updated MPICH to v3.1.3 on Linux and V1.4.1 on Windows
- Updated Microsoft MPI to V5.0.
- Rearranged the location of MPI libraries in the distribution to support unification across all Altair solvers.
- Discontinued support for mx-mpi and gm-mpi. This will be the last release for which these executables are shipped.
- Removed and discontinued support for Octigabay MPI.
- Removed and discontinued support for Cray MPI.
- Included redistributable ms-mpi installer package in Win 64 package.

### ▼ Changes in default behavior

The `AUTO_SOLUTION_STRATEGY` command now solves the mesh stagger first when performing simulations with mesh motion present.

#### ▼ 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 <code>rigid_body_external_force_type</code> , <code>rigid_body_external_moment_type</code> , and supporting options to enable specification of external forces on rigid bodies using user functions and curve fits.
VISCOSITY_MODEL	Added pressure to the list of available <code>curve_fit_variables</code> .
NODAL_INITIAL_CONDITION	Added <code>type</code> , <code>selection_type</code> , <code>precedence</code> , and supporting options to enable the definition of initial conditions based on user defined functions and curve fits. Note that initial condition commands can now be applied to a node subset in the model and any number of <code>NODAL_INITIAL_CONDITION</code> commands can be present for a single variable.
VOLUME_SET	Newly created command that is used to read in a collection of volume elements. The volume set is then referenced from other commands within the input file to provide support for mixed topology volumes.
ELEMENT_SET	Added the <code>volume_set</code> parameter to enable definition of mixed element topologies within a single element set.
ELEMENT_OUTPUT	Added <code>volume_set</code> and <code>type</code> parameters to enable definition of element output on mixed topology element sets.
SURFACE_SET	Newly created command that is used to read in a collection of surface elements. The surface set is then referenced from other commands within the input file to provide support for mixed topology surfaces.
FAN_COMPONENT	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single <code>FAN_COMPONENT</code> command.

HEAT_EXCHANGER_COMPONENT	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single HEAT_EXCHANGER_COMPONENT command.
HEAT_EXCHANGER_COMPONENT	Added <code>coolant_temperature</code> and <code>coolant_temperature_multiplier_function</code> to support <code>type=constant_coolant_temperature</code> .
ELEMENT_BOUNDARY_CONDITION	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single ELEMENT_BOUNDARY_CONDITION command.
SURFACE_INTEGRATED_CONDITION	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single SURFACE_INTEGRATED_CONDITION command.
FREE_SURFACE	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single FREE_SURFACE command.
TURBULENCE_WALL	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single TURBULENCE_WALL command.
PARTICLE_SURFACE	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single PARTICLE_SURFACE command.
RADIATION_SURFACE	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single RADIATION_SURFACE command.
SOLAR_RADIATION_SURFACE	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single SOLAR_RADIATION_SURFACE command.
INTERFACE_SURFACE	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single INTERFACE_SURFACE command.
GUIDE_SURFACE	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single GUIDE_SURFACE command.
INTERPOLATED_MOTION_SURFACE	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single INTERPOLATED_MOTION_SURFACE command.

EXTERNAL_CODE_SURFACE	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single <code>EXTERNAL_CODE_SURFACE</code> command.
SIMPLE_BOUNDARY_CONDITION	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single <code>SIMPLE_BOUNDARY_CONDITION</code> command.
SIMPLE_BOUNDARY_CONDITION	Added <code>temperature_coupling_type</code> to allow specification of the method used to couple the thermal fields between AcuSolve and an external code.
SURFACE_OUTPUT	Added <code>surface_sets</code> to enable definition of mixed surface element topologies within a single <code>SURFACE_OUTPUT</code> command.
MESH_BOUNDARY_CONDITION	Newly created command that is used to define the mesh motion constraints on a given surface. This command houses the collection of mesh motion boundary conditions from the <code>SIMPLE_BOUNDARY_CONDITION</code> , <code>FREE_SURFACE</code> , and <code>EXTERNAL_CODE_SURFACE</code> commands.

▼ **Summary of Deprecated AcuSolve Input File Commands**

The following table summarizes the AcuSolve input file commands that are deprecated in this release. Support will continue for legacy input files, but users should migrate their modeling process to leverage the new commands when possible.

AcuSolve Command Name	Comments
FREE_SURFACE	Deprecated by the <code>MESH_BOUNDARY_CONDITION</code> command
EXTERNAL_CODE_SURFACE	Deprecated by the <code>MESH_BOUNDARY_CONDITION</code> command

▼ **Summary of New/Changed AcuSolve Command Line Options**

The following table summarizes the changed or newly introduced AcuSolve command line arguments. Note that a full description of each option is available in the AcuSolve Programs Reference Manual.

Program	Option	Description
<b>AcuRun</b>	<code>-install_smpd</code>	Added support for the installation of message passing daemon service for Platform MPI. This is needed only when using distributed memory parallel operation across multiple Windows hosts.
<b>AcuRun</b>	<code>-translate_output_to</code>	Added an option of h3d as a supported output format at the conclusion of the run.
<b>AcuRun AcuRunTrace AcuRunFwh</b>	<code>-message_passing_type</code>	Removed mvapich, octigabay, and cray mpi support.
<b>AcuRun</b>	<code>-view_message_passing_type</code>	Removed mvapich, octigabay, and cray mpi support.
<b>AcuRun AcuRunTraceAc uRunFwh</b>	<code>-mpirun_options</code>	New option that allows passing of options directly to the mpirun/mpiexec command.
<b>AcuRun AcuRunTrace AcuRunFwh</b>	<code>-automount_path_remove</code>	Changed default setting to <code>_none</code> .
<b>AcuRun AcuRunTrace AcuRunFwh</b>	<code>-automount_path_name</code>	Changed default setting to <code>_none</code> .
<b>AcuSolve</b>	<code>-projection_eigenvalue_type</code>	New option that provides control over algorithm used to compute eigenvalues in the linear solver.
<b>AcuSolve</b>	<code>-linear_solver_error_output</code>	New option that provides control over whether debug information should be output when an error is encountered in the linear solver.
<b>AcuSolve</b>	<code>-mqm_reference_push_factor</code>	New option that provides control over whether the reference coordinates or the current coordinates should be used when computing the mesh quality metric for moving mesh simulations that use ALE.

<b>AcuSolve</b>	<code>-num_mmi_smoothing_passes</code>	New option that provides control over the number of smoothing passes used for interpolated mesh motion constraints.
<b>AcuSurf</b>	<code>-coordinates_file,</code> <code>-connectivity_file_list,</code> <code>-surface_file_list,</code> <code>-surface_node_file_list</code>	Added ability to read binary format for all mesh files.
<b>AcuMesh2Tet</b>	<code>-coordinates_file,</code> <code>-connectivity_file_list,</code> <code>-surface_file_list,</code> <code>-periodic_bc_file_list</code>	Added ability to read binary format for all mesh files.
<b>AcuProj</b>	<code>-coordinates_file</code>	Added ability to read binary format coordinates files.
<b>AcuPbc</b>	<code>-coordinates_file,</code> <code>-nodal_bc_1_file,</code> <code>-nodal_bc_2_file</code>	Added ability to read binary format for all mesh files.
<b>AcuPev</b>	<code>-coordinates_file,</code> <code>-node_file_list,</code> <code>-surface_file_list</code>	Added ability to read binary format for all fluid mesh files.
<b>AcuSurf</b>	<code>-coordinates_file,</code> <code>-connectivity_file_list,</code> <code>-surface_file_list,</code> <code>-surface_node_file_list</code>	Added ability to read binary format for all mesh files.
<b>AcuTrans</b>	<code>-moment_center</code>	New option that enables specification of which point moments are computed about for surface outputs.
<b>AcuTrans</b>	<code>-moment_center_mesh_motion</code>	New option that enables specification of a mesh motion to use when computing moments about a given location.
<b>AcuTrans</b>	<code>-flexible_body_output</code>	New option that controls the export of FLEXIBLE_BODY results.
<b>AcuTrans</b>	<code>-flexible_body_output_vars</code>	New option that controls the variables to export for the FLEXIBLE_BODY.
<b>AcuTrans</b>	<code>-flexible_body_output_sets</code>	New option that controls the FLEXIBLE_BODY output sets to convert.
<b>AcuRunTrace</b>	<code>-file_format</code>	New option that controls the internal file format.

<b>AcuTherm</b>	<code>-num_temperature_filter_passes</code>	New option controlling the number of filter passes on the temperature.
<b>AcuTherm</b>	<code>-num_reference_temperature_filter_passes</code>	New option controlling the number of filter passes on the reference temperature.
<b>AcuTherm</b>	<code>-num_film_filter_passes</code>	New option controlling the number of filter passes on the film coefficient.
<b>AcuTherm</b>	<code>-num_heat_filter_passes</code>	New option controlling the number of filter passes on the heat flux.
<b>AcuTherm</b>	<code>-film_calculation_type</code>	New option controlling the type of calculation to use for computing the reference temperatures and heat transfer coefficients.
<b>AcuTherm</b>	<code>-limit_reference_temperature</code>	New option controlling whether the reference temperature should be limited to the peak nodal temperature in the model.

## Multibody Solutions 14.0 Release Notes

The Altair multibody solution, MotionSolve, consists of the following capabilities:

- MotionView – the modeling environment
- MotionSolve – the solver
- Automotive Solutions – for suspension design, vehicle dynamics, ride comfort and durability
- HyperGraph/HyperView – the post-processing environment

The 14.0 release of MotionSolve contains three major capabilities that span the breadth of the multibody solution – modeling, simulation and post-processing.

- 3D Contact
- Nonlinear Finite Elements (NLFE) - Experimental
- Advanced Driver and Road for vehicle simulation

These are explained separately so you can understand the full range of the functionality. Other important, but relatively smaller features are binned by product.

[3D Contact](#)

[Nonlinear Finite Elements](#)

[Advanced Driver and Road Vehicle Simulation](#)



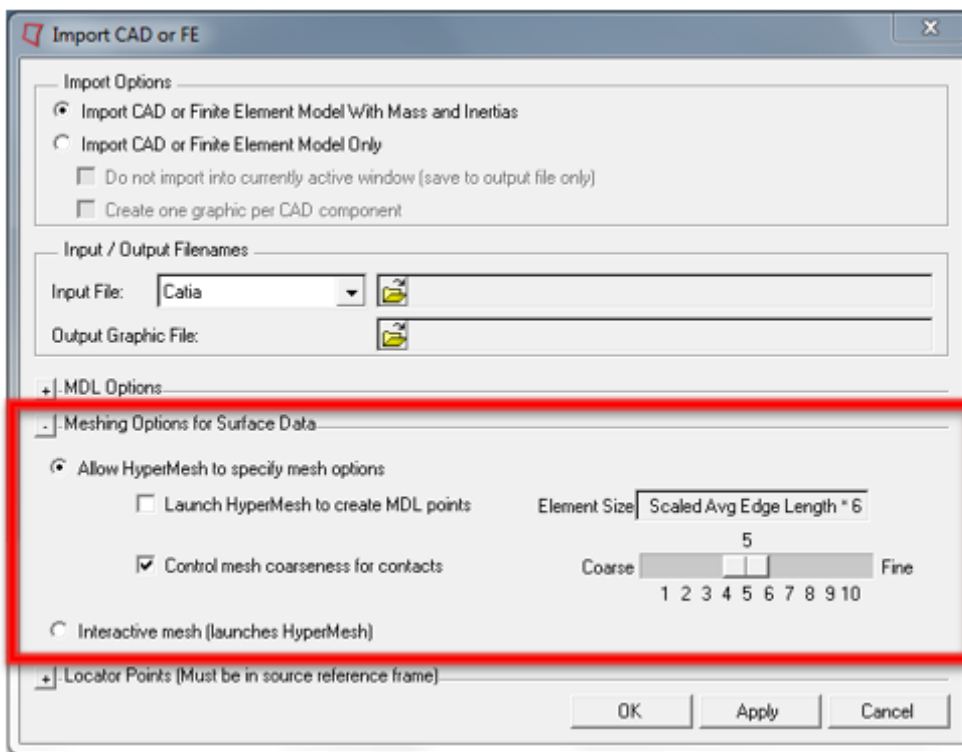
## 3D Contact

The 3D Contact capabilities in MotionSolve have been completely revamped. Pre-processing, Simulation, Post-Processing and Documentation have been significantly upgraded. The tables below summarize the key benefits you will obtain by using the new contact capabilities and how they have been implemented.

### Pre-processing

#### ▼ CAD Geometry can now be Imported Easily and Reliably to Provide the Geometry for 3D Contact

New meshing options are now available when importing CAD geometry. These enable you to control the mesh density in a very intuitive way.



Accurate contact force generation requires uniform meshes. New algorithms, more suited for uniform mesh generation are now used. This new meshing option is only enabled for input geometries and not for FE models.

#### ▼ Contact Models are Defined with Minimum of User Input

The Properties tab in the Contact panel has been reorganized to have sub-tabs for normal force and friction force.

Normal Force		Friction Force	
<input checked="" type="radio"/> Impact	Stiffness: <input type="text" value="100.0000"/>	Damping: <input type="text" value="0.1000"/>	
<input type="radio"/> Poisson	Exponent: <input type="text" value="2.1000"/>	Penetration depth: <input type="text" value="0.0500"/>	
<input type="radio"/> Volume Model			
<input type="radio"/> User Defined			

Normal Force		Friction Force	
<input type="radio"/> Disabled	MU Static: <input type="text" value="0.2000"/>	Stiction transition velocity: <input type="text" value="1.0000"/>	
<input type="radio"/> Dynamic Only	MU Dynamic: <input type="text" value="0.1000"/>	Friction transition velocity: <input type="text" value="1.5000"/>	
<input checked="" type="radio"/> Static & Dynamic			
<input type="radio"/> User Defined			

Reasonable contact and friction parameters are provided as defaults. You do not have to change these unless your contacts have special properties.

Parameters are checked for validity as they are entered and the user is alerted to invalid values before the model is run.

Output Requests to measure contact forces are automatically created.

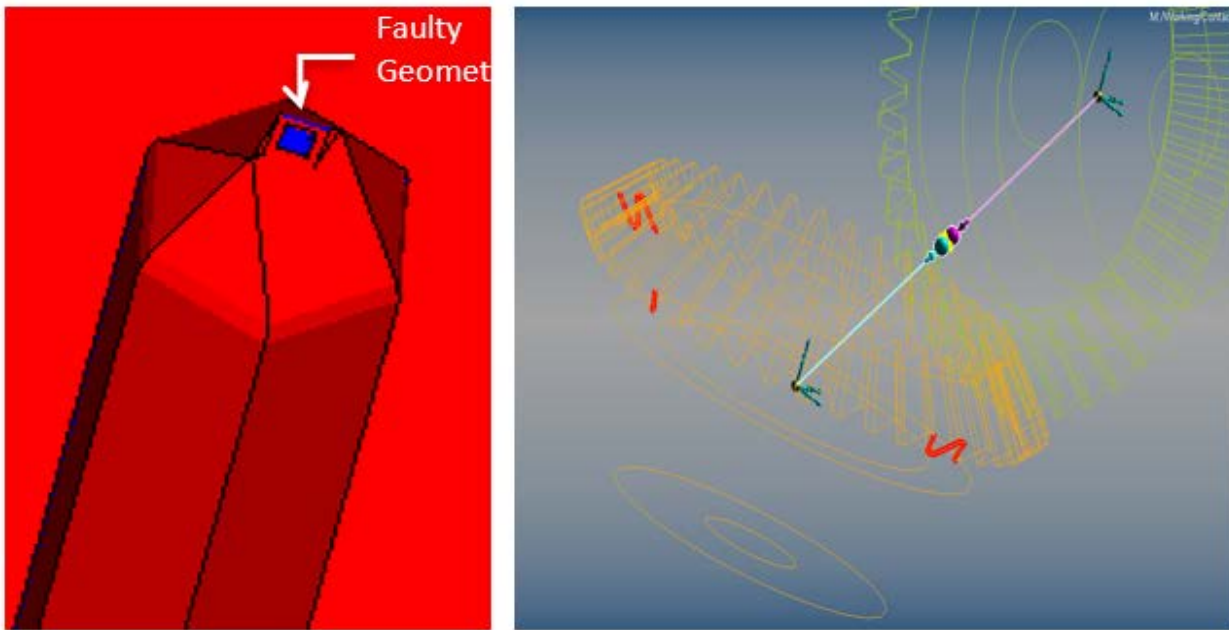
▼ **Extensive Geometry Checks are Performed Automatically to Avoid Potential Failures in the Solver**

All contact meshes examined and invalid or “dirty” geometry is clearly identified during modeling. Contact surfaces for all meshes can be visualized using “Highlight contact side” option in the Connectivity tab.

<input type="checkbox"/> Highlight contact side
<span style="color: red;">Contact side</span> <span style="color: blue;">Material side</span>

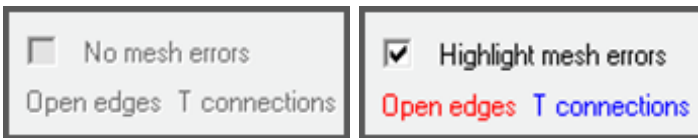
Surface normals can be inverted with just one operation. This is especially useful when contact is happening inside a selected closed geometry.

Red indicates outward normals and blue indicates the inward normals.



The normals for a mesh are automatically corrected when inconsistencies are detected. This is especially useful when tria-meshes are imported from STL files or Wavefront OBJ files.

Open edges and T connections, if any, can be highlighted as shown in the figure on the top right. If the mesh is clean (i.e. it forms a closed volume), the panel indicates “No mesh errors” else the “Highlight mesh errors” option becomes available.



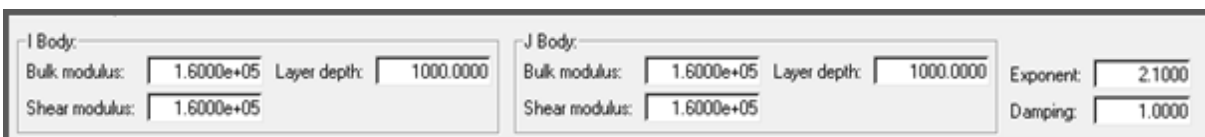
### ▼ Mesh Cleaning and Healing Tools

MotionView now includes a new mesh manipulation tool that allows you to clean previously unusable meshes for contact solution. The capability is accessed from the Import CAD or FE utility.

Import CAD or FE utility now provides additional cleaning options that enable you to fill holes, thicken surfaces, eliminate T-junctions and shrink-wrap unusable meshes. All of this is done with a minimum of user input.

### ▼ New Contact Models are Supported

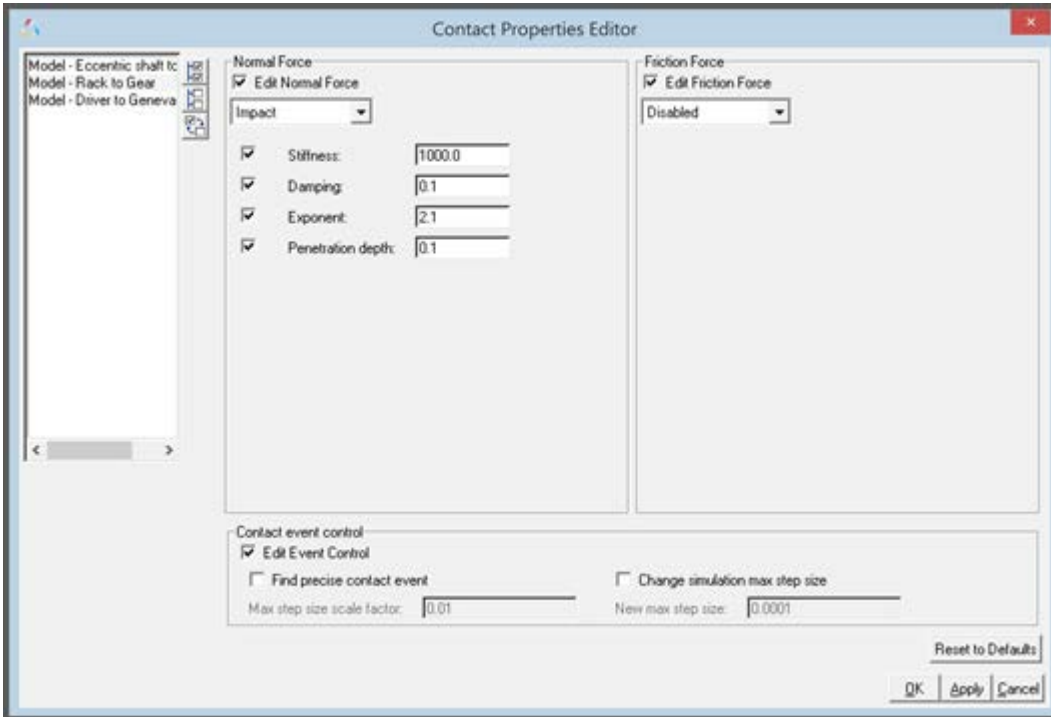
A new Volume Model for normal force calculations is now supported. User defined contact models for both normal and friction forces are also supported.



In addition, user-defined normal and friction force definitions are supported.

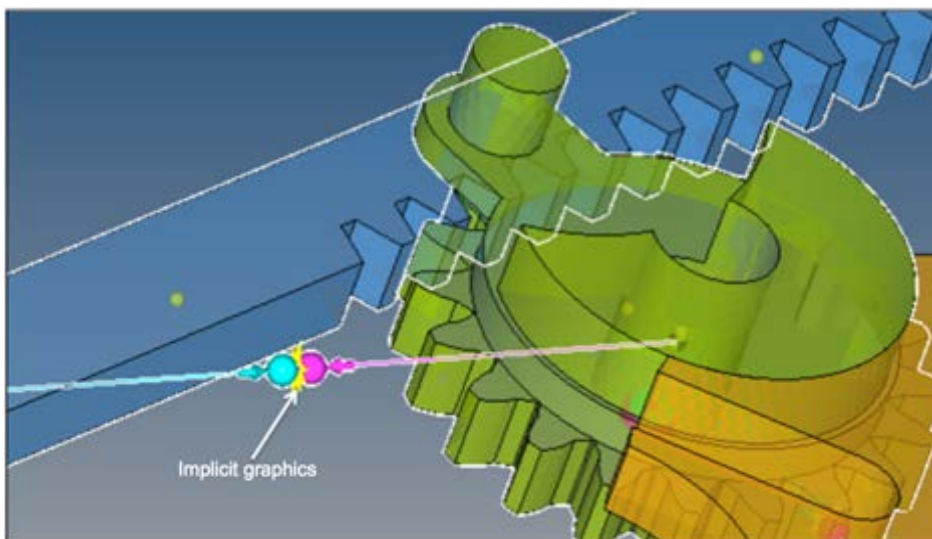
▼ **Updated Contact Properties Editor Macro**

This macro tool has been updated to take into account the new contact properties available in the solver and makes setting properties across multiple contacts quick and intuitive.



▼ **Contacts are Visualized During Modeling**

A new “implicit graphics” contact icon is now visible. You can visually see the geometries that have contact connections.



Associated graphics are highlighted when contact entity is selected.

#### ▼ **Precise Contact Detection is Supported**

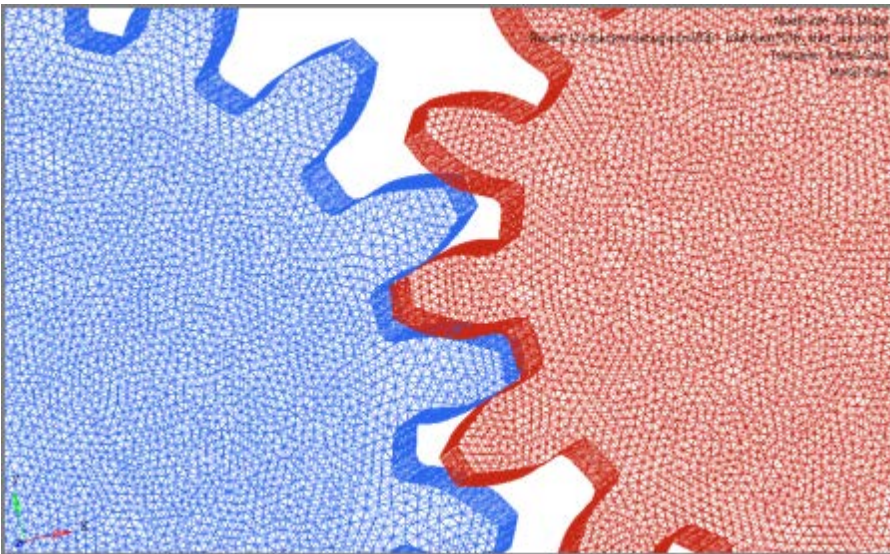
Advanced options are available to detect the precise event when contact occurs. An option is also available to change the maximum step size subsequent to contact detection. This option introduces a Sensor entity with a zero crossing attribute into the MotionSolve XML file. This Sensor will control integration step size so that the onset of contact is accurately detected.

Refer to Rigid to Rigid Contact section in MotionView User Guide to learn more about modeling contacts.

## **Solution**

#### ▼ **Improved Robustness of Mesh-to-Mesh Contact**

A new collision engine that vastly improves the robustness of mesh-mesh contact has been implemented.



#### ▼ **Contact Simulations are Much Faster Now**

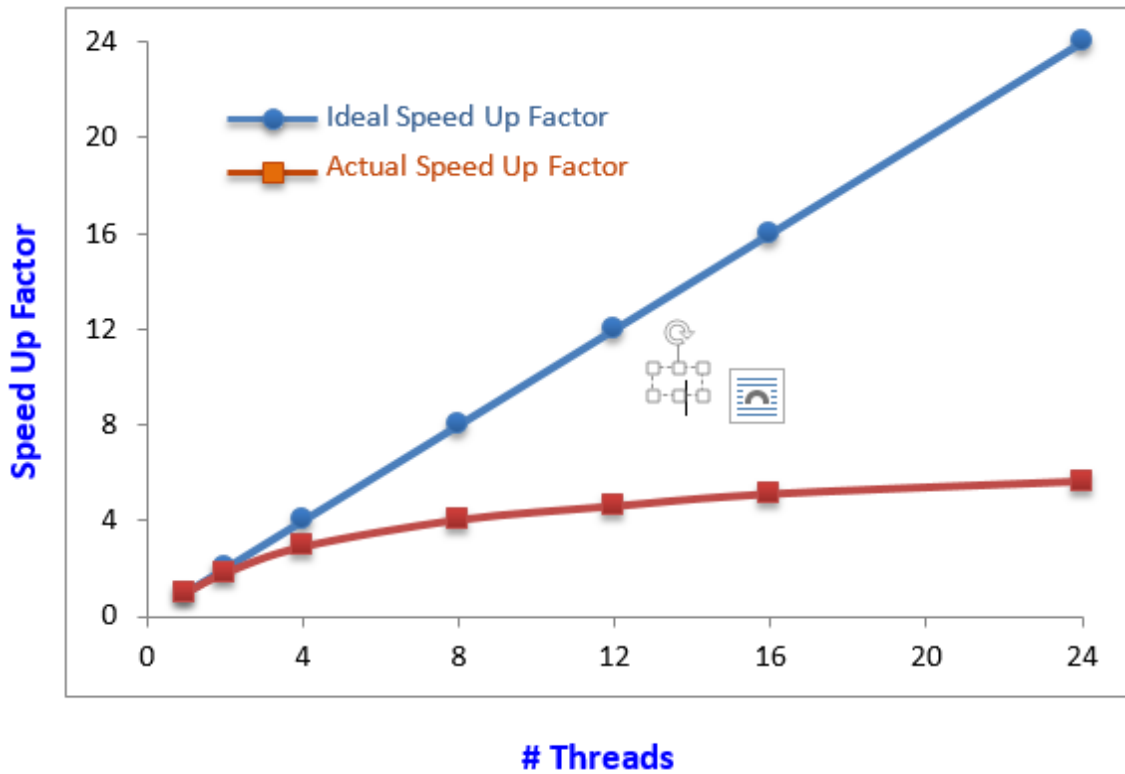
There are two main reasons for fast contact simulations:

- Parallel solution for all contacts
- Specialized contact algorithms for simpler shapes

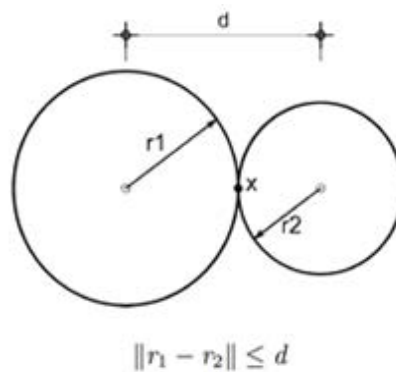
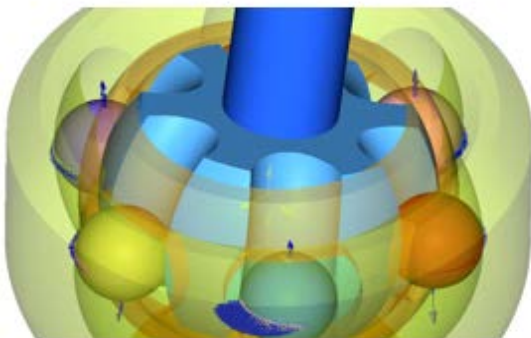
#### **Parallel solution for all contacts**

The contact kinematics between each pair of geometries is calculated in parallel in different threads. This means whenever there are multiple contacts in your model you can improve simulation time by using multiple cores in a machine. All contact pairs will be dynamically distributed onto the available cores.

The graph below shows how contact simulations speed up as a function of the number of threads used to solve for contact.



### Specialized contact algorithms for simpler shapes

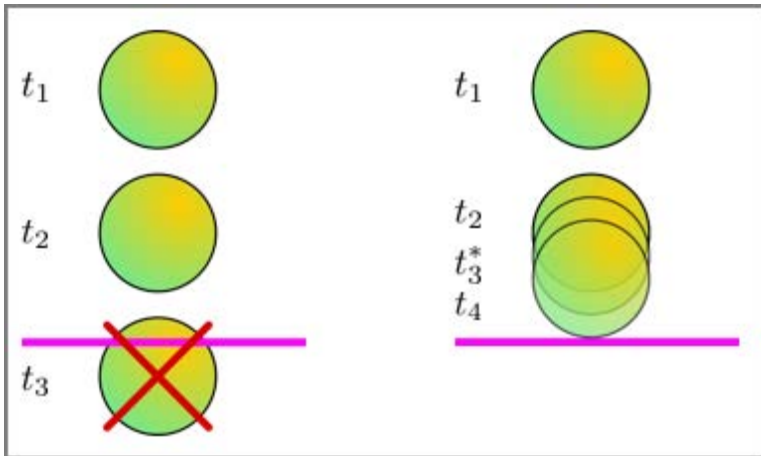


A much faster, specialized sphere-to-mesh contact algorithm is now available.  
 A much faster, specialized sphere-to-sphere contact algorithm is now available.  
 Both of these are automatically invoked when needed.



▼ **Large Initial Penetration Leading to Unrealistic System Behavior is Avoided**

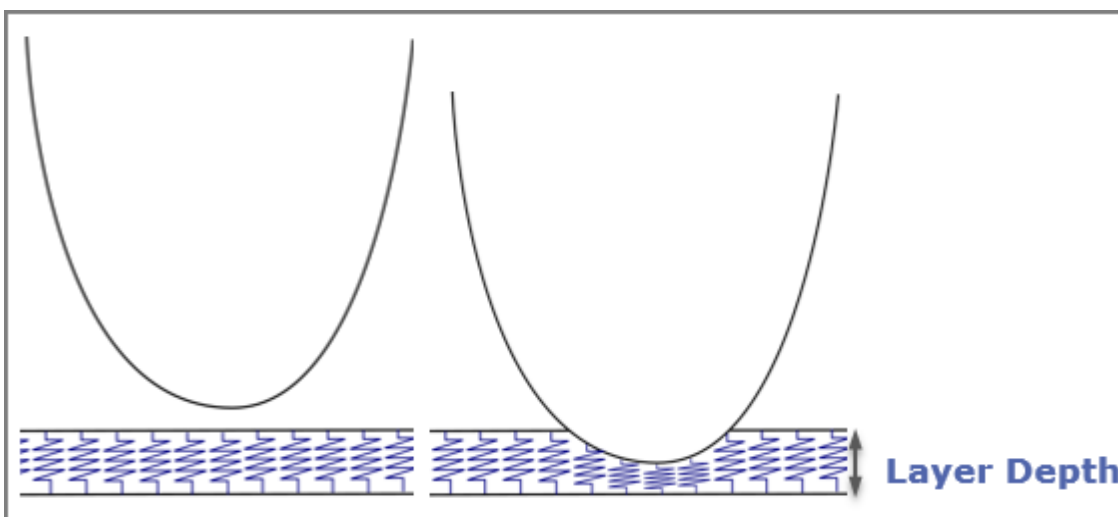
A zero crossing sensor that monitors the distance between contacting geometries is now available. The zero crossing sensor controls integrator step-size to precisely capture the contact event.



Note: This sensor is only meaningful for impulsive contacts. If a contact is persistent, then a zero crossing sensor is not needed.

▼ **Contact Parameters have a Physical Basis and can be Easily Defined**

A new “Volume” force model that relies on understandable physical parameters such as bulk modulus and shear modulus is now available.



The normal force is based on an *Elastic foundation* model. The rigid bodies in contact are assumed to be covered by thin elastic layers. If the tangential shear stress in the layer is ignored, a direct relationship can be found between the normal pressure and normal displacement. This relationship can be used to generate the force between two contacting layers. This model is well known in the literature for contacts.

▼ **Contact Outputs are Provided Even Though They Might Happen at “Irregular” Times**

An option to output the system state at the maximum penetration depth in-between two regular output steps is now available. When you run a simulation from MotionView, this is done automatically. You can filter out these “extra” outputs in HyperView.

▼ **Detailed Contact Behavior can be Exported from the Solver as Requests**

The CONTACT () function has been enhanced to provide many more details about the contact such as maximum penetration depth, number of contact patches, total number of contacts, area of contact, etc.

▼ **Key Contact Metrics are now Available. You no Longer have to Compute Them!**

Contact metrics tables are now printed in the MotionSolve log file. You can examine the metrics table to determine what the maximum penetration(s) and normal force(s) is/are in your model and when it occurred. Two tables are printed – one ordered by maximum penetration depth and another ordered by maximum contact force:

**Contact Overview TOP 5 (sorted by MAX\_PD)**

CONTACT	I_GRAPHIC	J_GRAPHIC	MAX_PD (TIME)	MAX_FORCE (TIME)
301014	90022	90001	7.436E-01 (0.000E+00)	4.567E+02 (0.000E+00)
301015	90042	90001	7.436E-01 (1.000E-05)	4.575E+02 (0.000E+00)
301018	90102	90001	7.436E-01 (1.000E-05)	4.575E+02 (0.000E+00)
301017	90082	90001	7.436E-01 (0.000E+00)	4.567E+02 (0.000E+00)
301011	90082	90000	7.232E-01 (1.198E+00)	4.058E+02 (1.199E+00)

**Contact Overview TOP 5 (sorted by MAX\_FORCE)**

CONTACT	I_GRAPHIC	J_GRAPHIC	MAX_PD (TIME)	MAX_FORCE (TIME)
301018	90102	90001	7.436E-01 (1.000E-05)	4.575E+02 (0.000E+00)
301015	90042	90001	7.436E-01 (1.000E-05)	4.575E+02 (0.000E+00)
301014	90022	90001	7.436E-01 (0.000E+00)	4.567E+02 (0.000E+00)
301017	90082	90001	7.436E-01 (0.000E+00)	4.567E+02 (0.000E+00)
301011	90082	90000	7.232E-01 (1.198E+00)	4.058E+02 (1.199E+00)

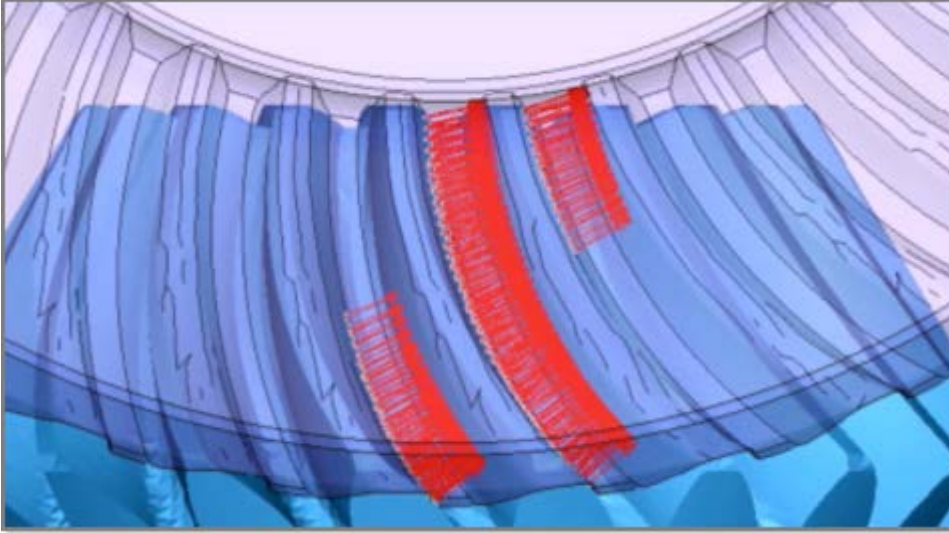
A contact report containing this table is also available in MotionView.



## Post-processing

- ▼ **Contact Behavior can be Examined in Detail in HyperView. You can Understand how the Contacts are Behaving During the Simulation.**

New vector plots are now available for visualizing normal force, friction force, penetration depth, and penetration velocity.

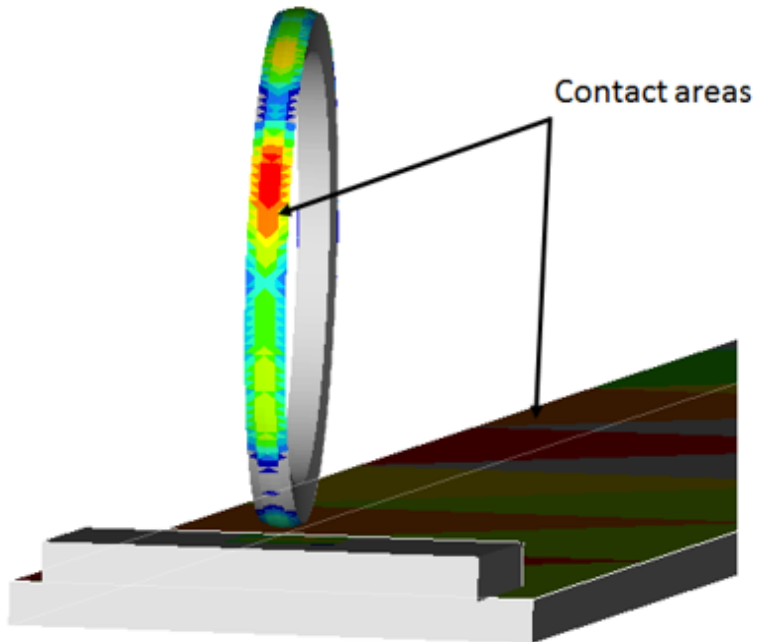


The coefficient of friction in the contact patches can now be visualized as a contour plot.

Contact forces by region can now be visualized.

- ▼ **The Range of Contact Occurrence on Geometry Across an Entire Simulation can now be Visualized**

A new contact summary frame is now available in HyperView. This summary frame will show you where contact has occurred during a simulation with a contour of the maximum penetration depth for all contact occurrences through the entire length of the simulation.

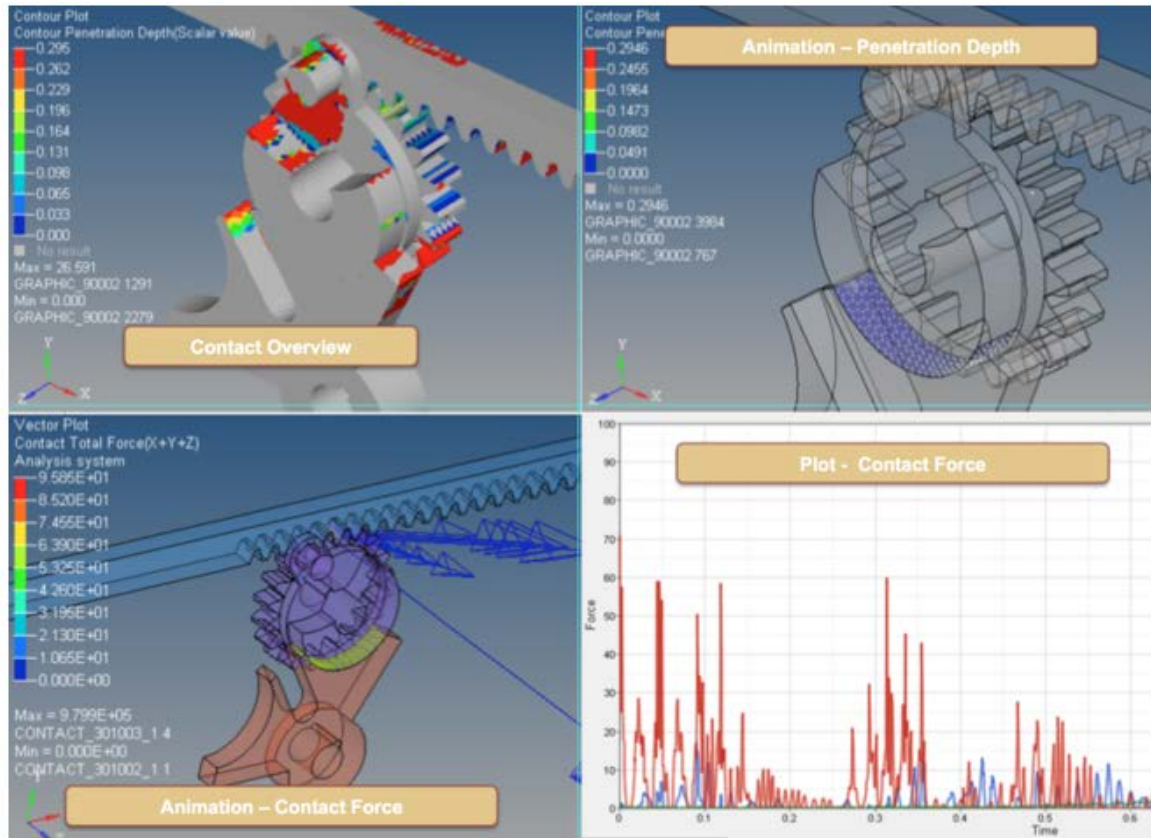


#### ▼ Contact Report for Easy Post Processing

MotionView now has capability to generate a Contact Report automatically to simplify the post processing of contacts.

For every model that involves contacts, the auto-generated report contains the necessary plots and animations that enable you to easily understand contact behavior in your model.

- A 'Contact Overview' is first generated so you can see where the contact has occurred in the model.
- Two animations of the model, one showing penetration depth and the second showing normal contact force are available for review
- Finally, the report also contains 'Overall Contact Force vs. Time' plot for every contact in the model.



### ▼ Detailed Contact Information can now be Plotted

New options have been added to the contact function. This enables you to create requests containing contact details that can be easily plotted.

### Change in Behavior

The new mesh-to-mesh contact algorithm requires Graphics that have a well-defined volume. Open meshes, i.e., meshes that do not have a volume, are not allowed. MotionView will detect open meshes and flag these as errors. You must fix such Graphics before using these for contact. For more information, refer to the Best practices for running 3D contact models in MotionSolve.

The new mesh-to-mesh contact algorithm is capable of handling very large meshes quite efficiently. In the past you had to take care to mesh only the regions in contact with a fine mesh. This is no longer required. A fine mesh that is not in contact, does not add much overhead to the calculations.

## Nonlinear Finite Elements

---

MotionSolve 14.0 introduces the “NLFE Body” which allows you to model non-linearly flexible components in your multi-body system. This is an experimental feature in the current version. The NLFE Body is based on the Absolute Nodal Coordinate Formulation (ANCF). Nonlinearity can occur for two main reasons: (A) Geometric nonlinearity and (B) Material non-linearity. The NLFE body supports both.

A non-linear modeling component is typically required in the following scenarios:

- A large deformation is expected in the flexible component, or an unusually large deformation is observed in the flexible component
- Stresses in your flexible components approach yield point of the material
- The flexible component is composed of non-linear hyper elastic materials like rubber or polyurethane. The material exhibits a nonlinear stress vs. strain behavior or is hyper-elastic
- A linearly flexible modelling component does not provide the fidelity that is desired

**Please note that the deformation is still assumed to be in the elastic region. Plastic deformations are not supported in this release.**

### Pre-processing

MotionView has new capabilities for modeling NLFE systems. A generic NLFE Body can be created using either the CABLE or BEAM element.

#### ▼ CABLE

The CABLE element does not resist shear or torsion forces. This implies that the cross section of this element does not change as the entity deforms.

This element can be used to model cables, wires, ropes, etc. Only the Linear Elastic material model is supported.

#### ▼ BEAM

The BEAM element resists all forms of deformation, which implies that the cross section can also deform with load. This can be visualized in HyperView while using this element.

Multiple cross sections are supported for the BEAM element:

- Circular (solid and hollow)
- Rectangular or Box (solid and hollow)
- Channel
- Cross

- Hat
- H, I, L, T and Z sections

This element can be used to model beams, springs, belts, rubber components etc. You may choose from either a Linear-Elastic or a Hyper-Elastic material.

Three Hyper Elastic materials are supported – Neo-Hookean Compressible, Mooney-Rivlin and Yeoh models. MotionView provides a set of recommended values for the model parameters. You may however, create new materials and enter your own parameters.

#### ▼ 3D Visualization

MotionView provides 3D visualization of the NLFE body based on the profile and cross section properties.

#### ▼ Defining Preloaded/Relaxed Configurations

An NLFE body can now have a pre-loaded position in addition to the model position. The user can import these in a CSV format. Upon import, the model position will be shown in wireframe mode and the loaded position in solid.

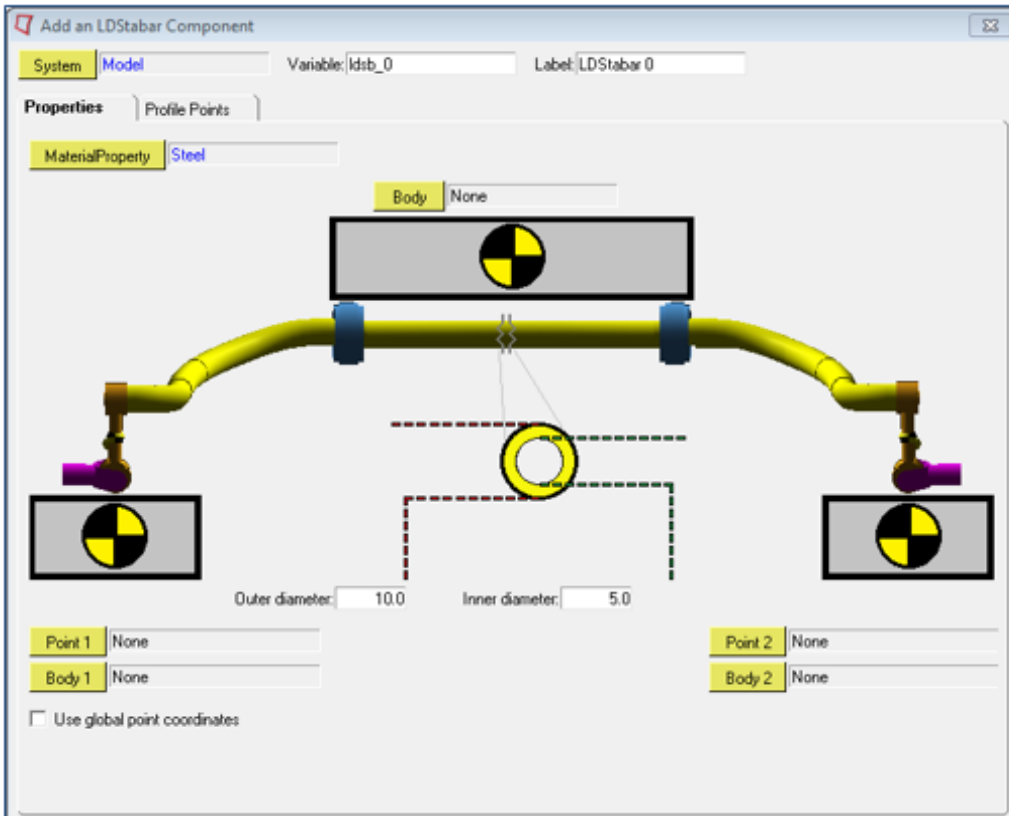
#### ▼ Saving Pre-load Configurations

With a combination Save command in MotionSolve and a Python script, it is now possible to save the configuration of the NLFE at the end of simulation into a CSV file. This csv file can be read into MV as preloaded configuration. An option is also available to visualize the unloaded and pre-loaded profile together.

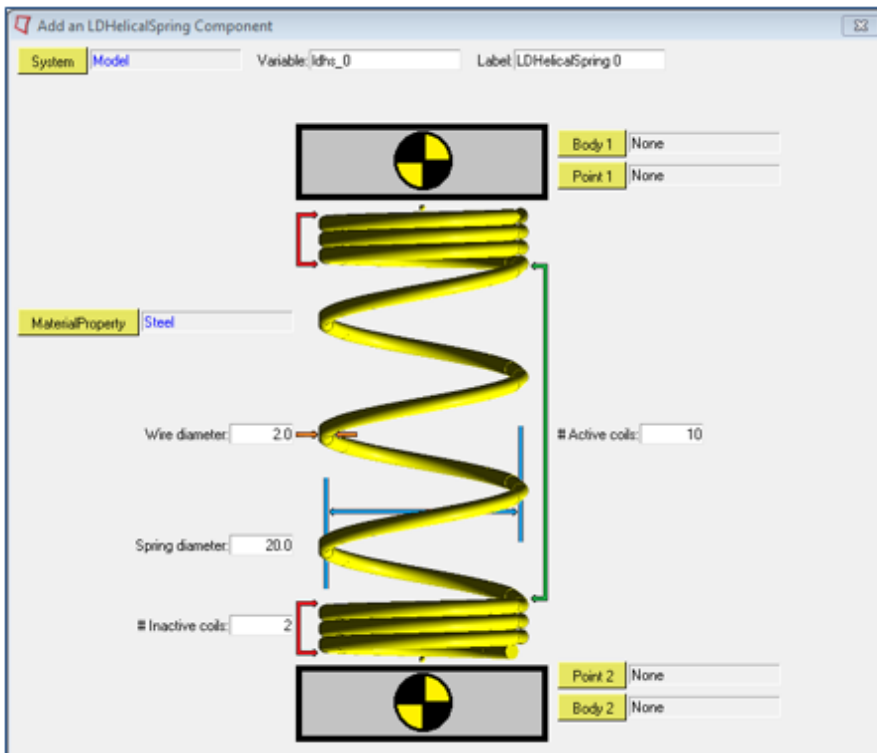
#### ▼ NLFE Subsystems

In addition to the generic Cable and Beam modeling elements, MotionView also supports a small set of NLFE subsystems that are specific to the Auto industry. These subsystems may be accessed from the vertical toolbar to the left of the graphics window. The following subsystems are supported.

##### **Stabilizer bar**



### Coil spring



Refer to NLFE Subsystems to learn more about using these subsystems in your model.

These subsystems are provided as examples to you to demonstrate how the built-in capabilities of MotionView may be used to create parametric subsystems that can contain NLFE elements. You can build your own subsystems in a similar manner. Source code for the built-in NLFE subsystems will be provided in the installation.

MotionSolve supports a much larger set of NLFE modeling elements. You can use these in a MotionView model by using a Templex template. Native MotionView support is not available at this time.

## Solution

NLFE subsystems are included in MotionSolve as a flexible body that is specified through an ANCF XML file. The MotionSolve XML file fragment below illustrates this.

```
<Body_Flexible
  id           = "30102"
  label       = "NLFE Body 0"
  full_label  = "Model-NLFE Body 0"
  ref_marker_id = "30101010"
  ancf_file   = "simo-spinning-beam4_30102.xml"
  rayleigh_damping = "0.5"
  num_i_marker = "3">
    30102021 30102101 30102102
</Body_Flexible>
```

The NLFE subsystem can consist of any combination of atomic MotionSolve NLFE elements. For the example shown above, the NLFE subsystem is specified in the file [simo-spinning-beam4\\_30102.xml](#). The contents of this file are shown below.

<ANCFModel1>

```

<UNIT force="NEWTON" mass="KILOGRAM" length="METER" time="SECOND"/>
<GRID id="301002" x="0.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0 1.0"/>
<GRID id="301003" x="0.5" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0 1.0"/>
<GRID id="301004" x="1.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0 1.0"/>
<GRID id="301005" x="1.5" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0 1.0"/>
<GRID id="301006" x="2.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0 1.0"/>
<GRID id="301007" x="2.5" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0 1.0"/>
<GRID id="301008" x="3.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0 1.0"/>
<GRID id="301009" x="3.5" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0 1.0"/>
<GRID id="301010" x="4.0" y="0.0" z="0.0" rx="1.0 0.0 0.0" ry="0.0 1.0 0.0" rz="0.0 0.0 1.0"/>
<BEAMC id="20000" pid="10000" g1="301002" g2="301003"/>
<BEAMC id="20001" pid="10000" g1="301003" g2="301004"/>
<BEAMC id="20002" pid="10000" g1="301004" g2="301005"/>
<BEAMC id="20003" pid="10000" g1="301005" g2="301006"/>
<BEAMC id="20004" pid="10000" g1="301006" g2="301007"/>
<BEAMC id="20005" pid="10000" g1="301007" g2="301008"/>
<BEAMC id="20006" pid="10000" g1="301008" g2="301009"/>
<BEAMC id="20007" pid="10000" g1="301009" g2="301010"/>
<PBEAMC id="10000" mid="3000006" ri1="0" ro1="0.020000" ri2="0" ro2="0.020000" nf="1" nx="5"
nr="4" nt="10" ngx="10" ngr="1" ngt="1"/>
<MAT1 id="3000006" e="2e+011" nu="0.3" rho="8000"/>

```

</ANCFModel1>

MotionSolve supports a comprehensive set of NLFE modeling elements. The table below summarizes the current implemented atomic NLFE entities in MotionSolve. **Note, that MotionView supports only a small subset of these. You can use these in a MotionView model by using a Templex template.**

Element Type	Supported Element Set	
Line Elements	BEAM9	(Gradient Deficient ANCF Beam Element)
	BEAMC	(Beam Element with Circular Cross Section)
	BEAM12	(Fully Parameterized ANCF Beam)
	CABLE	(Cable Element)
Surface Elements	QUAD12	(Rectangular Thick Plate Element)
	QUAD9	(Rectangular Thin Plate Element)
	TRIA12	(Triangular Thick Plate Element)
	TRIA9	(Triangular Thin Plate Element)



Element Type	Supported Element Set	
Solid Elements	HEXA12	(C1 Brick Solid Element)
	HEXA3	(C0 Brick Solid Element)
	PENTA12	(C1 Penta Solid Element)
	PENTA3	(C0 Penta Solid Element)
	TETRA12	(C1 Tetra Solid Element)
	TETRA3	(C0 Tetra Solid Element)
Scalar and Bushing Elements	ABUSH	(Bushing Element)
	LINE2	(Two nodes line spring element)
	LINE3	(Three nodes line spring element)
	LINE4	(Four nodes line spring element)
Mass Elements	CONGM	(Concentrated mass element connection)
	CONPM	(Concentrated point mass)
Connector Elements	CONN0	(Connector element)
	CONN1	(Connector element)
	CONN2	(Connector element)
	CONN3	(Connector element)
Linear Elastic Material Models	MAT1	(Continuum Mechanics Approach )
	MAT1LS	(Linear Strain Material Model)
	MAT6	(Elastic Line Approach)
Hyper-Elastic Material Models	MAT2	(Incompressible Neo-Hookean Material Model)

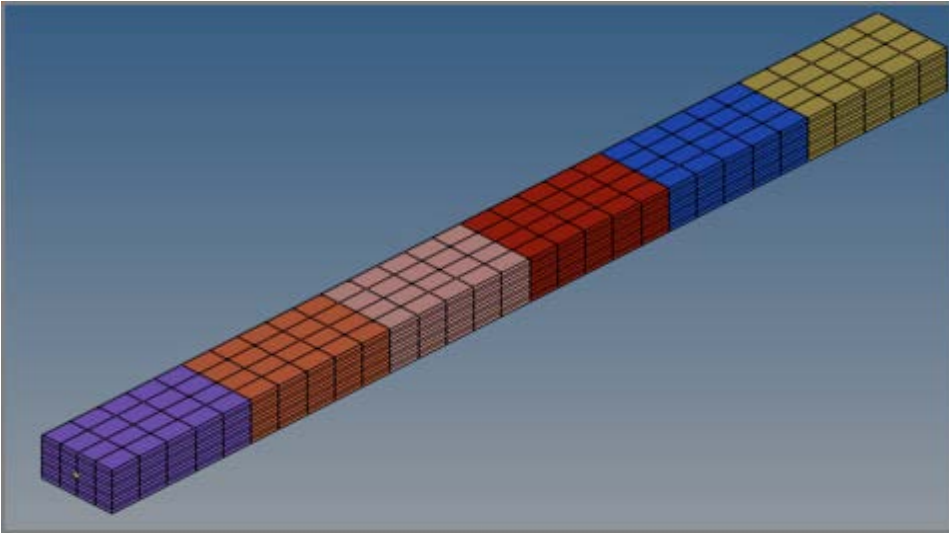
Element Type	Supported Element Set	
	MAT3	(Compressible Neo-Hookean Material Model)
	MAT4	(Mooney-Rivlin material model)
Anisotropic Material Model	MAT7	(Thin Plate Element Anisotropic Material Model)
	MAT7ORT	(Thin Plate Element Orthotropic Material Model)
	MAT9	(Anisotropic Material Model)
	MAT9ORT	(Orthotropic Material Model)
	MAT9TRA	(Transverse Isotropic Material Model)
Geometric Properties	PABUSH	(Bushing element properties)
	PBEAM9	(Arbitrary Cross Section Beam element properties)
	PBEAMA	(Arbitrary Cross Section Beam element properties)
	PBEAMC	(Circular beam element properties)
	PBEAML	(Property card of the ANCF beam elements)
	PCABLE	(Cable element properties)
	PLINE	(Line spring property)
	PSHELL	(Plate elements properties)
	PSOLID	(Solid elements properties)
Other Inputs	GRID UNITS	

For more information about these elements, refer to the NLFE Reference Manual.

## Post-processing

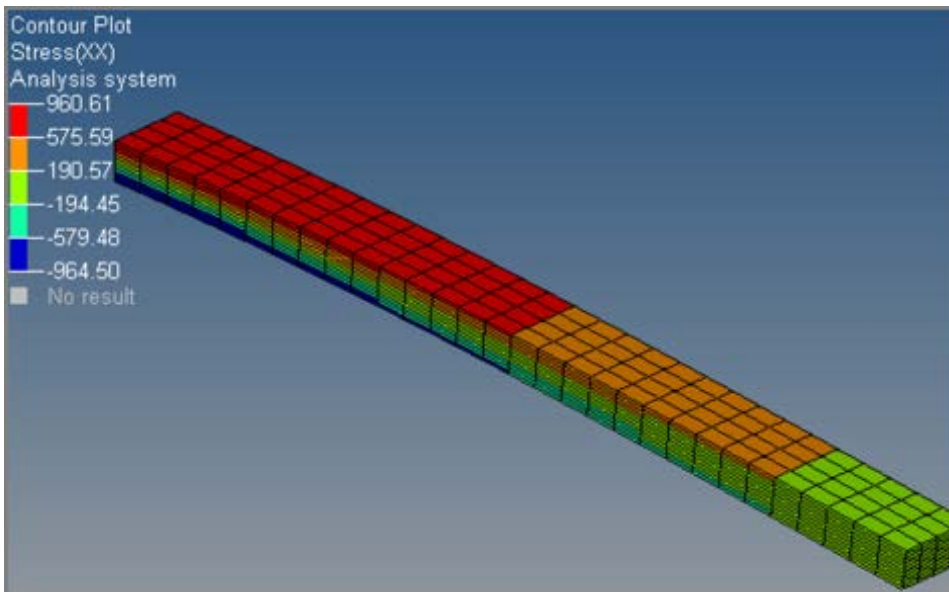
### ▼ Beam and Cable Elements can be Visualized in a 3D Form

The result H3D contains the NLFE elements in 3D form which can be visualized in HyperView.



### ▼ NLFE Body can be Post-Processed in Similar Way as any Finite Element Model

Displacement, stress and strain contours can be visualized on the NLFE body. Other operations such as section cuts can also be applied.



## CMS Flexible Body vs. NLFE Body: Pre-Processing Comparison

The NLFE Body and a CMS Flexible Body provide somewhat similar capabilities. There are, however, key differences in how the entities are defined, used and in the limitations they have. Key differences between the two are compared in the tables below.

Task	CMS flexible body	NLFE body
Creating flexible bodies	User meshing is required to represent the component.	No user meshing required for creating NLFE sub-systems in MotionView.
	An additional CMS analysis is required to generate a modal representation of the flexible component.	No analysis required to create an NLFE body. The NLFE Body can be created and modified directly in MotionView.
	Line, shell and solid elements are supported.	This release introduces support for line elements only.
	Connect to joints, forces etc. by defining interface nodes in CMS analysis.	All nodes can be used as interface nodes – no special handling is required.
Modeling flexibility in components	Only accurate in a linear deformation range, i.e. for small deformations; Must capture enough modal information to represent physical deformation accurately	Fully non-linear formulation allows accurate solution for small and large deformations without any reduction analysis

## CMS Flexible Body vs. NLFE Body: Solution Comparison

The solution characteristics of an NLFE Body are compared to the CMS flexible body in the following table.

Task	CMS flexible body	NLFE body
Modeling flexibility in components	Only accurate up to a linear deformation range, i.e. for small deformations. Must capture enough modal information to represent physical deformation accurately.	Fully non-linear formulation allows accurate solution for small and large deformations without any reduction analysis.
Modeling geometric non-linearity in flexible components	Cannot handle large deformations in general.	The BEAM and CABLE element allow you to model large deformations in your flexible components.

Task	CMS flexible body	NLFE body
Modeling material non-linearity in flexible components	Does not support non-linear material models.	You can model your flexible components with hyper elastic as well as linear elastic materials.

### CMS Flexible Body vs. NLFE Body: Post-Processing Comparison

	CMS flexible body	NLFE body
Recover stresses and strains	Yes	Yes. MotionSolve writes out a 3D representation of the Beam and Cable elements to enable stress, strain and displacement visualizations in HyperView.

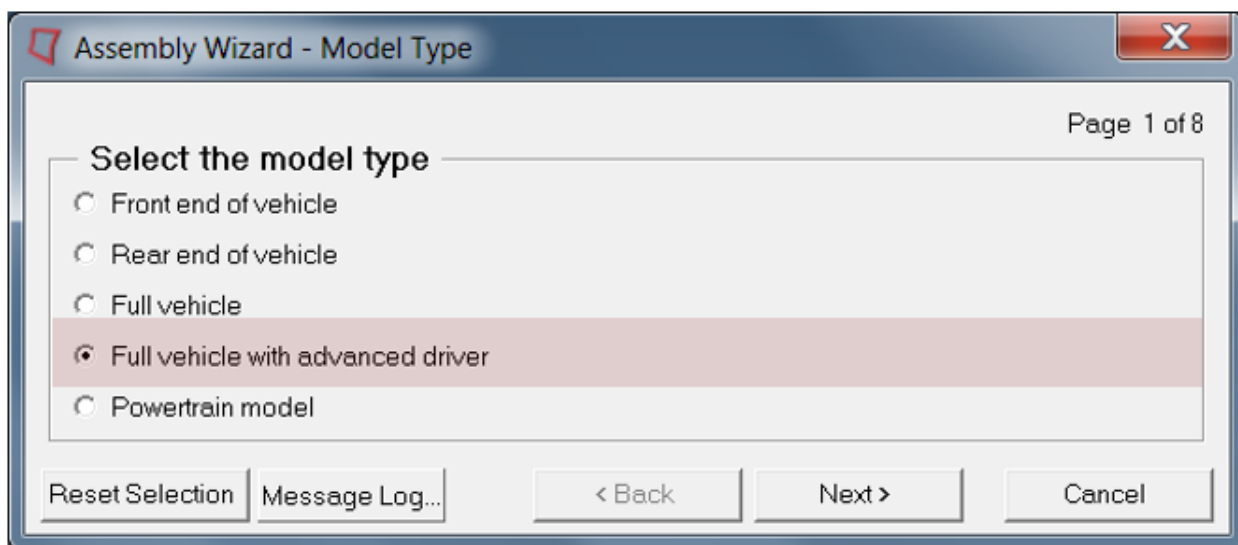
## Advanced Driver and Road Vehicle Simulation

---

### Driver

The Altair Driver is a set of MotionView models and libraries that drives virtual vehicles with closed loop and open loop inputs to perform a variety of events. Examples include driving on a rough, curving road while controlling speed for durability or ride comfort analysis, or driving a skid pad at increasing speed to predict understeer.

This is achieved by controlling the five vehicle inputs Steering, Throttle, Gear, Brake and Clutch so that the vehicle is able to accomplish the required maneuvers. The driver is completely integrated into MDLLIB. You can build a full vehicle for use with the Advanced Driver using MotionView's Assembly Wizard. You do this by selecting the **Full vehicle with advanced driver** option, as shown in the dialog below.



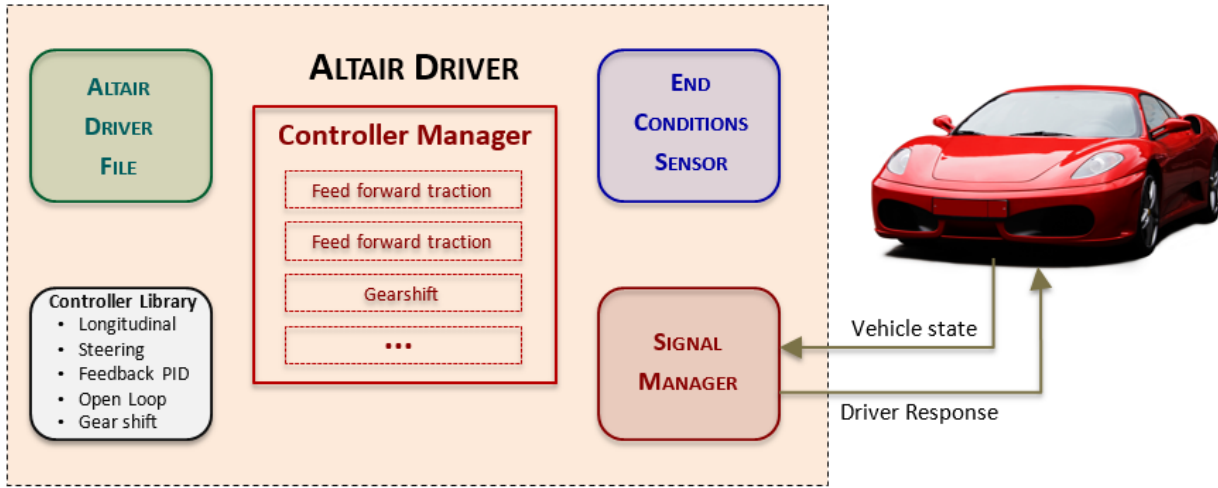
Once a vehicle + driver is built, the next step is to define what the combined system should do. You specify driver events to do this. In this release, the input to the driver model is an "Advanced Driver File". This is an ASCII TeimOrbit format file that scripts a simulation as a sequence of maneuvers. The Altair Driver and the Advanced Driver File support both open and closed loop events.

With Altair Driver File you can define any number of full vehicle events for the driver to execute.

- A complex simulation may be decomposed into many simpler "atomic maneuvers"
- For each maneuver
  - You can select the controllers for vehicle inputs
  - You can define termination conditions for each maneuver.
- You can define open loop, closed-loop or user-defined controllers to control:
  - Longitudinal speed or acceleration.
  - Vehicle path or lateral acceleration.
- Switching controllers during a simulation

- Defining path and speed profiles parametrically, in a table, or by referencing a data file.

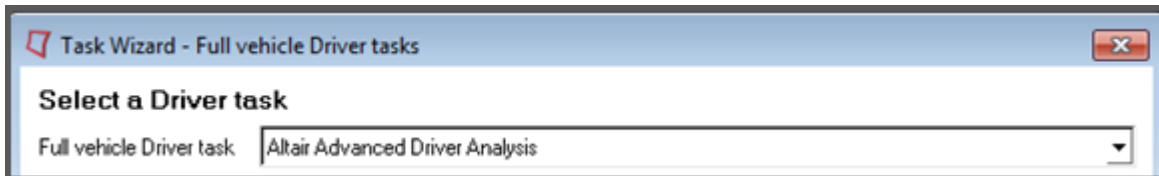
The overall relationship between the Altair Driver and the vehicle is shown in the figure below.



Full vehicle models built for use with the Advanced Driver include the following new systems:

- A four-wheel disc brake system for slowing the vehicle
- Either a linear torque map power train or an IC engine with clutch and manual gearbox for accelerating the vehicle.

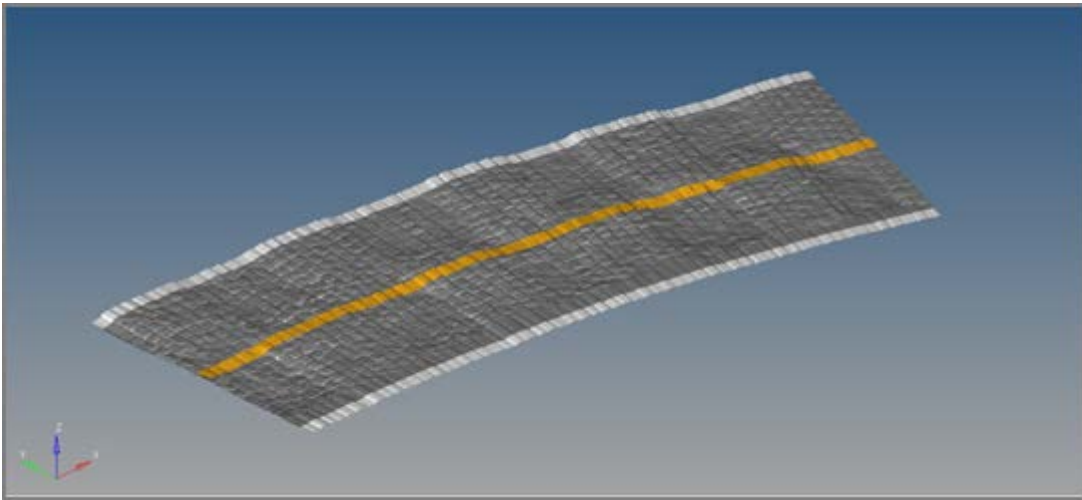
Once the full vehicle is assembled use the task wizard from the Analysis menu to add an Advanced Driver Analysis as shown below.



## Road

Use the Road Graphics Tools to build a MotionView graphic system to visualize the road or to extract a driver path from Curved Regular Grid (.crg), Regular Grid (.rgx) and 3D Spline Road files. Using the Road Graphics Tool you can resample the road to reduce or increase the number points in the driver path or the number of surfaces in the resulting graphics. You can also add lane divider lines and shoulders to the road, and set the colors of the road, the lane divider lines, and shoulders.

The road graphic system the tool creates is automatically imported into your model, but also saved to disk so you import it in any MotionView model.





## Other MotionSolve Enhancements

---

### ▼ New Python Based Lexicon

A new Python based lexicon for MotionSolve is provided in this release. Using this lexicon, one can now create Python applications that can:

- Create a MotionSolve model
- Execute the model
- Use any other Python utilities that are useful in this context.

For instance, you could: Perform a MotionSolve simulation and display live plots during the simulation with Matplotlib (a Python based plotting tool).

Along with the Python based lexicon, a new Python lexicon Reference Manual is also available.

### Supported Entities

ACCGRAV	ACTIVATE	ARRAY	BEAM	BEGIN
BUSHING	CONTACT	CONTROL	COUPLER	CURVE
CVCV	CVSF	DEACTIVATE	DEBUG	DIFF
DV	END	EQUILIBRIUM	FIELD	FLEX_BODY
FRICITION	GCON	GEAR	GFORCE	GRAPHICS
GSE	H3DOUTPUT	INTEGRATOR	JOINT	JPRIM
LSE	MARKER	MATE	MATRIX	MOTION
NFORCE	OUTPUT	PART	PFORCE	PINPUT
POINT_MASS	POUTPUT	PTCV	PTSF	RELOAD
REQUEST	RV	SAVE	SENSOR	SFORCE
SFSF	SIMULATE	SPLINE	SPRINGDAMPER	STOP
STRING	SURFACE	TFSISO	TRANSIENT	UCON
UNITS	VARIABLE	VFORCE	VTORQUE	

### ▼ Support for Multi-Iterative Coupling (MIC) in Co-Simulation with AcuSolve

With this release, MotionSolve supports multi-iterative coupling (MIC) while co-simulating with AcuSolve. This enhancement allows MotionSolve to re-compute its system states on the fly when needed during co-simulation. This is helpful while trying to solve “tightly” coupled problems more accurately. With this enhancement, your co-simulation results are more accurate and the co-simulation is more robust.

▼ **MS\_USERDLL\_DIR Supports Python Scripts**

You can use the MS\_USERDLL\_DIR environment variable to specify the location of your Python scripts (\*.py, \*.pyc) as well as your C++/FORTRAN user subroutines.

The environment variable MS\_USERDLL\_DIR can be set to a list of folder locations on your machine (separated by ;) where you want MotionSolve to search for your user subroutines (\*.dll, \*.so). This scenario is common when you don't specify the absolute path of the user subroutine in your model or if the user subroutine is not in the same location as the solver input XML file.

**Geometric Stiffening for CMS Flexible Bodies**

You can model geometric stiffening for CMS flexible bodies in MotionSolve. Geometric or stress stiffening is an effect most commonly seen in beams under tension where the frequency of the fundamental modes increases as the tension is increased. Such an effect can be observed in rotating beam structures like helicopter or wind turbine rotor-blades.

Include the stress stiffness data in your CMS flexible body by including the following in your FEM file:

```
CMSMETH, 1, CBG, , 15
```

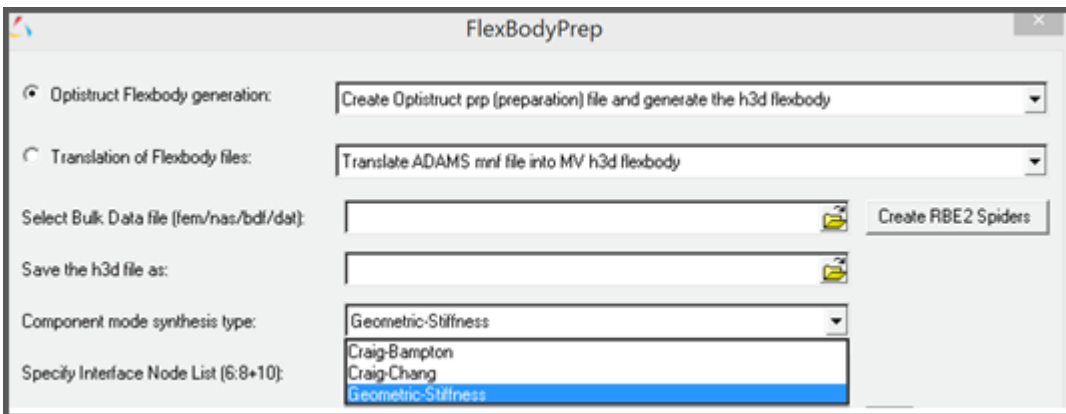
CBG is the keyword that instructs OptiStruct to create the CMS flexible body with additional geometric stiffness data.

You can then import this flexible body in your MotionView model as you would ordinarily. Note: Only the Craig-Bampton method is currently supported for calculating the geometric stiffness data.

This new MotionSolve capability is completely supported in MotionView also.

▼ **Flex-body Generation with Geometric Stiffness can be Done using the Flexprep Tool**

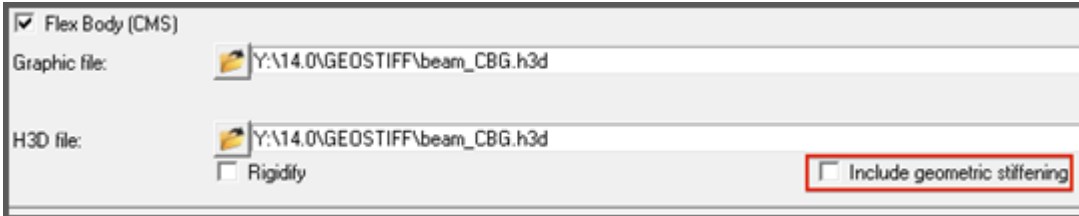
A GUI option is available to specify geometric stiffness.



Selecting the highlighted option would include the relevant CMSMETH statement of the form `CMSMETH, 1, CBG, , 15` into the FEM File.

▼ **Geometric Stiffness Option can be Specified in the Body (Flex body) Panel**

“Include geometric stiffening” checkbox is now available in the Flex-body panel. This option will only be activated when the flex h3d contains the geometric stiff (GEOSTIFF) data that is saved while generating the flexible body during the CMS process.

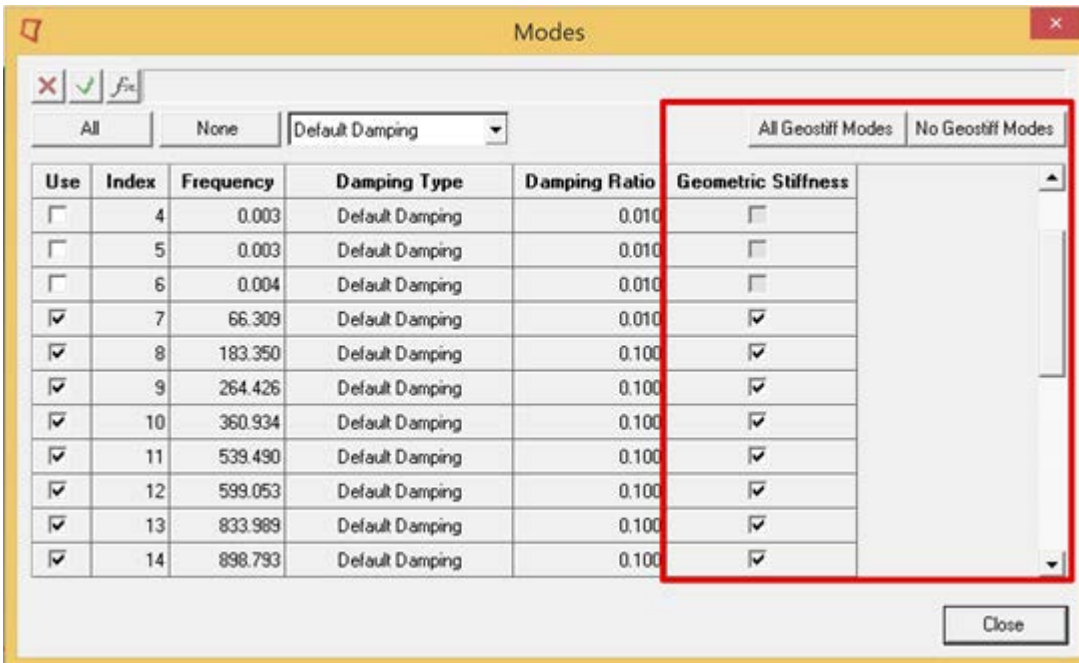


Checking on this option will enable export of the geometric stiffness data into MotionSolve XML file.

▼ **Geometric Stiffness Data can be Selectively Chosen by Mode for Export to MotionSolve**

The “Modes” dialog in the flex-body panel now contains an additional column to control export of geometric stiffness data for each mode.

Turning off geometric stiffness data for certain modes can significantly improve the solution time. However, you should be sure that the modes that are turned off have a minor influence on geometric stiffening effects.



▼ **Linking with msautoutils via the User Subroutine Build Tool**

The user subroutine build tool shipped with this release of MotionSolve now links with the msautoutils library on both Windows and Linux. You can now use access functions exported by the MotionAuto Utilities as well as the MotionSolve UserSub API. This substantially simplifies the effort

on your part to create your own road or other Auto functionality based on the published set of capabilities.

▼ **Updated ISTAT Codes for the GTCMAT Utility Subroutine**

The ISTAT return codes for the GTCMAT utility have been updated in MotionSolve.

ISTAT= 0	Normal return. Compliance matrix was calculated.
ISTAT= -1	Error return. $NM < 0$ .
ISTAT= -2	Error return. $NM > 100$ .
ISTAT= -3	Error return. $NC < 6 * NM$ .
ISTAT= -6	Error return. MKID refers to a non-existent REFERENCE_MARKER.
ISTAT= -7	Error return. The system has zero degrees of freedom.
ISTAT= -9	Error return. A singular Jacobian matrix was encountered during the calculations.
ISTAT=-11	Error return. A REFERENCE_MARKER is on GROUND. It cannot move.
ISTAT=-12	Error return. An initial static equilibrium was not performed.
ISTAT=-99999	Error return.

▼ **Linear Analysis Enhancements**

Many enhancements have been added to Linear Analysis. These include:

- A new option for “balancing” prior to eigenvalue analysis
- Support for strain/dissipative energy calculations
- Mode selection for linear analysis output

The updated statement is shown below with the 14.0 enhancements in red.

<Param\_Linear

```

anim_scale      = "real"
balancing       = {"TRUE" | "FALSE" | "AUTO"}
disable_damping = {"TRUE" | "FALSE"}
pinput_id      = "integer"
poutput_id     = "integer"
write_eig_info  = {"YES" | "NO"}
write_energy_dist = {"YES" | "NO"}
write_matlabfiles = {"YES" | "NO"}
write_simulinkmdl = {"YES" | "NO"}

{ke_modes_exclude = options}
{ke_modes_include = options}

{se_modes_exclude = options}
{se_modes_include = options}

{de_modes_exclude = options}
{de_modes_include = options}

```

Options:

- Select a single mode
- Select multiple modes
- Select a range of modes
- Select all modes

/>

### A new capability for “balancing” prior to eigenvalue analysis

For certain models, the eigenvalues calculated are sensitive to very minor changes in the state matrix used as input for the solution. This occurs when the eigenvector matrix is ill conditioned. MotionSolve detects an ill conditioned eigenvector matrix and “balances” the state matrix for a more robust eigenvalue solution.

Balancing refers to diagonally scaling the state matrix such that the row and column norms are numerically close to each other. You can control this behavior using the attribute “balancing” in the <Param\_Linear> model or command statement.

Set “balancing” to “TRUE” to force MotionSolve to perform balancing on the state matrix and “FALSE” to disable balancing. The default for “balancing” is “AUTO” which lets the solver decide when balancing is required based on the condition number of the eigenvector matrix. Please note, that balancing is a very inexpensive operation.

### Support for strain/dissipative energy calculations

For linear analyses, in addition to Kinetic Energy distribution, MotionSolve also calculates strain and dissipative energy for the following modeling elements:

- Force\_SpringDamper (TSPDP/RSPDP)
- Force\_Bushing (BUSH)

- Force\_Beam (BEAM)
- Force\_Field (FIELD)
- Force\_VectorOneBody/Force\_VectorTwoBody (VFORCE, VTORQUE, GFORCE)
- Force\_Scalar\_TwoBody (SFORCE)

The KE, strain and dissipative energy distribution is written to the log file, on the screen and the \*\_linz.mrf output file. To request MotionSolve to calculate these distributions, set the write\_energy\_dist attribute to "TRUE" in the <Param\_Linear> model or command statement.

### Mode selection for linear analysis output

You can now specify the modes for which MotionSolve writes the energy distribution on screen and to the log file. This is useful when the number of eigenmodes is large and you are interested only in a subset.

You may specify mode numbers to include or exclude, for each of the three distributions. Please see the documentation on <Param\_Linear> for more details on how this is done.

### ▼ Header Format Change for PLT Output Files

With this release, the header for the PLT output files has been changed to allow the PLT file to be imported into FEMFAT for fatigue analysis. The old and new PLT format are described below:

**Old format:** The old format contains 4 columns for each request and the version number is 8.1.

#### MotionSolve v8.1 Plot File

Generated on 12-AUG-2015 12:27:29, Altair Engineering, Inc.

```

19      MMKGS_N_RACA 1.0000000000000000e+003
70000005          4      30102080      30101080
rg1(Output Body1)
70000006          4      30102020      30103020
r12(Output Body1)
70000007          4      30103020      30102020

```

**New format:** The new format contains 5 columns for each request and the version number is 9.1. The fifth column is added as per the specifications from FEMFAT. If this value is non-zero, then the next line is treated as a comment line.

MotionSolve v9.1 Plot File

Generated on 12-AUG-2015 12:09:55, Altair Engineering, Inc.

```

19      MMKGS_N_RACA 1.0000000000000000e+003
70000005      4      30102080      30101080      5
rg1(Output Body1)
70000006      4      30102020      30103020      5
r12(Output Body1)
70000007      4      30103020      30102020      5

```

▼ Kinematic Analysis for “Structure” Models

Previously, MotionSolve did not solve models that have 0 DOF and time dependent loading. To solve such models you needed a workaround with “dummy” differential equations etc. With this release, MotionSolve has been enhanced to recognize 0 DOF models that have time dependent loading such that the simulation can continue. Such a scenario is useful when trying to determine joint reaction forces for models with 0 DOFs.

## Compatibility with ADAMS Language

▼ FX Function Compatibility

The definitions of the functions FX, FY, FZ, TX, TY and TZ have been made consistent with what is used in ADAMS. The table below lists the changes, using the FX function as an example:

Function	Arguments	Behavior
FX (I, J, K)	I, J and K markers	Computes the X component of all the forces acting between marker I and J resolved in the coordinate system of marker K.
FX (I, J)	I and J markers	Computes the X component of all the forces acting between marker I and J resolved in the ground coordinate system.
FX (I)	I marker	Computes the X component of all the resultant forces at marker I. To calculate the X component of action-only forces at the I marker, specify <code>frc_expr_action_only = "TRUE"</code> in the <code>&lt;ResOutput&gt;</code> command element in the XML input file.

### ▼ Graphic Statement Compatibility

MotionSolve now supports the adm statement GRAPHICS/... for displaying force vectors in the animation H3D. A sample statement is

```
GRAPHICS/10215001, FORCE, ETYPE=ALL, EM=10215001
```

The following ETYPE are now supported:

ETYPE	ALL, BEAM, BUSH, FIELD, GFORCE, JOINT, JPRIM, NFORCE, SFORCE, SPDP, VFORCE, VTORQUE
-------	---

### ▼ Correct Conversion of Output Steps to XML for Multiple Analyses

Previously, a model that had multiple SAVE/RELOAD statements and analyses with different output steps was not translated correctly to MotionSolve. An example of such a scenario is shown below:

```
RELOAD/SYSTEM, FILE=model.case5  
SIM/STA, END=5.0, STEP=20  
SIM/STA, END=6.0, STEP=5  
SIM/STA, END=6.1, STEP=5  
SIM/STA, END=6.5, STEP=4  
SIM/STA, END=6.9, STEP=4  
SIM/STA, END=7.0, STEP=5
```

This has been resolved within this release.

## Resolved Issues

### ▼ Solver Behavior Updated for Vehicle Models Containing Tires

With this release, the solver behavior for any model that contains tires has been updated to produce more accurate velocities and accelerations. The solver now recognizes force elements that model the tire and monitors the rotating wheel bodies so that the velocities and accelerations are computed more accurately.

For models that make use of the <[Force\\_VectorTwoBody](#)> modeling element to represent tires, you can expect to see smoother results slip ratio and tire rotational velocity.

### ▼ Using Python User-Subroutine Scripts on a Different Network Location than MotionSolve

Previously, MotionSolve was unable to access a Python script that was not located on the same machine as the MotionSolve installation. This issue has been fixed, now, you can run a simulation in a scenario where MotionSolve is installed on one machine and the scripts are located on another network location. Note: you need to have read access to the network location where your scripts are located.



### ▼ Flexbody Visibility of MotionSolve H3D in HyperView

Previously, if your model consisted of multiple flexible bodies generated using a combination of PLOTEL and non-PLOTEL elements, the flexible bodies would not be displayed properly in HyperView. This issue has been fixed within this release.

### ▼ Resolved Issues in Linear Analysis

Several issues have been fixed for the linear analysis within this release:

- If you request Simulink or MATLAB output files after a linear analysis, MotionSolve requires a valid [<Control\\_PlantInput>](#) model element that defines the input and a [<Control\\_PlantOutput>](#) model element that defines the output for the linearized system. If these IDs are not specified or specified incorrectly, MotionSolve will issue an error message and stop the simulation. These IDs can be specified via the [pinput\\_id](#) and [poutput\\_id](#) attributes in the [<Param\\_Linear>](#) model or command statement. See the documentation for [<Param\\_Linear>](#) for more details. For some models, MotionSolve wrote incorrect B and C matrices after a linear analysis when “[write\\_matlabfiles](#)” is set to [TRUE](#). This was determined to be an issue with numerical tolerance and has been fixed for this release.
- For a model that contained only flexible bodies, MotionSolve crashed while writing the Kinetic Energy distribution. This issue has been fixed within this release.
- While requesting a linear analysis on a model with no force elements (or zero total force), MotionSolve crashed during the linear analysis simulation. This was determined to be an error in the solver code and has been resolved starting with this release.
  - When the ADM/ACF deck contained more than one Linear command statements, the translation to XML resulted in a Param\_Linear statement which was not equivalent. This has been resolved within this release.

### ▼ Building User Subroutines by Using the GCC Compiler on Linux

In the previous release, you were unable to link your user subroutine code with the MotionSolve user subroutine API library ([ms\\_usersubapi](#)) using the GCC compiler. Starting with this release, this restriction has been removed. Now you can use both the GCC and Intel compilers to compile and link your user subroutine code.

### ▼ Incorrect Gear Ratio Calculated when Using the Constraint\_Gear Modeling Element

Previously, a gear modeled using the [<Constraint\\_Gear>](#) statement resulted in an incorrect gear ratio calculation in the solver. This has been fixed within this release.

### ▼ The SENVAL Function Returns an Incorrect Value when the Sensor is not Activated

The [SENVAL](#) function should return a value of 0 when the sensor it refers to is not active during the simulation; this behavior was not seen in the previous release. This has been fixed starting with this release.

▼ **SENSOR Causes a Crash if Deactivated before the First SIMULATE Command**

If your model contains a <Sensor\_Event> modeling element that is deactivated before the first <Simulate> command is executed, the simulation would crash with the previous release.

This was determined to be a bug and has been fixed with this release.

▼ **Post-Processing Module (MS-Post) Crashes with '0' Specified as a Marker ID**

Previously, the post processing module in MotionSolve crashed if your model contained an output request that referred to a '0' marker ID. This scenario may occur with older MotionSolve models. This has been fixed such that the post module simply ignores such a marker request.

▼ **PROXIMITY Function Referring to a Deactivated Sensor Crashed the Simulation**

Previously, if your model contained a PROXIMITY function that referred to a deactivated sensor in your model, the simulation would crash in such a scenario. This has been fixed such that the use of such a PROXIMITY function does not halt the simulation. Note: It is important to keep track of which sensors are deactivated in your model since the PROXIMITY function referring to a deactivated sensor will return a 0 value.

▼ **Rigid Body Contact for Box Graphic Defined with a Negative Length**

Previously, if you modeled 3D rigid body contact with at least one body being a box graphic defined by a negative length, the contact force was not generated as expected. This has been fixed with this release.

▼ **IF Function Doesn't Return User Defined Value**

Previously, if you defined a solver variable using multiple IF statements and queried the solver for this variable's value in a [Post\\_Request](#) using [VARVAL\(..\)](#), MotionSolve did not return the exact values for the variable as defined by the user. Specifically, the values were incorrect at the points where the gradient of the expression for the solver variable is discontinuous.

This issue can be resolved by specifying [dae\\_interpolation = "FALSE"](#) in the <[Param\\_Transient](#)> model or command statement in the MotionSolve XML input file.

▼ **Missing "XP" and "ZP" of Marker in XML**

Markers with direction cosine matrix very close to the identity matrix were not translated correctly to the MotionSolve XML input file. This was caused by a loose numerical tolerance in MotionSolve.

▼ **Sign of Velocity at Discontinuous Operating Point**

Previously, for a kinematically constrained model, if you specified a motion on a joint using multiple [IF](#) statements, the velocity of the marker associated with the joint showed a difference if the model was solved using the [VSTIFF](#) or [DSTIFF](#) integrator.

This has been fixed within this release – the choice of the integrator does not matter for such a kinematic model.

**Note:** Use of the [IF](#) function to define motions is not recommended in MotionSolve, since the IF function tends to produce discontinuities which make it hard for the solver to converge and can produce erroneous results in certain cases. Please refer to the documentation on [Motion\\_Joint](#) for more tips on defining smooth motion inputs.

#### ▼ **Incorrect Gear Ratio when Using “Constraint\_Gear” Model Element**

Previously, a gear modeled using the [Constraint\\_Gear](#) statement returned an incorrect gear ratio in MotionSolve. This has been fixed within this release.

#### ▼ **State Variable Doesn't Return User Defined Value Correctly, Especially in Transient Analysis**

Previously, a Motion defined as a saw-tooth function using a series of IF statements returned results inconsistent with the user input. This has been fixed within this release.

#### ▼ **Multiple Simulates with Time Dependent Motion Result in Incorrect Answers**

Previously, if your model contained multiple [Simulate](#) commands and a [Motion\\_Joint](#) modeling element was modified in between the analyses, MotionSolve did not update the [Motion\\_Joint](#) value correctly thus yielding incorrect results or simulation failure. Such a scenario is shown below:

```
<Motion_Joint
  id      = "1"
  expr    = "STEP(TIME,0,0D,1,360D)"
/>
<Simulate
  analysis_type = "Static"
  end_time     = "1."
/>
<Motion_Joint
  id      = "1"
  expr    = "360D+0.0*(TIME-1)"
/>
<Simulate
  analysis_type = "Static"
  end_time     = "2."
/>
```

This has been resolved with this release.

▼ **Suspension Design Factor (SDF) Outputs Caused Certain Models to Fail During GTCMAT Calls**

Previously, certain models that had SDF outputs defined in them failed the GTCMAT calls. This was determined to be an issue in the solver code and has been fixed with this release.

▼ **Save and Load Issue with Models Containing Joint Friction**

Previously, if you save and load your model that contains joint friction modelling elements, the reloaded model will not solve correctly. This has been identified as an error in writing the model upon saving.

## Known Issues

▼ **NLFE**

- The damping parameter for NLFE components is a coefficient for a Rayleigh damping model. Since this is a numerical damping, it is advised to start from a value of 0 in your model, since critical damping can be achieved with a small value and a larger value may yield unrealistic results.

▼ **Others**

- When using a custom element definition (including `<Force_JointFriction>`) that uses GFORCE with the ground being defined as the I body, the results may be incorrect. As a workaround, insert a dummy body between the ground and the J body.

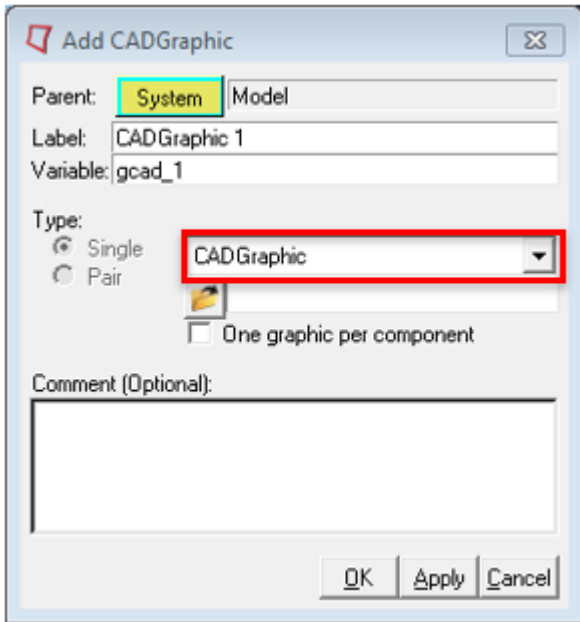
## Other MotionView Enhancements

---

This section documents other important enhancements that have been made to MotionView.

### ▼ Direct Geometry Import

In this release, MotionView introduces new functionalities based on the Parasolid geometry engine. A new ability to import geometry of type “CADGraphic” is now available. All common formats can be imported.

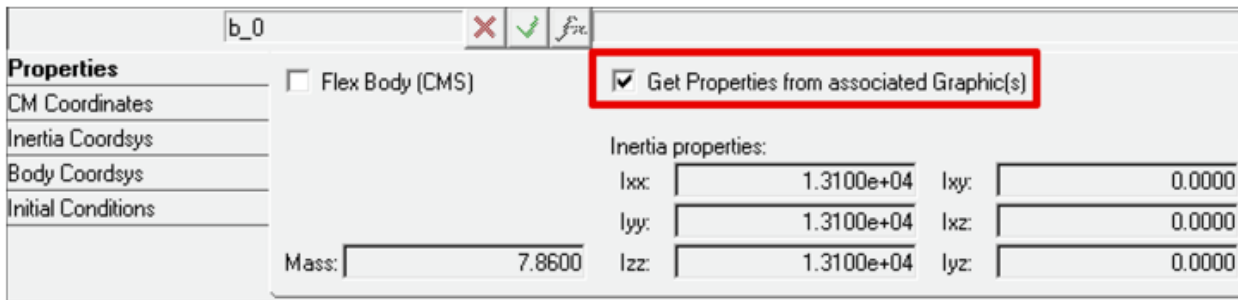


You will experience several benefits because of this capability.

- All common CAD file formats are supported and can be easily imported as Graphic statements
- CAD Geometry imported in this way has mass and inertia attributes
- The Body to which CADGraphics are attached, can automatically inherit the mass and inertia properties based on the geometry properties
- User can pick specific components in geometry and export them to solver. A separate parasolid file will be generated with the component being selected. No mesh data written to XML file.
- User has the flexibility to change mesh refinement level of geometry by adjusting the fidelity factor before exporting to MotionSolve.

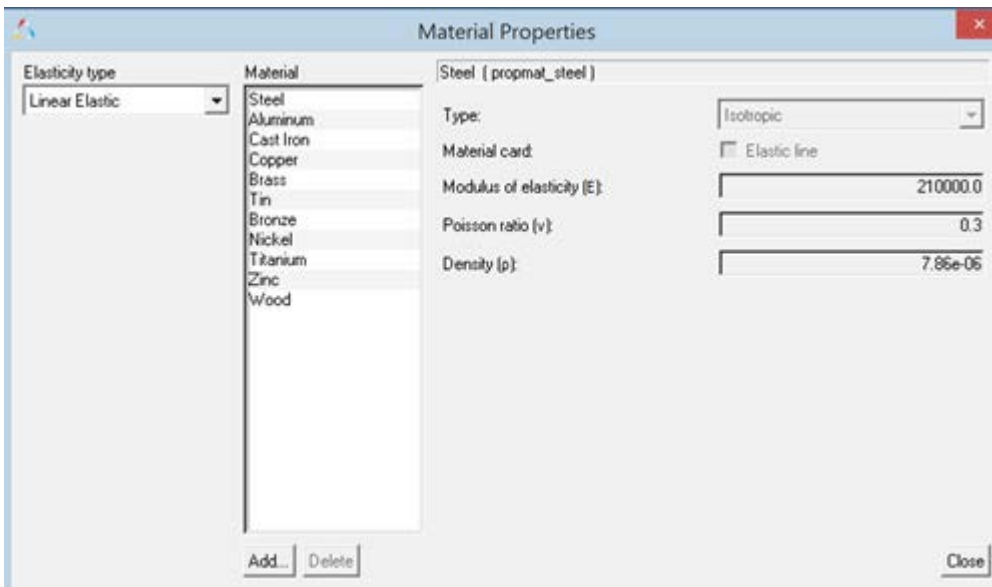
The underlying Parasolid geometry engine is also used to automatically compute mass and inertia properties for simple analytical shapes such as spheres, cylinders, boxes and frusta.

An option is provided in body panel using which user may choose to inherit mass and inertia properties from associated graphics.



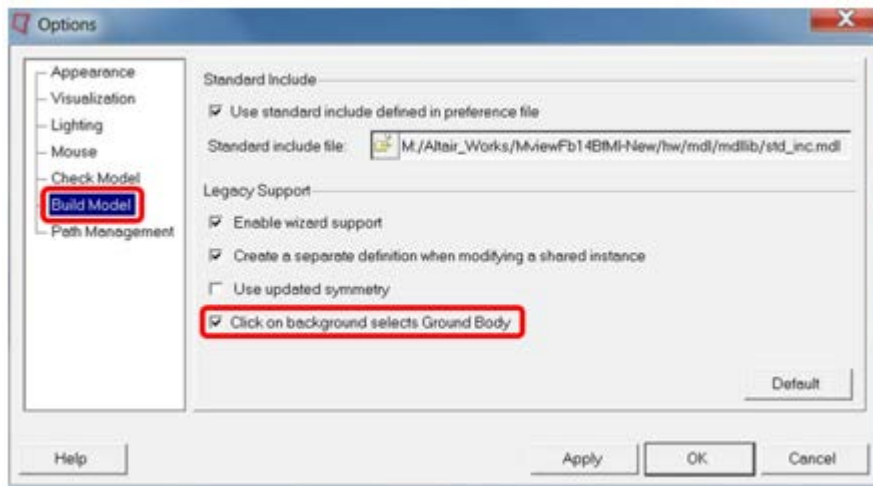
### ▼ Materials

A default set of materials with material properties is now available. These materials can be used either in an NLFE Body or CADGraphics. The Materials can be accessed using the menu **Model > Materials**.



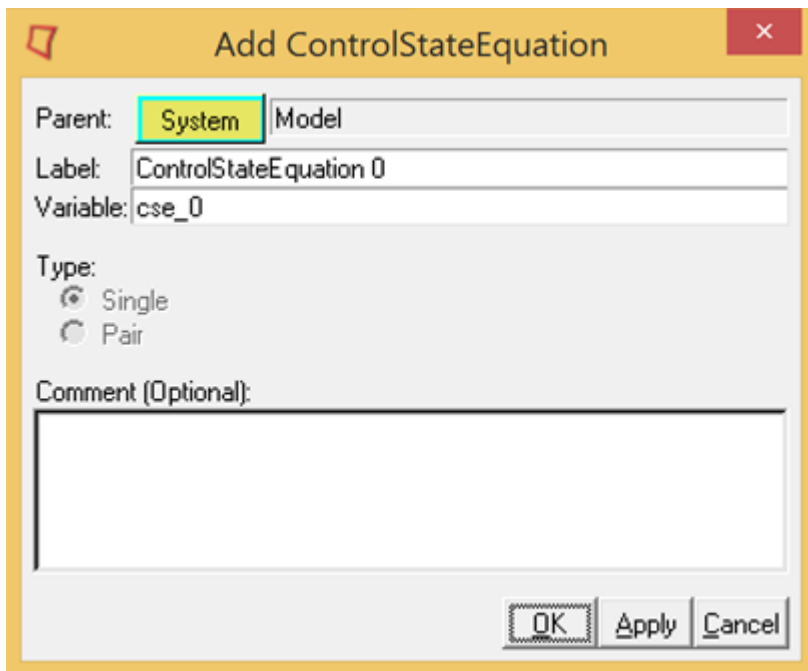
### ▼ Usability Improvements

- Multiple picking from Graphics area to cut/delete entities. Hold Shift key to select many entities.
- Merging multiple bodies from the graphics area. User can select two or more bodies (using the Shift key), use the right click to merge the bodies to a new body or to Ground Body
- A new option is now available in MotionView to select a Ground Body. When Body collector is active (to pick a body as a reference into an entity panel), clicking on empty space in the Graphics area picks Ground Body with following option ON.

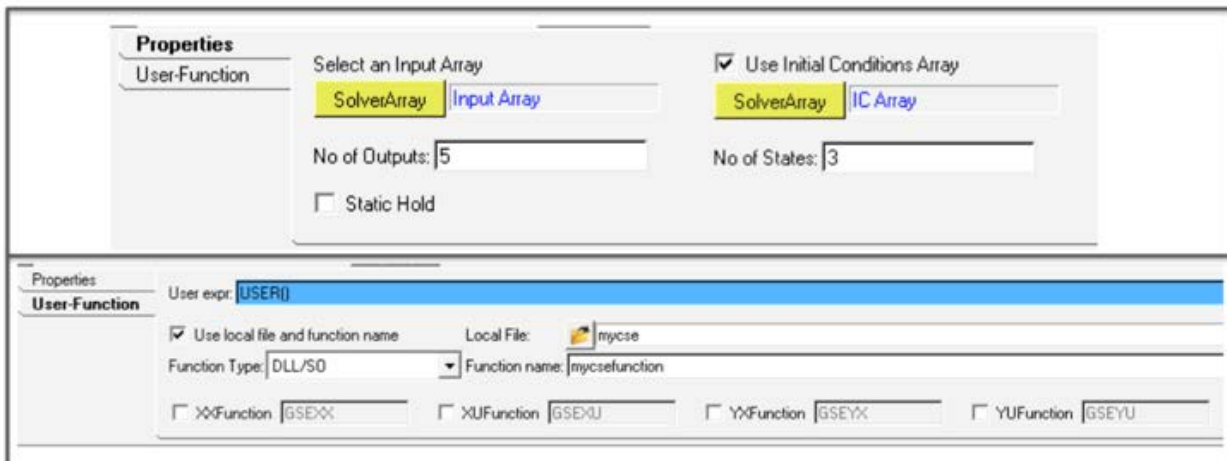


### ▼ General State Element (GSE)

The General State Element, also known as [CONTROL\\_STATEEQN](#) in MotionSolve, is now fully supported. This includes the user interface and complete MDL support. You no longer need a Templex template to use this capability. A Control\_StateEquation icon is available in the “Control Entity” toolbar to add the entity.

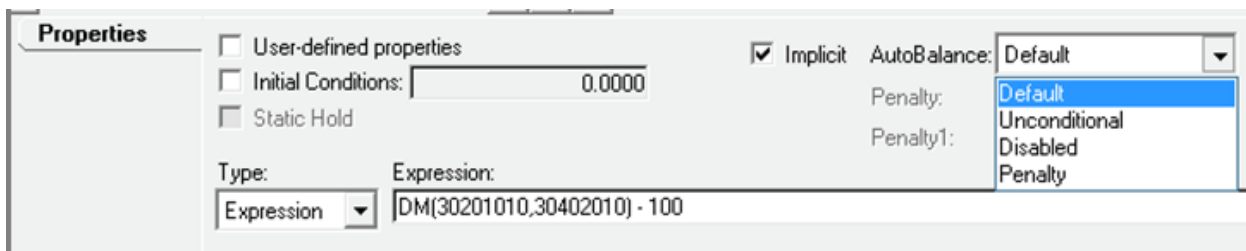


The two tabs in the panel can be used to define all the necessary inputs to the control state equation.



### ▼ Implicit Variables

This enhancement is applicable only in the MotionSolve mode. Traditionally, the [REFERENCE\\_VARIABLE](#) in MotionSolve has explicitly defined a new algebraic variable through an expression or a user-subroutine. Solver Variables are now extended so that they are *implicitly* defined. This means that [VARIABLES](#) can now be used to define constraints, and the [VARIABLE](#), instead of assuming the value of the expression, now becomes the Lagrange Multiplier required to enforce the constraint.

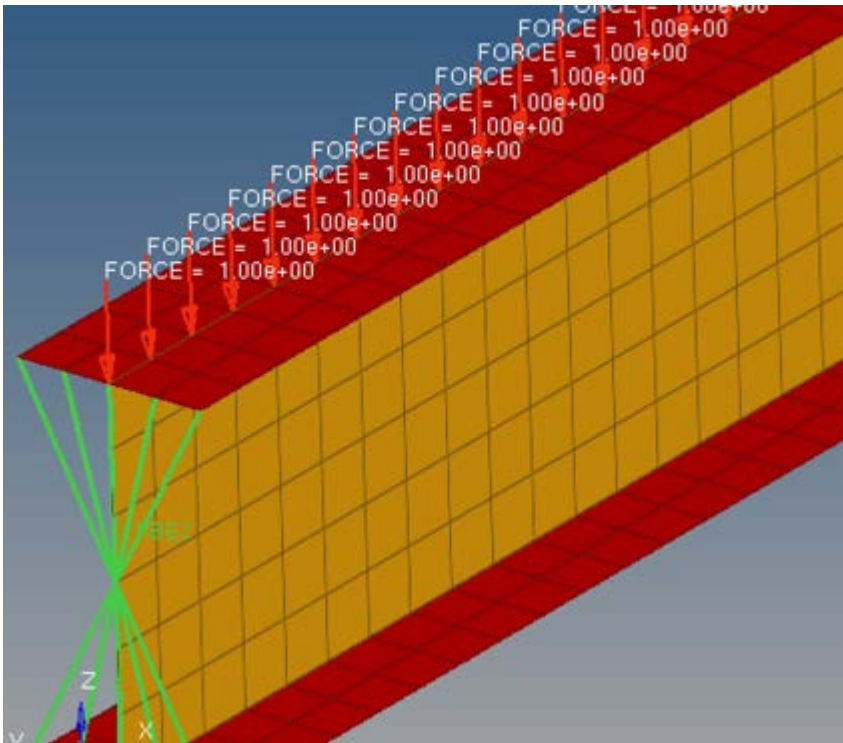


The SolverVariable panel along with the implicit option has additional controls to tell MotionSolve how the Lagrange Multiplier has to be handled.

### ▼ Modal Force

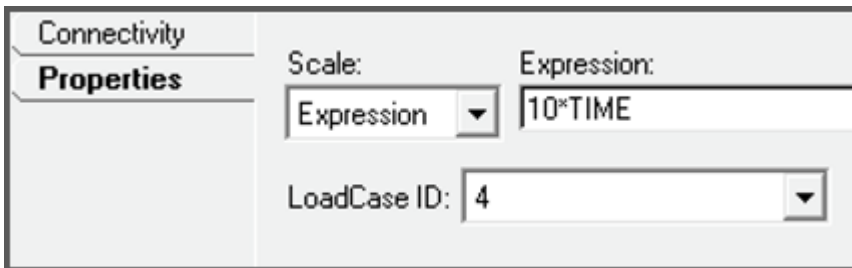
MotionView now supports adding a modal force on a flexible body (Force\_FlexModal in MotionSolve). A modal force can be a distributed force such as fluid pressure or aerodynamic load that can vary with regards to time time.





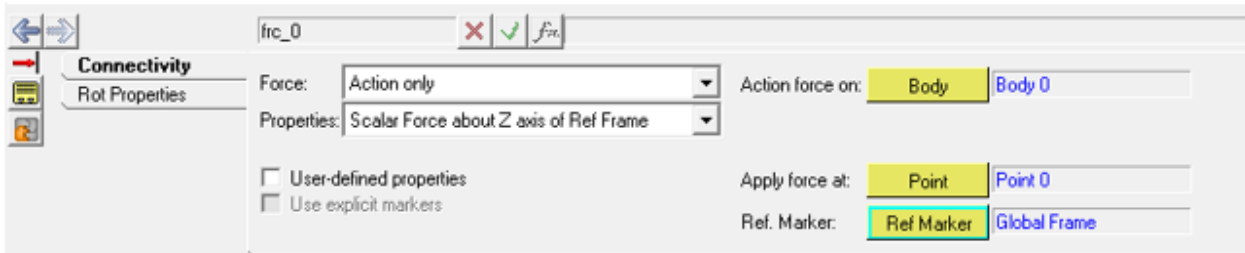
The distributed force should be applied on the FE model that would be used to generate the CMS flexible body. The force would be decomposed into modal form.

MotionView presents the modal force as load case ID as available in the flexible body H3D. The modal force can be scaled during simulation if needed.



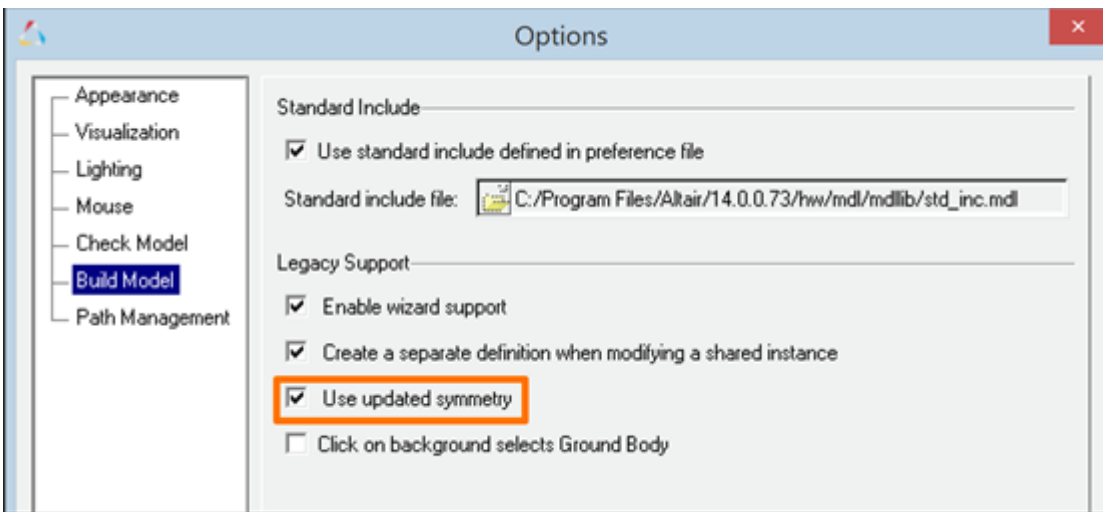
#### ▼ Scalar Action only Force and Torque

MotionView now supports a Scalar action only force or torque. This means that Action Only forces or torques can be applied in a direction specified by the z-axis of a Marker.

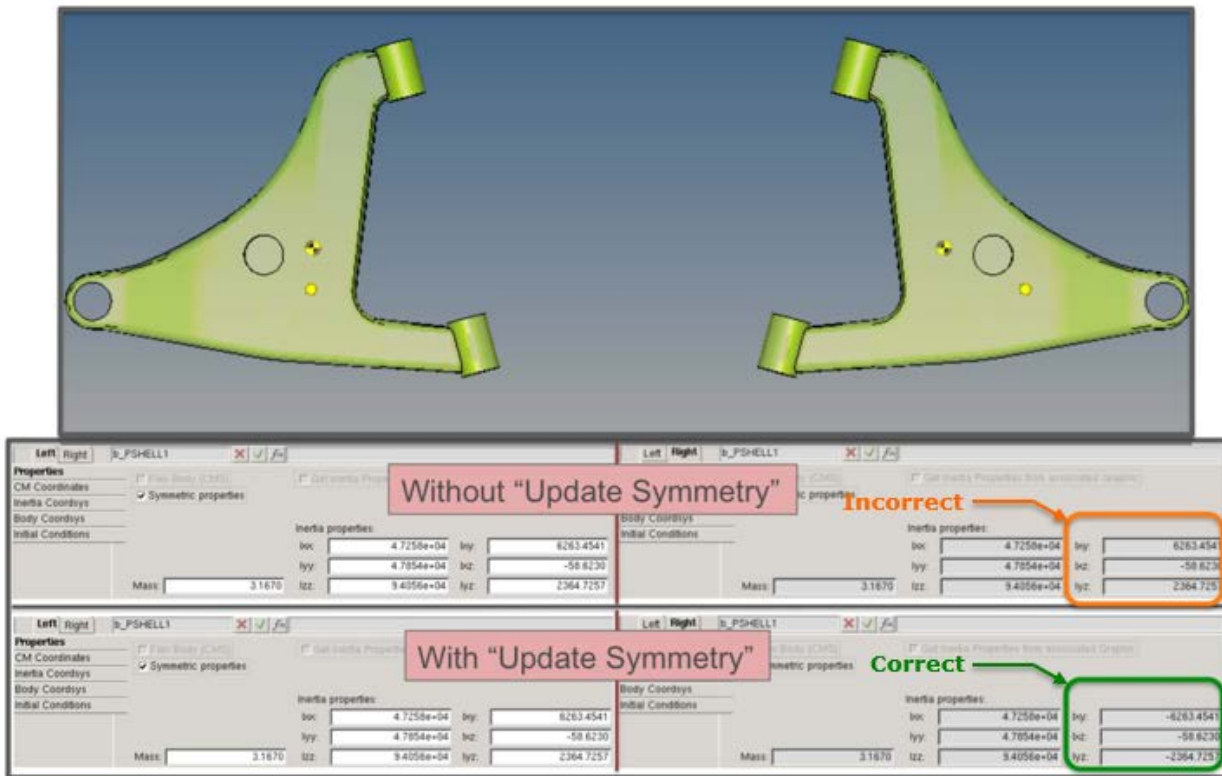


▼ **Mass and Inertia Value Correction for Symmetric Pair Bodies**

A new symmetry option is now available in MotionView to define symmetric body pairs. Cross products of inertias are properly negated (about the Global ZX plane) when a body is reflected. This new option can be turned on using the **Tools > Options > Build Model**. The symmetric pair's properties correspond to the bodies being represented as mirror reflections of each other.



The new calculation considers the offset between the CM marker and IM marker if any. Non-symmetry of the CM location is not considered. Also, the symmetry is not applied to initial conditions.



### ▼ "Body Deformable" Changed to "Flex Body (CMS)"

The option "Body Deformable" that is used to define a flexible body has been renamed to "Flex Body (CMS)" to distinguish it from the NLFE body. (The NLFE Body is added through the "Add Body" dialog)

The "Flex Body CMS" panel has additional options:

- Rigidify – to set the MotionSolve attribute `rigidified = "true"` in `<Body_Flexible`
- Include Geometric stiffening – To include geometric stiffening during solution of flexible body whose H3d has geometric stiffness data

### ▼ Import CAD or FE

- Now opens with mesh options expanded
- Graphic H3d files are automatically populated while selecting an input file
- Multiple confirmation messages related to successful import are eliminated.

### ▼ API for Calculating Aggregate Mass, CG and Inertia Properties of Systems

MotionView now provides a new TCL API, `GetAggregateCGAndInertia`, which allows you to calculate the aggregate mass, center of gravity, moment of inertia and product of inertias for multiple

systems or multiple bodies in your model. Documentation for this function is included with this release.

#### ▼ **Enhanced FatiguePrep Tool**

MotionView now supports translation of .plt to RPC format.

#### ▼ **Recent Files**

The File menu now lists the four most recent MDL files used.

## **Resolved Issues**

#### ▼ **Graphic Selection/Picking**

Linux/Mac Entity Selection: When a body is selected in a browser on Linux, all the mesh lines of the graphics of that body were highlighted. This is now resolved

Pick center choice with Ctrl+Middle mouse button was not being honored correctly. The graphics used to rotate about a different center than selected. This issue is fixed.

#### ▼ **ADM Import**

ADM files with expressions having large blank spaces from the beginning of the line leads to expressions being truncated. This issue is fixed in this release.

Import of ADM that did not contain MATRIX information for flexible body resulted in crash of application. This issue has been fixed.

#### ▼ **Flexprep**

Issues related to incorrect modes and interface nodes while converting .h3d files to .mnf files and .mnf to .h3d files are resolved within this release.

Erroneous GENSTIFF and SELNOD matrix was being written out while converting .h3d to .mtx when using kgf as the solver unit. This issue is fixed within this release.

An application error while using Create RBE2 spiders option in flex prep is fixed. The error appeared while closing HyperMesh button with the Close (x) button.

#### ▼ **Graphics Entity**

A custom ID specified for a graphic entity was lost on Check Model. This issue is fixed.

MotionView would not report error when components specified in \*Graphic statement of File type were missing. This issue has been fixed.

An application error while changing the mesh scheme for File Graphics under a Graphic system is fixed.

#### ▼ MDL

An issue with false referencing of entity due to use of same variable names in a system as well as at Model level is now fixed. When an entity (e.g. p\_0) that existed at Model level was referenced in a system that contained entity of same type and same variable name (p\_0) was being saved with a local address (p\_0 instead of MODEL.p\_0).

White spaces in the USER expression in Sensor entity evaluate subroutine resulted in the expression being truncated. This issue is resolved.

Setting of MAF export option was lost on model save. This issue has been resolved.

#### ▼ Miscellaneous

Statics simulation settings - Maximum # of iterations is now defaulted to 75

Write results data per time option is now set to LAST by default instead of ALL. This resolves issues in post-processing due to multiple results at certain time steps in a sequential simulation.

Outputs of type Entity sets on bodies used to result in many outputs with the same label. This issue has been resolved.

An application error in Curve panel while changing type from File to Values is fixed. The error was encountered when a file path was unresolved.

An application error encountered while using the Back/Forward button in any panel is resolved. The error used to occur while using the button after deleting an entity that was in the selection history.

## Known Issues

#### ▼ NLFE

- Providing a specific LPRF is not supported for NLFE Body
- Creating Helical Spring Subsystem with preload is not supported yet

#### ▼ Material Properties

- MotionView does not scale material property values when units are changed
- In the Deformable Surface panel and the Deformable Curve panel, the expanded table does not allow you to pick an entity into a collector.

#### ▼ Contact

- Switching between radio selections in Properties tab may be slower for large models

- Importing ADM – CONTACT statements are not converted into Contact entity in MotionView in some cases. The statement may end up in the Unsupported statements template.

▼ **Modal Force**

- Including Modal force during flexible body generation is not yet supported through Flex prep
- Modal Force does not have a graphic representation in MotionView

▼ **Geometric Stiffening**

- Unchecking and rechecking "Include Geometric stiffening does not retain the Geostiff modes checkbox

## Other Auto Solution Enhancements

---

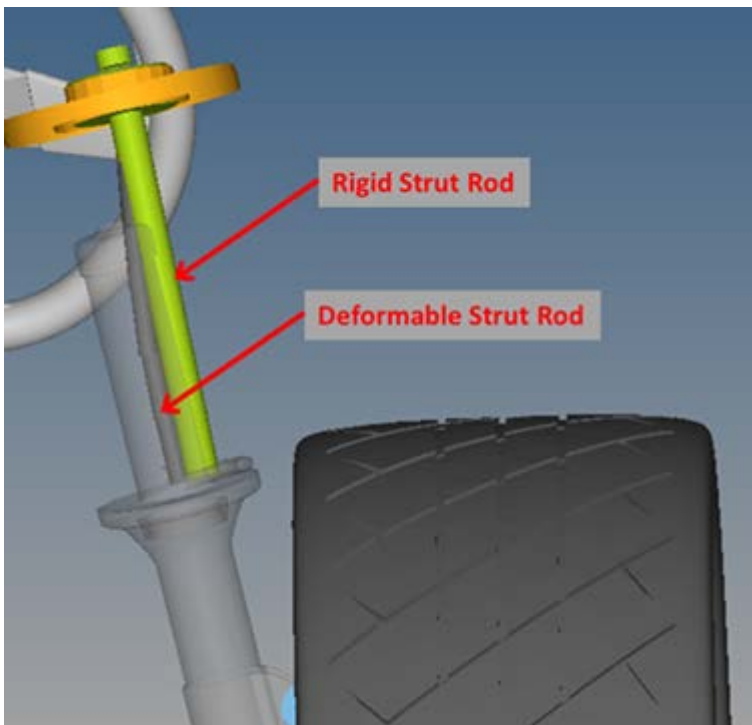
This section documents other important enhancements that have been made to the automotive simulation capabilities in MotionSolve.

### ▼ Deformable Strut

A new deformable strut is a new option available in the Assembly Wizard for MacPherson Strut and Quadra-Link Suspension in the MDLLIB. In the deformable strut, beams model the strut rod and a point to deformable curve constraint models the strut Tube to strut rod joint. This allows the strut to bend under load and captures the camber change due to lateral force common to strut suspensions.

Users with existing suspension or full vehicle models that wish to use the deformable strut can import the system definition, `sysdef_deformable_strut`, from:

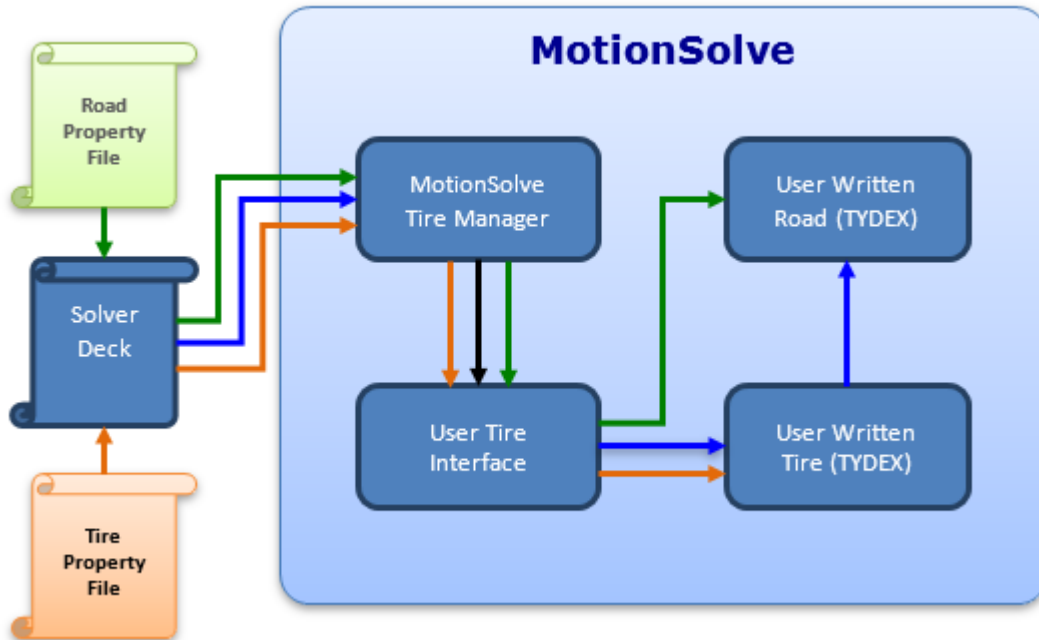
[\\$ALTAIR\\_HOME/hw/mdl/mdllib/Libs/Models/Frnt\\_strut/polybeam\\_strut.mdl](#)



### ▼ User-Defined Tire and Road Models

Tires and their interface with the road can be modeled in multiple ways. The Tyre Data Exchange (TYDEX) is an accepted standard for this purpose. Many commercially and openly available tire and road models are described in the TYDEX format. Any tire or road model that conforms to this format may now be used with MotionSolve.

The user written tire model connects to MotionSolve using the Altair User tire interface. The User tire interface accepts a tire model that is in the TYDEX format along with an accompanying user defined TYDEX road model. This is shown in the figure below.



▼ **Support for Version 6.2.0.2 of TNO Delft Tyre**

MotionSolve now supports MF-TYRE/MF-SWIFT Version 6.2.0.2 on the Windows and Linux platforms. Notable new features include a motorcycle version of MF-SWIFT for high inclination angles and support for OpenCRG release 1.0.5.

Note: The licensing of MF-SWIFT has changed in version 6.2.0.2. MF-SWIFT users must obtain a new license file and new license server software from TNO. MF-TYRE is not affected.

▼ **Support for Version 3.0 and 4.1 for CDTire**

This release includes support for version 4.1 of the CDTire model. Within this release, you may optionally select to run CDTire version 3.0.3 with MotionSolve by setting the environment variable CDTIRE\_VERSION to "303". If the environment variable is not set, then CDTire Version 4.1 is used.

▼ **Support for CDTire on Linux**

You are now able to run models that use CDTire on Linux. This release supports version 4.1 of the CDTire.

Note: A new license is needed to run CDTire v.4.1. CDTire version 303 is not supported on Linux.

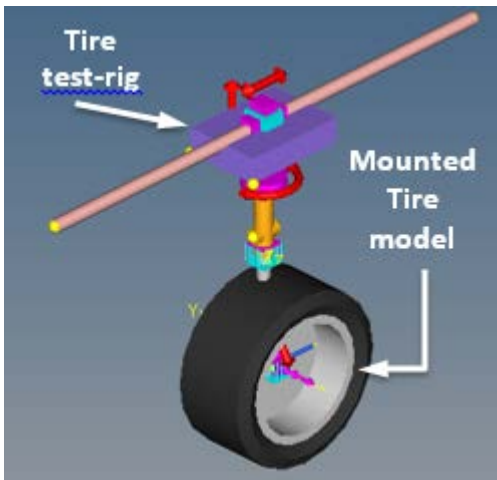


#### ▼ Support for Static Analysis using CDTire

With this release, support for version CDTire version 4.1 has been added. This version of CDTire supports static analyses in MotionSolve. This is supported on both, Windows and Linux platforms.

#### ▼ Tire test rig (experimental)

The Tire Test Rig is a system definition in MotionView that can be used to simulate an experimental test on a tire of one's choice. This Test rig along with Tire model (specified as a property file) is loaded on to the system from the Assembly Wizard and the analysis from the Task Wizard.



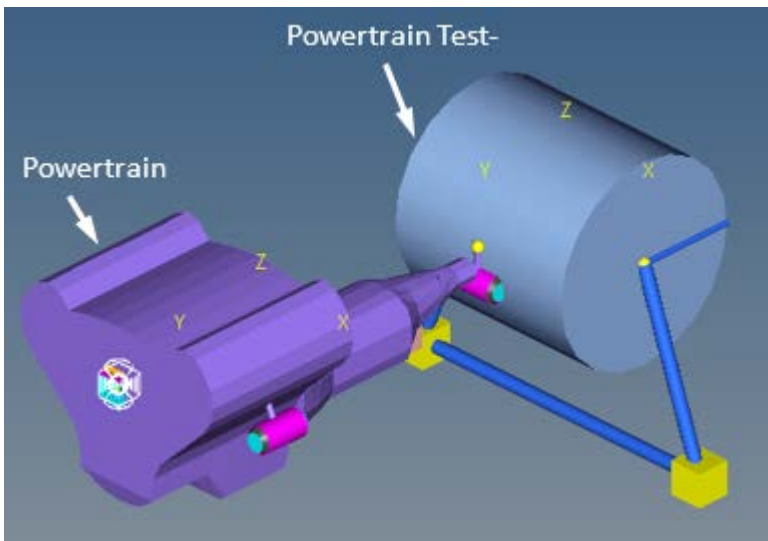
The following types of analyses are supported:

- Vertical Load Test
- Lateral Slip Test
- Longitudinal Slip Test

For each test, a standard set of results, sufficient to allow you to understand the tire behavior are generated.

#### ▼ Powertrain Test-rig (experimental)

The Powertrain Test-rig is a system definition in MotionView that can be used to simulate an experimental test on a powertrain of one's choice. This Test rig along with power-train model is loaded on to the system from the Assembly Wizard and the analysis of user's choice can be loaded from Task Wizard. The test-rig is commonly used to understand the behavior of a set of parameters describing the powertrain and tune these so they behave as desired.



Two types of tests are supported currently.

- Gear Shift Analysis
- User Defined Analysis

#### ▼ MDLLIB Updates

- Improved full vehicle kinematics and compliance test rig
  - Correlating results between the half vehicle and full vehicle suspensions is now much easier due to improvements to the full vehicle kinematics and compliance test rig.
- Improved model initial condition setup
  - All initial conditions on a vehicle are set to zero until an event from the analysis wizard is added.
- Improved vehicle weight distribution when including a powertrain
  - When a vehicle with a powertrain is assembled, the vehicle body location is updated to account for the increased weight. Thereby improving vehicle weight distribution.

## Resolved Issues

#### ▼ Auto Spring Using Force as Install Method

Choosing force as the install method for AutoSpring resulted in an incorrect spring force at design position during the simulation. This was due to an incorrect spring displacement offset being calculated. This issue has been resolved.

#### ▼ Auto Bumpstop Pair Impact Formulation Incorrect

Choosing “Impact Length” had no effect and the distance entered was treated as the clearance in the stop. Choosing “Impact Length” now works properly.

#### ▼ Auto CDTire Pair Tire Graphics not Symmetric

The tire graphics are now centered about the plane passing through the wheel center and perpendicular to the wheel spin axis. Previously, the changing the tire aspect ratio shifted the tire graphics along the wheel spin axis.

#### ▼ Ideal Steer Angle Incorrect

The ideal steer angle in the Static Design Factors report for half vehicle analyses is now output correctly. Previously, a spurious value was output.

#### ▼ CDTire v4.1: CDT20 - Integrator Failures Occur with the Default CDT20 Property File

Testing using the CDTire model 20 with road surface model 2000 (RSM2000) showed simulation failures. Because CDTire model 20 uses a point-follower contact and road surface model 2000 is three (3) dimensional, they should not be used together. To prevent inexperienced users from encountering such failures, the default property file for CDTire in MotionView has been changed to a CDT30 property file, which is appropriate for all road surface models. However, no changes to CDTire or MotionSolve were made to resolve this issue. In MotionView the following system definitions for CDTire now reference a CDT30 property file by default:

- [.../hw/mdl/autoentities/definitions\\_md1/sysDefCDTire.mdl](#)
- [.../hw/mdl/autoentities/definitions\\_md1/sysDefCDTirePair.mdl](#)

#### ▼ CDTire v4.1: CDT20, CDT30, and CDT40 Messages Truncated

Multi-line information and error messages for CDTire models 20, 30, and 40 were truncated on output when used with legacy road models and CDTire v4.0. Fraunhofer ITWM corrected the problem and improved the messages for models 20, 30, and 40.

For example, when the tire property file could not be found CDTire v4.0 output the following message:

```
USRMES:USER [1]
INFO: CDTire:: Using CDTire version 4.0.0

USRMES:USER [1]
INFO: CD Tire - Time Discrete Interface.
cdt CDTire R 4.0.0 -> TIRE 1 INITIALIZING
cdt TIRE MODEL 30 SELECTED
cdt CTRL FILE SELECTED:
cdt ERROR:
```

CDTire v4.1 now reports:

```
USRMES:USER [1]
  INFO: CDTire:: Using CDTire version 4.1

ERRMES:USER [1]
CDTire> CDTIRE_INIT_F: PRIME PAR FILE NOT FOUND AT:

ERRMES:USER [1]
CDTire> ./prop_files/tie_30.21
```

#### ▼ **CDTire v4.1: REQ838 - Adams Models Fail in MotionSolve**

The MotionSolve implementation of REQ838 incorrectly ordered the user subroutine input parameters causing Adams datasets for CDTire to fail when submitted to MotionSolve. The implementation of REQ838 in MotionSolve now orders parameters as documented by Fraunhofer ITWM for Adams. The MotionView definitions for CDTire now create output requests using REQ839 to avoid conflict with ADAMS.

#### ▼ **CDTire v4.1: REQ838 – Models Without Output Requests Calling REQ838 Fail**

Customers reported that models without output requests calling REQ838 sometimes failed. However, Altair was unable to reproduce the issue. The problem occurred in the MotionSolve code that interfaces CDTire. The code was updated to correct the problem.

#### ▼ **CDTire v4.1: RSM2000 - Does not work with CDT20, CDT30 and CDT40**

This release includes new CDTire new binaries from Fraunhofer ITWM that fix the issue.

#### ▼ **CDTire v4.1: Models 20, 30, and 40 Fail with Property File Read Error**

When the initial character of the first two lines (as well as any other comment line) of a CDT20, 30 or 40 tire property file began with a numeral, CDTire reported errors reading the file. Fraunhofer revised CDTire to ignore any comment lines in property files for models 20, 30, 40 and in control files to correct the problem.

#### ▼ **MF-SWIFT: Initial Velocity Reporting Issue in MF-SWIFT Tyre Model**

Previously MotionSolve incorrectly reported the initial velocities for tires using the MF-SWIFT model. MotionSolve now correctly reports the initial velocities for tires using the MF-SWIFT model.

#### ▼ **Using FIESUB 810 Causes Failure in Quasi-Static Model**

- With the previous version, using FIESUB 810 caused a solver failure for a Quasi-Static model shared with Altair. This was determined to be due to a programming error in the

code, where the row and column values for the FIELD Jacobian were interchanged. This has been fixed with this release.

- A side-effect of this change is that linear analysis of models containing FIESUB 810 may yield slightly different results. This is to be expected, since the bug-fix involves a correction to the Jacobian matrix of the system. The following changes may be seen for these models:
  - The set of independent coordinates may be slightly different
  - The eigenvalues and eigenvectors may be slightly altered
  - The ABCD matrices may be slightly different

#### ▼ Fixes to ACTCLC Routine

The contact patch velocities, slip angle and inclination angle were incorrectly computed in the previous version of the ACTCLC routine for the user tire. These have been corrected to be consistent with the documentation that has been shared previously with Honda.

#### ▼ MDLLIB: Tire Envelope Analysis on Half Vehicle Models now Support Property File Based Shock Absorber

This was non-functional in previous releases. This issue has been fixed.

#### ▼ MDLLIB: Support for Models using Legacy Steering Boost System

This was non-functional in previous releases. This issue has been fixed.

#### ▼ MDLLIB: Full Vehicle Analyses - Incorrect Initial Rotation Velocities Using SETIC/SETWIC

Previously when using the SETIC/SETWIC option for initial velocities, the initial rotational velocities of half-shafts, hubs and other bodies in the driveline were inconsistent with the initial rotational velocities of the driven wheels. The initial rotational velocities are now consistent.

#### ▼ MDLLIB: Full Vehicle KnC Analysis

KnC analysis results for the same suspension match between Full Vehicle and Half Vehicle Analyses. Previously, there were small differences.

#### ▼ MDLLIB: Steering Column System

The column support joint is moved to the lower column shaft center of mass. This makes the support joint easier to identify when, for example, attaching the steering column to a flexible body.

## Known Issues

### Road Tools:

- The road tools dialog should offer a means to abort road graphics generation because of the large time require for some road file.

### Driver:

- Feedforward Controller doesn't follow acceleration profiles well
- When using the driver, the tire longitudinal slip shows oscillations at time=0
- Zero vehicle speed causes steering controller to fail.
- Driver event UI is experimental: To enable the event editor button on the driver panel in MotionView set the environment variable HW\_MV\_EXPERIMENTAL to ADVANCED\_DRIVER.

### MDLLIB

- Full vehicle events with a powertrain often fail to solve because the suspension spring preloads are too small causing the excessive suspension jounce. We recommend increasing the spring preloads to compensate for the mass of the powertrain, and as always you should update models.
- Steering systems contain dummy bodies to support analyses, but these have no meaning to users.
- Some combinations of wizard selections do not build because referenced system definitions use the same varname.
- To reduce the length of the Constant Radius event the length of the initial straight section should be based on the initial vehicle velocity

## OptiStruct Enhancements Pertinent to Multibody Simulations

---

This section documents other important enhancements that have been made to OptiStruct that you should be aware of.

### ▼ MBD Analysis Results can now be Recovered for Flexible Bodies in OptiStruct

After a MotionSolve run, it is possible to recover displacements, velocities, accelerations, stresses, and strains for the flex-body in OptiStruct in order to create .op2 and .h3d results files for fatigue analysis. The procedure is explained as follows:

- First, you perform a MotionSolve simulation with a model that contains flexible bodies. MotionSolve creates a results file (\*.mrf) that contains the modal participation factors at each time step for the Flex Body.
- Next, in OptiStruct, perform a residual run to recover displacement, velocity, acceleration, stress, and strain results for interior girds and elements in the Flex Body based on the modal participation results from MotionSolve. In the residual run, the Flex Body filename\_recov.h3d file and the .mrf results file are specified using the ASSIGN data:
  - ASSIGN, H3DMBD, 30101, 'pfbbody\_1\_recov.h3d'
  - ASSIGN, H3DMBD, 30102, 'pfbbody\_2\_recov.h3d'
  - ASSIGN, MBDINP, 10, 'pfbbody.mrf'

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 Flex Body ID's in the .mrf file.

### ▼ New Parameter and Solution Control Data

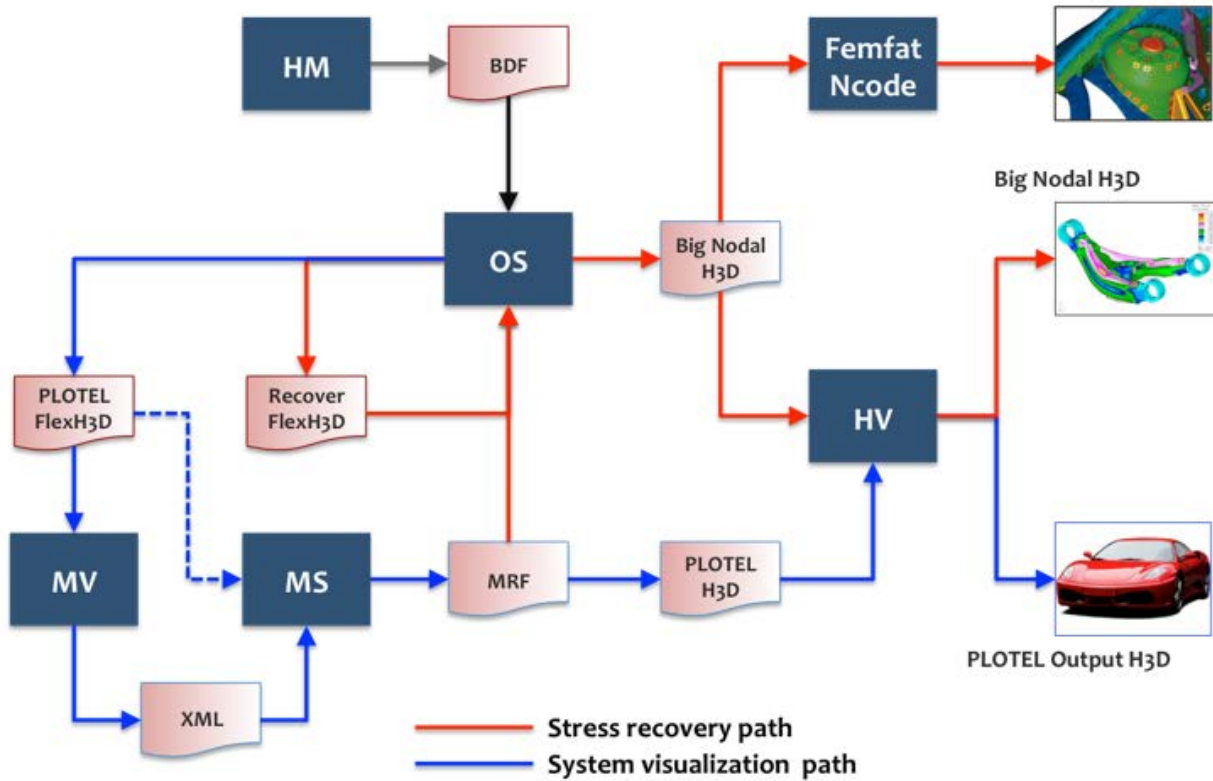
- PARAM, MBDREC: The parameter MBDREC can be used to create solver and recovery .h3d files during Component Mode Synthesis (CMS)
- PARAM, CMSDIRM: The parameter CMSDIRM can be used to allow or disallow flexible body generation when directional masses are defined in the input file
- ASSIGN, H3DMBD: Identifies an external nodal flexh3d file that contains the analysis recovery information for displacements, velocities, accelerations, stresses and strains.
- ASSIGN, MBDINP: Identifies an external file from which to read the modal participation factors calculated by MotionSolve for transient analysis
- CMSMETH, 1, CBG, , 15: CBG is the keyword that instructs OptiStruct to create the CMS flexible body with additional geometric stiffness data using the Craig-Bampton CMS method

### ▼ PARAM, MBDREC

Two H3D files are generated when the PARAM, MBDREC option is used.

- A PLOTTEL based FlexH3D file that contains a minimum set of graphics and all the necessary information required by MotionView and MotionSolve to perform a multibody simulation with CMS flexible bodies.
- A “Recover FlexH3D” file that contains all the information need by OptiStruct to recover the full set of stress and strains from a MotionSolve simulation.

The overall process for using this is illustrated below.



- A small file \*.h3d contains the minimum amount of information needed for MBD simulation in MotionSolve. FlexH3D files are significantly smaller (sometimes by 100x), and file management becomes much simpler.

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.



## Inspire 2016 Release Notes

The following features and capabilities have been added or modified for solidThinking Inspire® version 2016:

### PolyNURBS Tools

---

#### ▼ Create PolyNURBS

The new Create tool allows you to quickly create free-form solid geometry that is smooth and continuous.

#### ▼ Wrap

The new Wrap tool allows you to quickly generate polynurbs geometry by clicking along the contour of your existing shape.

#### ▼ Add/Remove

The new +/- tool allows you to add a new cage to the face of an existing cage, so you can quickly build the basic shape of a PolyNURBS object.

#### ▼ Loop

The new Loop tool allows you to draw an edge loop around a cage.

#### ▼ Bridge

The new Bridge tool allows you to connect two cages together. You can also use it to create a hole through an existing cage.

#### ▼ Sharpen

The new Sharpen tool allows you to sharpen the edges of the underlying geometry at the edge of the cage.

## Geometry Tools

---

### ▼ Midsurface

The new Midsurface tool allows you to find and extract surfaces from single thickness thin solid geometry.

### ▼ Mirror

The new Mirror tool allows you to reflect selected parts across a symmetry plane.

### ▼ Scale

The new Scale tool allows you to resize selected parts.

### ▼ Revolve

The new Revolve tool allows you to revolve a planar face around an edge, a line, or a cylindrical hole. You can also create a new part by holding down the Ctrl or Shift key as you pull.

### ▼ Push/Pull

The updated Push/Pull tool allows you to create a new part by holding down the Ctrl or Shift key as you pull.

## Setup Tools

---

### ▼ Enforced Displacements

The new Enforced Displacements tool can be used when the magnitude of a force applied to a part is unknown, but you do know how much the part displaces as a result of that force.

### ▼ Angular Velocity and Acceleration

The new Angular Velocity and Accelerations tool allows you to define the speed of rotation of the entire model and the axis about which it rotates.

### ▼ g-Loads

The new g-Loads tool allows you to simulate a model undergoing acceleration, which imparts a force on all of the parts in the model.

#### ▼ Temperature Loads

The new Temperatures tool allows you to simulate the effects of temperature changes on your model.

## Optimization and Analysis

---

#### ▼ Buckling Analysis

You can now analyze your model for buckling modes, which are used to predict when a part will bend or collapse under load.

#### ▼ Compare Results

The new Comparison tool allows you to compare the results of analysis or optimization runs.

#### ▼ Callouts

You can now add callouts to view analysis results at specific locations on your model.

## User Interface

---

#### ▼ Section Cuts

The new Section Cut tool allows you to view 2D cross-sections of your model.

#### ▼ Move Tool

The updated Move tool features updated graphic manipulators with larger pick targets. You can also double-click to reposition the tool.

#### ▼ Model Configuration

Press F5 to open the Model Configuration toolbar on the Model Browser.

▼ Standard Views

Standard views can now be accessed from buttons below the View Rotator in the lower left corner of the modeling window.

## File Input/Output

---

▼ **Product Data Management**

Inspire now supports product data management (PDM) via PTC Windchill and Siemens Teamcenter.

## Evolve 2016 Release Notes

The following features and capabilities have been added or modified for solidThinking Evolve® version 2016:

### User Interface and Workflow

---

#### ▼ Translate, Rotate, and Scale

The parameters in the Control Panel for the Translate, Rotate, and Scale tools have been simplified.

#### ▼ Snaps

Two new point snaps have been added: Snap to Intersections and Snap to Midaxis.

#### ▼ Camera Controls

The camera controls are now displayed by default. Click **Show/hide camera** icons to hide them.

The camera position can be locked using the new Lock Camera icon located above the camera controls, along the right side of each modeling view.

#### ▼ Console

Spacebar and Enter usage has been unified so that either key can be used to accept Console input for all tools.

#### ▼ Snapshot

A snapshot of the current model can be taken using the Snapshot icon located along the right side of each modeling view.

#### ▼ Image Browser

The Renderings Browser has been renamed the Image Browser because it now displays snapshots in addition to final renderings. It now supports image panning and zooming, and contains file and rendering information for rendered images.

#### ▼ Control Panel

The Control Panel has been redesigned so that parameters are grouped together more clearly.

#### ▼ Curve Plot

The color of the Curve Plot has been changed from white to gray to improve visibility.

#### ▼ Default Theme

The default theme has been updated to a lighter color palette to improve visibility.

#### ▼ Background Image

Interactive handles have been added to the Background Image tool, allowing you to translate and rotate reference images directly in the modeling views.

#### ▼ Open Folder in Finder (Mac)

On Mac, you can now determine the location of an open file by holding down the Command key and clicking on the file name in the title bar of the application.

## Modeling

---

#### ▼ Edit PolyNURBS Tool

The Edit PolyNURBS tool has several new features:

- Edges and faces can be bridged together using the new Create Bridge button. (There is no longer a separate Faces Bridge tool.)
- An Auto-align axis feature has been added to facilitate face editing. When you translate or rotate a face, the transformation operation automatically aligns to the face's local axes.
- New edge creasing options have been added, allowing you to make selected edges sharper or smoother.
- The new Advanced selection mode allows you to quickly select or deselect loops and adjacent rows of faces or edges.
- The new Info panel at the bottom of the Control Panel flags potential topology issues in red. Potential issues are also flagged in red in the modeling views.

#### ▼ Draft Analysis Tool

The new Draft Analysis tool allows you to visualize and identify problematic areas related to draft angles in the molding process.

#### ▼ **Divide Curve Tool**

The Divide Curve tool has been updated to allow you to divide a curve into multiple independent curves and the output is now contained in construction history.

#### ▼ **Intersect Curve Tool**

The new Intersect Curve tool replaces the former Split Curve tool and allows you to create trimmed curves by splitting selected curves into separate pieces and keeping only the desired pieces. The input curves do not need to physically intersect, as Evolve can create a visual intersection. The output is now contained in construction history.

#### ▼ **Rotate Tool**

The improved Rotate tool not only has more user-friendly interactive handles, it also allows you to rotate about a custom rotation axis, and rotate with reference to a start point and an optional end point on another object.

#### ▼ **Loft and Sweep Tools**

The updated Loft and Sweep tools offer a new feature called Auto-alignment, which lets you use profiles for reference only (i.e., the profiles and rails don't have to intersect). It comes with extra parameters that let you fully control the position and orientation of each profile.

#### ▼ **Modeling Tool Feedback**

Modeling tool feedback is now available for the Intersect Curve, Surface Offset, Shelling, Trim, Boolean Subtraction, Boolean Union, and Boolean Intersection tools.

When one of these modeling tool fails, an error icon is displayed next to the object's name in the World Browser. Hovering over the object's name in the World Browser displays information regarding the error.

#### ▼ **Rectangle, Cube, Plane, PolyCube, PolyPlane Tools**

The names of the Width, Thickness, and Height parameters for the Rectangle: corner, corner; Rect: 3 Points; Plane; Cube; PolyCube; and PolyPlane tools have been changed to Length, Width, and Height. In addition, you can now create a centered object by holding down the Ctrl key as you click the tool icon.

A new rectangle tool has been added, Rectangle: center, corner.

## Rendering

---

### ▼ Interactive Rendering

The Interactive Rendering display mode by default uses the Performance (GPU+CPU) rendering method and automatically stops when the specified samples per pixel has been reached.

Selecting Enable Caustics will switch to the Full rendering method.

Pause/Restart options have also been added and can be activated by clicking on the IR icon.

### ▼ Metallic Flakes

The new Metallic Flakes option allows you to add metallic flakes to the base color of Plastic materials. It includes controls for flake color, size, density, and glow.

### ▼ Final Renderings

Final renderings can be paused and resumed, and the estimated rendering time is displayed. Rendering Info has also been added to the rendered image in the Image Browser.

### ▼ Texture Positioning

The Texture Positioning tool was updated to support adjustments to brightness and contrast in texture maps.

### ▼ Environment Properties

Environment Properties parameters have been renamed and rearranged for ease of use.

The new Ground option allows you to insert a ground plane to catch shadows from objects in your scene.

.IBL format support has been added.

### ▼ Restore Default Settings

A Restore Default Settings button has been added to the Material Properties, Environment Properties, and Rendering Properties tools.

### ▼ Rendering Properties

A Swap button has been added to the Rendering Properties tool, allowing you to invert the width and height values of the final rendered image.

360° Panoramic has been added, allowing you to use your scene to create a HDR environment map.



## Input/Output

---

### ▼ sT-Adv

All formats are now listed individually.

## Operating Systems

---

### ▼ Improved Graphics

Enabled and forced High Performance Graphics with Optimus Intel/NVIDIA architecture.

Forced discrete graphics in AMD Switchable Graphics, AMD/Intel combo GPU architecture.

# FEKO 14.0 Release Notes

## Highlights

FEKO joins the Altair family and is included as part of the HyperWorks simulation software platform. In the 14.0 release FEKO boasts numerous extensions to both its user interface and the electromagnetic solver. Highlights include curvilinear segments, complete application automation for CADFEKO, script recording and extensions to higher order basis functions (HOBFB).

### ▼ Curvilinear Segments

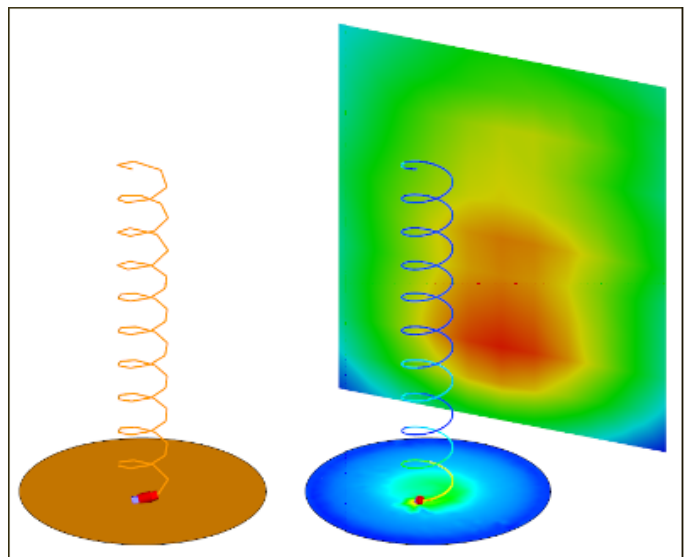
EKO now supports curvilinear segments for the method of moments (MoM) and multilevel fast multipole method (MLFMM) solvers.

This extension allows for models with fewer segments in comparison to linear segments to obtain the same geometrical accuracy.

Curvilinear segments are also used in cases where it is impossible to reduce the segment length without violating mesh segmentation rules.

FEKO supports the import and export of curvilinear segments in and to the following formats:

- Import from NASTRAN, ANSYS (CDB) and Abaqus file formats
- Export to NASTRAN, Gerber and AutoCAD (DXF) file formats



Comparison of linear segments (left) and curvilinear segments (right) with a near field, surface currents and wire currents.

CADFEKO and POSTFEKO fully support the display of curvilinear segments. This includes the display of currents and charges in a 3D view.

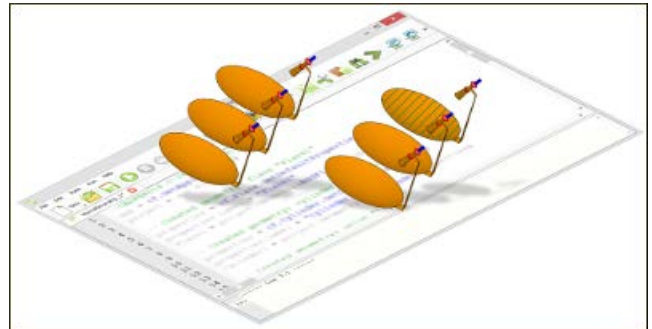
### ▼ HOBFB for Hybrid MoM/UTD and Hybrid MoM/RL-GO

Higher order basis functions (HOBFB) are now supported for the MoM hybridised with the following methods:

- Uniform theory of diffraction (UTD)
- Ray launching - geometrical optics (RL-GO)

### ▼ Complete Application Automation in CADFEKO

In the 14.0 release the application automation in CADFEKO is now complete. Any action performed manually in CADFEKO can now be done via scripting. The completed application automation is documented extensively and contains numerous examples.



### ▼ Script Recording in CADFEKO

CADFEKO now supports script recording. Start the recording process and manually perform actions in CADFEKO. Perform the actions that need to be recorded, stop the recording process and view the recorded script. Once a script is created, it can be modified to perform more advanced and repetitive actions. Analysing the generated script aids in learning the application automation syntax.

## Other Notable Changes

---

The GUI components have been extended to support all new features and solution techniques in the FEKO solver. In addition to supporting the new solver features, numerous other features and improvements have also been implemented. These features are listed in the sections that follow.

### ▼ CADFEKO - 3D CAD Modeler

- Improved automatic mesh settings reduce the cases where a user needs to specify the mesh size settings:
  - HOBf faces are automatically meshed with a refinement applied to multiple media junction edges. These edges are meshed according to the lowest order (0.5).
  - Metal faces which separate two surface equivalence principle (SEP) regions are now meshed according to the smallest wavelength on either side of the face. Non-metallic faces are meshed according to the largest wavelength on either side of the face.
- Extensions to the finite difference time domain (FDTD) solver includes:
  - Added support for multiple configurations in a single model.
  - Added support for perfect magnetic conductor (PMC) boundary conditions.
  - Added the ability to use an implicit wire radius definition (user does not need to specify a wire radius).
- Added the ability to rotate array elements when creating a cylindrical/circular array.
- Added support for the Vance braided shield definition for cable modelling.
- Added the ability to change the model extents even if it results in faulty parts. It allows the model to be modified and the (possibly) faulty elements to be fixed later using the CAD fixing tools.
- Extended the internal frameworks for the equivalent sources and receiving antennas to accommodate future extensions. Due to these modifications, layout changes are evident in the equivalent source dialogs and receiving antenna dialogs.
- Added the ability to allow users to select or deselect all layers when importing Gerber, ODB++ or AutoCAD files.
- Added support for the import of Gerber, ODB++ and 3Di formats on Linux. Previously the file formats were only supported for Windows.
- Added support for an advanced healing option on the Import geometry dialog. This option adds extensive healing operations to the import process. Due to the time-consuming nature of the healing process, it is not the default setting.
- Added the ability to set the 3D view font settings on the Default settings dialog.
- Added the ability to select either the model mesh or simulation mesh for export when exporting a mesh.
- Dialogs now allow referenced entities to be created on the fly with the creation button located next to the required entity.
- Updated the CAD exchange library to allow the latest CAD file formats to be imported and exported.

## ▼ POSTFEKO - Post-processor

The following extensions to the application programming interface (API) are shared between CADFEKO and POSTFEKO:

- Added support for callback functions on forms. Callback functions allow forms to respond dynamically to actions performed on the dialog.
- Extensions to the `Matrix` and `ComplexMatrix` objects:
  - Added support for additional construction methods to allow alternative initializations for matrices.
  - Added support for element-wise operations for standard math functions. The element-wise operations and element-wise comparisons allow calculations to be performed faster in comparison to performing the same operations within a loop. The `ComplexMatrix` object supports the transpose and element-wise transpose.
  - Added support for a `Find` function as an efficient way to search in matrices.
- Added support for currents and charges on surfaces and wires in the scripting environment to allow further post-processing. A currents and charges math script is included allowing customised calculations which can be displayed in the 3D view.
- Added the `UpdateStoredData` method and `Delete` method to simplify the modification and maintenance of stored results.
- Added the `Clone` method to the `DataSet` object. This new method simplifies `DataSet` creation.
- The `DataSet` object now allows faster processing and data storage.
- Power setting information is available in the scripting environment for further post-processing.
- Added the ability to execute a script in a non-interactive way by including the `--non-interactive` option in the command line.

Other enhancements:

- Added the *Received signal phase* quantity to the receiving antenna results.
- Added the ability to add math scripts to an empty project. When data in the form of math scripts or imported data is available, it can be displayed on a 2D or 3D graph.
- Added support to display the impedance data for loads and general networks on a Smith chart.
- Added the ability to hide a trace in the 2D legend by leaving its legend text empty. This feature allows control over the entries displayed in the 2D legend.

## ▼ EDITFEKO - Scripting Model Editor

- Added support at the BC card to specify perfect magnetic conductor (PMC) boundary conditions when using the FDTD solver.
- Added support at the SH card for the Vance braided shield type.

### ▼ Electromagnetic Solution Kernel

- Improved performance and stability of the kernel by updating to the latest libraries, math libraries and compilers.
- Added support for the GPU CUDA version 7.0 framework for both Windows and Linux.
- Upgraded the Intel MPI library for Windows to version 5.0.
- Extensions to the finite difference time domain (FDTD) solver include:
  - Added support for multiple configurations in a single model.
  - Added support for zero impedance loads.
  - Added the ability to treat a port without a source or load as a short circuit (similar to the MoM).
  - Added support for OpenMP parallelisation.
- Added a warning to the kernel that notifies a user if the modes of a waveguide port are evanescent when solved with the MoM.
- Added support to radar cross section (RCS) computations for active voltage sources in addition to the incident plane wave in a model.
- Processing time is reduced significantly when using impressed aperture sources in conjunction with RL-GO.
- Each solution request is now written to a separate \*.ffe/\*.hfe/\*.efe file in the format <FEKO\_base\_filename>\_<requestname>(k).extension.

### ▼ Support Components, Installation and Licensing

- Added the ability to launch the FEKO installer from the HyperWorks master installer.
- Added support for multiple concurrent installations of the same version of FEKO in Windows (this has always been supported in Linux).
- Added the ability to perform an installation on Windows without administrative rights. Although some features may still require administrative rights to be correctly set up, these can be configured after the installation if required.
- Integrated FEKO as a recognised solver in HyperStudy.
- Refined the Optenni Lab plugin and improved its integration with FEKO.
- Added a parameter sweep plugin that allows the configuring and launching of a parameter sweep in CADFEKO as well as processing the calculated results in POSTFEKO.
- Support for 32-bit architecture is being phased out and is not available for download. If you require 32-bit support, please contact the Altair Support team.
- The Altair Student Edition (with significantly increased limits in terms of number of mesh elements) replaces the FEKO LITE version.

# AcuFieldView 14.0 Release Notes

## Highlights

---

AcuSolve 14.0 includes a new version of AcuFieldView that is based on Intelligent Light's latest release of FieldView. This release includes several new post-processing capabilities allowing users to read data faster with better control, load layout restarts, compute streamlines from user defined seeds and gives easier access to user defined colormaps.

### ▼ Improved Performance and Additional Controls for AcuSolve Reader

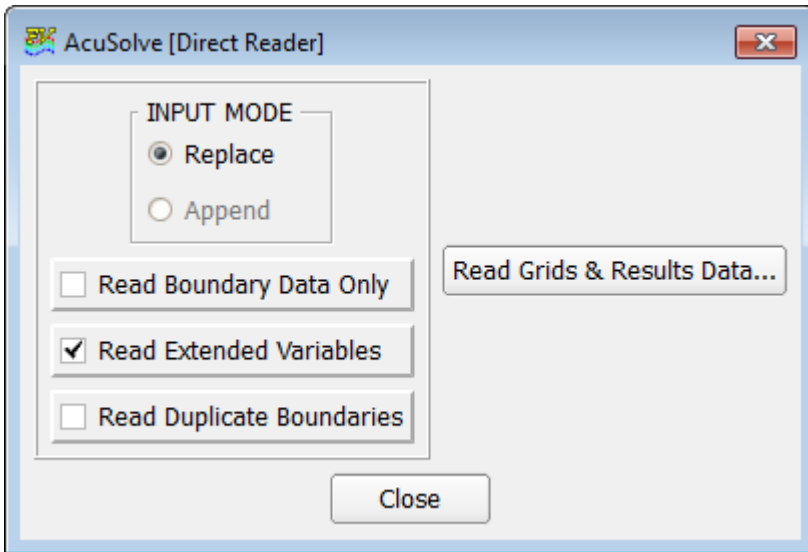
Users will first notice new direct reader options when loading AcuSolve results files. The AcuSolve direct reader includes new options to control the information read into AcuFieldView. Users have the option to read the extended output by toggling the "Read Extended Variables" checkbox and/or read duplicate boundaries by toggling the "Read Duplicate Boundaries" checkbox. By default, both of these options are off. This does represent a change in behavior with respect to previous releases. When loading results with more than one time step (transient dataset), the Time Step Selection panel now has the option to "Read as Steady State". These settings, along with coding optimizations within with AcuSolve direct reader reduce the time needed to read data into AcuFieldView. These optimizations have made the direct read times nearly equivalent to the time necessary to convert the data using AcuTrans and read via the FV-UNS format. The following table summarizes the read performance.

Model	Element Count	Read Time (Minutes)		Read Time (Minutes)	
		V13.0 Basic Variables	V14.0 Basic Variables	V13.0 Extended Variables	V14.0 Extended Variables
VWT Roadster - AcuSolve direct reader	5.54E+07	4.8	<b>4.6</b>	11.3	<b>5.2</b>
VWT Roadster - FV-UNS reader	5.54E+07	4.9	<b>4.4</b>	-	-

Performance of AcuSolve data read operations using AcuFieldView V13.0 and V14.0

### ▼ Reading Extended Variables

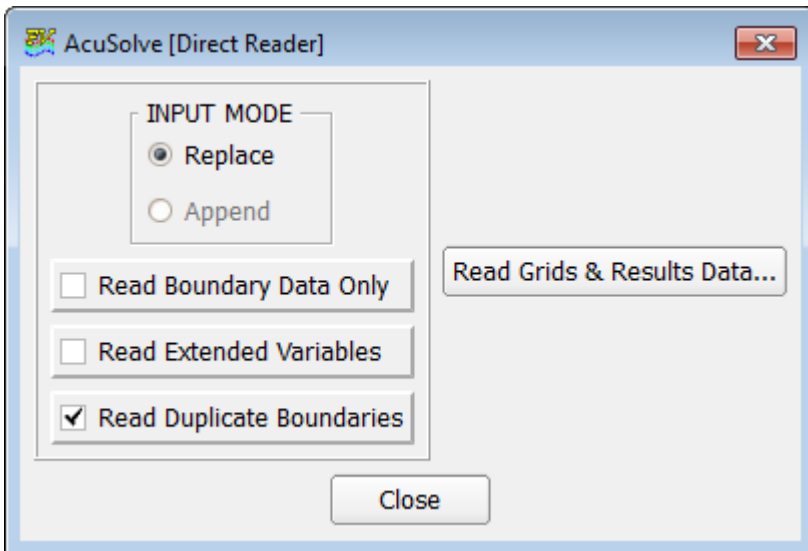
A "Read Extended Variables" button has been added to the AcuSolve direct reader data input panel. When turned on, "Read Extended Variables" gives users access to additional variables from the AcuSolve solution database. This allows post-processing and visualization of additional quantities, but comes at the expense of slower read times.



If active, the extended and derived quantities are added to the nodal output variables to form the variable list that is presented to the user. The extended variable list can be expanded further by setting the environment variable FV\_ACUSOLVE\_GRAD. This will cause AcuFieldView to read the gradients of the transport variables. By allowing the user to specify whether the extended variables should be read gives additional control, allowing for faster read times and more succinct data inquiry. Note that in releases of AcuFieldView prior to V13.0, the option to read extended variables was on by default and was controlled by an environment variable.

#### ▼ Read Duplicate Boundaries

The Acusolve direct reader data input panel now contains an option to read duplicate boundary surfaces.



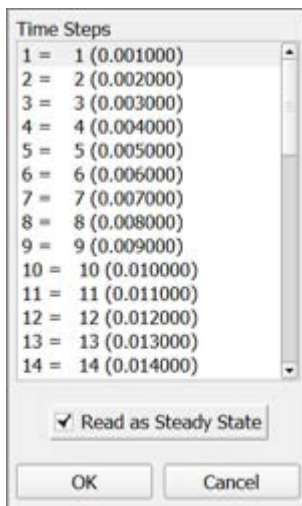
When turned on, AcuFieldView will create duplicate boundary surface entries for surfaces that are reproductions of each other. For example, if the collection of faces present within a simple boundary condition command and a surface output command are identical, these surfaces are assumed to be duplicates. By default, only one boundary surface entry will be created for the duplicated pair. If



the "Read Duplicate Boundaries" option is turned on, however, AcuFieldView will create a unique boundary surface entry for each instance of the duplicated surface. Note that in releases of AcuFieldView prior to V13.0, duplicate surfaces were always read. This option eliminates these duplicates, yielding simplified lists of surfaces in the boundary surface creation panel.

### ▼ Read as Steady State

A "Read as Steady State" button has been added to the Time Step Selection panel when using the AcuSolve direct reader. When active, AcuFieldView will load a specific time step of a multiple time step dataset, as selected by the user, then treat it as static data. This feature is added to provide consistent behavior between the FV-UNS data import route and the AcuSolve direct reader import route. When reading FV-UNS files, AcuFieldView checks for sequences in the file name, then prompts you to indicate whether the data is steady or transient. The "Read as Steady State" button provides this same functionality for the AcuSolve direct reader. Note that this option has implications for particle path display behavior as well as the availability of special annotation strings such as Time=%%T or Time Step=%%N. The dataset needs to be read as transient, with the "Read as Steady State" option turned off, to restore the behavior of the direct reader to that of releases prior to V13.0.



### ▼ Load Layout Restart Files

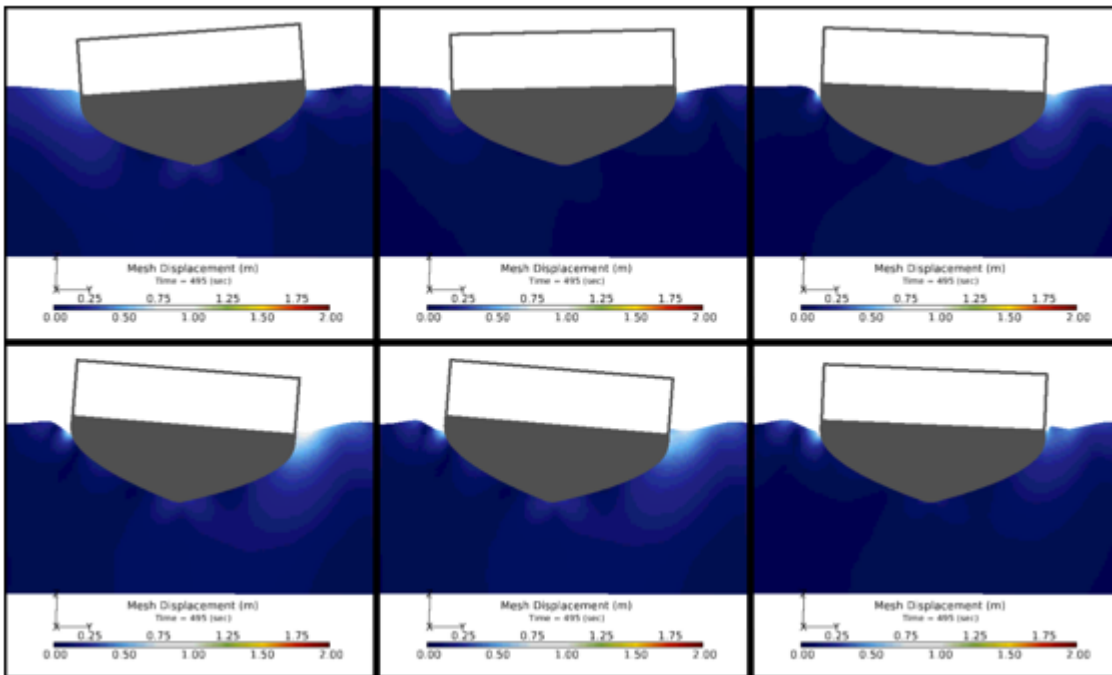
Starting with AcuFieldView 13.0, AcuFieldView has the ability to display multiple windows, called Multi-Window Layouts. The number, size and layout of the windows can be stored to a Multi-Window Layout Restart file, which can be opened during an AcuFieldView session.

Since defining complex layouts can be tiresome, especially when trying multiple windows of the exact same size, the AcuFieldView installation directory now provides a number of default Layout Restart files, under

`<Install Directory>/fv/fvx_and_restarts/layout_restarts`

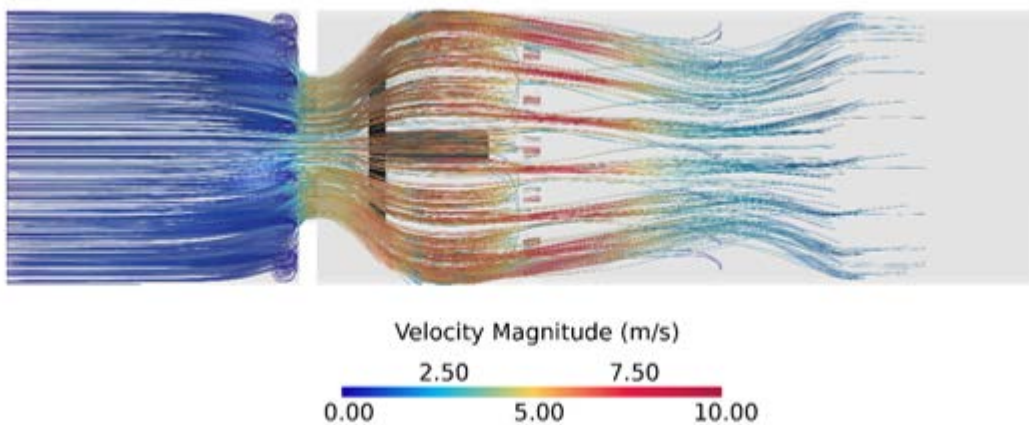
These Restart files can be applied to an AcuFieldView session through the following menu:

**File > Open Restart > Multi-Window Layout**



▼ "Growing" Display Type for Transient Streakline Particle Paths

Particles can now be rendered as growing lines, even for transient datasets. This will let users visualize not only the current particle's location but also its trajectory over time. This capability is limited to Particle Paths written to the STREAKLINE format, i.e., for which all time steps have been written to a single file. It is not supported for the high performance PARTICLE SET format, which stores no information about trajectories. For particles computed in AcuFieldView with the Streamlines panel, the environment variable `FV_SINGLE_FILE_STEAKLINES` can be set to write particles information to the single file STREAKLINE format. The growing lines are synced to the solution time. A sphere indicates the start of each growing line.



### ▼ User-Defined Seeding Locations for Surface Flows

A radio button 'From File' has been added to the Seed Density options for SURFACE RESTRICTED FLOW on the Vortex Cores/Surface Flows panel. When 'From File' is first selected, a file browser will come up automatically. To change from a previously read file, click the Browse... button to bring up the browser explicitly.

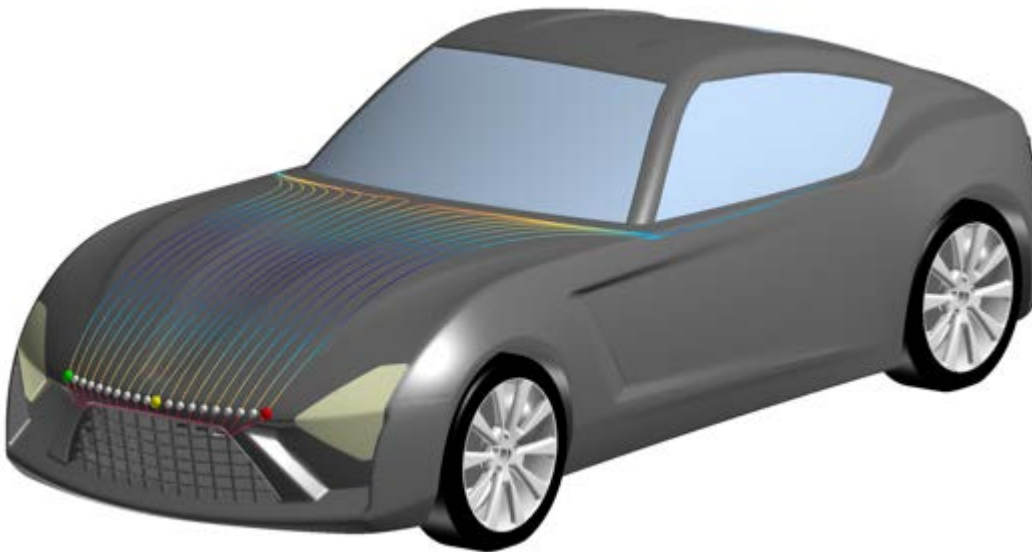
The seeds location file can be generated in a number of different ways:

- Outside of AcuFieldView, as a simple text file where each line represents a point location, with 3 columns for XYZ coordinates.
- From AcuFieldView, by:
  - Exporting a point probe location
  - Exporting the points created from an open path from the 2D Plot Panel

After the seeds location file has been selected with the browser, click Calculate on the Vortex Cores/Surface Flows panel to read it in and generate and display the flow lines.

Each surface flow object may have a unique seeds location file. Points that do not lie on a visible Boundary Surface will be ignored. File paths will be saved with restarts.

The maximum number of points for user-defined seeding locations for surface flows is the same as the limit for streamline seeds, currently 2000 streamlines per rake.



## Reduced Memory Usage for Rendering

---

Starting with this version, AcuFieldView will use Vertex Buffer Objects (VBOs) by default for rendering. This method will allow a significant decrease in memory required for rendering a given scene, while maintaining a similar level of performance, compared to the default method used in AcuFieldView 14 (display list).

The environment variable `FV_USE_FV14_GL_MODE` can be set to any value to revert to the GL mode used in AcuFieldView 14. AcuFieldView will then require more memory to render the same scene, but it may also perform faster on some older graphic cards or using older graphic drivers.

Both GL modes can be used in conjunction with AcuFieldView's various graphic modes:

- Local rendering
- Remote graphics with `-hrg` or `-srg`
- Batch with hardware or software rendering
- Reduced functionality graphics with `FV_RF_GRAPHICS`

### ▼ Colormap Selection Improvements

AcuFieldView has expanded and simplified the process for selecting predefined and user defined colormaps. Predefined colormaps are now shown directly in the colormap selector tab of a given coordinate surface, boundary surface, streamline rake, etc. The predefined colormaps are also accessible from `.fvx` scripts by specifying the colormap name.

### ▼ Easier Access to Predefined Colormaps

41 predefined colormaps have been added to the Colormap selector. They appear under the User... section following the original 8 colormaps. The scroll bar on the right accesses all options. AcuFieldView now displays an icon in front of each colormap, making it easier to identify them. This is especially beneficial now that the list includes 50 colormaps.

### ▼ Easier Access to User Colormaps

User defined colormaps can now be referenced by AcuFieldView by including the `.col` files in specific directories. At startup, AcuFieldView will look for two directories:

```
<Install Directory>\fv\data\colormaps
```

```
$HOME/FieldView/colormaps
```

- Under Windows, this is `C:\Users\<USER>\FieldView\colormaps`
- Under Linux, this is `/home/<USER>/FieldView/Colormaps`

AcuFieldView scans these directories for `.col` files and will include them in the colormap selector panel. One of the two directories mentioned above will always be found if AcuFieldView is installed. The basename of each `.col` file will be added to the **User** section of the Colormap selector.

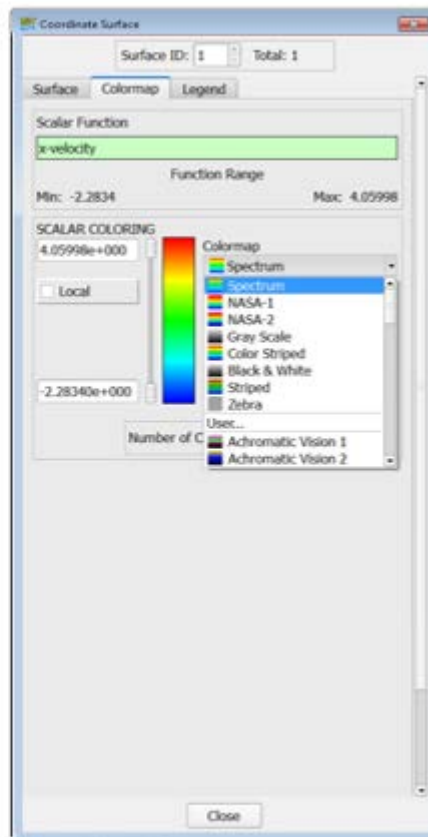
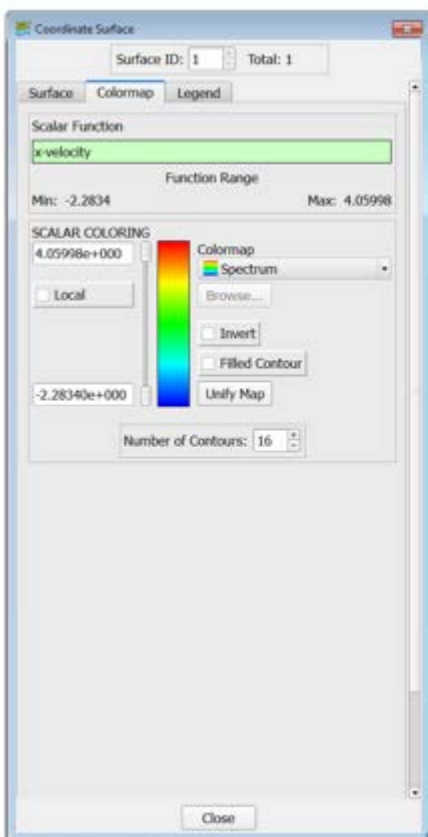
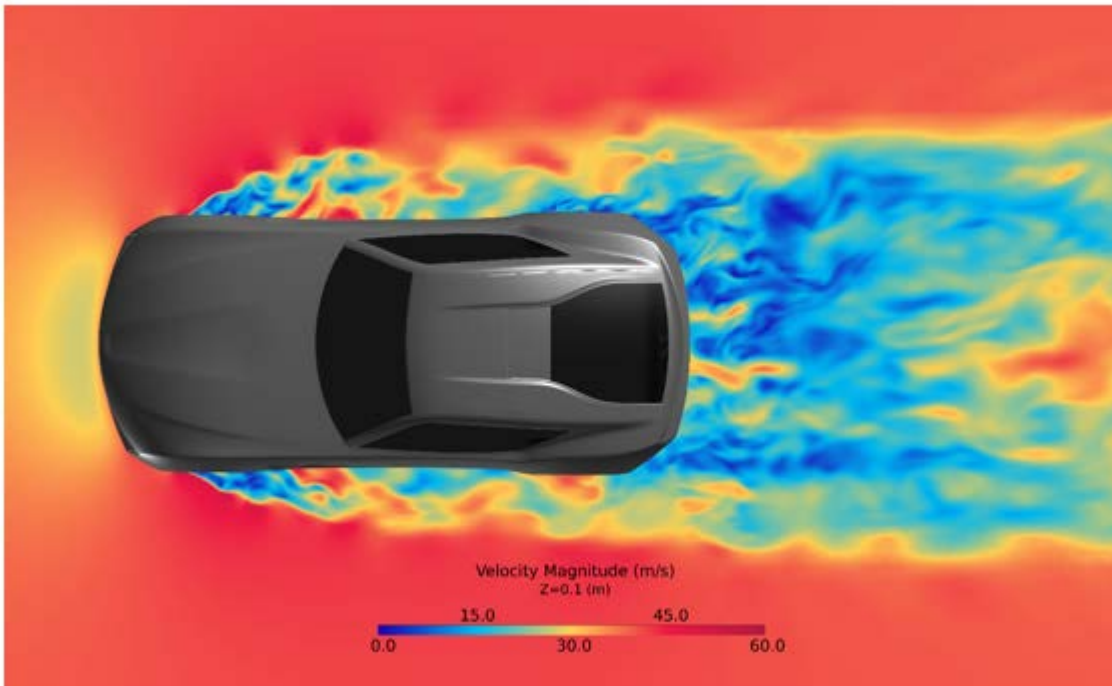
▼ **FVX Scalar Colormaps**

FVX support for defined scalar colormap names has been extended from 8 to 49 options.

If `scalar_colormap.name` does not match a defined colormap name, AcuFieldView assumes the string represents a filename, including path, of a colormap file.

The complete list of predefined colormap names is given in the following table.

'spectrum'	'chrome'	'big difference'	'radiation colors'
'nasa-1'	'cold'	'bio spectrum 1'	'red to blue diff'
'nasa-2'	'dark radiation'	'bio spectrum 2'	'red to purple diff'
'gray scale'	'dark spectrum'	'bloodflow doppler'	'relief map'
'color striped'	'dark green gradient'	'blue chrome'	'relief map land'
'black & white'	'flame transition'	'camouflage'	'relief map ocean'
'striped'	'gold'	'ccm blue red'	'simple flux'
'zebra'	'high contrast'	'ccm cool warm'	'small difference'
'achromatic vision 1'	'hot'	'ccm high contrast'	'spectrum diff gray'
'achromatic vision 2'	'hot to cold diff'	'ccm spectrum'	'spectrum diff white'
'banded blue to red dark'	'indigo flame'	'cd spectrum'	'steel blue'
'banded blue to red light'	'banded grayscale'	'leaf color'	'teal black gradient'
'cd striped'			



## SimLab 13.3 Release Notes

### ▼ System

- Support added for rotation center “Ctrl + Left” click for HM mouse settings.
- Support to highlight faces present in group when the group is selected.
- Delete entities performance improved for more number of groups.
- During Abaqus deck export, when the failure elements are found an element group is created

### ▼ Import Enhancements

#### Parasolid

- Parasolid version updated to 27

#### UG

- Support added to import Assembly Structure.

#### JT

- Support added to import cylindrical and circular attributes.

#### Abaqus

- Sub Assembly creation in Abaqus import if include file with model data is present.
- Support for reading Gasket element property for bodies that are grouped using \*Elset card in Abaqus.

#### Nastran

- The Transition elements present will be created as a separate element group.
- Support for reading of Analysis LCS. The Analysis LCS nodes that are not referred in any other LBC's will be created as Assign LCS.

#### OptiStruct

- The Transition elements present will be created as a separate element group.
- Support for reading of Analysis LCS. The Analysis LCS nodes that are not referred in any other LBC's will be created as Assign LCS.
- Support for reading gasket elements Gasket property and Gasket materials in OptiStruct.

### ▼ Export Enhancements

#### Parasolid

- Support added to export Body Name, Face color and Groups.

#### Facet Geometry

- Performance for large models is improved



### ▼ Graphics/User Interface

- Bodies option added in Define Box to draw box primitive by selecting body.
- Show | Entities | Bodies | By Plane option is added.
- Edge path option is added in Selection filter.
- "Select containing bodies" option added in right click for failure element display.
- Show | Distance | Face to Face and Show | Distance | Edge to Edge option enhanced to support CAD faces and edges.
- Show | Distance | Face to Face option enhanced to identify shortest distance.
- Delete element performance is improved.
- Delete bodies performance is improved.
- Create sub assembly option is added in right click – assembly browser.
- Tutor link support added in Parameter dialog to link the images.

### ▼ Geometry

#### Edge | Chaining

- Node based edge chaining option is added.

#### Face | Create | From Edges

- Direction input method support is added.

#### Face | Create | Polygon

- Support enhanced if the nodes are not perfectly lying on a plane.

#### Face | Face Cleanup | Identify Thin Regions

- Option enhanced to break the identified thin faces.

#### Face | Connect Surface

- Color pallet is added to differentiate Face List 1 and Face List 2.

#### Face | Face Extend

- Option added to extend the face.
- Group input is supported.

#### Body | Create | Block

- Support enhanced to update origin point location using node or vertex.
- Block creation performance is improved.

#### Body | Create | Cylinder

- Support enhanced to update origin point location using vertex.

#### Body | Create | Extrude

- Support enhanced to remember element type for face input.



- Support enhanced to maintain shared faces with tri element as input and tetra or penta element as output.

#### Body | Create | From Faces

- Support enhanced to update the Body mesh control from the newly created body using "Remove from parent body" option.
- "Duplicate and make shared nodes" and "Shared faces in selected bodies" options are added.

#### Body | Create | Bounding Box

- Support added to create bounding box using "Bodies" option.
- Body name input support is added.

#### Body | Merge

- Support added to update Body mesh control when bodies containing same body mesh control are merged.

#### Body | Break | Plane (Bounded)

- Support enhanced to break body or face using finite plane.

#### Body | Skin

- Option added to extract the outer and inner faces of the body.

#### Model | Transfer body

- Support added to move bodies to sub models.

#### Transform

- Support enhanced to update the group with faces from newly created body.
- Create side face option is added in Offset

### ▼ Meshing

#### Mesh Control | IsoLine

- IsoLine start point identification is enhanced to use primary axis.

#### Surface Meshing

- "Improve interior angle" option is added in advanced options.
- Imprint meshing (parasolid assembly) option is added in advanced options.

#### Meshing | Auto Cleanup | Fill Hole

- Support enhanced to collect filled faces and create group.
- Support enhanced to fill hole by picking non manifold edges.

#### Boundary Layer

- Boundary layer meshing option is added. Environment variable should be set as SET SL\_BOUNDARY\_LAYER=TRUE.

#### Volume Meshing

- "Mesh as a Single body" option is added in advanced options.
- "Retain 2D surface bodies" option is added to keep shell bodies along with solid.
- Support enhanced to retain existing solid mesh for Tet element type using "Mesh as a Single body" option.
- Aspect Ratio and Aspect Ratio: Nastran quality cleanup support is added.
- Meshing performance is improved for the CFD support.

#### Tools | Layered Elements

- Support enhanced to input all faces in the body.

#### Tools | Wrapper

- Advanced wrapper options are supported.
- Performance is improved for refinement.
- Local wrap performance is improved.

#### Tools | Create Pyramid

- Support enhanced to maintain shared nodes between bodies.
- Support enhanced to maintain assembly structure.

#### Qcheck

- Aspect Ratio auto cleanup support is added for Tet Element type.
- Auto cleanup support is added for Aspect Ratio – Nastran.
- Tet cleanup - Support enhanced for multiple bodies.
- Height quality parameter is supported for Tri and Quad element type

### ▼ Assembly

#### Join

- Matching faces (mesh) assembly performance is improved.

#### Shared Entities

- Performance is improved for large models.

#### Body Union

- Option enhanced to maintain shared faces during merge bodies.
- General – Retain faces on other bodies option is added.

### ▼ FEM

#### Tools | Remove Logo

- "Include Loop" Option added to collect the faces to be removed using the given edges.
- Undo/Redo support is added.

#### Nodes | Create | Coordinates:

- X, Y, and Z values are updated in edit box when user picks a node/vertex.

## ▼ Analysis - Structural and Thermal

Option added for Automatically creating Sets from Groups or Mesh Controls.

Connections | Bush

- Local coordinate support added for Bush creation. Added the local coordinate id under CBUSH card for Nastran.

Free Shape design Space is supported for optimization in OptiStruct.

Design Response Equation option added to support DRESP2 card in OptiStruct.

Export

- ABAQUS
  - Mass element support for writing in Abaqus None Analysis type.
- Nastran
  - BarL and BeamL property with various section types supported.
- OptiStruct
  - Param AUTOMSET in OptiStruct Excite Analysis.
  - Support for Global SPC writing in OptiStruct. Now the constraints that are not referred in Load case will be written as global SPC.
- ANSYS
  - Export enhanced to write Body Force load also supports for writing of Global Rotational Velocity in Solver Input dialog.
  - Ansys export enhanced to write Orthotropic material (Using Engineering Constants). Also supported MXPAND card writing in Modal Analysis.
  - Support for Bearing Pressure writing in Ansys CDB format.
  - Ansys Solver interface enhanced to directly solve Ansys model from SimLab and load the Ansys results in SimLab.

## ▼ Results

Import Results

- Support added for importing ANSYS results.
- ANSYS results supported using H3D reader

## ▼ Midsurface

Connect | Collapse Faces

- Undo/Redo support added.

## ▼ Bolt Modeling

### Import File

- The head and thread body will be identified automatically by comparing the diameter of bolt holes while importing the bolt definition file.

### Thread Pretension | Permas

- Supported PREDIR/SCREWDIR options in Permas Thread pretension creation.

## ▼ Excite

### Crank Fillet

- New Crank fillet pattern is added.
- Pattern 6 is redesigned to change the cross section of the fillet.
- Element type(Hex/Wedge or Tet) support added for Crank fillet.

### Crank Bay

- XML import and export is supported for added element type (Hex/Wedge or Tet) in crank fillet.

## ▼ Process Automation

Tutor link added Group Mapping dialog for linking the images.

## ▼ Trials

### Find Crack/Wedge

- Auto cleanup supported for predefined patterns.

## ▼ Newly Supported Automation Tools

### Geometry

- Body | Create | Bounding box | Box
- Body | Create | From Solid Elements | Box
- Body | Break | Box

### System

- Show | Bodies | By Plane

### FEM

- Meshing | Tools | Pipe Meshing Tools

Analysis -Structural and Thermal

- Automatic creation of Sets from Groups or Mesh Controls.

#### Import Results

- Import Results | Ansys results file

#### Special

- PowerTrain
- Special | PowerTrain | 1D Ball Bearing

#### Trials

- Script support added for update attributes.

#### System

- Show | Entities | Bodies | By size

### ▼ Enhanced Automation Tools

#### Geometry

- Body | Create | Extrude | Upto Surface

### ▼ New Scripting Functions

- GetNodePositionsFromNodeID - which will give the XYZ values of given node.
- GetBoxPointsForBodies - will give box points of selected bodies.
- Common script function added to get bodies with sub string
- Common function added to pop up message box in Auto play and interactive mode.
- Script function added to merge all bodies by giving the suffix name as input.

# Virtual Wind Tunnel 13.1 Release Notes

## Highlights

---

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.

### ▼ Highlights

- Supported physics
  - Transient/steady state
  - Turbulence
  - Rotating wheels, moving wind tunnel ground
  - Heat-exchanger
  - Fluid-Structure-Interaction (FSI)
- Based on AcuSolve CFD technology
  - Fast, robust and accurate solutions for transient and steady state applications
- Automatic and fast volume meshing
  - User defined volume mesh refinement zones
  - Boundary layer (BL) parameter definition on a part level
  - Exclude parts from BL generation to reduce total element count
- Dimension measurement utility to ensure model is in correct unit system
- Vehicle positioning tool
- Accurate frontal area calculation of vehicle
- Result output control
- Automatic reporting
  - Report generation containing aerodynamic results, such as drag coeff.
  - User defined section cuts and probes for post-processing

### ▼ New Features

- Definition of planar section cuts for result contour plotting, such as velocity magnitude on the symmetry plane of the vehicle. The plots will be included in the automatically generated report. A selection of different results type are available; for example, pressure or velocity. The legend range for each result type can be adjusted globally or individually for each section cut.
- User defined probe points to monitor the evolution of the flow field in a particular location, e.g. pressure fluctuations over time. The plots will be included in the automatically generated report. A selection of different result types are available, for example, pressure or velocity. The legend range for each result type can be adjusted globally or individually for each probe point.
- Support of user defined fluid material by specifying density and viscosity of the material.
- Implementation of a new algorithm to compute projected frontal areas of vehicles with high accuracy.
- Wind tunnel inflow speed can be defined by a velocity magnitude and an angle around the z-axis, to model situations where side wind effects occur.

### ▼ Enhancements

- Definition of number of cores to be used for the CFD analysis has been moved to the Run dialog.
- When clicking **Export** in the Run dialog the \*.stmod file of the current session will be exported to the run directory.
- Version number of the CFD solver AcuSolve used to perform the analysis is included in the report.
- Refinement zones are displayed in the graphic area only when the user has entered the refinement zone mode.
- Added a vehicle model for the tutorial into the installation directory, surface mesh and geometry.

### ▼ Resolved Issues

- If a surface mesh model, consisting of quad elements only, is not touching the wind tunnel ground, the surface model is removed from the analysis domain.
- Deleting an analysis from the Run History table does not delete all files generated during the corresponding CFD analysis.
- If the surface mesh model is not touching the wind tunnel ground, the ref. zones are ignored for surface meshing and the wind tunnel ground is meshed with a constant element size.
- In case the surface mesh model is cutting the top of the wind tunnel, the mesh intersection step fails and an error message is displayed: "can't perform mesh intersection".
- The default element size for refinement boxes of type *Around Part or Around Body* is 1m.

▼ **Note**

- It is necessary to install both the Virtual Wind Tunnel 13.1 package as well as the AcuSolve 13.302 package to access the new functionality of VWT.
- For certain configurations, VWT 13.1 requires the Altair License Manager version 12.1 to be installed.



# HyperXtrude 2015.1 Release Notes

## Highlights

---

Today's extrusion companies are forced to produce a higher mix of complex profiles in shorter product development cycles while simultaneously reducing their production costs. HyperXtrude is a simulation tool developed to analyze and validate the design of extrusion dies and process.

HyperXtrude enables production engineers to analysis material flow and heat transfer inside a die during extrusion to validate die designs which in turn helps to reduce or eliminate costly and time-consuming die trials.

HyperXtrude® 2015 introduced a new CAD interface for HyperXtrude solver and takes giant leap to a direct CAD to solution approach. This approach automated and hid the meshing and data deck creation; and completely relegated it to the background. Hence, after creating the CAD model user submitted the job for analysis. HyperXtrude 2015 supported only steady state metal extrusion analysis. In this release, this is extended to transient, tool deflection, and bearing optimization analysis.

### Supported Analysis Types

- Profile nose cone analysis (steady state)
- Multi-cycle transient analysis
  - Charge weld analysis
  - Billet skin tracking analysis
  - Starter billet analysis
  - Tapered billet heating analysis
- Tool deflection analysis with OptiStruct
  - With assumed loads
  - With loads mapped from extrusion analysis
- Bearing optimization analysis

### ▼ Choked Bearing

Interface can automatically handle bearing regions with choke.

### ▼ Variable Bearing Start

Models with bearing starting at different Z-coordinates (variable bearing start) is supported. This requires manual manipulation of the bearing region.

### ▼ Charge Weld Analysis

With a click of a button, any model can be submitted for transient analysis. Charge weld analysis is automatically enabled.

▼ **Billet Skin Tracking**

While creating billet, a skin layer can be added with no additional effort. This automatically enable billet skin tracking analysis, which is used for predicting back end defects, butt length, and any potential issues with the die design.

▼ **Starter Billets**

Multi-cycle analysis with different billet lengths for each cycle up to five cycles.

▼ **Tapered Billet Heating**

HyperXtrude 2015.1 now supports specification of tapered heating: axial, lateral, and radial. Specification used can be different from cycle to cycle up to a maximum of five cycles. For axial taper, five linear taper zones are supported.

▼ **Tool Deflection Analysis**

This release supports an easy to use tool deflection interface. You can import the model, specify constraints, and submit for analysis. Load surface will be automatically determined. This analysis supports mapped loads, linearly interpolated loads, and manually specified constant surface loads.

▼ **Bearing Optimization Analysis**

An interface to set up a bearing optimization analysis is available in this release.

▼ **Material Database**

An interface to add, delete, edit, and select material is available in the release

▼ **Advanced Parameters**

While submitting the job for analysis, users can control the tolerance and number of nonlinear iterations.

# Software and Hardware 14.0 Release Notes

## Hardware Platforms

---

### ▼ Operating System Additions

Support for the following platforms has been added:

- Red Hat Enterprise Linux 6.6 64-bit (x86\_64)
- SuSE Linux Enterprise Server 11 SP3 (x86\_64)
- Mac OS X 10.9 64-bit (x86\_64)

### ▼ Discontinued Hardware Platform Support Systems

Support for the following platforms has been dropped:

- Red Hat Enterprise Linux 6.2 64-bit (x86\_64)
- SuSE Linux Enterprise Server 11 SP2 (x86\_64)
- Windows Vista 64-bit (x86\_64)
- Mac OS X 10.8 64-bit (x86\_64)

## Installation

---

HyperWorks 14.0 suite contains several independent product packages. This gives you the flexibility to download desired packages as needed. Any HyperWorks 14.0 standalone SA update package can be installed either on top of an existing 14.0 installation, or can be installed independently without the prerequisite of a 14.0 base. Multiple product SA updates can now be installed concurrently with or without the 14.0 base.

HyperWorks 14.0 installer brings the robust capabilities and features of the HyperWorks installer as well as the additional support for new products including new GUI based HyperXtrude and FEKO applications.

### ▼ New Products, Tools, and Packaging

- New products and tools
  - FEKO product suite
  - New GUI based HyperXtrude
  - New Linux Compatibility Tool: HyperWorks 14.0 now includes a Linux utility for system diagnostics to help troubleshoot Linux issues
  - New GUI based license utility
  - HyperWorks Student Edition available for Mac OS X
  - New HyperWorks Toolkit
  - MotionSolve subroutine builder
- Standardized MPI implementations: The implementations of the Message Passing Interface (MPI) have been standardized across FEKO, AcuSolve, RADIOSS, and OptiStruct. The Intel, IBM Platform, Microsoft, and MPICH MPI implementations are now included. End users can still use their own MPI implementation.

### ▼ Installation

- Upgrade the HyperWorks Installer to InstallAnywhere 2014 SP1
- File Association: HyperWorks 14.0 on Windows now associate HyperWorks files with appropriate HyperWorks applications
- File Preview: HyperMesh files now have previews in the icons

### ▼ Invoking

- HyperMath is now available on on Mac OS X

### ▼ Resolved Issues

- Resolved issue encountered when non-HyperWorks product installers existing in the same directory as Altair HyperWorks installers
- Improved error reporting for HyperWorks installation

- Resolved an issue with help configuration in the installer
- Resolved graphics issues on Optimus based video card setups

#### ▼ Known Limitations

- In Mac OS X 10.9 the installer will still report “-0” on the Summary panel due to a bug in InstallAnywhere.
- 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.
- On Windows 8, desktop and Start Menu icons are blank for server and client installations. This can be resolved by changing the Windows File Explorer policy to “Allow the use of remote paths in file shortcut icons”.
- 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 (such as, 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_hw14.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.” KDE is not affected.
- On Linux, the installer does not run the icon installation by default; the user must run the icon installer located in the LinuxMenu directory.

# HyperWorks Licensing 14.0 Release Notes

## Highlights

---

Through its innovative HyperWorks licensing mechanism, Altair provides great value. With every release licensing may slightly be modified to address new usage scenarios our customers may face. Common throughout the existence of HyperWorks is the goal to maximize the functionality that can be accessed thru a HyperWorks license.

The highlights of HyperWorks 14.0 licensing are following:

### ▼ **New Solver Licensing Option - HyperWorks Unlimited Solver Node Licensing**

For solvers (AcuSolve, FEKO, HyperForm, HyperXtrude, MotionSolve, OptiStruct, and RADIOSS), two different licensing options exist. The two options are the Per-Job licensing (as available since HyperWorks 11.0) and the HyperWorks Unlimited Solver Node (Per-Node) licensing (New in HyperWorks 14.0). Selection has to be made at purchase/renewal of license.

### ▼ **Adding New Product License Features**

- **FEKO** suite: FEKO is a powerful 3D simulation package intended for the analysis of a wide range of electromagnetic radiation and scattering problems. The FEKO GUI applications require 21 HWU to run. FEKO Solver follows the HyperWorks Solvers licensing mechanisms.
- **HyperXtrude 2015** is a new CAD-based, easy to use interface for metal extrusion analysis. Users start with a 3D CAD model of die assembly and submit a run for analysis in just a few minutes. It draws 21 HyperWorks units to invoke this product.
- **HyperStudy Fit Excel Plugin**, a curve fitting tools for MS Excel, draws 5 HyperWorks units.

### ▼ **Changes in Licensing Behavior**

- Under the same user and host, HyperMesh when running in batch mode will level the licensing units against other HyperWorks products. In the earlier versions, HyperMesh in batch mode stacks the units on the second instance.
- Under the same user and host, HyperMesh BatchMesher will level the HyperWorks units against other HyperWorks applications. In the earlier versions, the BatchMesher stack the units.
- Removal of the OptiStruct Check and Radioss Check license features to run HW 14.0 OptiStruct and RADIOSS.

▼ **New License Utility Tool**

A new GUI license utility tool has been added. This tool has the capability to perform the server queries, unit draw lookups, borrowing set up, host ID lookup, and Hosted HWU management.

▼ **Altair License Management System Upgrade**

The Altair License System has been updated to version 12.2. HyperWorks 14.0 requires the Altair Licensing Manager 12.2 or higher.

▼ **Resolved Issues**

Resolved the issue when application took several minutes to start due to licensing lock file.