LS-DYNA R7.0.0 (R7.79069) released

Filed under: release note

New version of LS-DYNA is released for all common platforms.

Release notes for Version R7.0.0
Herein are summarized most of the new features, enhancements, and significant corrections made since the previous release, 971 R6.1.1. New features are listed first in no particular order. Enhancements and corrections are then listed by category.

I. LICENSE

- In the license file necessary to run R7.0.0,

1. the program names are LS-DYNA and MPPDYNA, and
2. The OPTION is "REVISION 7".

- Note that we have removed "971" from the program names.
- For example, here's an excerpt from a R7.0.0 license file:

LS-DYNA 12312013
OPTION: REVISION 7
MPPDYNA 12312013
OPTION: REVISION 7

II. DOCUMENTATION

- The R7.0.0 Keyword User's Manual can be downloaded from www.lstc.com/download/manuals.

III. NOTES

- Three solvers (EM, CESE, and ICFD) which are new in this release. Please note that these solvers are currently only supported in a subset of the double precision, R7.0.0 executables; please refer to http://ftp.lstc.com/user/ls-dyna/R7.0.0/README_first (SMP) and http://ftp.lstc.com/user/mpp-dyna/R7.0.0/README (MPP) for details.
- A new volume mesher supporting the ICFD and CESE solvers.
- Other new features, enhancements, and significant corrections not included in the previous LS-DYNA release, version 971 R6.1.1

.

New R7 Electromagnetic (EM) solver

- Keyword family: *EM_
  - The keywords starting with *EM refer to and control the Electromagnetic solver problem set up
- Solver Characteristics:
  - Implicit
  - Double precision
  - Dynamic memory handling
  - SMP and MPP
  - 2D axisymmetric solver / 3D solver
  - Automatic coupling with structural and thermal LS-DYNA solvers
  - FEM for conducting pieces only, no air mesh needed (FEM-BEM system)
  - Solid elements for conductors, shells can be insulators
- Solver Main Features:
  - Eddy Current (a.k.a Induction-Diffusion) solver
  - Induced heating solver
  - Resistive heating solver
  - Imposed tension or current circuits
  - Exterior field
  - Magnetic materials (beta version)
  - Electromagnetic contact
  - EM Equation of states (Conductivity as a function of temperature)
- Solver Applications (Non-exhaustive):
  - Electromagnetic forming
  - Electromagnetic welding
  - Electromagnetic bending
  - Inductive heating
  - Resistive heating
  - Rail-gun

- Ring expansions

.

New R7 Compressible CFD (CESE) solver

- Keyword family: *CESE_
  - The keywords starting with *CESE refer to and control the Compressible CFD solver problem set up
- Solver Characteristics:
  - Explicit
  - Double precision
  - Dynamic memory handling
  - SMP and MPP
  - 3D solver / special case 2D solver and 2D axisymmetric solver
  - Automatic coupling with structural and thermal LS-DYNA solvers
  - Eulerian fixed mesh or moving mesh (Either type input with *ELEMENT_SOLID cards or using *MESHcards)
- Solver Main Features:
  - CESE method (Conservation Element / Solution Element) used for CFD solver
  - Highly accurate shock wave capturing
  - Cavitation model
  - Embedded (immersed) boundary approach or moving (fitting) approach for FSI problems
  - Coupled stochastic fuel spray solver (See *STOCHASTIC keywords)
  - Coupling with chemistry (See *CHEMISTRY keywords) solver
- Solver Applications (Non-exhaustive) :
  - Shock wave capturing
  - Shock/acoustic wave interaction
  - Cavitating flows
  - Conjugate heat transfer problems
  - Many different kinds of stochastic particle flows, e.g, dust, water, fuel.
  - Chemically reacting flows, e.g, detonating flow, supersonic combustion.

.

New R7 Incompressible CFD (ICFD) solver

- Keyword family: *ICFD_
  - The keywords starting with *ICFD refer to and control the incompressible CFD solver problem set up
- Solver Characteristics:
  - Implicit
  - Double precision
  - Dynamic memory handling
  - SMP and MPP
  - 2D solver / 3D solver
  - Makes use of an automatic volume mesh generator for fluid domain (See *MESH keywords)
  - Coupling with structural and thermal LS-DYNA solvers
- Solver Main Features:
  - Incompressible fluid solver
  - Thermal solver for fluids
  - Free Surface flows
  - Two-phase flows
  - Turbulence models
  - Transient or steady-state problems
  - Non-Newtonian fluids
  - Boussinesq model for convection
  - Loose or strong coupling for FSI (Fluid-structure interaction)
  - Exact boundary condition imposition for FSI problems
- Solver Applications (Non-exhaustive) :
  - External aerodynamics for incompressible flows
  - Internal aerodynamics for incompressible flows
  - Sloshing, Slamming and Wave impacts
  - FSI problems

- Conjugate heat transfer problems

.

New R7 Volume Mesh generator

- Keyword family: *MESH_
  - The keywords starting with *MESH refer to and control the tools for the automatic volume mesh generator.
- Mesh Generator Characteristics:
  - Automatic
  - Robust
  - Generic
  - Tetrahedral elements for 3D, Triangles in 2D
  - Closed body fitted mesh (surface mesh) needs to be provided for volume generation
- Mesh Generator Main Features:
  - Automatic remeshing to keep acceptable mesh quality for FSI problems (ICFD only)
  - Adaptive meshing tools (ICFD only)
  - Anisotropic boundary layer mesh
  - Mesh element size control tools
  - Remeshing tools for surface meshes to ensure mesh quality
- Mesh Generator Applications :
  - Used by the Incompressible CFD solver (ICFD).
  - Used by the Compressible CFD solver (CESE).

.

Airbag

- Add new parameter VNTOPT to *AIRBAG_HYBRID, that allows user more control on bag venting area calculation.
- Allow heat convection between environment and CPM bag (*AIRBAG_PARTICLE) bag. Apply proper probability density function to part's temperature created by the particle impact.

- *AIRBAG_PARTICLE + *SENSOR_SWITCH_SHELL_TO_VENT allows user to input load curve to control the venting using choking flow equation to get proper probability function for vents. Therefore, this vent will have the same vent rate as real vent hole.
- Add new option NP2P in *CONTROL_CPM to control the repartition frequency of CPM particles among processors. -- MPP only
- Enhance *AIRBAG_PARTICLE to support a negative friction factor (FRIC or PFRIC) in particle to fabric contact. Particles are thus able to rebound at a trajectory closer to the fabric surface after contact.
- Use heat convection coefficient HCONV and fabric thermal conductivity KP to get correct effective heat transfer coefficient for heat loss calculation in *AIRBAG_PARTICLE. (h_effective = 1/(1/H + shell_thickness/kP)) If KP is not given, H will be used as effective heat transfer coefficient.
- Extend CPM inflator orifice limit from 100 to unlimited (*AIRBAG_PARTICLE).
- Support dm_in_dt and dm_out_dt output to CPM chamber database (*DATABASE_ABSTAT) to allow user to study mass flow rate between multiple chambers.

.

Boundary

- Previously, the number of ships (rigid bodies) in *BOUNDARY_MCOL, as specified by NMCOL, was limited to 2. Apparently, this was because the code had not been validated for more than 2 rigid bodies, but it is believed that it should not be a problem to remove this restriction. Consequently, this limit has been raised to 10, with the caveat that the user should verify the results for NMCOL>2.
- Implemented a structural-acoustic mapping scheme (*BOUNDARY_ACOUSTIC_MAPPING), for mapping transient structural nodal velocity to acoustic volume surface nodes. This is useful if the structure finite element mesh and the acoustic boundary/finite element mesh are mismatched.

.

Control

- *ALE_REFINE has been replaced and expanded upon by the *CONTROL_REFINE family of commands. These commands invoke local mesh refinement of shells, solids, and ALE elements based on various criteria. Shells or solids in a region selected for refinement (parent element) are replaced by 4 shells or 8 solids, respectively.
  - *CONTROL_REFINE_SHELL applies to shells.

- *CONTROL_REFINE_SOLID applies to solids.
- *CONTROL_REFINE_ALE and *CONTROL_REFINE_ALE2D applies to ALE elements.

Each keyword has up to 3 lines of input. If only the 1st card is defined, the refinement occurs during the initialization. The 2nd card defines a criterion CRITRF to automatically refine the elements during the run. If the 3rd card is defined, the refinement can be reversed based on a criterion CRITM. All commands are implemented for MPP.

- *CONTROL_REFINE_MPP_DISTRIBUTION distributes the elements required by the refinement across the MPP processes.
- Fixed bug in *CONTROL_STAGED_CONSTRUCTION whereby staged construction was not working with the combination of Hughes-Liu beams and *CONTROL_ACCURACY.

.

Constrained

- Added keyword *CONSTRAINED_MULTIPLE_GLOBAL for defining multi-node constraints for imposing periodic boundary conditions.
- Enhancement for *CONSTRAINED_INTERPOLATION_SPOTWELD (SPR3): calculation of bending moment is more accurate now.
- If *CONSTRAINED_NODAL_RIGID_BODY nodes are shared by several processors with mass scaling on, the added mass is not summed up across processors. This results in an instability of the NRB. -- MPP only

.

Contact

- *CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURACE_ORTHO_FRICTION c an now be defined by part set IDs when supplemented by *DEFINE_FRICTION_ORIENTATION. Segment sets with orientation per *DEFINE_FRICTION_ORIENTATION are generated automatically.
- Contact force of *CONTACT_ENTITY is now available in intfor (*DATABASE_BINARY_INTFOR).
- Fix for *CONTACT_AUTOMATIC_..._TIEBREAK options 9 and 11: element size dependent failure stresses (NFLS/SFLS<0) were not working correctly in MPP.
- Fix for *CONTACT_AUTOMATIC_..._TIEBREAK options 7 and 10 in MPP: no more unexpected high forces and subsequent instability.

- *CONTACT_FORCE_TRANSDUCER_PENALTY will now accept node sets for both the slave and master sides, which should allow them to work correctly for eroding materials. BOTH sides should use node sets, or neither.
- Added option to create a backup penalty-based contact for a tied constraint-based contact in the input (IPBACK on Card E of *CONTACT).
- New option for *CONTACT_ENTITY. If variable SO is set to 2, then a constraint-like option is used to compute the forces in the normal direction. Friction is treated in the usual way.
- *CONTACT_ENTITY: allow friction coefficient to be given by a "coefficient vs time" load curve (input < 0 -> absolute value is the load curve ID). Also, if the friction coefficient >= 1.0, the node sticks with no sliding at all.
- Fix for MPP implementation of *CONTACT_ORTHO_FRICTION.
- Minor tweak to the way both MPP and SMP handle nodes sliding off the ends of beams in*CONTACT_GUIDED_CABLE.
- Fix MPP groupable tied *CONTACT energy calculations. The contact was working correctly, but the energy calculation was incorrect since the changes of r71470.
- Frictional energy output in sleout (*DATABASE_SLEOUT) supported for *CONTACT_..._MORTAR.
- Tiebreak damage parameter output as "contact gap" in intfor file for*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_MORTAR, OPTION=9.
- Added MPP support for *CONTACT_2D_AUTOMATIC_SINGLE_SURFACE and*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE.

.

Database

- Eliminate automatic writing of a d3plot plot state after each 3D tetrahedral remeshing operation (*CONTROL_REMESHING) to reduce volume of output.
- Generate disbout output (*DATABASE_DISBOUT) for MPP and SMP binout files.
- Extend *DATABASE_MASSOUT to include option to output mass information on rigid body nodes.
- Bug fixes for D3PLOT:
  - Fixed bug in title of d3plot for double precision and LSTC_BINARY=ieee32 format.

- Fixed bug in d3plot if a solid part is removed.
- Fixed bug in d3plot if any part is removed for 10-noded tetrahedral element
- Fixed bug where hiding a part from d3plot removed its effect from the model. A removed part was also removed from contact definition.
- Fixed bug in binary file output (d3plot, d3drlf, ...) if the size of first (geometry) file is bigger than the defined size.
- Fixed improper warning message if any part is removed from d3plot.

- Added new keyword *CHANGE_OUTPUT for full deck restart to override default behavior of overwriting existing ASCII files. For small restart, this option has no effect since all ASCII output is appended to the result of previous run already.
- Added new option (NEWLENGD) to 2nd field of 3rd card of *CONTROL_OUTPUT. to write more detailed legend in ASCII output files. At present, only rcforc and jntforc are implemented.
- Fixed bug in writing dyna.inc (*INCLUDE_TRANSFORM) for *ELEMENT_DISCRETE,*ELEMENT_SOLID, and *ELEMENT_TSHELL.
- Fixed bug in d3part files (*DATABASE_BINARY_D3PART) triggered by *MAT_ADD_EROSION (SMP only).
- Fixed bug in d3part output triggered by MAXINT<0 in *DATABASE_EXTENT_D3PART.
- Fixed bug in ANSYS output, associated with adaptivity.
- Fixed bad report of beam integration rules in d3hsp.
- Fixed bug for encrypted data in adativity run if *KEYWORD line exists in the include files.
- Fixed bug in intfor database (*DATABASE_BINARY_INTFOR) if geometric *CONTACT_ENTITY exists.
- Fixed bug in small restart whereby intfor database is overwritten.
- Fixed bug in status.out file if no sw2. or glstat (*DATABASE_GLSTAT) is requested.
- Fixed bug in Material Model Driver if number of load curveis is less than 9.
- Fixed bug whereby correct IRID of *INTEGRATION_BEAM was not output to d3hsp.
- Increased default binary file size scale factor (x=) from 7 to 1024. That means the default binary file size will be 1 Gb for single version and 2 Gb for double version.
- Add echo of new "max frequency of element failure summaries" flag (FRFREQ in *CONTROL_OUTPUT) to d3hsp file.

- Support LSDA/binout output for new pllyout file (*DATABASE_PLLYOUT, *ELEMENT_BEAM_PULLEY) in both SMP and MPP.

.

Element

- Allow degenerated hexahedrons (pentas) for cohesive solid elements (ELFORM=19, 20) that evolve from an extrusion of triangular shells. The input of nodes on the element cards for such a pentahedron is given by: N1,N2,N3,N3,N4,N5,N6,N6
- Fixed error in reading *ELEMENT_SHELL_COMPOSITE.
- Fixed bug in *SECTION_SHELL for seatbelt where SECID was input as a label.
- Add new option to activate drilling constraint force for shells in explicit calculations. This can be defined by parameters DRCPSID (part set) and DRCPRM (scaling factor) on *CONTROL_SHELL.
- Add SMP ASCII database "pllyout" (*DATABASE_PLLYOUT) for *ELEMENT_BEAM_PULLEY.

.

FREQUENCY_DOMAIN

- *FREQUENCY_DOMAIN_ACOUSTIC_BEM:
  - Added an option to output real part of acoustic pressure in time domain.
  - Enabled BEM acoustic computation following implicit transient analysis.
  - Implemented coupling between steady state dynamics and collocation acoustic BEM.
  - Implemented Acoustic Transfer Vector (ATV) to variational indirect BEM acoustics.
  - Enabled boundary acoustic mapping in BEM acoustics.
- *FREQUENCY_DOMAIN_ACOUSTIC_FEM:
  - Added boundary nodal velocity to binary plot file d3acs.
  - Implemented pentahedron elements in FEM acoustics.
  - Enabled using boundary acoustic mapping in FEM acoustics.
- *FREQUENCY_DOMAIN_FRF:
  - Updated FRF to include output in all directions (VAD2=4).
  - Added treatment for FRF with base acceleration (node id can be 0).

- *FREQUENCY_DOMAIN_RANDOM_VIBRATION:
  - Updated calculating PSD and RMS von Mises stress in random vibration environment, based on the Report of Sandia National Laboratories, 1998.
- *FREQUENCY_DOMAIN_RANDOM_VIBRATION_FATIGUE:
  - Implemented an option to incorporate initial damage ratio in random vibration fatigue.
- *FREQUENCY_DOMAIN_RESPONSE_SPECTRUM:
  - Implemented double sum methods (based on Gupta-Cordero coefficient, modified Gupta-Cordero coefficient, and Rosenblueth-Elorduy coefficient).
  - Updated calculating von Mises stress in response spectrum analysis.
  - Implemented treatment for multi simultaneous input spectra.
  - Improved double sum methods by reducing number of loops.
- *FREQUENCY_DOMAIN_SSD:
  - Added the option to output real and imaginary parts of frequency response to d3ssd.
  - Added the option to output relative displacement, velocity and acceleration in SSD computation in the case of base acceleration. Previously only absolute values were provided.
- Implemented keyword *FREQUENCY_DOMAIN_MODE_{OPTION} so that user can select the vibration modes to be used for frequency response analysis.
- Implemented keyword *SET_MODE_{OPTION} so that user can define a set of vibration modes, to be used for frequency response analysis.
- Implemented keyword *FREQUENCY_DOMAIN_PATH to define the path of binary databases containing mode information, used in restarting frequency domain analysis, e.g. frf, ssd, random vibration.

.

INITIAL

- Compute normal component of impulse for oblique plates in *INITIAL_MINE_IMPULSE. The feature is no longer limited to horizontal plates.
- Disable license security for *INITIAL_IMPULSE_MINE. The feature is no longer restricted.
- Fix a bug of *INITIAL_STRESS_SOLID for 4-noded tetrahedral elements.

- Enabled hourglass type 7 to work well with *INITIAL_FOAM_REFERENCE_GEOMETRY so that initial hourglass energy is properly calculated and foam will spring back to the initial geometry.

.

Load

- Accommodate erosion of thin shells in *LOAD_BLAST_ENHANCHED.
- *LOAD_VOLUME_LOSS has been changed such that after the analysis time exceeds the last point on the curve of volume change fraction versus time, the volume change is no longer enforced.
- *LOAD_BODY_POROUS new option AOPT added to assign porosity values in material coordinate system.
- Added *LOAD_SEGMENT_FILE.

.

Sensor

- Add new sensor definition, *SENSOR_DEFINE_ANGLE. This card traces the angle formed between two lines.
- *SENSOR_DEFINE_NODE can be used to trace the magnitude of nodal values (coordinate, velocity or accleration) when VID is "0" or undefined.
- Add two new parameters to *SENSOR_DEFINE_ELEMENT, scale factor and power, so that user can adjust the element-based sensor values (strain, stress, force...)

.

Material

- Change history variables 10-12 in *MAT_054/*MAT_ENHANCED_COMPOSITE_DAMAGE (thin shells only) to represent strains in material coordinate system rather than in local element coordinate system. This is a lot more helpful for postprocessing issues. This change should not lead to different results other than due to different round-off errors.
- New features and enhancements to *MAT_244/*MAT_UHS_STEEL:
  - Added implicit support for *MAT_244.
  - Changed the influence of the austenite grain size in *MAT_244 according to Li et al.
  - Changed the start temperatures to fully follow WATT et al and Li et al.

- Hardness calculation is now improved when noncontinuous cooling is applied. i.e., tempering.
- Added temperature dependent Poisson ratio and advanced reaction kinetics.
- Added new advanced option to describe the thermal expansion coefficients for each phase.
- Added option to use Curve ID or a Table ID for describing the latent heat generation during phase transormations.
- Added support for table definition for Youngs modulus. Now you can have one temperature dependent curve for each of the 5 phases

- Bugfixes to *MAT_244:
  - Fixed a bug related to thick shell formulation 1.
  - Fixed an error in load curves when using temperature dependent variables (Youngs modulus and poissons ratio).
  - Fixed a problem when using dynain files.
- Added support for implicit to *MAT_188.
- Added material model *MAT_273/*MAT_CDPM/*MAT_CONCRETE_DAMAGE_PLASTIC_MODEL. This model is aimed at simulations where failure of concrete structures subjected to dynamic loadings is sought. The model is based on effective stress plasticity and has a damage model based on both plastic and elastic strain measures. Implemented for solids only but both for explicit and implicit simulations. Using an implicit solution when damage is activated may trigger a slow convergense. IMFLAG = 4 or 5 can be useful.
- Added an option in *MAT_266 (*MAT_TISSUE_DISPERSED) so that the user can tailor the active contribution with a time dependent load curve instead of using the internal hardcoded option. See ACT10 in the User's Manual.
- Fixed a problem related to the shell thickness update (ISTUPD in *CONTROL_SHELL) and *MAT_255.
- *MAT_173/*MAT_MOHR_COULOMB is available in 2D.
- Fix a bug affecting evolution type failure, IDAM of *MAT_ADD_EROSION<0, is applied to shell element of*MAT_015.
- Fix a bug in the general Darcy's law pore air flow calculation triggered when the pore air pressure gradient fell outside the range of the input curve (*MAT_ADD_PORE_AIR).
- Fix a bug in the user-defined failure subroutine matusr_24 (optional for *MAT_024 and others) whereby the failure flag fail(i) was stored incorrectly.

- Enable *MAT_103 and *MAT_104 to discretize the material load curves according to the number of points specified by LCINT in *CONTROL_SOLUTION.
- Implement Prony series up to 18 terms for shells using *MAT_076/*MAT_GENERAL_VISCOELASTIC.
- Added *DEFINE_STOCHASTIC_VARIATION and the STOCHASTIC option for *MAT_010, *MAT_015,*MAT_024, *MAT_081, *MAT_098 for shells, solids, and type 13 tets. This feature defines a stochastic variation in the yield stress and damage/failure of the aforementioned material models.
- Minor modification for *DEFINE_CONNECTION_PROPERTIES:
  - PROPRUL=2: thinner weld partner is first partner,
  - PROPRUL=3: bottom (nodes 1-2-3-4) weld partner is first partner.
- Add spotweld area to debug output of *DEFINE_CONNECTION_PROPERTIES which is activated by*CONTROL_DEBUG.
- Add support of *MAT_ADD_EROSION option NUMFIP<0 for standard (non-GISSMO) failure criteria. Only for shells.
- Improve implicit convergence of *MAT_ADD_EROSION damage model GISSMO by adding damage scaling (1-D) to the tangent stiffness matrix.
- Provide plastic strain rates (tension/compression, shear, biaxial) as history variables no. 16, 17, and 18 for *MAT_187.
- Add new variables to user failure routine matusr_24 (activated by FAIL<0 on *MAT_024 and other materials): integration point numbers and element id.
- Add new energy based, nonlocal failure criterion for *MAT_ADD_EROSION: parameters ENGCRT (critical energy) and RADCRT (critical radius) after EPSTHIN. Total internal energy of elements within a radius RADCRT must exceed ENGCRT for erosion to occur. Intended for windshield impact.
- Add new option to *MAT_054 for thin shells: Load curves for rate dependent strengths and a rate averaging flag can be defined on new optional card 9.
- Add new option for *MAT_MUSCLE: Input parameter SSP<0 can now refer to a load curve (stress vs. stretch ratio) or a table (stress vs. stretch ratio vs. normalized strain rate).
- Expand list of variables for *MAT_USER_DEFINED_MATERIAL_MODELS by characteristic element size and element id.
- Enable *MAT_USER_DEFINED_MATERIAL_MODELS to be used with tetrahedron element type 13. New sample routines "umat41_t13" and "umat41v_t13" show corresponding pressure calculation in the elastic case.

- Fix for *MAT_119 with initial displacements IUR, IUS, IUT, IWR, IWS, IWT which were erroneously applied twice.
- Fix for *MAT_083 with damage/hysteresis (TBID<0 or HU>0). Combination with FMATRX=2 on*CONTROL_SOLID (default in implicit) could lead to uninitialized energies and consequential incorrect unloading behavior.
- Fix for *MAT_104. For FLAG = -1 (anisotropic damage), it was possible that the load curve number of LCSS was changed internally.
- Fix for combination of *INCLUDE_TRANSFORM with FCTTIM.ne.1.0 and material tables with logarithmic strain rates. The transformation of these logarithmic strain rates is correct now.
- Add a new feature to *MAT_125 allowing C1 and C2 to be used in calculation of back stress. When plastic strain < 0.5%, C1 is used, otherwise C2 is used as described in Yoshida's paper.
- Extend non-linear strain path (_NLP_FAILURE) in *MAT_037 to implicit.
- *MAT_173/*MAT_MOHR_COULOMB now works in ALE. A new option has been added to suppress the tensile limit on hydrostatic stress - recommended for ALE multi-material use.
- Upgraded *MAT_172/*MAT_CONCRETE_EC2.
  - Corrections to DEGRAD option.
  - Fix bug affecting shells (not beams). When an element has two cracks, and one crack is under compression while the other is still widening, the element could erroneously shrink in the compression direction. Another symptom of the same bug would be that if two cracks are opening at the same time in the same element, one or both might not follow the expected stress-crack opening curve.
  - Concrete and reinforcement types 7 and 8 have been added to reflect changes to Eurocode 2.
  - Extra history variables for reinforcement stress and strain are now output as zero for zero-fraction reinforcement directions.
- Added RCDC model for solid *MAT_082.
- Added Feng's failure model to solid *MAT_021.
- Added *MAT_027 for beams.
- Added *DEFINE_HAZ_PROPERTIES and *DEFINE_TAILOR_WELDED_BLANK for modifying material behavior near a spot weld.
- Added fourth rate form to viscoplastic Johnson-Cook model (*MAT_015).

- Added option to *MAT_224 to not delete the element if NUMINT=-200.
- New damage initiation option 3 in multi fold damage criteria in *MAT_ADD_EROSION. Very similar to option 2 but insensitive to pressure.
- Rotational resistance in *MAT_034/*MAT_FABRIC. Optionally the user may specify the stiffness, yield and thickness of and elastic-perfectly-plastic coated layer of a fabric that results in a rotational resistance during the simulation.
- FLDNIPF<0 in *MAT_190/*MAT_FLD_3-PARAMETER_BARLAT for shell elements means that failure occurs when all integration points within a relative distance of -FLDNIPF from the mid surface has reached the fld criterion
- A computational welding mechanics *MAT_270/*MAT_CWM material is available that allows for element birth based on a birth temperature as well as annealing based on an annealing temperature. The material is in addition a thermo-elasto-plastic material with kinematic hardening and temperature dependent properties.
- *MAT_271/*MAT_POWDER is a material for manufacturing (i.e., compaction and sintering) of cemented carbides. It is divided into an elastic-plastic compaction model that is supposed to be run in a first phase, and a viscoelastic sintering model that should be run in a second phase. This model is for solid elements.
- For IHYPER=3 on a *MAT_USER_DEFINED_... shell material, the deformation gradient is calculated from the geometry instead of incremented by the velocity gradient. The deformation gradient is also passed to the user defined subroutines in the global system together with a transformation matrix between the global and material frames. This allows for freedom in how to deal with the deformation gradient and its transformations in orthotropic (layered) materials.
- The Bergstrom-Boyce viscoelastic rubber model is now available in explicit and implicit analysis as*MAT_269/*MAT_BERGSTROM_BOYCE_RUBBER. The Arruda-Boyce elastic stress is augmented with a Bergstrom-Boyce viscoelastic stress corresponding to the response of a single entangled chain in a polymer gel matrix.
- Added a new parameter IEVTS to *MAT_USER_DEFINED_MATERIAL_MODELS (*MAT_041-050). IEVTS is optional and is used only by thick shell formulation 5. It points to the position of E(a) in the material constants array. Following E(a), the next 5 material constants must be E(b), E(c), v(ba), v(ca), and v(cb). This data enables thick shell formulation 5 to calculate an accurate thickness strain, otherwise the thickness strain will be based on the elastic constants pointed to by IBULK and IG.
- Implemented enhancements to fabric material (*MAT_034), FORM=14 as follows:
  - Stress-strain curves may include a portion for fibers in compression.

- When unload/reload curves with negative curve ID are input (curve stretch options), the code that finds the intersection point now extrapolates the curves at their end rather than simply printing an error message if an intersection point cannot be found before the last point in either curve.

- Fix bug in output of crack data for *MAT_084. -- MPP only

.

ALE

- Map 1D to 3D by beam-volume averaging the 1D data over the 3D elements (*INITIAL_ALE_MAPPING).
- In a 3D to 3D mapping (*INITIAL_ALE_MAPPING), map the relative displacements for the penalty coupling in *CONSTRAINED_LAGRANGE_IN_SOLID
- The .xy files associated with *DATABASE_ALE_MAT are now created when sense switches sw1, sw2, quit, or stop are issued.
- *ALE_ESSENTIAL_BOUNDARY is available in 2D.
- *DATABASE_FSI is available for 2D (MPP).
- *ALE_ESSENTIAL_BOUNDARY implemented to apply slip-only velocity BC along ALE mesh surface.
- *CONTROL_ALE flag INIJWL=2 option added to balance initial pressure state between ALE Soil and HE.

.

SPH

- Include SPH element (*ELEMENT_SPH) in time step report.
- Time step and internal energy of 2D axisymmetric SPH elements are calculated in a new way more consistent with the viscosity force calculation.
- Only apply viscosity force to x and y components of 2D axisymmetric SPH element, not on hoop component.
- MAXV in *CONTROL_SPH can be defined as a negative number to turn off velocity checking.
- Improve calculation of 2D axisymmetric SPH contact force in *DEFINE_SPH_TO_SPH_COUPLING.
- Added the following material models for SPH particles:
  - *MAT_004/*MAT_ELASTIC_PLASTIC_THERMAL (3D only)

- *MAT_106/*MAT_ELASTIC_VISCOPLASTIC_THERMAL
- Added a new parameter DFACT for *DEFINE_SPH_TO_SPH_COUPLING. DFACT invokes a viscous term to damp the coupling between two SPH parts and thereby reduce the relative velocity between the parts.
- Added *BOUNDARY_CONVECTION and *BOUNDARY_RADIATION for explicit SPH thermal solver.
- Fix bug for SPH symmetric boundary conditions (*CONTROL_SPH).

.

EFG

- *CONTROL_REMESHING_EFG:
  - Add eroding failed surface elements and reconstructing surface in EFG adaptivity.
  - Add a control parameter for monotonic mesh resizing in EFG adaptivity.
  - Add searching and correcting self-penetration for adaptive parts in 3D tetrahedron remeshing.
- Enhance 3D axisymmetric remeshing with 6-node/8-node elements (*CONTROL_REMESHING):
  - Use RMIN/RMAX along with SEGANG to determine element size.
  - Remove the restriction that the reference point of computational model has to be at original point (0,0,0).
  - Rewrite the searching algorithm for identifying the feature lines of cross-sections in order to provide more stable remeshing results.
- Improve rigid body motion in EFG shell type 41.
- Support EFG pressure smoothing in EFG solid type 42 for *MAT_ELASTIC_VISCOPLASTIC_THERMAL.
- Add visco effect for implicit EFG solid type 42
- Add new EFG solid type 43 (called Meshfree-Enriched FEM, MEFEM) for both implicit and explicit. This element formulation is able to relieve the volumetric locking for nearly-incompressible material (eg. rubber) and performs strain smoothing across elements with common faces.
- EFG shell adaptivity no longer requires a special license.
- Application of EFG in an implicit analysis no longer requires a special license.

.

Implicit

- Fix sign for resultant moments for joints using implicit (jntforc).
- Fix numerous problems with stress recovery for mode shapes especially for intermittent eigenvalue analysis.
- Fix output of forces pertaining to prescribed motion (bndout) for implicit.
- Fix *INTERFACE_LINKING problem for SMP implicit.
- Fix implicit output of spcforc for *CONSTRAINED_COORDINATE.
- Fix implicit *BOUNDARY_SPC with birth time and death time.
- Add *SENSOR_CONTROL for prescribed motion constraints in implicit.
- Update *INTERFACE_LINKING_NODE in implicit to catch up with explicit, including adding scaling factors.
- Add support for *DATABASE_RCFORC_MOMENT for implicit.
- Enhance Iterative solvers for Implicit Mechanics.
- Add, after the first implicit time step, the output of projected cpu and wall clock times. This was already in place for explicit. Also echo the termination time.
- Fix an issue with implicit storing the resultant forces for *CONSTRAINED_SPOTWELD.

.

Thermal

- Add variable MXDMP in *CONTROL_THERMAL_SOLVER to write thermal conductance matrix and right-hand side every MXDMP time steps.
- Add keyword *CONTROL_THERMAL_EIGENVALUE to calculate eigenvalue(s) of each thermal conductance matrix.
- New thermal material keyword --> *MAT_THERMAL_ORTHOTROPIC_TD_LC This is an orthtropic material with temperature dependent properties defined by load curves.
- Change in structured file format for control card 27 (first thermal control card). Several input variables used i5 format limiting their value to 99,999. A recent large model exceeded this limit. The format was changed to i10. This change is not backward compatible. Old structured input files will no longer run unless control card 27 is changed to the new i10 format. This change does not affect the KEYWORD file.
- Fix coupled thermal solution for problems using thick thermal shells.

- Fix computation of assembled stiffness matrix for the unsymmetric case. Should only affect coupled thermal/implicit mechanical problems in MPP.
- Thermal material *MAT_T07/*MAT_THERMAL_CWM for welding simulations to be used in conjunction with *MAT_270/*MAT_CWM mechanical counterpart.

.

MPP

- Modify decomposition costs of *MAT_181 and *MAT_183.
- Fix transfer of mes* files at code termination.
- Fix for possible problem in *CONTROL_MPP_DECOMPOSTION_ARRANGE_PARTS.
- Fix for MPP *CONTACT_GUIDED_CABLE in case some processor has no slave nodes.
- Fixes to MPP reporting of element controlling the time step.
- Introduction of new timing routines and summary at termination.
- Echo "MPP contact is groupable" flag to d3hsp
- For MPP groupable tied contact only: output of part IDs for MAT_100_DA spotwelds (*DEFINE_CONNECTION_PROPERTIES) was missing.
- Bodies using *MAT_RIGID_DISCRETE were never expected to share nodes with non-rigid bodies, but this now works in MPP.
- Fix variable initialization that resulted in MPP writing bad intfor data (*DATABASE_BINARY_INTFOR) on some systems.
- There is no longer any built-in limitation on the number of processors that may be used in MPP.
- Echo contents of the MPP pfile (including keyword additions) to the d3hsp and mes0000 files.
- Add new keyword *CONTROL_MPP_PFILE, which allows for insertion of text following this command to be inserted into the MPP pfile (p=pfile).
- Change in MPP treatment of *CONSTRAINED_TIE-BREAK. They now share a single MPI communicator, and a single round of communication. This should improve performance for problems with large numbers of these, without affecting the results.
- Fix MPP decomposition error for *LOAD_BODY_GENERALIZED that could result in these forces being applied incorrectly in some cases.
- Fix summation order for *CONSTRAINED_LAGRANGE_IN_SOLID, CTYPE=2 for rebar. This will avoid the round off error causing slightly different shared nodal velocity on echo processor. -- MPP only

.

Forming

- Added two input variables for *CONTROL_FORMING_ONESTEP simulation:
  - TSCLMIN is a scale factor limiting the thickness reduction.
  - EPSMAX defines the maximum plastic strain allowed.
- Added output of strain and stress tensors for onestep solver *CONTROL_FORMING_ONESTEP, to allow better evaluation of formability.
- Improved *CONTACT_AUTO_MOVE. Before: Changes the termination time, and it causes problems when several tools need to be moved. Now: Does not change the termination time, but changes the current time. In this way, several tools can be moved without the need to worry about the other tool's move. This is especially useful in multi-flanging and hemming simulations.
- Made improvements to previously undocumented keyword *INTERFACE_BLANKSIZE, including adding the options _INITIAL_TRIM, and _INITIAL_ADAPTIVE. This keyword was developed for blank size development in sheet metal forming. Generally, for a single forming process, only the option _DEVELOPMENT is needed, and inputs are an initial estimated blank shape, a formed blank shape, and a target blank shape in either mesh or boundary coordinates. Output will be the calculated/corrected initial blank shape. Initial blank mesh and formed blank mesh can be different (e.g. adaptive). For a multi-stamping process involving draw, trimming and flanging, all three options are needed. Related commands for blank size estimation are *CONTROL_ FORMING_ONESTEP, and for trim line development,*CONTROL_FORMING_UNFLANGING.
- Made improvements and added features to previously undocumented keyword*CONTROL_FORMING_UNFLANGING. This keyword unfolds flanges of a deformable blank, e.g., flanged or hemmed portions of a sheet metal part, onto a rigid tooling mesh using the implicit static solver. using the implicit static solver. It is typically used in trim line mapping during a draw die development process. The 'roots' of the flanges or hemmed edges are automatically processed based on a user input of a distance tolerance between the flanges/hemmed edges and rigid tool. It includes the ability to handle a vertical flange wall. Other keywords related to blank size development are,*CONTROL_FORMING_ONESTEP, and *INTERFACE_BLANKSIZE_DEVELOPMENT.
- Added keyword *CONTROL_FORMING_OUTPUT which allows control of d3plot output by specifying distances to tooling home. It works with automatic position of stamping tools using*CONTROL_FORMING_AUTOPOSITION_PARAMETER.

- Added the LOCAL_SMOOTH option to *INTERFACE_COMPENSATION_NEW which features smoothing of a tool's local area mesh, which could otherwise become distorted due to, e.g.,
  - bad/coarse mesh of the original tool surface,
  - tooling pairs (for example, flanging post and flanging steel) do not maintain a constant gap,
  - several compensation iterations.
  - This new option also allows for multiple regions to be smoothed. Local areas are defined by*SET_LIST_NODE_SMOOTH.

.

Discrete Element Method

- Added output to rcforc for *DEFINE_DE_TO_SURFACE_COUPLING.
- Implement traction surface for *DEFINE_DE_TO_SURFACE_COUPLING.
- Add keyword *DATABASE_BINARY_DEMFOR with command line option dem=dem_int_force. This will turn on the DEM interface force file for DEM coupling option. The output frequency is controlled by the new keyword.
- Add new feature *DEFINE_DE_INJECTION to allow DEM particle dropping from user defined plane.
- Add new option _VOLUME to *ELEMENT_DISCRETE_SPHERE. This will allow DEM input based on per unit density and use *MAT card to get consistent material properties.

.

Isogeometric Elements

- Added FORM=-4 for *ELEMENT_SHELL_NURBS_PATCH. Rotational dofs are automatically set at control points at the patch boundaries, whereas in the interior of the patch only translational dofs are present. This helps for joining multiple nurbs patches at their C0-boundaries.
- Disabled FORM=2 and 3 for *ELEMENT_SHELL_NURBS_PATCH. These formulations are experimental and not fully validated yet.
- Added energy computation for isogeometric shells (*ELEMENT_SHELL_NURBS_PATCH) to matsum.
- Allow isogeometric shells (*ELEMENT_SHELL_NURBS_PATCH) to behave as rigid body (*MAT_RIGID).

- Fix for degenerated isogeometric shells (*ELEMENT_SHELL_NURBS_PATCH) that share multiple control-points at the same physical location, which results in a not well defined element normal. This caused stability problems and error terminations.

.

Miscellaneous

- Fix a bug affecting *INCLUDE_TRANSFORM that refers to a transformation (*DEFINE_TRANSFORMATION) involving SCALE.
- Added "g" as abbreviation for gigawords in specification of memory on execution line, e.g, memory=16g is 16 billion words.
- Suppress non-printing characters in *COMMENT output.
- Add command line option "pgpkey" to output the current public PGP key used by LS-DYNA. The output goes to the screen as well as a file named "lstc_pgpkey.asc" suitable for directly importing into GPG.
- When reading the NAMES file, allow a '+' anywhere on a line to indicate there will be a following line, not just at the end. This was never intended, but worked before r73972 and some customers use it that way.
- Check for integer overflow when processing command line arguments and the memory value on the*KEYWORD card.
- Added new capability for *INTERFACE_LINKING_NODE to scale the displacements of the moving interface.
- Several fixes for internal CASE driver.
- Fix problem of LSDA file names getting mangled in the restart file which was causing file reopening problems during restarts.
- Support for *KEYWORD_JOBID with internal *CASE driver.
- *DAMPING_FREQUENCY_RANGE now works for implicit dynamic solutions. An error check has been added to ensure that the timestep is small enough for the damping card to work correctly.
- Added new option *DAMPING_FREQUENCY_RANGE_DEFORM to damp only the deformation instead of the global motion.
- Fixed bug affecting models containing both *DAMPING_GLOBAL and*DAMPING_FREQUENCY_RANGE. This combination is not recommended (and generates a warning message) but such models could previously generate unpredictable error messages.
- Added *DEFINE_VECTOR_NODES. A vector is defined using two node IDs.

- Add sense switch "prof" to output current timing profile to messag (SMP) file or mes#### (MPP) files. Also, for MPP only, collect timing information from processor and output to prof.out when sense switch "prof" is detected.