HyperWorks Solvers
12.0.223 Release Notes

RADIOSS 12.0.223 Release Notes

MotionSolve 12.0.223 Release Notes

RADIOSS 12.0.223 Release Notes

This is mainly a bug fix version, including also several enhancements for submodels, airbags and Cut Section Method.

Main enhancements:

- Cut Section Method allows you to reduce the size of your model focusing on the area of interest. The size of the files defining the boundary conditions (SC file) for the “cut model” might become huge depending of the number of nodes stored and the frequency sampling. In case of multiple sections, it is now possible to read/write multiple sections into multiple SC files.

- Contact TYPE7: auto-impact for shells with gap larger than mesh size

- Airbags: user-friendly input for multi-chambers airbags; 4N membrane element with full integration, compatible with total strain formulation and with fabrics material laws.

- SUBMODEL: compatibility with all crash keywords; enhanced support of multiple input versions

- Main fixes in this version: Gravity setting in Implicit

Introduction

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 for Crash and Safety

GENERAL

AMS

- Possible instabilities with AMS have been corrected; WARNING: in v12.0.223, tolerance value must be set to 1E-03
- AMS & /DAMP + Kinematic conditions
- /DT/AMS : cannot keep the target time with /MAT/VOID + tied interface

Cut Section Method

- Read/write multiple sections into multiple SC01 files
- Cut Model cannot find SC01 file

SUBMODEL

- Support of multiple input deck versions below a master deck version: Version inside submodel block (in /BEGIN cards) overwrites //SUBMODEL/submodel_ID/Vxxx
- Model with more than 101 submodels and TRANSFORM definitions fails
- Make all crash keywords compatible with the submodel block.
  - Enable the usage of /SENSOR/TY11, 11 and 12.
  - Implement new airbag cards into //SUBMODEL
  - Option /FAIL/CONNECT is not supported in submodel
  - Compatibility with INTER/TY2
  - Encrypted safety tool model cannot be included inside of //SUBMODEL
  - /GRNOD/BOX not supported correctly into //SUBMODEL
  - FAIL/SPALLING: engine's floating overflow when included in SUBMODEL
  - Error with /MONVOL/AIRBAG1, /PROP/INJECT1 & 2, MAT/GAS in the submodel
  - Enable the usage of /SPH in the submodel option

- Issue applying /TRANSFORM/ROT on a /SUBMODEL having /BOX - /BOX are not correctly rotated
- /TRANSFORM submodels transformations should be applied before the group/surface evaluation

HyperWorks Solvers 12.0.223 Release Notes
- Units of the /RWALL parameters are not transformed.
- Error for the offset of the function ID in /PROP/SPR_PUL
- Errors with /MAT/FOAM_TAB material (LAW70)

PARAMETER
- Starter errors out if templex.exe does not respond within a given delay
- Precision for parameter expression

Engine
- Time step imposed in /DT/INTER/DEL for interface TYPE7, 20, 22 is not respected
- Engine fails at initialization time when using rigid body activation and de-activation by sensor. Affected version: 12.0.210
- /DT/NODA/1 is not allowed
- Accidentally in case of /DT/NODE/CST (not /DT/NODA/CST), /DT is used instead of issuing an error

- RADIOSS Starter memory usage optimization (reduced stacksize usage with large models)
- Get error when use /SECT/CIRCLE and /SECT/200 at the same time
- If SKEW_ID > 214748364 in IMPVEL IMPDISP IMPACC, Starter errors out (ERROR ID: 137 ** ERROR IN SKEW/FRAME REFERENCE)
- /TRANSFORM/MATRIX: transformations using global matrix
- /SURF/PLANE: Possible Starter core dump
- /BOX/RECTA: starter error
- Possible Starter crash when total number of items in nodal and element groups, surfaces and lines is greater than representation capacity of 32 bit integer address variable

CONTACTS
TYPE7:
- Variable gap (Igap=1): wrong estimation of kinematic time-step could lead to loose contact
- Wrong stiffness calculation with /MAT/VOID
- Possible issue with Icurv=2 and convex surfaces

HyperWorks Solvers 12.0.223 Release Notes
Possible crash on Mac OS/X

**TYPE11:** dtmin value not scaled in case of unit change in the starter

**TYPE7:** wrong behavior with Igap=3

**TYPE6:** Hflag=2 was not behaving correctly (nonlinear behavior with constant unloading stiffness)

**TYPE24:** possible NAN on Linux64 with Single Precision version

**TYPE18:** possible Starter segmentation violation in SPMD

**TYPE2:**
- In RADIOSS version previous to v12, the slave nodes projected outside the master segment were not deactivated from interface, even for big projection errors. v12 enforces projection check for all spotflags and deactivates slave nodes in case of error. This change reverts this behavior (except spotflag 25), to avoid result differences in some models compared to previous builds.
- Coating with SH3N TETRA10 elements, degenerate master segments of TYPE2 interface

**AIRBAGS**

- Contact **TYPE23:** FPENMAX parameter resolves “zombie” motion when penetrated nodes are very close to the master surface. Double precision version is recommended.
- QBAT and QEPH compatibility with /XREF + total small strain formulation (Ismstr=11)
- QBAT compatibility with LAW58
- /LAW58 compatibility total small strain formulation (Ismstr=11)
- /MONVOL/COMMU1 input is similar to /MONVOL/AIRBAG1; Grafele porosity option added
- /MONVOL/FVMBAG1 output of number of FV in TH file
- /MODIF/FVMBAG: gives warning and ignores input when several airbags available in model;
- FVMBAG: non-linear terms (CPb and CPC) for initial gas and injectors are generating pressure drop (results are wrong if CPb and/or CPC are different from zero)
- Airbag internal surfaces are ignored by FVM
- /MONVOL/AIRBAG1 numeration of jetting functions wrong
- Possible negative volume in folded CAB with internal sock (with FVM based on TETRA)
- Default value for Pext set to 1atm
HyperWorks Solvers 12.0.223 Release Notes

Proprietary Information of Altair Engineering

- Compatibility of /MONVOL/AIRBAG and SUBMODEL
- EREF, XREF, REFSTA: single precision version starter fails to initialize the reference geometry of a shell component when there are tetra elements available
- /EREF compatibility with TETRA4
- Results of interface type 23 not PARITH/ON
- Possible issue with LEAK/MAT
- Possible issue with MONVOL/FVM1
- Airbag Monitored Volume: no outputs in TH in case of multiple TH output requests.

MATERIALS

- LAW36: input supports more than 10 curves
- LAW59: small strain option (now default in prop TYPE43) prevents time step drop after releasing the spotweld element from its support. The elemental stress is now computed using the initial area, as opposite to the current area
- LAW65: corrected loading and unloading behavior in elastic region. The material was not following the prescribed elastic loading curve and unloading was wrong. For material stability, some membrane viscosity is added by default, depending on shell formulation.
- LAW76 (SAMP): strain rate filtering added
- LAW24, NAN in the Engine, due to unassigned default value for S0; current default value = 5/4
- LAW0 (VOID): possible segmentation violation
- LAW77: relative velocity (VF-VS) for computing the Darcy force instead of the flow velocity.
- LAW38: if small strain is activated (Ismstr=10) strain rate computed is wrong
- LAW42: unstable behavior of Isolid=17+1cpres=1; introduction first internal Icpres=11
- LAW13: possible Starter segmentation fault
- BARLAT: possible Engine crash
- LAW72 (HILL_MMC): wrong damage output & possible core dump

COMPOSITES

- Young’s modulus in direction 33 (E33) is ignored for /MAT/LAW25 CRASURV
- Post treatment of stresses and strain in orthotropic directions for composite shells and bricks, /ANIM/ SHELL/PHI/N
• User failure criteria compatible with composites (law12 and 25) + all laws < 28 and law 49 (solid). Also for law shell < 28 + law 32

FAILURES
• /FAIL/WILKINS failure criteria delivers wrong results when W2 is used (beta <> zero)
• /FAIL/TAB: Wrong results as scaling factors and strain rate values were not read correctly
• Failure not working for 4nodes and 3nodes shells with Iform=1. Version(s) concerned: 13.0, 12.0
• /FAIL/LAD_DAMA_ 13 and 23 modes are not working
• XFEM: not PARITH/ON in SPMD

PROPERTIES
• Variable friction in /PROP/TYP12 pulley spring
• /PROP/KJOIN (TYPE33) doesn't work if used without /RBODY at both ends of the spring
• /PROP/KJOIN2 (TYPE45)
  - Friction doesn't work properly if property is used without stopping displacement
  - Possible starter crash
• Spring TYPE4 with H=7: spring does not follow the prescribed force/displacement curve
• Starter fails with single /XELEM element model
• /XELEM doesn't work correctly with SMP version
• Instability with shell elements in LAW25
• /PROP/SPR_AXI rupture inactive for Ifail2=2
• Starter crash with Isolid=14+Ntp=222 for tetra10
• Shell with drilling dof correction
• Change default dn=0.01 for QBAT when npt=1, and optimization

MULTI-DOMAIN
• /INTER/TYP2 with SPOTFLAG = 25 gives very small time step for the subdomain
• Possible Starter crash with TETRA10 elements
• Compatibility with cylindrical joints
• Compatibility with /SECT
• Output improvement - consistency of /TH and /ANIM generated by full and subdomains
• Automatic consistency check between /SUBDOMAIN in Starter and /RAD2RAD/ON in Engine

**ALE**

• LAW51: Possible Engine failure at cycle 0 on Windows. Issue occurring when using plasticity models with MM-ALE LAW51.
• LAW51: no strain rate dependency with JCOOK criteria
• LAW51: outlet does not work properly with explosives
• LAW51: 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 the previous one)
• /INIVOL: possible failure on windows
• Drucker-Prager material law 10 and 21. Artificial Viscosity was not taken into account while computing energy integration from EOS numerical solving. Might affect numerical result unless if user set artificial viscosity to $0 : qa qb = 1e^{-20}$
• LAW10, 21, 51: wrong Drucker-Prager behavior in tension: 1- Yield surface was symmetrized in tensile domain, while there shouldn’t be any plasticity in tension. 2- If Yield surface has no root, user is warned. This might affect numerical results since Yield surface was fixed
• LAW20: detonation wave velocity 10% faster than expected results. ALE Rezoning issue was fixed. It was computed only if second submaterial in input card required it.
• SPH: possible segmentation violation if mass is not defined; error message (ID=138) is introduced for an easy localization of the issue

**IMPLICIT**

• Gravity setting: Removing automatically from the impact candidates list the node/segment pairs (TYPE7, TYPE24) which have been defined in TYPE2 already
• /PROP/SPR** (TYPE4, TYPE8, TYPE12, TYPE13): Coefficient for strain rate effect in tension "A" does not work with nonlinear spring with implicit solution
• NaN implicit results with Nonlinear springs

HyperWorks Solvers 12.0.223 Release Notes
• Add error out if implicit ISOLV=2 and EXEC SMP
• LAW28 with /PROP/TYP=6 does not converge: issue with solid elements and orthotropic laws
• Not initial zero velocity values in some case (if RADIOSS add the one more Anim file at the end of run), if the restart run followed a static implicit. This might the effect on the restart run with the high initial velocity.
• Avoid incorrect implicit result with very small Tol (1.0E-12) input
• Implicit NL converging issue using shell composite with offset (asymmetric positions) (QBAT+T3)
• Slow convergence of nonlinear implicit using QEPH with LAW27
• Zero dof of implicit crashed in spmd (np>1) using Mumps.
• Run crashed with implicit+Rbe2+contact
• Compatibility implicit with LAW12, 14 elastic isotropic used for LAW12, wrong modular matrix for LAW14
• RADIOSS crashes: if contact with T_start(int7)=T_stop(run)

**INITIAL STATE**

• Possible engine crash with /MAT/LAW01 + /STATE/SHELL/AUX/FULL
• Possibility to have values in degree for /INISHE/ORTH_LOC
• Initialization of orthotropy with the /INISHEL_ORTHO_LOC card: some elements were deleted at the beginning of calculation in the engine.
• TETRA10 compatibility with .sta files
• Due to large time steps (AMS is used) .sta file is not written at the end of forming
• Possibility write tensors in global coordinate system in card /INIBRI
• Strain output in .sta file correction
• Wrong Isolid flag in INIBRI cards for prop SOLID, SOL_ORTH
• When mapping results from a previous simulation using /INISHE/STRS_F the thickness was not read by the starter, and not taken into account in the next simulation.
• RADIOSS is not able to read INIBRI/AUX it writes itself for prop SOL_ORTH and isolid=17
• /INIBRI/ORTHO lets crash the STARTER in V12.0.210 and newer V12_main

**OUTPUT**

Anim

HyperWorks Solvers 12.0.223 Release Notes
● External force in ANIM file are wrong
● Wrong output /ANIM/SHELL/USR - PROP/TYPEx6 + QBAT and in INSHE/ORTH_LOC
● Stress and strain output for orthotropic solid/thickshell elements
● /ANIM/SHELL/FAIL does not work with stack and plies (prop 17 and 19)
● Forces of interface type2 to animation files
● LAW58, wrong output of stress tensor in animation file
● Engine crashes with /ANIM/TENS/STRESS/ALL
● Mat LAW25 + PID9 - /ANIM/SHELL/TENS/STRESS/UPPER and ../2 and ../3 don’t work
● /LOAD/PFLUID force is wrong
● Anim stress out wrong with HSEPH+PID22

TH
● New output for elements groups
● Wrong output in ABF file
● /TFILE/3, /ATFILE/4 wrong interaction between binary and ascii output
● /TH/GAUGE fails in starter (affected version: 12.0.210)
● Wrong Energy balance with RBE2

MESSAGES
● Change the error message for a missing subsets into a warning message for
  - /GR..../....
  - /SURF
  - Entities like groups and surfaces not defined in the model are ignored, allowing to run a generic load case where some entities might be missing in a specific model.
● Engine error messages - statistics on nodes and parts which are possibly the source of divergence
● Starter estimates wrong time step for Ishel=12 (Engine is correct)
● /TRANSFORM/ROT card, no error message when only 1 node defining rotation axis is prescribed
● Output message in /PROP/SPH: skew_ID is not correct.
● /SUBSET: change the error message for missing subsets to warning message.
• Starter writes wrong pressure values to 0000.out file
• Error trapped message after a warning for incompatible value for Ish3n
• WARNING ID 432 in STARTER reports wrong property ID
• TYPE2 + Ignore (too many WARNING ID 147 and 870
• Wrong BCS value "9" instead of 0 or 1 is not detected by starter
• /IOFLAG : Print the part mass and inertia any time (with Ipri=0)
• Engine output for part and node PID with highest KE, IE, velocity is wrong
• Message error of INTER/TYPE2 + Spotflag=25 w/ implicit
• Out file 8 digit function IDs
• Improve error messages for RADIOSS to give more specific information (title of the option in warning/error messages)
• /INTER/TYPE19: wrong error message
• Incorrect information on Gravity function ID in 0.out file vs. 0.rad
• Warning ID 147 is missing in starter's summary
• INFO 864: non correct message display
• Wrong error and warning information for interface TYPE21
• Engine should warn about a non-existing node group used in /DT/NODA
• In /SPHGLO output message, LVOISPH and KVISPH should be changed in the code to be consistent for the doc
• /INTER/TYPE5: add the node number on the initial penetration message
• ERROR ID: 727 - ERROR IN MEMORY ALLOCATION
• /PROP/TYPE28 - wrong parameter printout in starter output file

12.0.222 Release Notes

This is a bug fix version, based on 12.0.221, including few fixes for airbags, contact interface TYPE7 and TYPE2, material LAW65. Bugs fixed in this version:

• FVMBAG: Possible segmentation fault
• Contact TYPE7: possible issue with initial penetration computation
• Tied contact TYPE2: slave nodes are abnormally removed from contact
• LAW65: material does not follow prescribed loading/unloading curve
12.0.221 Release Notes

This is a bug fix version, based on 12.0.210, which resolves several issues concerning //SUBMODEL, airbags, TYPE6 contact and others. Bugs fixed in this version:

- //SUBMODEL
  - Compatibility with: /SKET/NOE, /MONVOL/AIRBAG1, /PROP/INJECT1 & 2, MAT/GAS
  - Offset not working for function_ID in /PROP/SPR_PUL
- Airbags: incorrect TH output for airbag venting
- FVM: possible engine failure with NaN
- Contact TYPE6: formulation flag=2 is not taken into account
- Section definition: SECT/PARAL: element not found if paral plane X=cst is defined
- /INIBRI/AUX for tet10 elements is not working
- Printout: /IOFLAG : Print the part mass and inertia any time (with Ipri=0)

12.0.210 Release Notes

RADIOSS for Crash and Safety

AIRBAGS – Contact TYPE23
Penalty formulation specifically designed for airbags fabric self-contact. Initial penetrations and intersections are allowed and resolved automatically during the airbag inflation.

AIRBAGS – Lost Heat Flow
Heat transfer coefficient allows you to compute lost heat flow due to convection.

AIRBAGS – UP
- /EREF: Elements based reference state
- /MONVOL/COMMU1: for multi-chamber airbags, injector properties can be defined referring to /PROP/INJECT and gas mixtures referring to /MAT/GAS
- /LEAK/MAT: leakage models (Nporsurf in /MONVOL/AIRBAG1)
AIRBAGS – FVM
- Injection can be defined on volume internal surfaces also
- /DT/FVMBAG time step stability control for FVM to reduce time step drop
- /FVMBAG/MODIF to redefine along the engine phase the merging criteria and parameters originally defined in the starter which is particularly useful for airbags with internal surfaces like dual chamber side airbags
- /LEAK/MAT: leakage models (Nporsurf in /MONVOL/FVMBAG1)

AIRBAGS – Reference state based on elements
- New input for reference state, based on elements IDs.

Fabric - MATERIAL LAW19
Possibility to define a porosity for an internal surface in FVM application

CONTACT TYPE24
Penalty formulation contact designed to work with large time-step. Main features are: large time-step (it can't drop more than a factor 1.4x); zero gap; improved(*) robustness even with models with initial penetrations and intersections.
Known limitations in 12.0.210 patch:
- No advanced treatment of initial penetrations
- MPI parallelization is available but not fully optimized yet: load unbalancing might deteriorate scalability on large number of cores
- Not available for single precision version

(*) vs. TYPE7

CONTACT TYPE7 – Self Contact with Gap Larger than Mesh Size
- Resolves self contact when mesh size is smaller than gap value; neighbors nodes to a master segment are automatically removed from the list of contact candidates
- Icurve: now available for both concave and convex master surfaces

CONTACT TYPE11 – Friction with Stiffness Formulation
- Incremental (stiffness) formulation allows larger time step and improves stability at the same time

CONTACT TYPE2 - Flag Ignore=3
HyperWorks Solvers 12.0.223 Release Notes
New flag for search distance computation for tied contact.

**Multi-Domain – MPI Parallelization of the rad2rad**

- SPMD parallelization of the rad2rad has been achieved showing tremendous speed-up for high number of cores compared to previous SMP rad2rad. It can also run in Hybrid SMP/SPMD mode.
- Kinematic conditions compatibility has been extended to cylindrical joints.
- Several post-processing improvements are also available, especially for TH files to make output consistent between main and subdomain T01 files. Sections are now available for output.

**XFEM (beta)**

12.0.210 includes a brand new and more powerful X-FEM formulation; nevertheless actual implementation has not been fully tested yet. Therefore it should be considered a beta option.

- XFEM technology is available for multilayered 3 and single layer (PID1) shells but not mixed together, yet.
- The format was changed and simplified for a new XFEM formulation compatible with failure criteria /FAIL: Teuler-Butcher /TBUTC; Johnson-Cook /JOHNS; Forming Limit Diagram FLD and tabulated /TAB.
- HMPP parallelization is available
- Known limitations:
  - PARITH/ON is not ensured for SPMD parallel mode
  - LEVSET is not plugged yet with the new XFEM formulation

**Foams - MATERIAL LAW77 (beta)**

This is an advanced material law for foams which takes into account the behavior of the air inside the foam cells. The air flow thru the cells affects the foam properties, stiffness, viscosity, etc. Special boundary conditions are provided to limit the cells porosity in case of contact with non-porous parts.

Although comparisons with experiments show good correlation, the implementation in version 12.0.210 shall be considered as a beta: so far, only academic and basic tests have been considered, like porosity tests and simple drop tests.

**Foams - MATERIAL LAW70**

Items: tension curve can be prescribed.

**Composites - MATERIAL LAW25**

HyperWorks Solvers 12.0.223 Release Notes
Viscosity has been identified as a critical property for results accuracy of numerical models with composite materials in crash applications. Viscosity is now available in LAW25. Prony viscosity (/VISC/PRONY) is now compatible with LAW25.

**Composites – FAIL RATIO for Hashin, TSAI-WU and CRASURV**

To improve stability, a shell element can be deleted if a user defined proportion of layers are failed.

**Polymers - MATERIAL LAW76**

This is a semi-analytical material model for polymers; strain-rate dependency has been implemented.

**Resolved Issues**

- Geometrical contact surfaces definition Icurve.
- Wrong direction for alpha values output in .sta files (LAW58 + PROP16).
- /LAW59: nodes belonging to deleted elements still set the time step
- /GRNOD/BOX not supported in //SUBMODEL
- /ANIM/SHELL/PHI returns wrong layer with RADIOSS 12 and 12.0.2
- Moving frame didn't work with nonlinear implicit solution
- /INTER/TYPE2 + Spotflag=25: possible instability when distance between slave node and master segment is large
- /INTER/TYPE2 : slave nodes ignored whatever the Ignore flag.
- /INTER/TYPE7: kinematic time step was computed using always Gapmin, ignoring variable gap options
- Improved stability of elastic material laws 1 and 19 for shells
- /LAW68: strain output in the time history was null
- Improved stability for solid elements with Isolid=17 + Icpre=1 + Ismstr=10:
- /ANIM/BRICK/TENS/STRESS/0j0 + Isolid=15: Stress values were not computed when j = 1, 2 and 4

**RADIOSS for Metal Forming**

**HOT-FORMING - MATERIAL LAW80**

HyperWorks Solvers 12.0.223 Release Notes
• Material law to predict microstructure and hardness after quenching.

**CONTACT TYPE21**

• Improved performance
• Non-uniform nodal temperature can be prescribed
• Thickness defined at /PART level is taken into account

**Resolved Issues**

• LAW78: improved stability
• LAW73: engine stop when extrapolation from tables was giving yield < 0; now Radioss always consider yield >= 0

**RADIOSS for Blast Simulation**

**P-alpha - MATERIAL LAW75**

• Porous material law to take into account the effect of porosity on the equation of states

**MATERIAL LAW51**

• Improved silent boundaries formulation for blast applications

**Post-processing: Material Tracking (LAW37 and LAW51)**

• /ANIM/BRIC/VFRAC - allows to track material evolution; it’s targeted to replace global density contour which was not always adequate.

**Grid Velocity Formulation Parameters**

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

• /ALE/DONEA
• /ALE/DISP
• /ALE/SPRING
• /ALE/ZERO
• /ALE/STANDARD
Resolved Issues

- LAW51: multi-material outlet doesn't work when detonation occurs in air

RADIOSS - General

SENSORS Improvements

- Activation/deactivation of CONTACTS
- Possibility to stop a job - Allows tool positioning

Single Precision – Solid Elements

Improved accuracy of single precision with solid elements.

TRANSFORM MATRIX

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

Starter Warnings and Error Messages

Title of the option is printed in the out file.

Starter errors out in case:

- an empty group is detected
- a vent hole surface is not defined

Engine Error Messages

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

/DAMP/INTER

Sensitivity analysis

/SURF/GRBRIC/EXT & /SURF/GRBRIC/FREE

HyperWorks Solvers 12.0.223 Release Notes
Extracts the external (or free) surface of a group of solid elements

**Integrated Beams**

In `/PROP/TYP`e18 it is possible to refer to predefined sections (rectangular or circular)

**AMS**

Performance improvement for `/DT/INTER/AMS`

Compatibility between `/DT/AMS` and `/DT/NODA/CST` allowing the application of AMS only to a group of parts and a classical time step control to the remaining parts of a model for optimized global performances.

**Resolved Issues**

- Domain decomposition fails if `NUMSPH*KVOISPH > 2exp31` for large SPH models
- Running in SPMD a first job with `/DT/NODA/CST` and then switching to AMS caused engine failure
- AMS and Penta6 elements caused Engine failure
- Incorrect orthotropy results using `/INIBRI/ORTHO`
- `/INISHE/ORTH_LOC`: values in degrees were not available
- `/INIVEL/AXIS` and `/INIVEL/TRA` incompatibility
- RBE3 with many independent nodes crashes
- RBE3 brings added mass from independent nodes to dependent ones
- `/NLIMPL+` moving skew caused an error for the implicit solution

**12.0.202 Release Notes**

**RADIOSS for Noise and Vibrations (OptiStruct)**

**PARAM,DISJOINT for AMLS Solver Interface**

`PARAM,DISJOINT` can now be used to allow AMLS to deal with disjoint structures. This only works with version of AMLS that are 4.2r22 or newer. This `PARAM` must be set to a value that is one larger than the number of disjoint parts. For older versions of AMLS, `PARAM,AMLSUCON` can be used.

**New Parameters**

- **PARAM,DISJOINT**: Setting this parameter to one larger than the number of disconnected parts allows new versions of AMLS (4.2r22 or newer) to
solve problems with disconnected parts. The value must be set to one larger than the number of disconnected parts.

**New Subcase Information Entries**
- **RADSND**: Used to specify RADSND bulk data for radiated sound calculations.

**New I/O Option Entries**
- **SINTENS**: Used to request radiated sound intensity to the .h3d file.
- **SPOWER**: Used to request radiated sound power to the .h3d file.
- **SPL**: Used to request radiated sound pressure to the .h3d file.

**New Bulk Data Entries**
- **RADSND**: Used to specify panels and microphone locations for radiated sound calculations.

---

**12.0.201 Release Notes**

**RADIOSS for Stiffness, Strength, Stability (OptiStruct)**

**Geometric Nonlinear Solutions**
The following enhancements have been made for the geometric nonlinear solutions, i.e. the NLGEOM, IMPDYN and EXPDYN analysis types.
- The Force-Deflection Curve from Snap Through analysis can be generated using PARAM,SNAPTHRU,YES.

**New Parameters**
- **PARAM,SNAPTHRU**: PARAM,SNAPTHRU,YES is used to control the generation of a Force-Deflection curve from Snap Through analysis. The default is NO.

**Enhanced Subcase Information Entries**
- **TSTRU**: In addition to using the resulting thermal field from a steady state heat transfer analysis for a thermal load on a structural analysis, the thermal field from the last time step of a transient heat transfer analysis can be used for the thermal load.
RADIOSS for Noise and Vibrations (OptiStruct)

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

A file to allow optimization of CMS Super Elements can be generated
Using the DMIGIDV continuation of the CMSMETH data, the contents of an include file used for optimization of the Super Element modal frequencies and their corresponding structural viscous damping values is specified.

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

Enhanced Bulk Data Entries
- CMSMETH: Using the DMIGIDV continuation of the CMSMETH data, the contents of an include file used for optimization of the Super Element modal frequencies and their corresponding structural viscous damping values is specified.

RADIOSS for Kinematics and Dynamics (OptiStruct)

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

RADIOSS for Crash and Safety (Block Format)
HyperWorks Solvers 12.0.223 Release Notes
Airbags

Several enhancements to improve results quality and solution robustness:

1. Injectors can be positioned on internal airbag surfaces
2. /MONVOL/COMMU1 is supported; similarly to /MONVOL/COMMU:
   - Gas materials can be prescribed in separate /MAT/GAS cards
   - Injectors can be prescribed in separate /PROP/ for injectors

In addition to the above, the option ACOM (t) allows you to scale the area between communicating chambers as function of time or relative pressure.
MotionSolve 12.0.223 Release Notes

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

This document describes changes that have occurred to MotionSolve since version 12.0. All of these improvements are available in the 12.0.223 release.

12.0.223 Update

Models using DSTIFF integrator with index 2
With this hot fix release, for any models using the DSTIFF integrator with the index set to 2 (SI2), the solver will instead use the DSTIFF integrator with index set to 1 (SI1). Velocity states are not checked for integration error (dae_vel_ctrl = “FALSE”) in this case.

12.0.221 Resolved Issues

Default option for output plot file *.plt
The default option for the attribute “plt_file” in the command statement <ResOutput> is changed to TRUE.

That is, MotionSolve will now always write the ASCII result file *.plt, unless explicitly specified. If the model does not have any requests, the *.plt will not be written.

12.0.220 Release Notes

Support for DSTIFF Integrator Index 2 (SI2) Removed
Starting with this release, the DSTIFF integrator will no longer support index 2 (SI2) as an option. You may choose between 1 (SI1) and 3 (I3) for the DAE index.

HyperWorks Solvers 12.0.223 Release Notes
Models that made use of the stabilized index 2 will be setup to solve using stabilized index 1 within the integrator, with relaxed error tolerance on acceleration related constraint equations.

### Change in Static Solver Parameters

Starting with this release, MotionSolve no longer supports the attributes TRANS_LIMIT and ROT_LIMIT that were used to specify the maximum translational and rotational perturbations that the static solver is allowed to make to the model configuration in order to find a static equilibrium.

In their place, MotionSolve now uses a line search algorithm to find the optimal perturbation step.

### 12.0.220 Solver Enhancements

#### Improved Workflow for Co-simulation Using Simulink Coder™

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

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

#### Faster Simulation Times While Using SI1

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

#### Enhancements to Assembly Analysis

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

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

#### Kinetic Energy Distribution for Linear Analyses

MotionSolve now writes out the modal kinetic energy distribution for Linear Analyses. This information is available both in the solver log file as well as in the *._linz.mrf output file that HyperWorks Solvers 12.0.223 Release Notes
is generated at the end of the analysis. You can set the write_energy_dist attribute to TRUE in Param_Linear to enable this feature.

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

**Enhancements to Point to Deformable Surface (PTdSF) Model Element**

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

- The first change improves the robustness of models containing PTdSF elements. It was observed that in the corrector iterations, the surface parameters U and/or V could go out of the range defined for the surface and the simulation would stop. In the enhanced implementation, if the U or V parameters begin to go out of range, MotionSolve prints out a warning and continues with the simulation. The U or V parameters are held fixed at the start or end limits until they come back into range. If the model is well defined, the U or V parameters will come back into range.

- The second change increases the generality of PTdSF. PTdSF SUB – the user subroutine definition for the PTdSF model element has been enhanced to accept SYSFNC and SYSARY calls. This means that the force in a PTdSF element may now be a function of other model states. This is particularly useful for defining complex force models inside the PTdSF SUB. For instance, the coefficient of friction may be calculated using a differential equation. This state dependent value may be subsequently accessed in the PTdSF SUB.

**Enhancements to Animation .h3d**

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

- RIGID elements in the flex h3d are now written out in the animation h3d for better visualization

- Shell thickness information in the flex h3d is transferred to the animation h3d for better visualization

**Enhanced Gravity Modeling**

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

**Python User Subroutine Library**

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

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

- Improved error messages if non-matching parenthesis are detected in any function expression
- More understandable messages when redundant constraints are detected in the model
- More understandable messages when a U or V parameter goes out of range in a PTCV, PTdCV, PTSF, PTdSF etc. modeling element
- More understandable messages when a negative stiffness or damping value is specified for a spring damper or bushing modeling element

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

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

**Visualizing Rigid Body Contact**

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

**Automotive Extensions to the Solver**

**MBD – Vehicle Dynamics Tools**

The MBD – Vehicle Dynamics Tools in MotionSolve and MotionView are a set of modeling entities for bushings, bump-stops, dampers, rebound-stops, springs and tires that extend the capabilities of MotionSolve.

Common to all of these are the following:

- An equal but opposite force is applied on two bodies that these entities are defined between
- The force applied depends on the displacement and velocity of one body relative to the other.

- Outputs specific to each entity are supported, for example tire lateral slip angle for the tire entity.

- The parameters (for example stiffness), used to compute the force are stored in a property file that is independent of the MotionView MDL file and the MotionSolve input deck. The property files are:
  - Text files and may be viewed and edited using any common text editor
  - Compatible with Adams/Car™ – property files from Adams/Car™ may be reused with the Auto Entities in MotionView
  - Read by MotionSolve just prior to analysis.

The table below gives more information about these entities.

<table>
<thead>
<tr>
<th>Entity</th>
<th>Function</th>
<th>Features</th>
<th>Uses</th>
</tr>
</thead>
</table>
| Bushing (autoBush) | Elastically mount suspension arms, links, sub-frames, engines, gearboxes, etc. to vehicle chassis | • Exerts force and torque all directions.  
• Non-linear force/torque verses deflection interpolated using Akima’s method from tabular data  
• Linear damping | • Suspension kinematic and compliance analysis  
• Vehicle Dynamics |
| Bump-stop (autoBumpstop) | Limit suspension bump or jounce travel | • Exerts a force along a line to repel two bodies when the distance between the bodies is less than a threshold  
• Non-linear force verses deflection interpolated using Akima’s method from tabular data  
• Linear Damping | • Vehicle Dynamics,  
• Durability |
| Damper (autoDamper) | Dissipate energy stored in springs and damp chassis motions. | • Exerts a force along a line between two bodies  
• Non-linear force verses velocity interpolated using Akima’s method from tabular data  
• Linear Damping | • Vehicle Dynamics  
• Ride Comfort  
• Durability |
| Rebound-stop (autoReboundstop) | Limit suspension rebound travel | • Exerts a force along a line to attract two bodies with the distance between the bodies exceeds a threshold  
• Non-linear force verses deflection interpolated using Akima’s methods from tabular data  
• Linear Damping | • Vehicle Dynamics  
• Durability |
| Spring (autoSpring) | Limit forces in suspension components | • Exerts a force along a line between two bodies. | • Suspension Kinematics and |
and isolate chassis by absorbing energy from vertical road inputs

- Non-linear force verses deflection interpolated using Akima's methods from tabular data
- Optionally set the initial load or length of the spring at vehicle input position

Compliance
- Vehicle Dynamics
- Ride Comfort
- Durability

<table>
<thead>
<tr>
<th>Tire – CTI (autoTireCTI)</th>
<th>Generate forces for steering, accelerating, and braking. Isolate the vehicle from small road irregularities</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>• COSIN FTIRE (flexible ring) and Pacejka 2000 Magic Formula tire models</td>
</tr>
<tr>
<td></td>
<td>• 3D and 2D Road Models including Curved Gridded Road (CRG) and COSIN RGR.</td>
</tr>
<tr>
<td></td>
<td>• FTIRE: Ride Comfort &amp; Durability</td>
</tr>
<tr>
<td></td>
<td>• Pacejka 2000: – Vehicle Dynamics</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tire – CD (autoTireCD)</th>
<th>Generate forces for steering, accelerating, and braking. Isolate the vehicle from small road irregularities</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>• Supports family of CD-Tire models (flexible ring)</td>
</tr>
<tr>
<td></td>
<td>• 2D and 3D Road Models including CRG</td>
</tr>
<tr>
<td></td>
<td>• Vehicle Dynamics, Ride Comfort, Durability</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Tire – TNO (autoTireTNO)</th>
<th>Generate forces for steering, accelerating, and braking. Isolate the vehicle from small road irregularities</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>• MF-TYRE</td>
</tr>
<tr>
<td></td>
<td>• SWIFT-TYRE (rigid-ring)</td>
</tr>
<tr>
<td></td>
<td>• 2D and 3D Road Models, including CRG</td>
</tr>
<tr>
<td></td>
<td>• MF-Tyre: Vehicle Dynamics</td>
</tr>
<tr>
<td></td>
<td>• SWIFT-Tyre: Vehicle Dynamics &amp; Ride Comfort</td>
</tr>
</tbody>
</table>

More information on these entities can be found here.

ADAMS Compatibility

Additional Support for Model Units
More options for choosing the model units are now available. Some of these include MegaNewton, Poundal, US Ton, Yard, Nanosecond, Day etc.

Modal Force Translation
This release contains a fix for translating the MFORCE element from an adm/acf deck when the SCALE attribute is defined as a function expression.

Joint Friction Model Definition
Some attributes within the FRICTION definition in an adm/acf deck were wrongly translated to the MotionSolve XML input deck. This has been resolved.
Resolved Issues

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

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

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

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

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

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

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