HW-11.0-230-HWSolvers Release Notes

HW-11.0.230-HWSolvers provides many updates to the HyperWorks 11.0 solver suite.

RADIOSS	Finite element solver for linear and nonlinear problems A summary of new features in release 11.0-230: • AMSES solver for EIGRA is even faster • Boundary conditions can change in a continuation SUBCASE • Support of elasto-plastic materials for composite analysis • Option added to not overwrite previous analysis results files
MotionSolve	Multi-body simulation solver
OptiStruct	Design and optimization software using finite elements and multi-body dynamics A summary of highlights in release 11.0-230: • Move limits implemented for topology optimization with nonlinear responses
HyperXtrude	Finite element solver for metal extrusion, polymer processing, metal rolling, and friction stir welding processes.

RADIOSS

RADIOSS is a state-of-the-art finite element solver uniting implicit and explicit integration schemes for the solution of a wide variety of engineering problems, from linear statics and linear dynamics to complex nonlinear transient dynamics and mechanical systems. This robust, multidisciplinary solver enables designers to maximize performance related to durability, NVH, crash, safety, manufacturability, and fluid-structure interaction, in order to bring innovative products to market faster.

RADIOSS (Bulk Data Format)

New Features

Boundary conditions can change for continuation subcases	-230	The SPC set can change from the previous subcase when the continuation subcase (CNTNLSUB) is used for nonlinear analysis. This allows for previously SPC'ed DOF to be released.
Internal (MPCF) and External (OLOAD) forces written to the .op2, .pch, and .h3d files for nonlinear analysis	-230	MPCF and OLOAD analysis results are now written to the `.op2`, `.pch`, and `.h3d` files for NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.
Support of elasto-plastic orthotropic material with TSAI-WU and CRASURVT yield criteria for composite shell materials	-230	MATX25 material definition has been added for NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.
The SYSSETTING command	-230	The new command SYSSETTING(SAVEFILE=ALL) will save the

SAVEFILE has been added to save analysis results from previous runs		previous run's results files (.h3d, .op2, .pch, etc.) by renaming the old file with a numeric extension as is currently done with the .stat and .out files. This can also be set in the configuration file (hwsolver.cfg).
-compress option now works with a tolerance	-230	A tolerance can be specified with the –compress option to combine property data with the same or similar values.
Mechanical strain written to the HyperMesh .res file	-230	Under thermal analysis, the separate mechanical and thermal strain components are written to the HyperMesh .res file.
Added PARAM,AMSESLM so the Large Mass Method can be used with EIGRA	-230	SPCD should be used for enforced motion for frequency response. If the old Large Mass Method is used for enforced motion for modal frequency response and EIGRA is used to calculate the modes, then PARAM,AMSESLM,1 should be specified.
Added PARAM,GE_MOD to override all material, element, and property GE data	-230	The value of GE_MOD will replace the value of GE on all MATx, CELAS2, PELAS, PELAST, PBUSH, PBUSHT, PCOMP, PCOMPP, and PCOMPG data. This includes blank values of GE. GE_MOD can be set to zero to remove all structural damping from the model.
Bolt Pretension	-220	Pretensioned bolts can be modeled with 1D elements. The pretension analysis can be applied to linear and nonlinear static subcases. These static subcases can then be used in pre-loaded dynamic analysis.
Solution Control AUTOSPC Command	-220	The new Solution Control AUTOSPC command replaces the AUTOSPC and PRGPST PARAM data.
Use of SUBCOM and SUBSEQ to combine Static Analysis SUBCASE results	-220	The new SUBCOM data can be used to generate results that are a linear combination of preceding linear and nonlinear static SUBCASE results. The new SUBSEQ data is used to specify the weighting factors.
Equivalent Radiated Power (ERP)	-220	The Equivalent Radiated Power (Square of the normal velocity integrated over the area) from a panel is calculated and output to the .pch file.
Modal STRESS and DISPLACEMENT for all modes and residual vectors written to the .op2 and .pch files	-220	Using the keywords STRESS(MODAL) and DISPLACEMENT(MODAL) in an eigenvalue, modal FRF, or modal transient analysis SUBCASE will cause the modal stresses and displacements to be output. These modal results include both the structural modes and the residual vector modes. The .op2 or .pch file generation can be specified using the OUTPUT command (OUTPUT=OUTPUT2 or OUTPUT=PUNCH) or in the result specification: STRESS(MODAL,OP2), DISP(MODAL,PUNCH), etc.
	-220	These results can be used when third party fatigue codes combine the stress modes using modal participation coefficients from a transient analysis to calculate transient stresses.
GENEL Element	-220	The General Element (GENEL) for which the stiffness matrix for an arbitrary number of grids has been added. Either the stiffness or flexibility matrix can be input by the user.
Output of Applied Loads for Frequency Response and Transient Analysis	-220	Applied load output (OLOAD) is now available for Frequency Response, Coupled Fluid-Structure (NVH) Frequency Response, and Transient Analysis. The OLOAD information can be output to the .pch, .op2, and .h3d

		results files.
Modeling and Analysis of Seam Welds	-220	CSEAM and PSEAM data have been added for the modeling and analysis of seam welds.
GPFORCE output available for eigenvalue analysis	-220	Grid Point Force Balance information is now available for structural modes in `.h3d`, `.op2`, and `.pch` results files. These results can be used when third party fatigue codes combine the modal forces using modal participation coefficients from a transient analysis to calculate transient forces in welded, riveted, or bolted connections.
Ability to control the output of results from interior points and elements in Super Elements	-220	The OUTPUT options NODMIG, DMIGALL, and DMIGSET are used to specify that no results, all results, or a SET of results are output for interior points and elements of a CMS Super Element in a residual run. The SET is defined in the corresponding output request data.
Output of modal grid point (GPSTRESS) and element corner stresses for the CMS modes to the .op2 file	-220	During the CMS Structural Super Element generation, the CMS modal stresses can be written to the `.op2` file. These include the element centriodal and corner stresses for plates and solids, as well as the grid point stresses for grids of a solid mesh.
Ply based composite definition can now be used in nonlinear analysis	-220	PCOMPP ply based composite property data can now be used in the NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.
Ply based composite definition can now be used in MBD analysis	-220	PCOMPP ply based composite property data can now be used in the Multi-body Dynamic analysis using rigid bodies (PRBODY) and flexible bodies (PFBODY).
No available license, no worries	-220	If a RADIOSS run is submitted with `-licwait`, but there are not enough HyperWorks Units available for the job to start, the job will wait until enough HyperWorks Units are available, and then it will check them out and start running.
SMP and SPMD parallel analysis now available for nonlinear analysis	-220	The SPMD parallel option can now be used for analysis types NLGEOM, IMPDYN, and EXPDYN through the use of the `–rnp` and `–rnt` command line options. `–rnt` specifies the number of SMP parallel threads, and `–rnp` specifies the number of SPMD parallel processors.
Plasticity with Kinematic and/or Mixed Hardening now available for Solid Elements	-220	The MATS1 data now allows specification of Kinematic and/or Mixed Hardening rules through expanded options for the HR data.
RBAR data supported in nonlinear analysis	-220	RBAR rigid bar data can now be used in the NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.
Rigid Element Force (MPCF) results written to the .op2 file for nonlinear analysis	-220	MPCF analysis results are now written to the `.op2` file for RBE2, RBE3, and RBAR elements used in NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.
Bulk Data SET1 and SET3 formats supported	-220	SET1 and SET3 set definitions are now supported for defining sets in the Bulk Data section. These are internally converted to the more powerful and flexible Bulk Data SET data.
Passing of metadata from the	-220	If the keywords METADATA and ENDMETADATA are present in the

input file to a resulting XML file

input file, the content between the keywords is written in a separate <basename>_metadata.xml file. This allows passing of metadata information seamlessly through the solver.

Automated Multi-level Substructuring Eigenvalue Solver -201

A state-of-the art automated multi-level sub-structuring method for eigenvalue extraction is now available in RADIOSS.

This feature is available for normal modes, modal frequency response and modal transient response solution sequences, as well as CMS super element generation.

This solution method is activated when the EIGRA Bulk Data entry is referenced via the METHOD data selector in the Subcase definition. It may also be activated by adding PARAM,AMSES,YES or via the command line argument –amses.

Pre-stressed Buckling Analysis -201

It is now possible to perform a linear buckling analysis of a structure in a pre-stressed state.

STATSUB(PRELOAD) is used to identify the pre-stress subcase. STATSUB(BUCKLING) is used to identify the loading subcase.

Both references are required for a pre-stressed buckling analysis.

Hybrid damping for direct dynamic analysis -201

Hybrid damping is now available in RADIOSS.

This is a means of applying modal damping in a direct frequency response or direct transient response analysis.

This is recommended mainly for residual models, where a large part of the structure is represented by external superelements. This is because of the computational expense of this feature.

It is activated via the HYBDAMP subcase information entry, which in turn references a HYBDAMP Bulk Data entry.

Additionally, the DMIGMOD entry has been expanded to allow hybrid damping to be defined for external superelements only. (HYBDAMP will apply damping to both residual structure and external superelements).

New 3D orthotropic material definition -201

A solid orthotropic material definition using engineering properties rather than stiffness matrix input is now available in RADIOSS.

This is an alternative input format for the MAT9 anisotropic material definition and is translated on reading. The echoed material definition will use the MAT9 definition format.

Frequency dependent spring property -201

A frequency dependent spring property is now available in RADIOSS.

Similar to the existing PBUSHT frequency dependent bushing property definition, a PELAST frequency dependent spring property definition has been added. It may be referenced by spring elements (CELAS1, CELAS3).

Automatic input update -201

An automated mechanism for updating input definitions from an external file is now available in RADIOSS.

This allows optimization output from OptiStruct (e.g. .grid file, .prop file) to be used in a follow-on run without the need to regenerate the input file.

The I/O Option ASSIGN has a new sub-option UPDATE, which allows an external file to be referenced for the purpose of updating input. Input definitions in the referenced file are then used in place of the definitions in the source file.

		Another I/O Option, UPDATE, has several parameters to control the behavior of ASSIGN,UPDATE.
Long-field free format input	-201	RADIOSS now accepts Bulk Data input using the long-field free format. Long-field free format consists of four comma separated input fields per line. This input format is identified by an asterisk in the first column or (for the line with the card name) at the end of the card name.
Compressed input files	-201	RADIOSS now accepts compressed input (using GZIP format) directly. The main input file and/or any include files may be compressed using the GZIP format. Compressed files are recognized by the `.gz` extension. It is not necessary to modify INCLUDE lines in the input deck with this `.gz` extension. **Note**: Files compressed using Windows ZIP format are not recognized.

Enhancements

EIGRA solution time reduced for large eigenvalue problems	-230	The solution time has been reduced when EIGRA is used for calculation of a large number of modes.
Support of Platform MPI Version 8 on Windows	-230	RADIOSS will now run MPI parallel on Windows machines that have Platform MPI version 8.
CB and CC Flexbody Generation Time Reduced	-220	The CB generation time is reduced for problems with large number of ASET DOF. The CC generation time is greatly reduced, if AMSES or AMLS is used.
EIGRA can be used to solve larger problems	-220	The number of modes that can be calculated when using EIGRA has been increased for large models.
Modal Frequency Response solution time reduced for problems without structural or viscous damping	-220	If there is only modal damping (no structural or viscous damping), then the resulting modal Frequency Response problem is diagonal. In this case, because a diagonal solution algorithm is used, the solution time is reduced significantly.
MPCFORCES output written to the .h3d file for static analysis	-220	In addition to the `.op2` file, MPCFORCES results for static analysis are now available in the `.h3d` file.
Unreferenced SET data written to the .op2 file	-220	Even if a SET is unreferenced by the input data, the SET data is written to the `.op2` file. This allows SET data to be passed to and used by post-analysis programs, such as those that calculate fatigue life.
SET data written to the .h3d file	-220	SET data (both referenced and unreferenced) is written to the `.h3d` file.
Negative values for LDM and Scale Factors now allowed in FATLOAD data	-220	This allows more flexibility in defining fatigue loading parameters.
Element types, properties, and materials can now be used to	-220	The Bulk Data SET definition can include element types (PLOTEL, CBEAM, etc.), property types, and materials to define a set of GRID

define sets of grids using Bulk Data SET		points associated with those elements, properties, or materials.
Drilling stiffness of shells can be turned off for nonlinear analysis	-220	The data IDRIL has been added to shell element property parameter data (XSHLPRM) to run off the drilling stiffness of shell elements for nonlinear analysis types NLGEOM, IMPDYN, and EXPDYN.
Composite Laminate added to account for offset elements	-220	The PCOMP/PCOMPG/STACK now support offset elements in laminate option (LAM) for smeared laminates. If the elements are offset (Z0 not equal to one half of the lay-up thickness), use the new laminate option SMEARZ0.
AVL/EXCITE Interface	-201	The AVL/EXCITE interface has been enhanced to allow users to obtain results for interior points in a residual model run. After running an AVL/EXCITE analysis an .INP4 file is generated that contains the modal participation factors. The .INP4 file can be used along with the original H3DDMIG file to recover stresses, strains, displacements, velocities, and accelerations from a residual run.
Geometric Nonlinear Solutions	-201	The following enhancements have been made for the geometric nonlinear solutions, i.e. the NLGEOM, IMPDYN and EXPDYN analysis types. • It is no longer a constraint that continuing geometric nonlinear subcases must have the same boundary conditions (same SPC reference). • 2D orthotropic materials are now supported (MAT8). • Output of internal forces is now supported (GPFORCE). • Interdependent coordinate systems are now supported (coordinate systems that reference other coordinate systems via the RID field). • Added time history output for rods, bars and beams (XHIST). • Added output time step control to XHIST Bulk Data entry, allowing output time step control for individual XHIST requests. (XHIST). • Added option to enable small displacement assumption (NLPARMX, TSTEPNX). • Added option to enable springback analysis (NLPARMX). • Added support of fixed time points that the automatic time step control will adhere to. (NLPARMX, TSTEPNX). • Expanded material library to include LAW0 (MATX0), LAW21 (MATX21), LAW43 (MATX43), LAW60 (MATX60) and LAW68 (MATX68). • Added support of contact definitions for 10-noded tetra elements. • 2D laminated composite definitions are now supported (PCOMP, PCOMPG). • Added support of enforced displacements (SPCD) for NLGEOM analysis type. • Adjusted the default convergence criteria defined on the TSTEPNL card from PW to UPW. • Added support for rotational displacement output requests

(DISP(ROTA)).

RIGID Option for Stiffness Entry on Bushing Property Definition -201

A new keyword option has been added for stiffness entries on PBUSH property definition.

Instead of entering a real value for any of the stiffness components, the user may now enter the keyword RIGID.

When RIGID is entered, a very high relative stiffness value (quasi-rigid) is used for that stiffness component.

Allow Craig-Chang Superelements in Residual Run Using AMLS/AMSES -201

It is now possible to use superelements generated using the General Method (GM) with free boundary interfaces (the Craig-Chang approach), in a residual run with AMLS/AMSES.

Improved SMP Parallelization for Iterative PCG Solver -201

Improvements were made to the performance of shared memory parallelization for the iterative PCG solver.

Solver Run Manager GUI -201

Several enhancements have been made to the HyperWorks Solver Run Manager GUI.

- Checkboxes were added to facilitate SMP and SPMD parallel runs.
- GUI will remember options used for Bulk Data input and Block input separately, and based on the selected input file type the appropriate previously used options will be selected and displayed.
- Added a “Results” button that appears once the job has completed. This button will launch HyperView and load the resulting `.mvw` file. There are also additional options under the “View” menu, to view the resulting `.html` summary file and the resulting `.h3d` results file.

Default Projections and Checks for CWELD Elements -201

Several adjustments were made to the default behavior of the CWELD element to enhance its ease of use.

Time Varying W3 and W4 for Transient Response Analysis -201

The TSTEP card has been enhanced to include fields for W3 and W4, such that these quantities can be defined differently for each set of time increments.

Modal Participation Ratio Output -201

Modal participation ratios are output to the `.out` file when PARAM,EFFMASS,YES is defined.

1D Element Forces Included in Flexbody Output -201

The FORCE output request may be used in a flexbody generation run to obtain 1D element force results in the flexh3d file.

AutoSPC for AMLS Solver Interface -201

PARAM,AUTOSPC is now used to control autospc functionality for AMLS solver interface too.

The parameter previously used to control this, PARAM,AMLSAPC, is now defunct.

New Parameters

PARAM,AMSESLM -230

SPCD should be used for enforced motion for frequency response. If the old Large Mass Method is used for enforced motion for modal frequency response and EIGRA is used to calculate the modes, then PARAM,AMSESLM,1 should be specified.

PARAM,GE_MOD -230

The value of GE_MOD will replace the value of GE on all MATx, CELAS2, PELAS, PELAST, PBUSH, PBUSHT, PCOMP, PCOMPP, and PCOMPG data. This includes blank values of GE. GE_MOD can be set to zero to remove all structural damping from the model.

New SYSETTING Entries

SAVEFILE=ALL	-230	The new command SYSSETTING(SAVEFILE=ALL) will save the previous run's results files (`.h3d`, `.op2`, `.pch`, etc.) by renaming the old file with a numeric extension as is currently done with the `.stat` and `.out` files.

New I/O Option Entries

ERP	-220	Used to request ERP results to the `.pch` file.
UPDATE	-201	Controls the behavior of the input update feature where an external file can be selected to update input definitions in the current file (ASSIGN,UPDATE).

New Solution Control Entries

AUTOSPC	-220	Replaces the AUTOSPC and PRGPST PARAM data.

New Subcase Information Entries

SUBCOM	-220	Specifies that results are calculated as a linear combination of preceding static SUBCASE results.
SUBSEQ	-220	Specifies the weighing for the linear combination of preceding static SUBCASE results for the SUBCOM.
PRETENSION	-220	Used to select and activate a pretensioning bolt load.
HYBDAMP	-201	Hybrid damping data selector. This data selector references a HYBDAMP Bulk Data entry.

New Bulk Data Entries

MATX25	-230	MATX25 material definition for support of elasto-plastic orthotropic material with TSAI-WU and CRASURVT yield criteria for composite shell materials has been added for NLGEOM, IMPDYN, and EXPDYN nonlinear analyses.
CSEAM	-220	Defines the seam weld element connection data.
PSEAM	-220	Used to define the seam weld element property data.
ERPPNL	-220	Used to define the panels for the ERP calculation.
PANELG	-220	Used to define panels for the PFPANEL, PFMODE, and ERP calculations.
PRETENS	-220	Defines the ROD, BAR, or BEAM element used to model the pretensioned bolt.

PTADD	-220	Defines a pretension load as a linear combination of load sets defined via PTFORCE, PTFORC1, PTADJST and PTADJS1 entries.
PTADJS1	-220	Defines adjustment (additional shortening) on a set of pretension sections.
PTADJST	-220	Defines adjustment (additional shortening) on pretension section.
PTFORC1	-220	Defines pretensioning force on a set of pretension sections.
PTFORCE	-220	Defines pretensioning force on pretension section.
GENEL	-220	Used to define a general finite element stiffness matrix to be added to the analysis.
METADATA / ENDMETADATA	-220	The content between the keywords METADATA and ENDMETADATA in the input file is written to the `<basename>_metadata.xml` file. This allows passing of metadata information seamlessly through the solver. METADATA/ENDMETADATA pairs can exist in both the Solution Control Section, as well as the Bulk Data section of the input file.
EIGRA	-201	Requests real eigenvalue extraction using the Automated Multi-level Substructuring Eigenvalue Solver (AMSES).
HYBDAMP	-201	Defines the application of modal damping in a direct frequency response or direct transient response analysis.
MAT9ORT	-201	A solid anisotropic material definition using engineering properties rather than stiffness matrix input.
MATX0	-201	Void material property extension for geometric nonlinear solutions.
MATX21	-201	Rock-concrete material property extension for geometric nonlinear solutions.
MATX43	-201	Hill orthotropic material property extension for geometric nonlinear solutions.
MATX60	-201	Elastic plastic piecewise nonlinear material property extension for geometric nonlinear solutions.
MATX68	-201	Honeycomb material property extension for geometric nonlinear solutions.
PARAM,AMSES	-201	Switches eigenvalue extraction method from Lanczos to AMSES.
PCOMPX	-201	Laminated composite property extension for geometric nonlinear solutions.

Enhanced I/O Option Entries

ASSIGN	-201	AVL/EXCITE, in order to recover results for interior grids and elements.
	-201	Added UPDATE option to reference updated property input.

	-201	Added EXCINP option to reference modal participation factors from
SYSSETTING	-201	Added H3DVTAG option.

Enhanced Subcase Information Entries

STATSUB	-220	Added type PRETENS to specify bolt pretension.

Enhanced Bulk Data Entries

XSHLPRM	-220	The data IDRIL has been added in order to run off the drilling stiffness of shell elements for nonlinear analysis types NLGEOM, IMPDYN, and EXPDYN.
PCOMP/PCOMPG/STACK	-220	There is a new laminate option (LAM) for smeared laminates. If the elements are offset (Z0 not equal to one half of the lay-up thickness), use the new laminate option SMEARZ0.
MATS1	-220	The MATS1 data now allows specification of Kinematic and/or Mixed Hardening rules through expanded options for the HR data.
DRAPE	-220	Integer values for T and THETA are now read in correctly as real values.
DRESP1	-220	Added FRERP for defining equivalent radiated power as a response.
PANELG	-220	Generic panel definition for ERP or panel participation output.
ERPPLN	-220	Panel definition for equivalent radiated power output.
DTI,UNITS	-201	Expanded the set of available units.
NLPARMX	-201	Added options to control small displacement assumption, springback analysis and fixed time points.
PBUSH	-201	Added RIGID keyword option for stiffness.
TSTEP	-201	Added W3 and W4 fields allowing these values to be defined differently for each set of time increments.
TSTEPNX	-201	Added options to control small displacement assumption and fixed time points.
XHIST	-201	Added DDTHM field, allowing output time step control for individual XHIST requests.
	-201	Expanded the available types, adding BAR, BEAM and ROD.

Resolved Issues

Incorrect shell element corner stresses	-230	In some cases, corner stresses for shell elements could be incorrect when there were also solid elements in the model and corner stresses were requested for those elements, as well.
Incorrect results when SPCD	-230	Imposed displacement (SPCD) in a local coordinate system was

was used with a local coordinate system in nonlinear analysis		implemented incorrectly for nonlinear analysis types NLGEOM, IMPDYN, and EXPDYN.
Contact Pressure wrong	-230	The calculated contact pressure was incorrect for certain meshes.
Modal transient analysis with enforced motion (SPCD) gave poor results	-220	A new algorithm was implemented for modal transient analysis with enforced motion (SPCD). Now the results match those from direct transient analysis.
GPSTRESS not available for multiple SUBCASE in the .res file	-220	In the past, only the GPSTRESS for the first SUBCASE was available in the `.res` file. Now, the GPSTRESS for every SUBCASE is available.
Sign of MPC Force incorrect in the .op2 file	-220	The signs of the MPC Forces had been reversed in the `.op2` results file.
GPKE wrong for AMLS/EIGRA analysis	-220	Unless DISP(ROT)=ALL was requested, the GPKE values were wrong when AMLS or EIGRA was used. This has been corrected.
Wrong data input when DRAPE items T and THETA are integers	-220	The DRAPE data items T and THETA should be real numbers. In the previous release, if integer numbers were used they were not correctly converted to real numbers.
PCOMPP referenced by EXCLUDE ignored	-220	If the Solution Control data EXCLUDE referenced PCOMPP properties, these were not actually excluded from the buckling analysis. Now, PCOMPP properties referenced by EXCLUDE data are excluded from the buckling analysis.
No corner stress output to .op2 file if only CTRIA3 elements exist	-220	Since the corner stress is the same as the centriod stress for CTRIA3 elements, output of corner stress to the `.op2` file was skipped if the model only contained CTRIA3 elements. Now, corner stress results will be written to the `.op2` file, even if the model only contains CTRIA3 elements.
PS Field on GRID Bulk Data Entry Was Ignored for Geometric Nonlinear Solutions	-201	PS field is now recognized for the geometric nonlinear solutions, i.e. the NLGEOM, IMPDYN and EXPDYN analysis types.
Loads with Magnitude 0.0 Were Incorrectly Interpreted in Geometric Nonlinear Solutions	-201	Forces, moments, pressures with magnitude 0.0 are now interpreted correctly for the geometric nonlinear solutions, i.e. the NLGEOM, IMPDYN and EXPDYN analysis types.
Wrong Thickness Used for Contact Involving Laminated Composite Materials	-201	The correct thickness is now being used for contacts involving laminated composite materials.
Error Termination When Unused Property References a Non-existing Material	-201	We no longer terminate that solution when an unused property references a non-existing material. Instead we just issue a warning.

RADIOSS (Block Format)

New Features

Airbags	-230	/MONVOLFVM1: Injector properties can be defined referring to /PROP/INJECT card (as for /AIRBAG1); gas mixtures and properties

can be prescribed referring to /MAT/GAS cards.

Meshing parameters for Finite Volumes can be prescribed referring to a given frame.

AcuSolve Coupling -230 Primary applications targeted by this capability include:

- Automotive: Hydraulically damped rubber mount, door seals, shock absorbers, design of valves and rubber diaphragms, antilock braking systems
- Oil/Gas: Long marine risers, moorings, free spans, drilling risers
- Aerospace: Wing aero-elasticity, UAV, MAV
- Wind turbine: Large deformation of blades
- Consumer Goods: design and packaging
- Bio-medical

Note: Currently available on Linux64 platform only.

Starter Performance -230 RADIOSS Starter SMP parallelization. Currently available for Linux64 and Win64 only.

Multi-domains -201 Multi-domains technique allows running simultaneously several domains; each one can have its own time-step.

-201 RADIOSS 11.0.201 Starter now has the capability to automatically split a single input deck into two domains (decks) based on domain definition given in the deck itself. The master file for the coupling is also automatically generated by RADIOSS Starter.

-201 The new single input file together with 11.0.201 RADIOSS Manager GUI makes it much easier to run multi-domains jobs: the GUI recognizes the Starter single-input file and runs simultaneously the master file and the multi-domains Engine files.

Elements -201 /TETRA10 compatibility with small strain formulation.

Contacts -201 /TYPE18 with master/slave formulation gives enhanced precision.

Materials and Failure Models -201 New Equations of State have been added: Sesame table and Puff EOS

-201 /XFEM: bending failure criterion has been added.

Initial Volume Fraction -201 /INIVOL: Describes the initial volume fractions of different materials in multi-material ALE elements.

Implicit -201 Drilling DOF for shells (available with Batoz, QEPH, SH3N C0)

-201 Small displacement option

-201 Rayleigh damping with dynamic implicit

-201 20 nodes solid element

Enhancements

Materials	-230	/LAW70: improved stability and material behavior, especially at high compression level. **Warning**: Results obtained with previous versions may be incorrect and differ from actual results.
Failure criteria	-230	/FAIL/ENERGY compatible with all material laws (including LAW25).
Composites	-230	Conservative strategy for element deletion: element is deleted only when all integration points have fulfilled failure criteria (Isolid=2). Compatible with Hashin, Puck and Ladeveze failure models.
	-230	/ANIM/SHELL/PHI/N: outputs the angle between fibers and element skew for composite shells; the target is to plot stress and strains in fibers direction.
	-230	Plastic work per ply and per direction.
Reference state	-230	/XREF compatibility with tetra elements.
Joints	-230	Starter checks and ensure consistency of skew systems defined in the joint.
Documentation	-230	Input card examples for material laws.
User interface	-230	Compatibility with SPMD.
Contacts	-230	/INTER/TYPE7, TYPE11 and TYPE19: automatic memory resizing (MultiMP)
	-230	/INTER/TYPE2: search distance is computed automatically with Ignore=2
	-230	/INTER/TYPE21: Shear friction model Heat conductivity depending on contact pressure
	-201	/INTER/TYPE2: Penalty formulation (Spotflag = 25) allows automatically solving incompatible kinematic conditions. Optionally, stiffness (Stfac) and critical damping coefficient (Visc) can be tuned.
	-201	/INTER/TYPE2: improved user-friendliness: new flag "Ignore=1" allows selecting tied nodes based on a distance.
	-201	/INTER/TYPE20: new option to exclude from contact treatment penetrating nodes, with penetration > user defined value.
	-201	/INTER/TYPE7 and /TYPE11 have a new gap min formulation (Igap=3) taking into account mesh size. gap = min (gap_max, %mesh_size, max(gap_min, (gm+gs))
	-201	/INTER/TYPE21: add damping with reference to another tool
Airbags	-230	LAW58 and /LAW19 + reference metrics + inflation time > 0: the new flag "Isens" allows activating both reference metrics and inflation at a given time, using the same sensor.
	-201	New vent hole model based on the modified St Venant - Wantzel formulation.
Implicit	-230	/IMPL/LRIGROT: improves results quality in case of large rotations.

	-201	New option in /IMPL/SOLVER/N (N=7): Automatic selection between direct solver and PCG.
Advanced Mass Scaling	-230	AMS compatible with mesh adaptively.
	-230	Error Message Improvement: the node (or list of nodes) source of the issue is localized. Example: "*AMS IS LIKELY DIVERGING... CHECK FOR NODE ID* ="
	-201	/DT/AMS: compatible with /INTER/TYPE20, penalty joints.
	-201	/DT/INTER/AMS: contact stability time-step switch to AMS.
	-201	/DT/AMS: reproducibility versus number of threads
Performances	-230	Improved performance for Single Precision version.
	-201	Contact algorithm optimization: up to 2x a speed increase with a high number of cpu.
	-201	/MAT/LAW36: performance optimization for shells and solids.
Elements	-201	Small strain formulation available for TETRA10 Irot=1.
Input/Output	-201	Cross section output request just by defining a plane and part set.
	-201	RADIOSS Starter tries to detect all errors without stopping at the first error detected.
	-201	Animation file can be output according to SENSOR activation, for example when distance between two of nodes reaches a given value.
	-201	New options to prescribe non-structural (added) mass.
	-201	/INIBRI/ORTHO: initial state compatibility with orthotropic solids
Failure models	-201	Tabulated failure criterion available for solids and shells.
Sensors	-201	/SENSOR/INTER and /SENSOR/RWAL: activation based on interface or rigid wall contact.
	-201	/SENSOR/RBODY and /SENSOR/SECT: activation based on rigid body or section force.

New Starter Keywords

/MONVOL/FVMBAG1	-230	Injector properties can be defined referring to /PROP/INJECT card (as for /AIRBAG1)
/SECT/CIRCLE	-230	Section definition through a plane with circular bounds
/SECT/PARAL	-230	Section definition through a parallelogram
/EOS/PUFF	-201	Linear polynomial equation of state
/EOS/SESAME	-201	SESAME table of equation of state
/FAIL/TAB	-201	Strain failure model

/PROP/TYPE26	-201	Tabulated spring
/SURF/PLANE	-201	Infinite plane surface definition
/SUBDOMAIN	-201	Specifies parts belonging to a given subdomain for a multi-domains run.
/TIMEOUT	-201	Specifies the value of the timeout parameter for the establishment of the connection between RAD2RAD and the RADIOSS Engine(s) for a multi-domain run.

New Engine Keywords

/IMPL/LRIGROT	-230	Large rotation option for non-linear implicit
/IMPL/DYNA/FSI	-230	Implicit dynamics fluid structure interaction parameters
/DTSDE	-201	Increases time-step for 6-nodes degenerated solid elements
/STATE/NODE/TEMP	-201	Nodal temperature
/STATE/BRICK/ORTHO	-201	Orthotropy direction for orthotropic solids
/ABF	-201	Time History output in Altair Binary Format
/ANIM/LSENSOR	-201	Writes animation file at sensors activation times, similarly to /ANIM/SENSOR; input type is a list of sensors.
/DT/INTER/AMS	-201	Contact interface time step control

Modified Starter Keywords

/MAT/LAW19	-230	Reference state is activated through a sensor.
/MAT/LAW25	-230	Max plastic work per ply per direction.
/INTER/TYPE2	-230	Ignore=2: search distance is computed automatically
	-230	VISC flag (for spotflag=25): Critical damping coefficient on interface stiffness
	-201	Spotflag=25 switches to penalty formulation; Ignore = 1: all slave nodes which dot find a projection on master segment in the search distance defined in Dsearch, are ignored. Warning will be printed.
/INTER/TYPE21	-230	Heat conductivity depending on contact pressure.
	-201	Damping (translational and rotational) with reference to lab or another tool (TYPE21).
/ADMAS	-201	New input types for non-structural (added) mass: input by single node, by group of nodes, by surface, by part and group of parts.
/DEF_SHEL	-201	Flag to activate stiffness for drilling Degree Of Freedom, available for: • /PROP/TYPE1 (SHELL)

- /PROP/TYPE9 (SH_ORTH),
- /PROP/TYPE10 (SH_COMP)
- /PROP/TYPE11 (SH_SANDW)

/INTER/TYPE7 AND TYPE11	-201	Igap=3 - Gapmin is computed taking into account mesh size.
/INTER/TYPE20	-201	Fpenmax flag allows removal from contact penetrated nodes when penetration is higher than user defined value.
/EOS/GRUNEISEN, /EOS/POLYNOMIAL, /EOS/TILLOTSON	-201	Reference density input.
/INITEMP	-201	Outputs in `sta` file initial nodal temperature. Input by node list.
/INIVOL	-201	Allows defining Initial Volume Fractions of different materials in multi-material ALE elements.
/MAT/LAW58	-201	Reference state is activated at sensor time.

Modified Engine Keywords

/INIV/ROT, /INIV/ROT/Keyword3/1, /INIV/TRA/Keyword3/1	-201	Input by list of nodes
/IMPL/SOLVER/N	-201	N=7 for automatic solver choice
/DT/NODA/Keyword3/1	-201	Allows selection of node groups.
/DYREL/1	-201	Allows selection of node groups; input format has a new line (Beta and T are moved to next line).

Removed Engine Keywords

/DT/NODA/AMS	-230	/DT/AMS replaces /DT/NODA/AMS

Resolved Issues

Time Step	-230	RADIOSS Starter estimation of time step does not take into account damping (dm, df) for /PROP/BEAM; the effect of df on time step may be important for thick beams.
Parallel arithmetics	-230	INACTI = 5, 6 or 7 + TSTART are incompatible in MPP: PARITH/ON is not working.
	-230	TYPE20 + variable gap (Igap=1): PARITH/ON is not working.
Composites	-230	/PROP/SOL_ORTH + Ip=13: stress results not oriented correctly
	-230	/PROP/TYPE22 (thick shell) + Number of layers > 7: Engine might fail with NaN
	-230	/MAT/LAW25 (CRASURV): the maximum plastic work per direction does not have any effect on the layers deletion.

Multi-domains	-230	Possible instability for strain-rate dependant material laws.
	-230	Possible memory allocation issue with beam/shell connection across domains.
Joints	-230	KJOINT TYPE2 and TYPE8: no translational stiffness
Single precision	-230	LAW36 + global integration + iterative projection: Engine could fail with NaN
	-230	Bad performance with Intel compiler versions (Windows and Linux).
ALE	-230	SPMD parallel performance
SPH	-230	SPH + inlet/outlet + single precision
	-230	Starter memory allocation issue with huge models (> 10M SPH cells).
AMS	-230	/DT/AMS + CYLINDRICAL JOINTS
	-201	/DT/AMS + /INTER/TYPE2: slow convergence
	-201	/DT/AMS + /INTER/TYPE20 + Inacti =5 and 6
	-201	/DT/AMS + TETRA10 and TETRA4
	-201	/DT/AMS + degenerated 8 nodes bricks
	-201	/DT/AMS + LAW59
	-201	/DT/AMS + /DT/BRICK/CST
	-201	/DT/AMS + KJOINT and RLINK
Contacts	-230	TYPE20 + Fpenmax + slave node on master segment
	-230	INACTI = 6, but initial contact forces are non-zero.
	-230	TYPE5 + /SURF/PART/EXT: wrong segments are found.
	-230	/TYPE5 + /SURF/PART/EXT + TETRA10: wrong segments are found
	-230	/TYPE2 + rupture (Spotflag=21): Starter failed
	-201	TYPE11 + TETRA10
	-201	TYPE21 + zero gap
Airbags	-230	FVMBAG1 potential instability.
	-201	Shift time dependent porosity function with TTF.
	-201	FVMBAG + CPB,CPC different from zero
	-201	Multi-chambered Airbags: communication between chambers
Output	-230	/ANIM/SHELL/PLY + /ANIM/SHELL/TENS/STRESS/ALL: Engine could fail

	-230	/ANIM/VECT/CONT2 + /INTER/TYPE2 with Spotflag=10, 11, 12, 20, 21 and 22
	-230	/STATE/NODE/TEMP + SPMD may lead to an error termination
	-201	/SECT output in `*SC01`
	-201	/CTH
	-201	SPMD + SPH: output in TH
	-201	SH3N mass calculation in TH with ruptured elements.
	-201	Animation file generated with option /ANIM/SHELL/TENS/STRESS/ALL
Material and Failure Models	-230	/FAIL/TENSSTRAIN + LAW10 and LAW12: all the solid elements are deleted at time=0
	-230	/MAT/LAW35: potential instability when Open cells foam flag (Iflag) <>0
	-230	/MAT/LAW51: several fixes
	-230	MAT/LAW68 + //SUBMODEL + scale factor on yield function
	-230	/MAT/LAW70: several fixes
	-230	/MAT/LAW72: elastic behavior
	-201	/MAT/LAW38 + element deletion (Cut-off stress): possible stack overflow
	-201	/MAT/LAW69 with N_pair = blank
	-201	Hyperelastic materials + shells
Initial State	-230	/INIBRI + Isolid=12
	-201	/INISH3
	-201	Strain tensor initialization with /INIBRI/STRA_F
	-201	/INIBRI/AUX bad integration points number
	-201	Nodal temperature in the state file
Implicit	-230	Implicit dynamic: incorrect kinetic energy
	-230	Implicit nonlinear analysis: RADIOSS Engine could fail in SMP mode.
	-230	/INTER/TYPE5 and /INTER/TYPE7: Tstart does not work
	-230	TETRA10 + /ANIM/GPS/TENS (and ANIM/GPS/VONM): Engine failed with segmentation fault
	-230	Implicit NL + shell composite with offset (asymmetric positions) might not converge.

	-230	Imposed displacement and boundary condition coupling defined in local skew with implicit solution: incorrect solution.
	-230	When void or rigid material is used with /IMPL/AUTOSPC/OFF, the nodes associated with these materials will not be constrained automatically.
	-201	Implicit computation may hang at the end of computation when executed on more than one thread.
	-201	Crash using BCS solver for Buckling analysis on certain platforms.
	-201	Mix solver with SPMD
	-201	Implicit dynamic with initial velocity
	-201	Imposed disp and /BCS on the same node in local skew.
Stamping Analysis	-201	Implicit spring back after trimming

MotionSolve

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

Enhancements

Closed deformable curves and surfaces now supported in MotionSolve	-230	Deformable curves, used by the Point-to-Deformable-Curve entity and deformable surfaces, used by the Point-to-Deformable-Surface entity, can now be closed. This allows for a continuous curve/surface description where the constraint can apply without interruption.
Improved contact force visualization	-230	Normal and tangential forces for each of the contact locations are now optionally saved in the MotionSolve H3D output file. These forces can subsequently be viewed for each time frame in the simulation in HyperView. A new `contact_gra_output` attribute was added to the `ResOutput` command to enable this feature.
Linear Analysis	-230	Users can control the size of their output files by optionally disabling damping and modal output to the `mrf` file, if these data blocks are not of interest.
Craig-Bampton and Craig-Chang component mode synthesis speed greatly increased	-220	Component mode synthesis in RADIOSS, when using the Craig-Bampton method, is much faster now for problems with a large number of interface nodes (ASET DOF).
	-220	Dramatic speed improvement is also seen for the Craig-Chang method when using AMSES or AMLS eigenvalue methods. This change, even though it is in RADIOSS, significantly affects the usability of MotionSolve for models containing flexible bodies.
MotionSolve now supports TNO MF and TNO SWIFT tire models	-220	MotionSolve now supports the TNO MF and TNO SWIFT tire models. These add to the list of tires available via MotionSolve for vehicle dynamics, ride and durability analyses.

	-	MF-Tyre is the Delft-Tyre standard implementation of the renowned Pacejka Magic Formula that includes the latest developments. With MF-Tyre the engineer can simulate validated steady state and transient behavior up to 8Hz, making it the ideal tire model for handling and control prototyping analyses. Pure cornering and braking, as well as combined conditions are accurately modeled. For more information on MF-Tyre, please visit http://www.tno.nl.
	-	MF-Swift is the high frequency (100Hz) extension to MF-Tyre. MF-Swift adds generic 3D obstacle enveloping and tire belt dynamics to MF-Tyre's tire-road contact force and moment calculations. Typical application areas for MF-Swift are vehicle comfort analysis, suspension vibration analysis, development of vehicle control systems, such as ABS or ESP, and handling and stability analysis, like braking and power-off in a turn on an uneven road. For more information on MF-Swift, please visit http://www.tno.nl.
	-	To obtain a license for these tires, please contact tno@tno.com or your Altair account manager.
AeroDyn libraries now certified to work with MotionSolve	-220	MotionSolve now supports the AeroDyn libraries for aero-elastic analysis of wind turbines. The National Renewable Energy Laboratories (NREL) originally developed AeroDyn, an aerodynamics software library for use by designers of horizontal-axis wind turbines. The aerodynamics model in AeroDyn is a detailed analysis that includes blade-element/momentum theory (with modifications to improve accuracy in yawed flow), dynamic stall, and an optional dynamic inflow theory. AeroDyn has been tested with models published by NREL, and NREL has certified its use with MotionSolve. AeroDyn is now included in the MotionSolve installation.
A revamped linear analysis module is now available in MotionSolve	-220	Linear analysis in MotionSolve has been significantly enhanced. It now supports all modeling elements. Through a linear analysis, users may examine system level vibration modes or extract Multiple Input-Multiple Output transfer functions as ABCD matrices for any operating point in the analysis.
Assembly improvements	-220	For transient analysis of models containing initial constraint violations, the initial assembly is significantly more robust.
General State Equation enhanced to support DAE	-220	SET_GSE_ALGEBRAIC_EQN is provided as new callable utility subroutines for defining DAE. This extends the range of phenomena that can be modeled with a Control_StateEqn (GSE) entity in MotionSolve.
General State Equation enhanced to support sparse partial derivative matrices	-220	SET_GSE_NONZERO_ENTRY is provided as new callable utility subroutines for defining the non-zero entries of the GSE derivative matrices in a sparse form. This method will yield significant performance improvements when the derivative matrices are sparse, i.e. most of the entries are zero.
MotionSolve now supports CD-Tire 3.0.3	-201	MotionSolve now supports CDTire version 3.0.3 from Fraunhofer LBF. CDTire is suitable for comfort and durability applications, but it can also be used for handling maneuvers. Depending on the application area, there are three basic CDTire sub-models that the user can select: • CDTire 20 – a model with rigid belt suited for handling applications on smooth surfaces • CDTire30 – a flexible belt tire with a brush type contact and

ability to model stick-slip behavior. Suitable for in-plane only obstacle enveloping.

- CDTire40 – a 3D flexible shell carcass with a brush type contact and ability to model stick-slip behavior. Suitable for full obstacle enveloping.

CDTire also offers two transitional sub-models. The sub-model transition is defined in the tire control file.

- CDTire2030 – transitional model between 20 and 30
- CDTire2040 – transitional model between 20 and 40

CDTire needs tire and road property files. The generation of tire property files requires a set of force/moment measurements, as well as topology data, cleat measurements.

CDTire supports various road property files, including digitized road data surfaces and parametrically described road surfaces, as well as user defined road models. The supported road files are categorized, as follows:

- 1000 – parametric road surface description
- 1002 – rolling drum road surface
- 1100 – user defined road
- 2000 – parametric and digitized road data in binary format

For more information about the various road models, please consult the CDTire documentation.

A license to enable CD-Tire can be obtained from your Altair account manager.

MotionSolve now supports RMOD-K Tire 709 tire model

-201

MotionSolve now supports RMOD-K Tire version 7.0.9 from T-Systems. This is a high fidelity tire model that can accurately predict tire response over a wide range of excitation frequencies and driving maneuvers. RMOD-K is typically used for durability, low frequency NVH and ride comfort simulations.

Depending on the scale of the road disturbances, two separate RMOD-K models are available to the user:

- Rigid Belt Model – adequate for excitation frequencies up to 100Hz and ground disturbance wavelengths greater than 100mm
- Flexible Belt Model – adequate for road excitations in excess of 100Hz and for very small ground disturbance wavelengths (down to millimeters). With flexible belt model, the user can select the number of nodes representing the mesh density of the tire carcass structure.

A third model suitable for misuse simulation is available on demand from T-Systems.

RMOD-K needs tire and road property files. The generation of tire property files requires a set of force/moment measurements as well as topology data, cleat measurements.

RMOD-K supports various road property files, including:

- C_ROAD analytical obstacles
- Regular Grid Roads

- Binary format road files representing digitized road surfaces.

RMOD-K is available on Windows platforms and requires a license. To obtain a license for RMOD-K tire, please contact admin@rmod-k.com or your Altair account manager.

The optimization function FMIN_SLSQP is available as a MotionSolve utility subroutine	-201	MotionSolve now supports an optimization algorithm, FMIN_SLSQP as a utility subroutine that may be called from any MotionSolve user subroutine. FMIN_SLSQP is the interface function for the SLSQP optimization function originally implemented by Dieter Kraft. SLSQP is a Sequential, Least SQuares Programming algorithm.
	-	Optimization can be done in MotionSolve through the use of a CONSUB user subroutine together with FMIN_SLSQP. This can be used for model identification or mechanism optimization. The CONSUB may be written in Fortran/C/C++ or Python/Matlab scripts.
Save/Reload functionality now provided as utility functions that can be invoked from user subroutines	-201	SAVE_MODEL and RELOAD_MODEL are provided as new callable utility subroutines. These can be used in conjunction with the utility optimization subroutine FMIN_SLSQP for faster system optimization. It eliminates the need to perform multiple simulations.
ADAMS dlls are now compatible with MotionSolve	-201	Existing user subroutines compiled for ADAMS and available as dlls or shared libraries can now simply be reused with MotionSolve. MotionSolve is able to work with these dlls, provided the publicly documented user-subroutine interface is strictly adhered to. There is no need to recompile source code or re-link the object files.
ADAMS models with flexible bodies can now be directly handled by MotionSolve	-201	ADAMS/Solver models containing flexible bodies and `mtx` files can be translated directly to the MotionSolve `xml` solver input file. If one is not interested in graphical display of the flexible body, there is no need to translate the significantly larger modal neutral file (`mnf`) file to `h3d`.
Performance improvements for rotating bodies	-201	Both the accuracy and performance of MotionSolve have significantly improved for models containing states that have a relatively high rate of rotational speed. Speeds in excess of 3x and improved accuracy have been seen for many full vehicle handling simulations.
Improved contact simulations in MotionSolve	-201	Models containing contact elements will provide smoother results, due to improved handling of discretized geometry.
TCP/IP available for MotionSolve Simulink co-simulation	-201	Co-simulation between MotionSolve and Simulink no longer need to be on the same machine or even on the same process. Co-simulation through TCP/IP sockets is available. This means that the computer hosting MotionSolve can be geographically separated from the computer hosting Simulink. This is easily accomplished by associating IP addresses with the input signals to MotionSolve and Simulink.
H3D Output	-201	Flex-body component and assembly information is now transferred all the way from HyperMesh via RADIOSS and MotionSolve to HyperView. Flex-body component and assembly information is now written to the MotionSolve output H3D file. This enables visibility of flex-bodies to be controlled by component or assembly in HyperView.
	-	Nodal velocities and accelerations are now written to the MotionSolve H3D output file.

Resolved Issues

Deformable surface	-230	The Deformable Surface entity contains a `flip_normal` attribute. The normal for the surface was calculated assuming no penetration to the surface at t=0. This assumption was not always correct and the calculation has been corrected.
Python splines	-230	AKISPL, CUBSPL and BISTOP expressions inside Python user subroutines would return a type of vector instead of scalar; thus, making these expressions unusable. This has been fixed.
H3D animation speed in 11.0	-220	H3D animation performance issues, seen with 11.0 in HyperView for results containing flex-bodies have been resolved.
Run termination	-220	If a MotionSolve run initiated by MotionView was stopped, the MotionSolve process would continue being active, possibly corrupting the result files. This issue has been corrected.
Flex-bodies	-220	Stress/strain results for flex-bodies issue related to unmatched units between RADIOSS and MotionSolve has been resolved.
	-	If the time unit used in the MotionSolve input deck was not seconds, the model may have incorrect transient behavior. This has been corrected.
HTire tire model licensing	-201	The HTire tire model was checking for an FTire license. This has been resolved.

OptiStruct

Altair OptiStruct is a finite element and multi-body dynamics software that can be used to design and optimize structures and mechanical systems. OptiStruct uses the analysis capabilities of RADIOSS and MotionSolve to compute responses for optimization.

New Features

Equivalent Radiated Power (ERP)	-220	Equivalent Radiated Power (ERP) is now available as a response (DRESP1) for all optimization types.
Parallelized Nonlinear Solution	-220	The nonlinear solution can now be run in parallel while performing a nonlinear response optimization.
Optimization History Output	-220	HyperGraph ASCII results file, HGDATA is now output for nonlinear response optimization.

Enhancements

Move Limit Controls	-230	Move limit controls have been implemented for topology optimization with nonlinear responses via the nonlinear geometry solution. This was added to improve robustness of the optimization solution.
Shape Design Variable Output	-230	Shape design variables are output for optimization with heat transfer subcases, so analysis results can be plotted on the updated shapes.
OUTPUT,FSTOSZ	-220	OUTPUT,FSTOSZ has been rewritten using a new pure ply based approach: • Sub-stacks and interfaces are now supported.

- The advanced algorithm is now available for output of 2-, 4- and 8-ply bundles. Previously, only 4-ply bundles were supported through the advanced algorithm.
- The thickness of ply bundles can be multiples of the manufacturable ply thickness.
- Advanced control is available to ignore elements in a given ply orientation when the thickness is less than 5% of the maximum thickness. Additionally, elements are deleted when they do not belong to any of the ply bundles.
- SMCORE is properly handled in the FSTOSZ output.

OUTPUT,PROPERTY	-220	OUTPUT,PROPERTY has been enhanced to support “ANY” and “DESIGN” as arguments. OUTPUT,PROPERTY,ANY specifies that all properties should be output (including non-design properties), while OUTPUT,PROPERTY,DESIGN specifies only designable properties be output. “ANY” remains the default behavior.
Support for unsupported Block data in the Bulk data input format solution for nonlinear response optimization	-220	The ASSIGN command has been enhanced to support two new file types; viz.; Starter and Engine file types, for nonlinear analysis based on Bulk data input run via the nonlinear geometry solution. These files are specified using ASSIGN,STARTER and ASSIGN,ENGINE control card options. Simple modifications are allowed in the internally generated Block data file, using the content in these files through INSERT and REPLACE. This is useful for advanced nonlinear response optimization studies.
LTID and UTID	-220	LTID and UTID are now supported for DRESP2 and DRESP3 responses on the DCONSTR and DOBJREF cards.
Reported MINMAX/MAXMIN Value	-220	Previously, the reported MINMAX/MAXMIN value was always the value of the first objective function (whether it was retained or not), instead of being the actual minimum/maximum value. The output has been enhanced by looping through the retained objectives and selecting the appropriate minimum/maximum value.
DDVOPT,2	-220	DDVOPT,2 is now automatically activated when SHAPEOPT,2 is used along with discrete design variables.
Improved Error Messaging	-	Improved error message handling in the following cases:
	-230	• OptiStruct appropriately errors out when the normal velocity of grids referenced as responses, cannot be calculated. This may occur due to the grids not being connected to any shell or solid elements, or not being located on the surface of the model.
	-220	• Global Search Option, for unreasonably large amount of potential starting points.
	-220	• When multiple sub-ranges are defined for the same response for DCONSTR or DOBJREF.
	-220	• Checking for manufacturing constraints that cannot be satisfied because they do not involve designable (or existent) plies

Resolved Issues

Non-Design Plies	-220	Non-design plies are now correctly accounted for in the volume fraction calculation.
Penalty for Mass and Inertia Calculations	-220	The penalty is now correctly ignored for mass and inertia calculations in free sizing optimization of shells.
Nodal Thickness	-220	Nodal thicknesses are now accounted for in formulations involving stress/strain constraints.
Element Groups	-220	Element groups (EG#) defining zones for zone-based free-sizing optimization are now correctly read as detailed in the documentation for GROUP, the optional continuation line for zone-based free-sizing optimization, on the DSIZE card.

HyperXtrude

New Features and Enhancements

General

Strain Rate Data in Results	-230	HyperXtrude will print the strain rate table in the output file when the parameter "`WriteStrainRateTable`" is set to "yes". In addition, when enabled via H3D Optional results, HyperXtrude will export strain rate tensor to the H3D file.
User Defined Functions	-230	There are few upgrades to this module. The maximum number of user-defined nodal and element results is now increased to 10. In addition to the primary variable, now all the computed data is sent to the UDF routines. These include rate of deformation tensor, temperature gradients, billet interface, skin interface, and stresses.
Output Filenames	-230	HyperXtrude output file naming convention has changed. Now the current output files will always have the name `<jobname>.<ext>`. If that folder has previously computed results, those files will be renamed.

Metal Extrusion

Tube Extrusion – Glass Pad	-230	HyperXtrude can model glass pad used for lubrication in extrusion of super-alloys. This setup is supported by the Tube Extrusion Wizard in the user interface.
Hensel-Spittel Constitutive Model	-220	Hensel-Spittel model is implemented in HyperXtrude for modeling the work piece. The Hensel-Spittel model describes the flow stress as function of temperature, strain rate, and strain. Including the dependency on strain improves the accuracy of flow stress calculation. Hence, the predicted results are more accurate and agree well with experiments. HyperXtrude now has four built-in constitutive models for work piece: • Sine Hyperbolic Inverse • Metal Forming Power-law • Norton-Hoff • Hensel-Spittel models
Particle Traces using AcuPT	-220	HyperXtrude can now create particle traces using AcuPT, which is

		part of AcuSolve software. In this release, this feature is limited to steady state solution. Generated particle trace data will contain history of velocity, pressure, temperature, and strain for each particle.
Profile Shape Factor	-220	Computes the shape factor and prints this data in an `.out` file.
Indirect Extrusion Results	-220	Indirect extrusion module will now consider the shape factor of the profile in the friction calculations. This enhancement will increase the accuracy of the results.
Tool Deflection Analysis	-201	Support to export pressure loads from the material flow analysis in HMASCII format for use in the Tool Deflection Wizard is implemented.
Bearing Optimization	-201	A built-in bearing profile optimization module with the HyperXtrude solver. This direct and faster approach does not require HyperStudy. This approach groups the bearing control points into design, fixed, and dependent points. By adjusting the design points based on the material flow dynamics, it balances the material flow. Unlike the HyperStudy analysis, you have to only categorize the points; any additional/involved setup is not required.
Indirect Extrusion – Moving/Fixed Die	-201	HyperXtrude will now support both variants of indirect extrusion. In the fixed die scenario, which is the default indirect extrusion, the dummy block and container move at ram speed. In the movie dies case, the die moves at ram speed in the negative extrusion axis.

Bug Fixes

General

Heat Balance Table	-230	Heat balance table printed in the `.out` file will include tool surface BC.
User Defined Functions for Constitutive Models	-220	Minor issues in the UDF hooks for material data have been fixed by rigorously comparing all material models implemented via UDF with built-in material models.
UDF Data Export	-220	Errors in the unit conversion for strain rate and other data in the UDF module has been fixed.
Unit Conversion	-220	Bug fixes to error in unit conversion in different modules.
FEM Export	-201	Errors in exporting loads (missing loads), has been fixed.

Metal Extrusion

Starter Billet	-220	Automatic scaling of variable time step data based on the billet length has been implemented. This will allow the data to be specified based on the meshed length of billet and scaled appropriately in each cycle based on the billet length.
Nose Cone Computation	-220	Bug fixes to creation of data structure in bearing and profile 3D region.
Choked Bearing	-220	Bug fixes to choked bearing computation model.

Polymer Processing

Temperature BCs	-220	Improvements to SUPG implementation to handle insulated BCs for non-hexahedral elements.

HyperForge Solver

New Features and Enhancements

New Solver

-220

This is a new solver and targeted for extrusion related open die forging applications. In this release, this solver can model round-to-round and square-to-round billet forging. This comes with an easy to use interface (for setting up the model).

This interface will be formally released in 11.0-130. Interested users can, however, download this interface and install it on top of HyperXtrude User Profile 11.0-101.