

Version 971 R4.2 offers the following new features and enhancements not offered in the previous release R3.2.1.

The items in this list are presented in no particular order and the list is not completely comprehensive. Certainly, not every minor bug fix and small improvement is mentioned.

Loading and Boundary conditions

Steady state rolling analysis is a generalization of *LOAD_BODY, allowing the user to apply body loads to part sets due to translational and rotational accelerations in a manner that is more general than the *LOAD_BODY capability. This capability is useful for initializing the stresses and velocity of tires during dynamic relaxation, and rolling processes in manufacturing. Still undergoing development. New keywords:

- *LOAD_STEADY_STATE_ROLLING
- *CONTROL_STEADY_STATE_ROLLING

Added empirical pressure loading module for treating multiple blast sources. Also now possible to define the location of the blast using a node ID. Also, a death time for blast can be specified. (*LOAD_BLAST_ENHANCED)

Added airburst blast loading which takes into account ground reflected waves and formation of a Mach stem wave. (*LOAD_BLAST_ENHANCED)

Added blast loading from missile delivered warhead (shaped charge or explosively forged projectile) which produces a non-spherical blast front. (*LOAD_BLAST_ENHANCED)

Added binary database for visualizing blast pressures applied to structures. (*DATABASE_BINARY_BLSFOR)

Added orientation of a rigid body by specifying the time history of components of a body fixed vector. The remaining free rotational degree of freedom (rotation about the vector axis) can be also be specified as either zero, constant or a function of time. (*BOUNDARY_PRESCRIBED_ORIENTATION_RIGID)

Added optional death time for *BOUNDARY_PRESCRIBED_ORIENTATION_RIGID.

Added new options for *DEFINE_CURVE_FUNCTION:

- ELHIST: monitors elemental quantities such as stress and strain
- RCFORC: monitors contact interface forces
- Possible to prescribe the acceleration of a rigid body as function of model response.

Add Birth and Death time to *BOUNDARY_SPC option.

Added option to *LOAD_MOVING_PRESSURE to scale the pressure based on the distance between the nozzle and the surface.

Pore fluid treatment can be added to any LS-DYNA material. The fluid pressure is calculated independently of the material model and added to the total stress. (*boundary_pore_fluid)

Material

*MAT_ADD_EROSION can now be applied to all nonlinear shell, thick shell, and fully integrated solids

Fixed *MAT_ADD_EROSION to work properly with multiple failure criteria.

Changes to *MAT_077 (*MAT_HYPERELASTIC_RUBBER, *MAT_OGDEN_RUBBER):

- Enable to work with tetrahedron formulation 13.
- Added reference geometry option (Card 1, Column 8).

Enable hourglass control type 6 for solid element material *MAT_123 (*MAT_MODIFIED_LINEAR_PIECEWISE_PLASTICITY).

Added RTCL damage to *MAT_123/*MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY.

Improvements to *MAT_036 and *MAT_133 (*MAT_3-PARAMETER_BARLAT and *MAT_BARLAT_YLD2000):

- Added Chaboche-Roussilier kinematic hardening.
- Added consistent viscoplasticity from tables.

Improvements to *MAT_103 (*MAT_ANISOTROPIC_VISCOPLASTIC):

- Multiphase description, i.e., the material can consist of several phases where each has its own set of constitutive parameters.
- Added consistent viscoplasticity (directly) from tables.
- User damage available (not only failure).

Added viscoplastic model to *MAT_124 (*MAT_PLASTICITY_COMPRESSION_TENSION).

Added option to *MAT_83 (*MAT_FU_CHANG_FOAM) to reload on original loading curve when damage is active.

Changed damping in *MAT_181 (*MAT_SIMPLIFIED_RUBBER/FOAM) foam option to correct nonphysical transverse deformation under compression.

*MAT_100, added beam spot weld failure option -9 which is like 9 except that beams do not fail.

*MAT_171 (*MAT_STEEL_CONCENTRIC_BRACE) – hysteretic algorithm improved to avoid the possibility of a step change of result for small changes of input. New input field EPTCRIT, also, fixed bug in loadcurve option.

*MAT_169 (*MAT_ARUP_ADHESIVE) – added new optional input parameter BTHK (bond thickness – by default, this is taken from the element dimension). Also, fixed bug that would have prevented wedge elements from working correctly. Also added input parameter THKDIR to allow thickness direction to be taken from element topology rather than smallest element dimension.

*MAT_172 (*MAT_CONCRETE_EC2) – less conservative timestep calculation; enable stiffness-method hourglass control; new option for concrete compressive behaviour following Mander's algorithm (TYPE6=6); new input parameter UNLFAC controlling unload stiffness; shear capacity calculations; reinforcement directions may be specified using AOPT inputs.

End-releases for *MAT_191 (*MAT_SEISMIC_BEAM) now tolerant of being connected to constrained nodes.

New *MAT_202 (*MAT_STEEL_EC3) for use in fire analysis – integrated beam elements only.

Modifications for *MAT_017 (*MAT_ORIENTED_CRACK). Crack propagation to adjacent elements via two new input parameters: SOFT and CVELO.

Improvements to *MAT_187 (*MAT_SAMP-1):

- Handling of strain rates and damage.
- Bug fix when used with pentahedron elements.
- New parameter RBCFAC (Card 1, Column 7) is the ratio of yield in biaxial compression vs. yield in uniaxial compression.

Added new option to cohesive *MAT_138 (*MAT_COHESIVE_MIXED_MODE): parameters T and/or S can now be negative. The absolute value then defines a load curve for now be negative. The absolute value then defines a load curve for peak traction(s) vs. element size. This should be helpful in case of coarse meshes.

Read reference geometry flag (REF) for *MAT_183 (*MAT_SIMPLIFIED_RUBBER_WITH_DAMAGE): Card 3, Column 2.

Extension of *MAT_120_JC (*MAT_GURSON_JC): new material parameters KW, BETA, and M are read on new optional Card 7, Columns 1, 2 and 3. KW and BETA are used in Hutchinson model (void growth under shear) and M is for enhanced JC failure (nonlinear damage evolution).

New options for *MAT_054 (*MAT_COMPOSITE_DAMAGE_ENHANCED) for shells:

- New parameter PFL (Card 7, Column 1) defines the percentage of layers which must fail until crashfront is initiated (reduced strengths in neighbor elements).
- Linear damage for transverse shear defined by 3 new input parameters: damage initiation strain, final rupture strain, and transverse shear maximum damage.

Fix for *MAT_089 (*MAT_PLASTICITY_POLYMER), solids were not eroded after failure.

Material parameters defined by load curves in *MAT_006 (*MAT_VISCOELASTIC).

Added *MAT_UHS_STEEL, an advanced material for hot stamping simulations including 5 phases, 4 phase transitions and trip effects. This model is available for both for solids and shells.

Fix for *MAT_133 (*MAT_BARLAT_YLD2000) load curve option.

Fix spall options 1,2, and 3 in *MAT_015 (*MAT_JOHNSON_COOK) for shell elements.

Added rate effects to *MAT_003 (*MAT_PLASTIC_KINEMATIC) for beam elements for the option VP=0.

Fix for *EOS_GASKET related to unloading option.

Fix for *MAT_105 (*MAT_DAMAGE_2) to fail element on plastic strain as documented in the users manual.

Added *MAT_220/*MAT_ORTHOTROPIC_ADVANCED_DAMAGE, for solid elements.

Perfectly matched layer material for absorbing outgoing waves from a computational domain, thus simulating wave propagation in unbounded domains. Available as

- *MAT_230/*MAT_PML_ELASTIC (Elastic material)
- *MAT_230_FLUID/*MAT_PML_FLUID (Elastic fluid material)
- *MAT_237/*MAT_PML_HYSTERETIC (for Biot hysteretic material)
- *MAT_231/*MAT_PML_ACOUSTIC (Acoustic media)

Biot hysteretic material for modelling frequency-independent damping.
(*MAT_232/*MAT_BIOT_HYSTERETIC)

Elements

Added new keyword *ELEMENT_DEATH_TIME to remove elements from the calculation at a given time, valid for solids, shells, tshells and beams.

New fully integrated solid elements to reduce transverse shear stiffness for elements with poor aspect ratio (element types -1 and -2).

Added a 6x6 symmetric nodal mass matrix *ELEMENT_MASS_MATRIX. With this element a nodal point can have different inertia in different directions.

Nonzero through thickness stress for shell elements 2 and 16 when used in contact.

Added thick shell formulation 5 (ELFORM=5 on *SECTION_TSHELL). It has 1 integration point per layer and calls 3D stress update routines. It uses an assumed strain field for accurate bending stiffness and improved behavior with layered anisotropic materials.

Improvements to solids type 3 and 4, extended to treat large strains and rotations

Added capability for a "skew angle" option in slirings for better friction modeling. It requires the input of an orientation node, and a second friction coefficient (or a user defined friction function), and computes friction based on the wrap angle (as seen when looking "down the slirping axis", i.e. the cylinder the belt passes over) and the skew angle (how much the belt twists as it passes over the slirping).

Fix for problem with MCID option on *ELEMENT_SHELL definition.

Added bulk viscosity for beam types 1 and 11 (BTYPE on *CONTROL_BULK_VISCOSITY)

Fix for energy conservation problem with shell type 16 when warping stiffness is active.

Contact

Added segment based (SOFT=2) contact options:

- DEPTH=13. Behavior is similar to DEPTH=3, but it has been tuned to improve energy conservation
- DEPTH=23. Behavior is similar to DEPTH=3, but contact detection uses a new algorithm intended to improve robustness.
- Automatically split quad shell segments into 2 triangular segments.
- include added mass from mass scaling and mass elements in the contact stiffness calculation (*CONTROL_CONTACT, PSTIF=1)
- Report penetrations of shell mid-planes.
- Use a moving average of the current time step in the penalty stiffness calculation rather than the initial time step.

Write contact forces in RCFORC in a local coordinate system (*CONTACT. Card C, CID_RF)

Added friction and energy calculations to *CONTACT_GUIDED_CABLE. Friction is an option in the manual, but was not implemented in the coding.

Added new option to automatic tiebreak contact opt 9/11 (Dycoss): Parameters NFLS and/or SLFS can now be negative. The absolute value then defines a load curve for peak traction(s) vs. segment size. This should be helpful in case of coarse meshes.

Bug fix for thick shells when used with contact entities.

For the *CONTACT_AUTOMATIC_..._TIEBREAK family, write node set of untied nodes in messag file for easy viewing in LS-PrePost (SMP only)

Mapping

Added optional thickness scaling for *INCLUDE_STAMPED_PART (new real parameter on Card 3, Column 8).

Fix for symmetric mapping (*INCLUDE_STAMPED_PART_... with ISYM>0). When mirroring the stress/strain tensor, 2 of the 3 shear stresses change sign. It was changed for all 3 shear stresses before, which was wrong.

Added option to enable a 'mapping only' run by setting INCOUT=4 in *INCLUDE_STAMPED_PART.

ALE

Added coupling of ALE ambient type 5 elements with *LOAD_BLAST_ENHANCED wherein pressure and velocity BC applied to ambient ALE elements are provided by *LOAD_BLAST_ENHANCED.

Added *ALE_AMBIENT_HYDROSTATIC and *INITIAL_HYDROSTATIC_ALE for initializing and prescribing hydrostatic pressure in ALE liquids due to gravity.

Implemented new ALE 2D capabilities and mapping to ALE 3D (*CONTROL_ALE2D)

EFG

Added EFG shell elements, *SECTION_SHELL, elform=41 and 42

Improvements to EFG (Element Free Galerkin):

- Added adaptivity for solid EFG and shell EFG in MPP.
- Implemented an explicit version of stabilized EFG method for 8-noded and 6-noded integration cells.
- Implemented a formulation switch from stabilized method to conventional EFG method controlled by time and other parameters.
- Implemented an EFG fracture formulation for 4-noded integration cell.
- Added MPP for adaptive EFG method for solid formulation and shell formulation.
- Included MPP thermal solver in the adaptive EFG formulation for solids.

SPH

Added a Lagrangian kernel for 3D SPH elements (IFORM=7 and IFORM=8 for the renormalized lagrangian kernel).

Interface

Added an optional _ID to *INTERFACE_COMPONENT_*.

Added command *INTERFACE_COMPONENT_FILE to specify name and format (LSDA is now the default format) of interface database that is written. "z=" on execution line will override this command. Similarly, added command *INTERFACE_LINKING_FILE to specify name of interface database that is read. "l=" on execution line will override the command.

Airbag

Improvements to *AIRBAG_PARTICLE:

- Support of heat convection in CPM to account for heat loss to ambient.
- Performance improvement for CPM method.
- Added option for dynamic particle radius scaling.
- Added ELA support for MAT_FABRIC.
- Added particle-to-structure momentum transfer option

Output

Improve GM filter (*CONTROL_OUTPUT, IACCOP=2) to prevent round off errors in single precision

Added a new database, *DATABASE_ATDOUT for the automatic tiebreak damage option.

Fix for beams with end-releases written to dynain file – the topology written to dynain is now the beam's original topology.

Bug fix for dynain writing of thick shell initial stresses.

Added rigid surface contact output to rcfrc file.

Write dynain file when terminating on cycle number rather than time.

Fix for rigidwall energy calculation when two rigid bodies impact.

Fix incorrect mass reported for seatbelt parts.

Fix for secforc calculation when tet type 4 is used in the meshing.

Added d3hsp output of DTSTIF from Contact Optional Card C for SOFT=1.

Added IOOPT option for *DATABASE_BINARY_INTFOR (Card 3).

Added new keyword *DATABASE_EXTENT_D3PART to control the output of d3part.

*DATABASE_EXTENT_BINARY, 4th optional card, 1st field:

dttdt=0, no action

dttdt=1, dump nodal temperature rate dT/dt to d3plot

Add t=0 state to d3drif database.

Fix for missing internal energy for linear solid element.

Added warning message if Part and nodal rigid body share the same ID's.

*DATABASE_MASSOUT writes a list with all nodal masses

Miscellaneous

Fix for reading of *INITIAL_STRESS_SOLID with EOS history variables.

New large format for *INITIAL_STRESS_SHELL.

Added new keyword *COMMENT.

Added optional _TITLE for *PARAMETER and *PARAMETER_EXPRESSION.

Added part option to dynamic relaxation, IDRFLG=3.

Hardwire contact deathtime to 1.e+20 during dynamic relaxation.

Added damping energy calculation for *DAMPING_FREQUENCY_RANGE

Added SET option to *DAMPING_PART_STIFFNESS and *DAMPING_PART_MASS.

Fix for *PART_MOVE so it works correctly with 10 node tetrahedron elements.

Fix for *PART_MOVE when used with a part defined by PART_COMPOSITE.

Reference surface offsets now work for *PART_COMPOSITE.

Added a new option c=-cputim. The absolute value of cputim will be used to terminate the restart job based on the cpu time of the current run and not the cumulative cpu time. This negative cputim can be input as a command line option or in the first field of *CONTROL_CPU.

Added autojump controls for superplastic forming.

MPP

Hybrid implementation, where SMP is used in SMP cores and MPP between thread

You can run 4 smp with 2 mpp threads, total 8 cores, by:

```
mpirun -np 2 mpp971_hybrid i=input ncpu=-4
```

Groupable contact, define with pfile only : contact { groupable 1 }

Added MPP support for self piercing rivet. (*CONSTRAINED_SPR, *CONSTRAINED_SPR2)

Implicit

Simple one-step, implicit solution for gravity loading of models which may even include contacts and unattached parts is now possible. Available in SMP and MPP.

(*CONTROL_IMPLICIT_FORMING (may be renamed later))

Segment to segment mortar contact for implicit contact analyses.

Added the ability to generate superelement representations of parts using Static Condensation, see *CONTROL_IMPLICIT_STATIC_CONDENSATION.

Extended *CONTROL_IMPLICIT_MODES to include eigenmodes and optionally generate the superelement representation of the part. This enables the LS-DYNA to build the Craig-Bampton linear representation of a part using constraint and eigenmodes.

Superelements constructed by *CONTROL_IMPLICIT_STATIC_CONDENSATION and *CONTROL_IMPLICIT_MODES are written in either the Nastran extended format for DMIG files or a LS-DYNA binary format, which reduces the size of the file. *ELEMENT_DIRECT_MATRIX_INPUT has been extended to read the binary format.

*CONTROL_IMPLICIT_TERMINATION has been extended to allow user control over termination the simulation in ways other than the end time. Energy based tests have been added to the original displacement control.

Enhanced the robustness and speed of the nonlinear solution process for implicit mechanical transient simulation. An area of special focus has been metal forming applications. We have added a contact penetration checking option (see *CONTROL_IMPLICIT_SOLUTION) and a new keyword *CONTROL_IMPLICIT_FORMING.

Consistent mass matrix for hex, tet and shell elements (*CONTROL_IMPLICIT_CONSISTENT_MASS)

For eigenvalue analysis, optional calculate stresses based on the eigenmodes displacements (*CONTROL_IMPLICIT_EIGENVALUE, MSTRES=1)

Thermal

Thermal composite shells available in *PART_COMPOSITE.

Consistent thermal contact algorithm for mechanical contact SOFT=2 and in particular for contact with 6-noded segments from 10-noded tetrahedra.

*CONTROL_CONTACT, 6th card, 3rd field:

- ithcnt < 0: conduction evenly distributed (pre-R4)
- = 0: default set to 1
- = 1: conduction weighted by shape functions, reduced integration
- = 2: conduction weighted by shape functions, full integration

Possibility to mix thick and thin thermal shells within one model.

Added thermal effects for 2D and 3D SPH elements (*MAT_THERMAL_ISOTROPIC).

*CONTROL_THERMAL_SOLVER, 2nd optional card, 8th field:

TSF = thermal speedup factor (default 1.)

This parameter will scale all thermal velocity terms (e.g., heat transfer coefficient) for use with mechanical time scaling (e.g., artificially increasing the punch velocity).

*CONTROL_THERMAL_SOLVER

SOLVER = -1: symmetric direct solver (ACTCOL)

SOLVER = 1: reset to use solver 11

New keyword *LOAD_THERMAL_VARIABLE_BEAM – similar to *LOAD_THERMAL_VARIABLE_SHELL – allows temperatures in beam elements to vary piecewise-linearly over the cross-section.

Echo *MAT_ADD_THERMAL_EXPANSION data into the d3hsp file.

Geotechnical

Staged construction – dormant elements now written to d3plot file as “deleted”, so user can see only the active elements at any time; data for penalty-method tied contacts is now written to the dynain file allowing smooth restarting from that dynain file.

Pore pressure analysis for geotechnical models, allowing modeling of drained or undrained pore water, time-dependent consolidation, or calculation of steady-state pore pressures. New keywords:

- *BOUNDARY_PORE_FLUID
- *BOUNDARY_PWP_option
- *CONTROL_PORE_FLUID
- *DATABASE_PWP_OUTPUT
- *DATABASE_PWP_FLOW
- *MAT_ADD_PERMEABILITY
- *INITIAL_PWP_DEPTH

Nonlinear soil-structure interaction invoked via

- *INTERFACE_SSI
- *BOUNDARY_FREE_FIELD_GROUND_MOTION

Acoustics

A collocation boundary element method is developed based on Burton-Miller formulation, to solve the irregular frequency problem for exterior acoustics. This method is available now for SMP and MPP. (*CONTROL_VIBRO_ACOUSTIC, *LOAD_VIBRO_ACOUSTIC)

Two preconditioners are adopted for boundary element acoustic solvers, thus the solution process is now faster.

Extended Rayleigh acoustic method to MPP.

Programming

Added information about element ID in cohesive usermat routine.

New argument for user friction interface USRFRC: plastic strain at slave node as average value from surrounding solid or shell elements.

Combine user material models with user equations-of-state (subroutine ueos21 - ueos30)