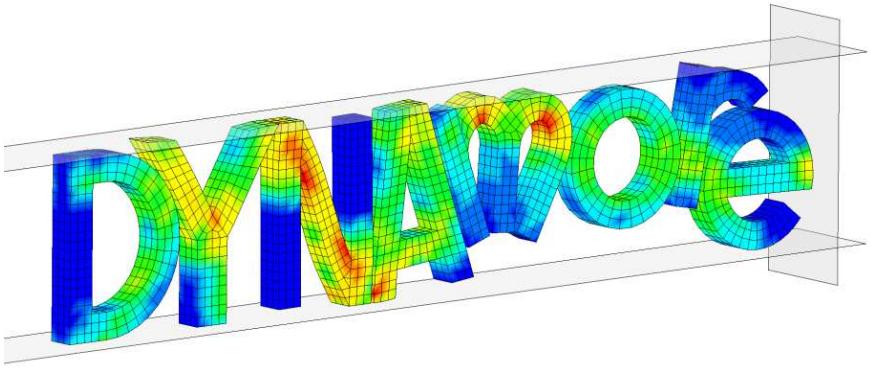
New features in LS-DYNA R8.0.0

- Newest release - published in January 2015
- Robust production version is R7.1.2 (97170)
- This presentation about major new solid mechanics features

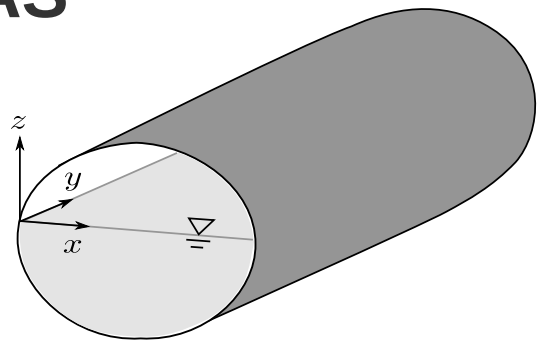


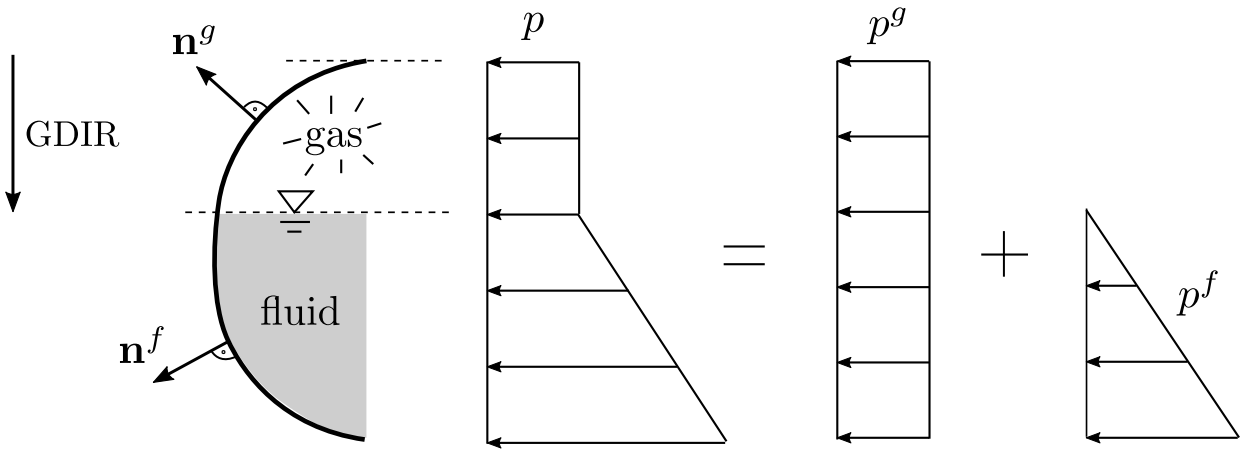
*AIRBAG

- Add RDT option for *AIRBAG_SHELL_REFERENCE_GEOMETRY: time step size based on reference geometry after *MAT_FABRIC birth time
- Several enhancements for *AIRBAG_PARTICLE
  - New blockage (IBLOCK) option for vents
  - Better MPP performance with *DECOMPOSITION_BAGREF/ARRANGE_PARTS
  - External work done by inflator gas to the structure is reported to glstat
  - Enhance segment orientation checking of CPM bag and chambers
  - Allow user to excluded some parts surface for initial air particles
  - Support compressing seal vent which acts like flap vent
  - Support Anagonye and Wang porosity equation through *MAT_FABRIC
  - Add keyword option _MOLEFRACTION
  - Calculate heat convection (HCONV) between environment and airbag in consistent fashion when TSW is used to switch from a particle airbag to a control volume
  - Add ENH_V=2 option for vent hole such that two-way flow can occur, i.e., flow with or against the pressure gradient

*AIRBAG_FLUID_AND_GAS

- Geometric description of fluids and gases: ”Energetically equivalent pressure loads”
- Based on work by Haßler, Maurer, Schweizerhof
- Implemented in 2010, now finally documented
- For quasi-static FSI problems: slow loading
- So far, only implemented for SMP explicit



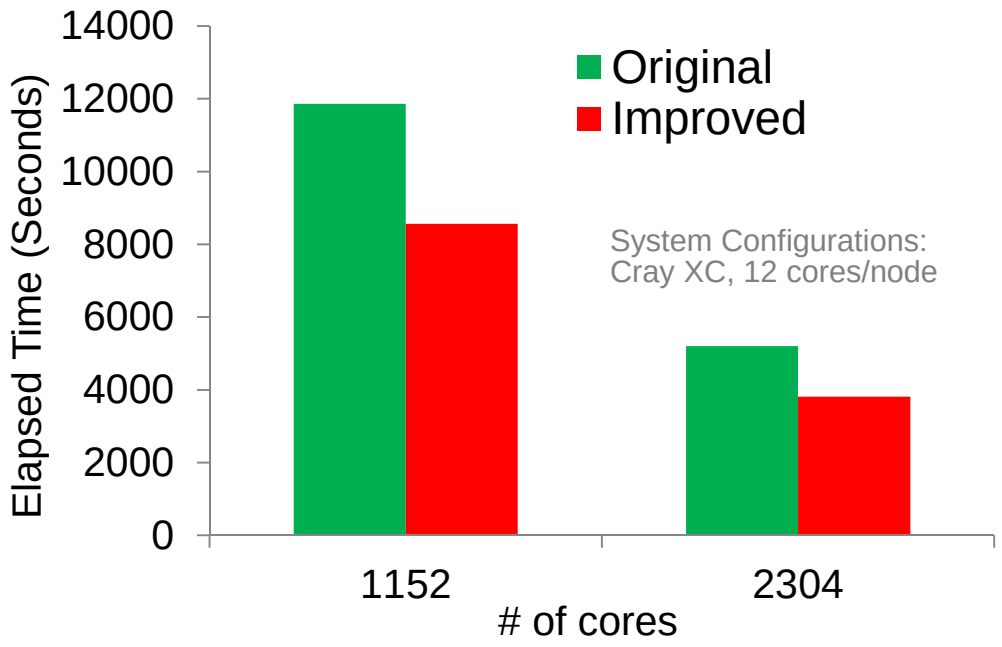


*CONTACT

- Add frictional energy calculation for beams in *CONTACT_AUTOMATIC_GENERAL
- Add keyword *DEFINE_CONTACT_EXCLUSION to allow for nodes tied in some contacts to be ignored in certain other contacts
- Add EDGEONLY option to *CONTACT_AUTOMATIC_GENERAL to exclude node-to-segment contact and consider only edge-to-edge and beam-to-beam
- Enable user defined friction for MPP contact SOFT=4
- Friction factors are now a function of temperature for *CONTACT_..._THERMAL_FRICTION

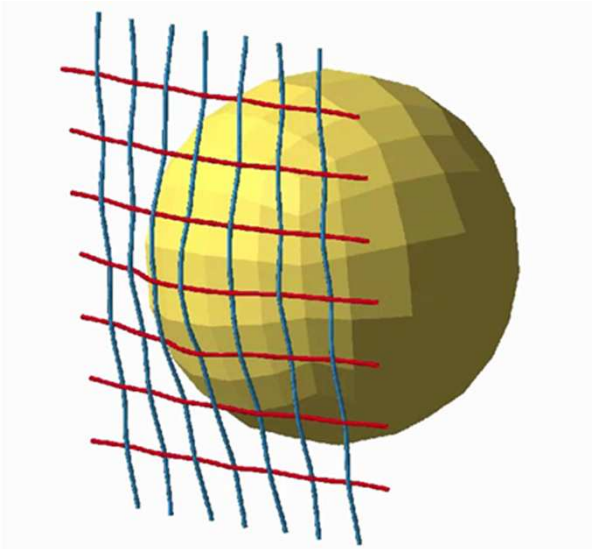
*CONTACT_ERODING_...

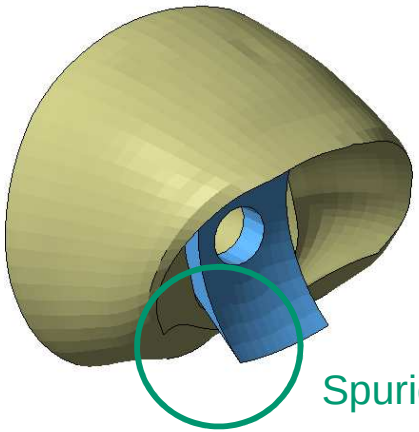
- Major rewrite to improve performance (MPP)
- The new algorithm uses a completely different approach to determining the contact surface. The old algorithm started from scratch when identifying the exterior of the parts in contact.
  The new algorithm is smarter about knowing what has been exposed based on what is eroded, and is faster.
- Example:
  Fan blade out problem ⟶



*CONTACT_..._MORTAR

- Efficiency improvements in contact search significantly improves execution
- Supports friction options FS.LT.0
  - Friction coefficients on *PART_CONTACT
  - Friction table
- Various bug fixes
  - Look ahead adaptivity reintroduced
  - Single surface beam contact
  - Reduce initialization time for forming contact
- Using IGNORE.LT.0 for single surface mortar contact will ignore penetrations of segments that belong to the same part. This will hopefully facilitate removal of initial penetrations in large models





Spurious self-contact

*CONTACT_ADD_WEAR

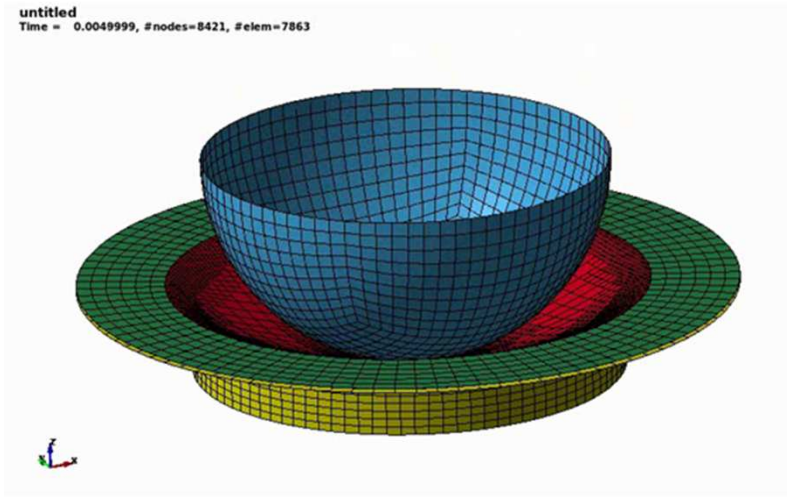
- Hot forming is hard on tools, wear much higher than in cold forming: Simulating wear is of interest for improving tool design

- New keyword *CONTACT_ADD_WEAR
  - Refer to a forming contact
  - Specify wear law, currently only ”Archard”

- Keyword *DATABASE_EXTENT_INTFOR
  - Parameter NWEAR to get wear depth to intfor file for post-processing

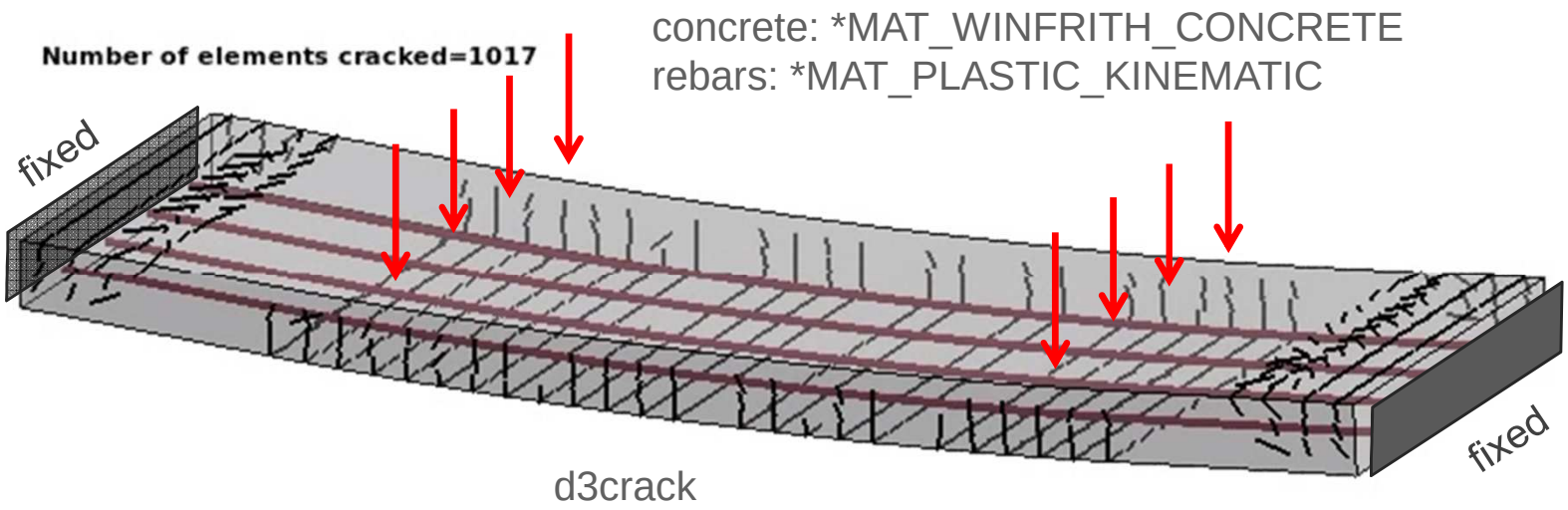
- No coupling to simulation, only post processing

- User wear interface to be implemented → R8.1



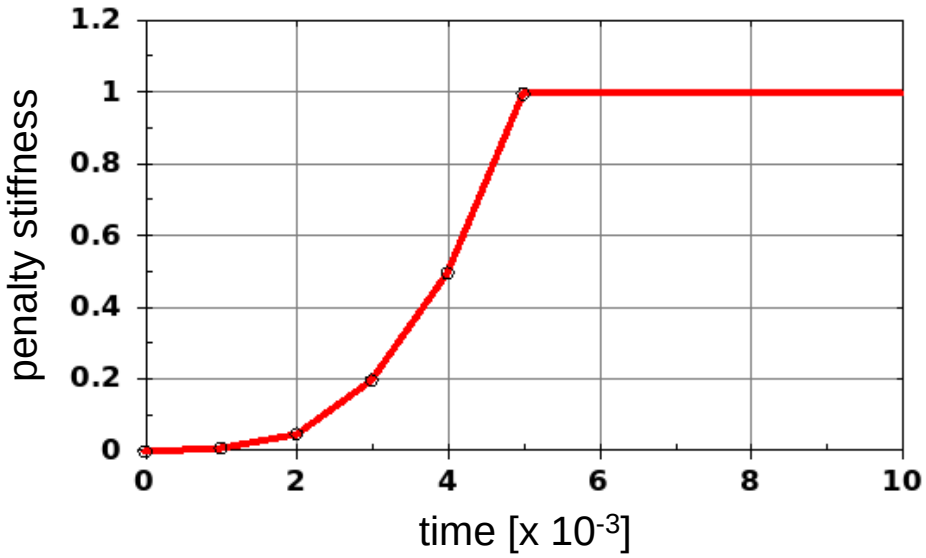
*CONSTRAINED

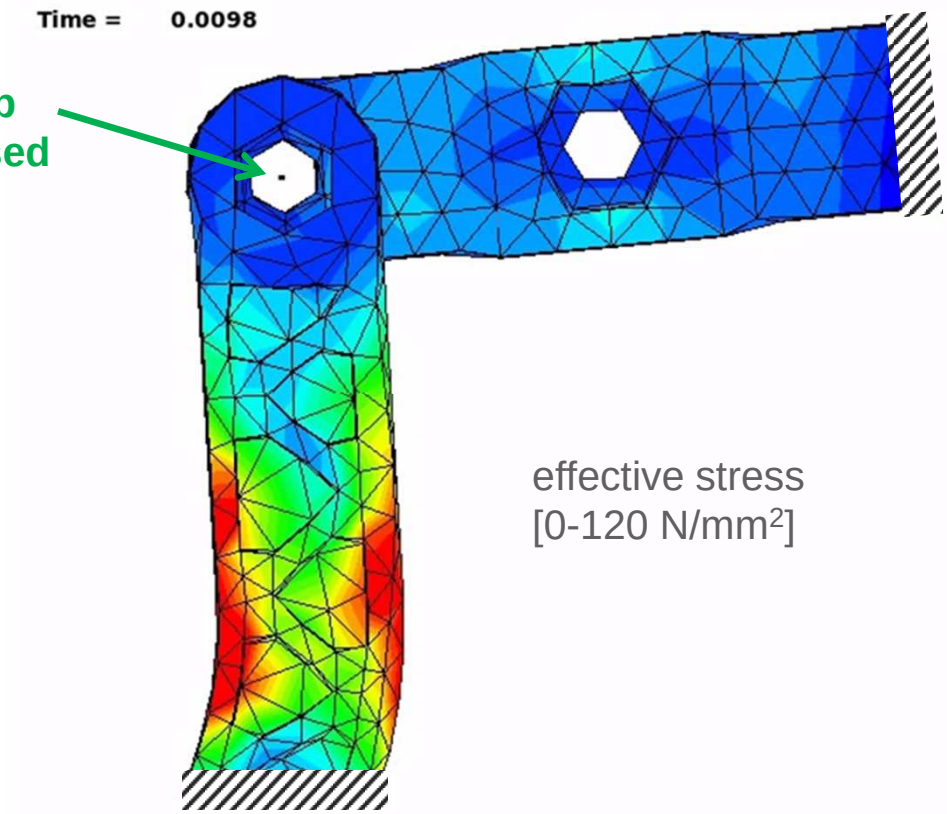
- New keyword *CONSTRAINED_BEAM_IN_SOLID:
  - This feature is basically an overhauled constraint coupling between beams and Lagrangian solids that includes features that make it more attractive in some cases than *CONSTRAINED_LAGRANGE_IN_SOLID, for example, in modeling coupling of rebars in concrete.
  - Allows coupling only in normal directions (optional)
  - De-bonding process via axial coupling force (*DEFINE_FUNCTION: force vs. slip)
  - Correct energy balance



*CONSTRAINED_JOINT

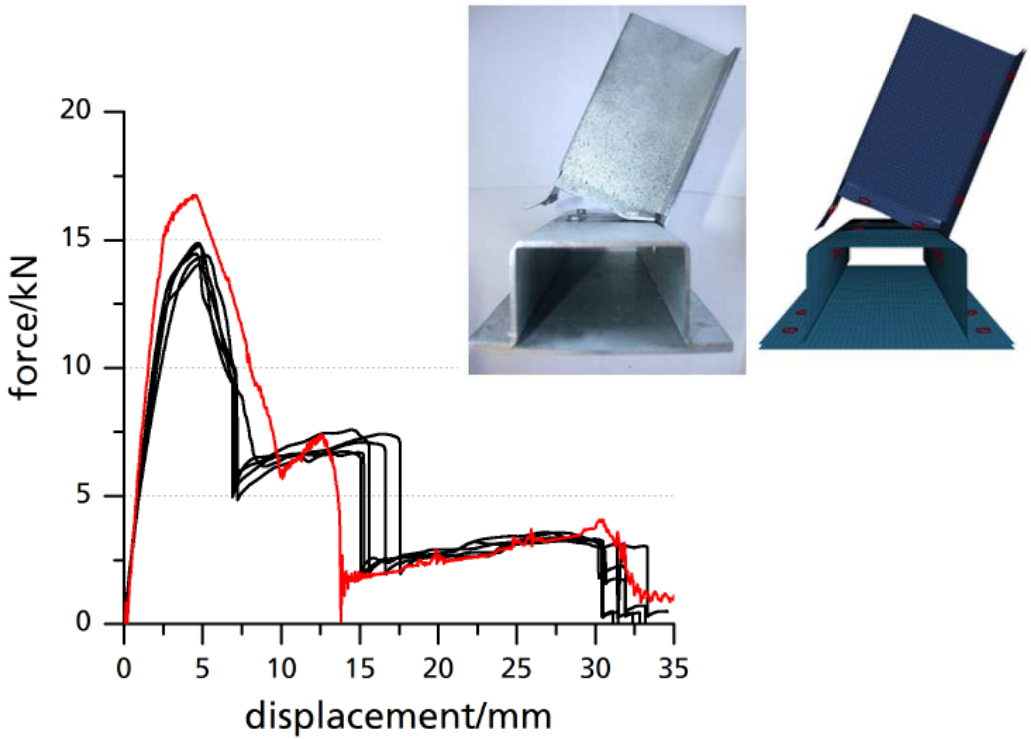
- For penalty-based joints, relative penalty stiffness can now be defined as time dependent value given by load curve (option RPS<0)
  - Nodal points of connected parts must not coincide initially anymore
  - For pre-stressing of joint connections
  - Works for options… SPHERICAL, REVOLUTE, CYLINDRICAL





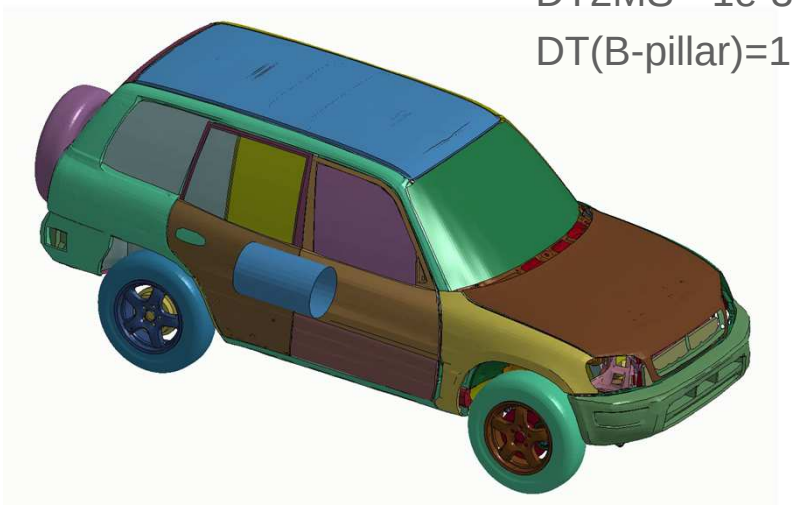
*CONSTRAINED_SPR2/3

- Add failure reports and more d3hsp output
- Add new option to *CONSTRAINED_INTERPOLATION_SPOTWELD: “SPR4”
  - There is a new parameter MODEL to select the new method.
- Add new feature MODEL.GE.10 to *CONSTRAINED_SPR3
  - This allows parameters STIFF, ALPHA1, RN, RS, and BETA to be defined as *DEFINE_FUNCTIONs of thicknesses and maximum engineering yield stresses of connected sheets.



*CONTROL_SUBCYCLE

- New subcycling scheme activated for *CONTROL_SUBCYCLE_K_L (subcycling) and *CONTROL_SUBCYCLE_MASS_SCALED_PART (multi-scale)
  - Simplified input: time step ratios for internal and external force calculations
  - Multi-scale: User may manually designate parts to be integrated at specific time steps
  - Improved robustness and performance

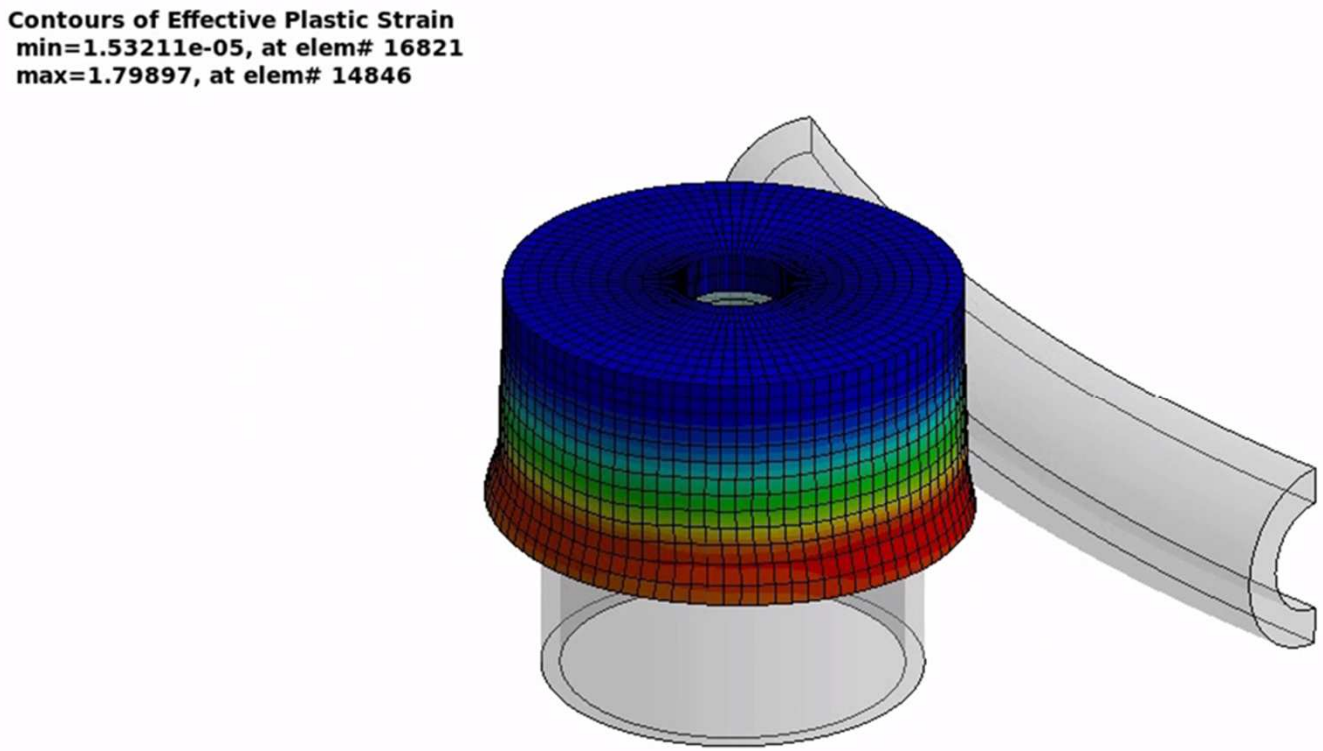


B-pillar refined
DT2MS=-1e-3
DT(B-pillar)=1.3e-4

CPU timings (s)	Subcycling, K=64, L=4	Multi-scale, L=4	Default
Contacts	133	133	288
Elements	194	206	636

*CONTROL_REMESHING

- New method for axisymmetric remeshing (*PART with ADPOPT=3)
- Works with hexahedral and pentahedral elements (quads and trias in sections)
- Number of elements can increase and decrease

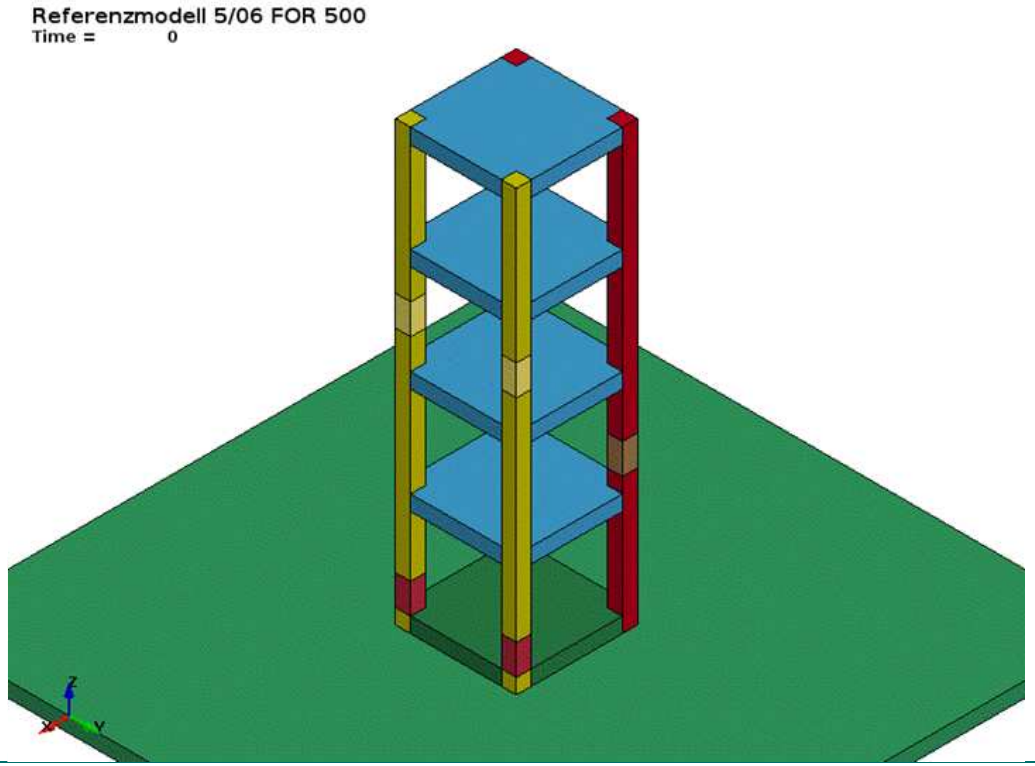


Application area:

3-D orbital forming

*DEFINE_ADAPTIVE_SOLID_TO_DES

- Adaptively transform a Lagrangian solid Part or Part Set to DES particles when the Lagrangian solid elements comprising those parts fail.
- One or more DES particles (elements) will be generated for each failed element.
- The DES particles replacing the failed element inherit all of the properties of the failed solid element, e.g. mass, kinematic variables, and constitutive properties.



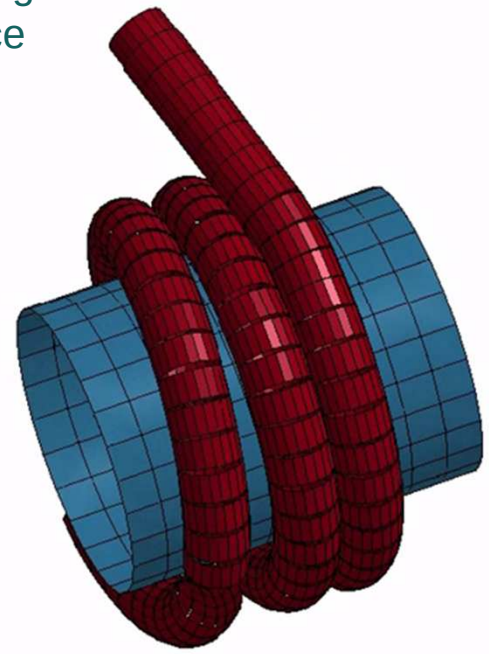
Courtesy of
Institut für Mechanik

Universität Karlsruhe (TH)
Forschungsuniversität · gegründet 1825

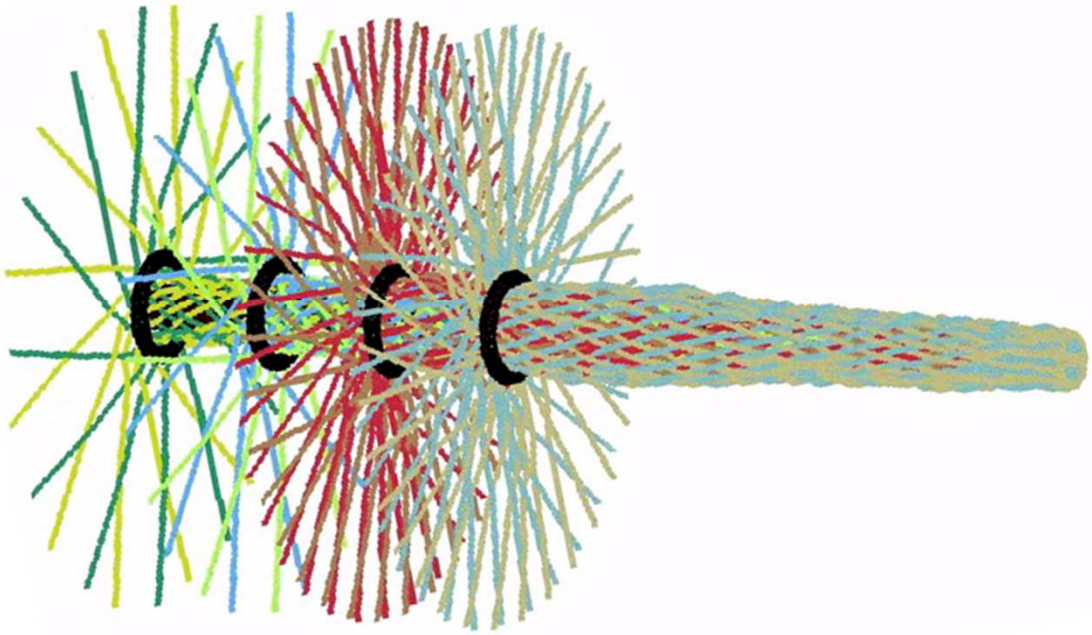
*ELEMENT_BEAM_SOURCE

- New keyword to define a point source (node) where a cable / thread / yarn with pre-defined length can be pulled out
- Input parameters are node id, number of elements to be drawn out, beam element fed length, pull-out force, and minimum beam element length
- Application: e.g. yarn feeders for braiding/weaving processes

moving source



176 moving sources

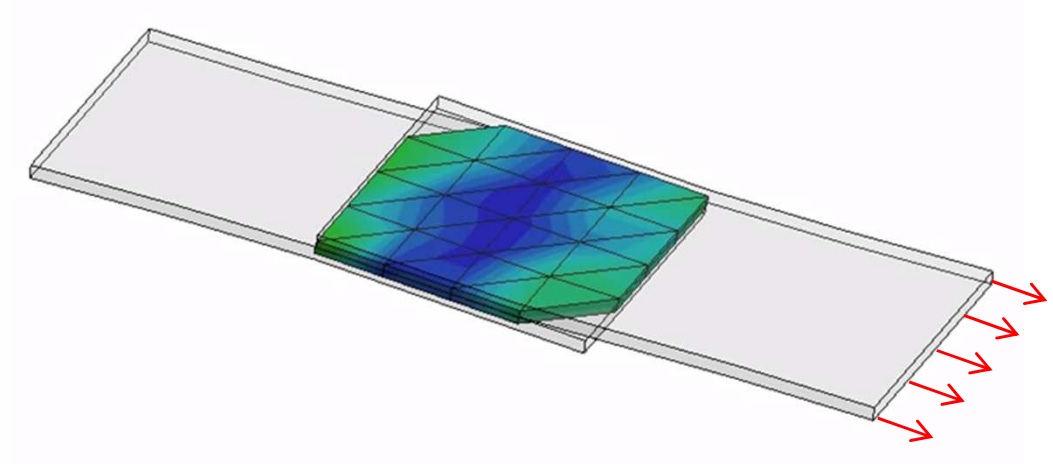


*ELEMENT_TSHELL_BETA

- Works in a similar way like *ELEMENT_SHELL_BETA
- Allows direct thickness extrusion SHELL → TSHELL without loss of material orientation

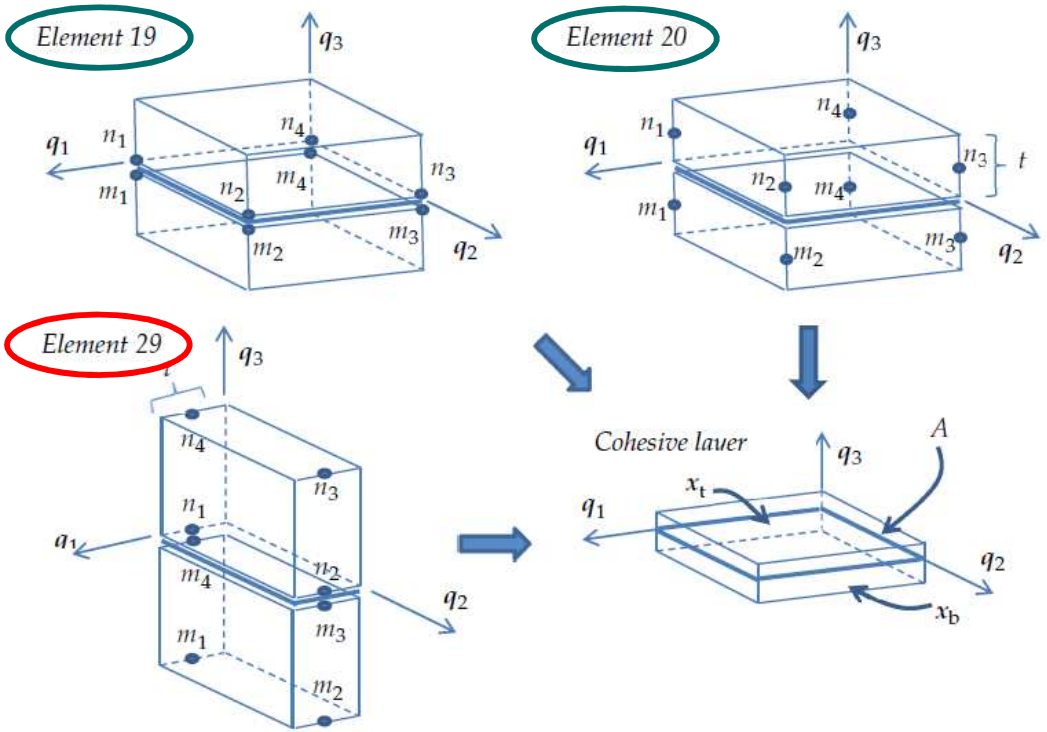
Cohesive Elements (*SECTION_SOLID, ELFORM=19-22)

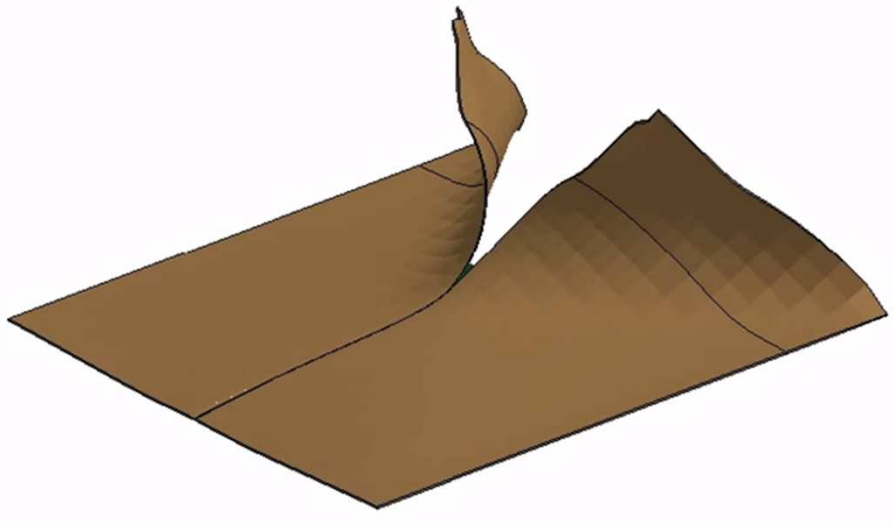
- Improve stability for ELFORM=20 (with offsets for use with shells)
  - New incremental formulation to properly handle large rotations
- Add pentahedron elements
  - ELFORM=21 (6-noded pentahedron)
  - ELFORM=22 (6-noded pentahedron with offsets for use with shells)
  - ESORT.gt.1 in *CONTROL_SOLID automatically activates element sorting of pentahedron solid elements



*SECTION_SHELL, new ELFORM=29

- New cohesive shell element for edge-to-edge connectivity between shells
- Takes bending into account, supports MPP and Implicit
- Can be used with usual cohesive material laws (138, 186, 240, 252)
- Presentation will be given in Würzburg by Jesper Karlsson





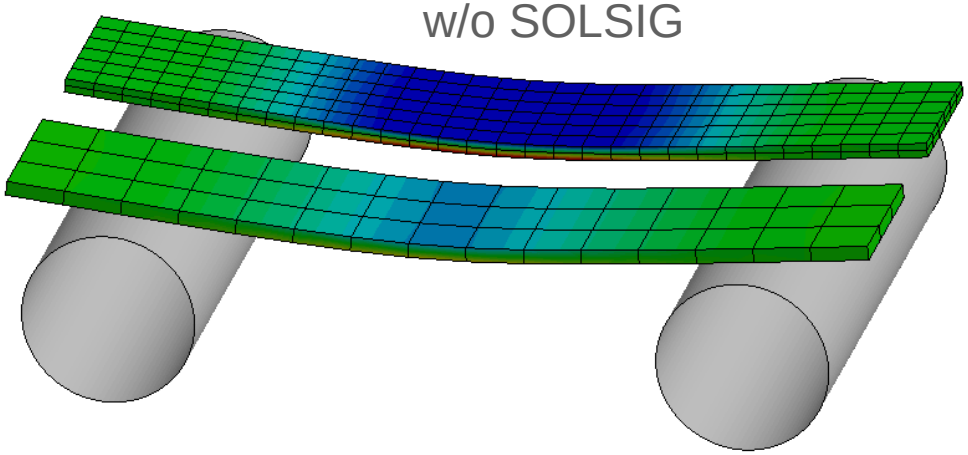
ripping of pre-notched clamped plate

Postprocessing of Solid Element Results

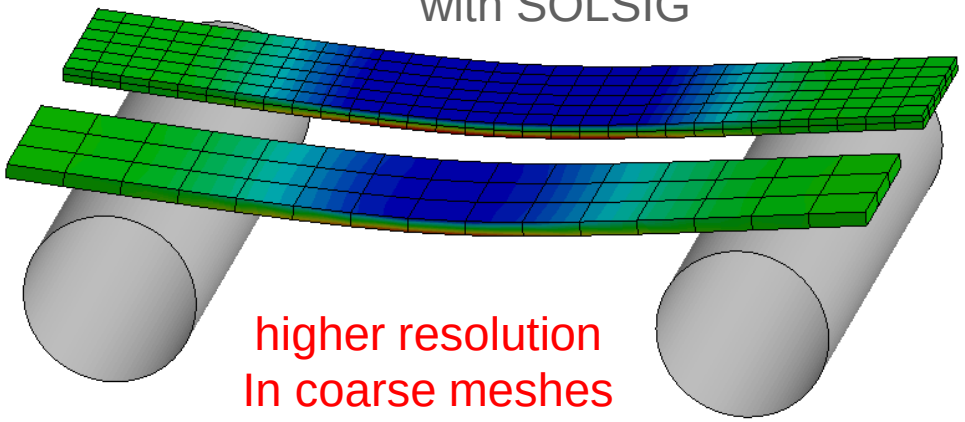
- Add option SOLSIG to *CONTROL_OUTPUT which will permit stresses and other history variables for multi-integration point solids to be extrapolated to nodes.
  - These extrapolated nodal values replace the integration point values normally stored in d3plot. NINTSLD must be set to 8 in *DATABASE_EXTENT_BINARY when a nonzero SOLSIG is specified.
  - Supported solid formulations are: -1, -2, 2, 3, 4, 16, 17, 18, and 23.
  - Warning: Do not use "Setting - Extrapolate" in LS-PrePost when SOLSIG is nonzero.

Example: 3-pont bending, longitudinal stresses

w/o SOLSIG



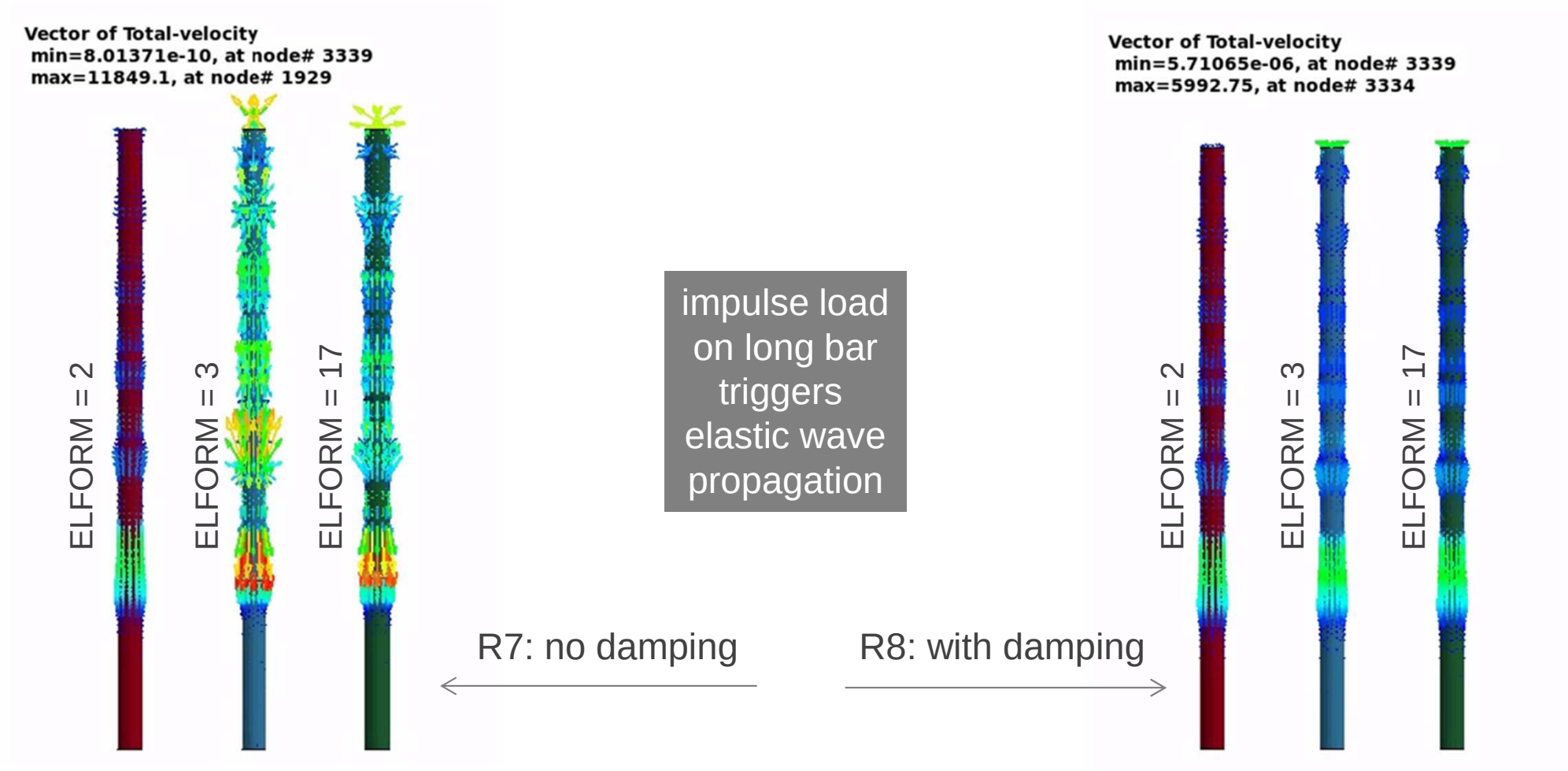
with SOLSIG



higher resolution
In coarse meshes

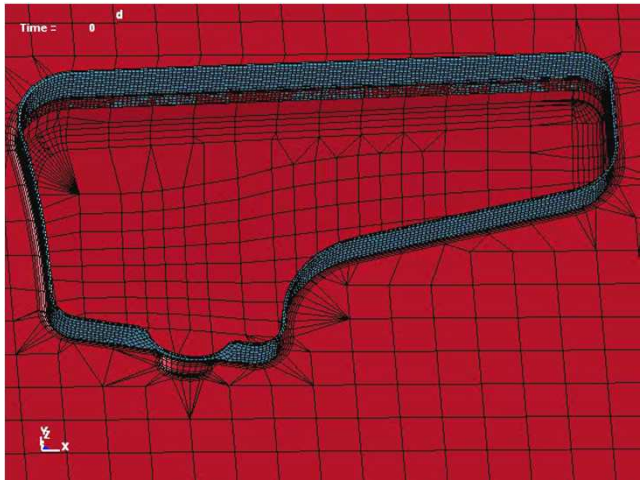
*DAMPING_PART_STIFFNESS

- Add Rayleigh damping for triangular shell elements 3 and 17 (ESORT = 2)



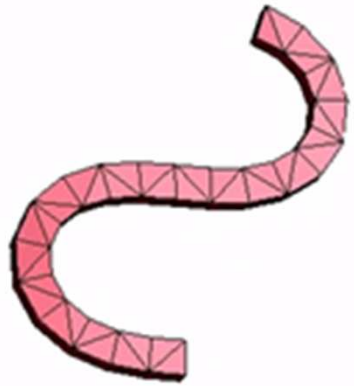
Forming related features

- *CONTROL_FORMING_AUTOCHECK
- *CONTROL_FORMING_MAXID
- *CONTROL_FORMING_ONESTEP
- *CONTROL_FORMING_OUTPUT
- *CONTROL_FORMING_SCRAP_FALL
- *CONTROL_FORMING_TRIM_MERGE
- *CONTROL_FORMING_TRIMMING
- *CONTROL_FORMING_UNFLANGING
- *DEFINE_CURVE_TRIM_3D/NEW
- *ELEMENT_LANCING
- *INTERFACE_COMPENSATION_NEW
- *INTERFACE_BLANKSIZE_DEVELOPMENT
- …

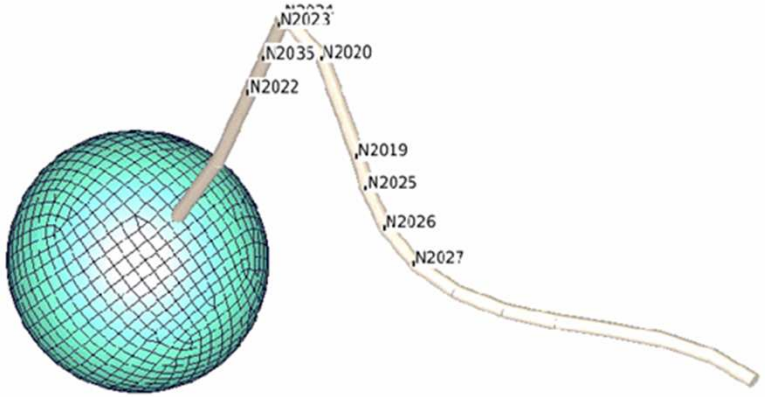


New implicit features

- Dumping of system matrices, see MTXDMP in *CONTROL_IMPLICIT_SOLVER
  - Can now dump the damping matrix (in addition to stiffness and mass matrices)
  - Extend matrix dumping capability to MPP
  - MTXDMP<0 will terminate the run after dumping of matrices



- *CONTROL_IMPLICIT_SOLVER
  - Nonsymmetric linear solver can be used by specifying LCPACK=3
  - Allow unsymmetric terms to the assembled stiffness matrix from some implicit features (nonuniform follower loads, user elements)



- *MAT_FABRIC now available
- *MAT_SEATBELT now available

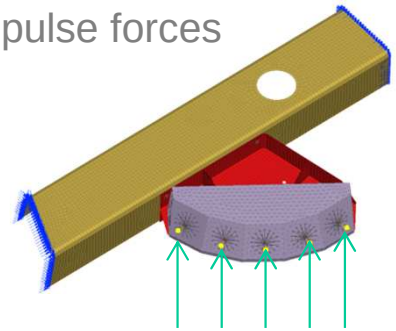
*CONTROL_IMPLICIT_MODAL_DYNAMIC

- Modal Dynamics is used to reduce the cost of analysis. Element and Material computations are replaced by two multiplications by Φ

- This has been in LS-DYNA since LS-DYNA 960 using *CONTROL_IMPLICIT_DYNAMICS with IMASS = 2 or 3

- Now, Modal Dynamics has been enhanced with new features and keywords.
  - *CONTROL_IMPLICIT_MODAL_DYNAMICS
    To activate modal dynamic analysis
  - *CONTROL_IMPLICIT_MODAL_DYNAMICS_MODE
    To select a subset of modes to use in the analysis
  - *CONTROL_IMPLICIT_MODAL_DYNAMICS_DAMPING
    To specify damping coefficients for modal damping.

Instrument bracket
with impulse forces



- Talk about that topic given
  by Roger Grimes in Detroit 2014

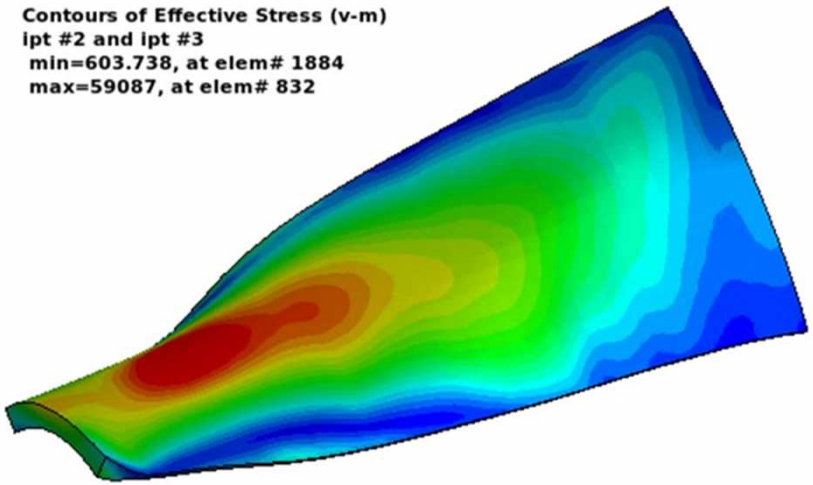
Transient Explicit: 220 min (single precision)
Transient non-linear implicit: 265 min
Transient modal dynamics: 4 min

*CONTROL_IMPLICIT_ROTATIONAL_DYNAMICS

- New keyword is added to study Rotordynamics using the implicit time integrator
- Linearized equilibrium equation in the rotating coordinate system includes gyroscopic damping (Coriolis contribution) and centrifugal stiffness

$$\mathbf{M}\mathbf{u} + (\mathbf{D} + 2\Omega\mathbf{C})\dot{\mathbf{u}} + (\mathbf{K} - \Omega^2\mathbf{K}_G)\mathbf{u} = \mathbf{F}$$

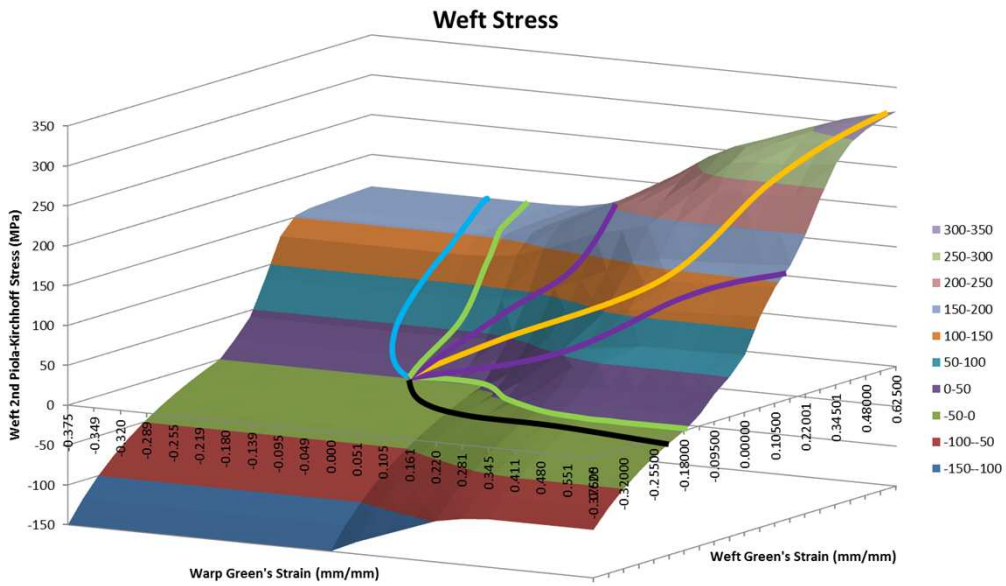
- Applications: transient and vibration analysis of rotating parts such as turbine blades, propellers in aircraft, and rotating disks in hard disk drives.
- It is available for beam, shell, solid and thick shell elements.
- Currently only SMP double, MPP under development.



*MAT_FABRIC (*MAT_034)

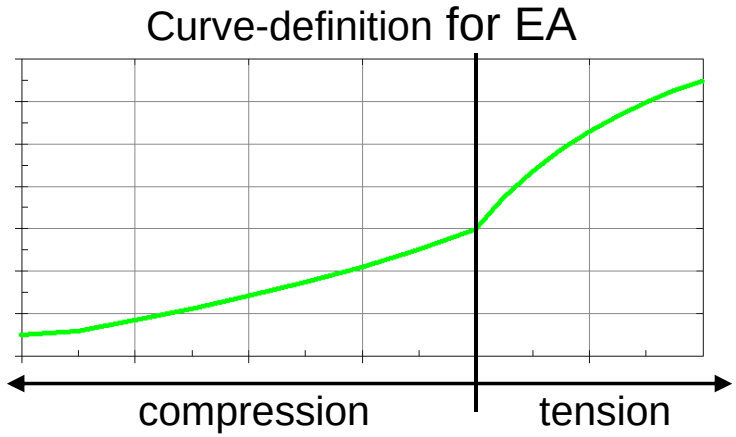
- Add new formulation FORM = 24 as modified version of FORM = 14
  - The main improvement is that the Poisson's effects work correctly with the nonlinear curves for fiber stress.
  - Also, the output of stress and strain to d3plot are engineering stress and strain instead of 2nd PK stress and Green's strain.

- New material model *MAT_FABRIC_MAP
  - stress response is given exclusively by tables, or maps, and where some obsolete features in *MAT_FABRIC have been deliberately excluded to allow for a clean input and better overview of the model
  - Upcoming presentation in Würzburg by Thomas Borrvall



*MAT_LAMINATED_COMPOSITE_FABRIC (*MAT_058)

- Added possibility to define „arbitrary“ uniaxial elastic stress vs. strain behavior using curve definitions (valid for EA, EB, GAB)
  - non-linear elastic behavior
  - different stiffness in tension and compression



- Strain rate dependent stiffness using table definition (stress vs. strain vs. strain rate)

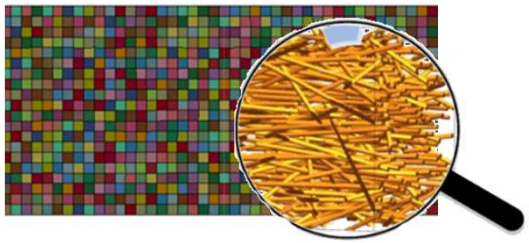
*MAT_SPOTWELD (*MAT_100)

- Add option to use yield curve or table for solid elements
  - if SIGY.LT.0 is used

*MAT_ANISOTROPIC_ELASTIC_PLASTIC (*MAT_157)

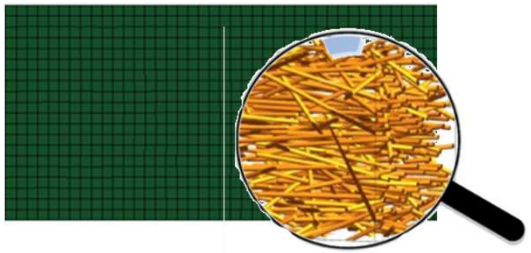
- Well suited for short-fiber reinforced composites due to anisotropy in elastic and plastic regime
- Added VP=1 (viscoplastic formulation) for shells
- *MAT_157 implemented for solids (including VP=1)
- Possibility to initialize various anisotropic material properties via *INITIAL_STRESS_SHELL/SOLID on a per-element basis (IHIS)

In material card



Drawback: inhomogeneous distribution (e.g. from previous short fiber filling simulation) in component needs individual part definition for every element

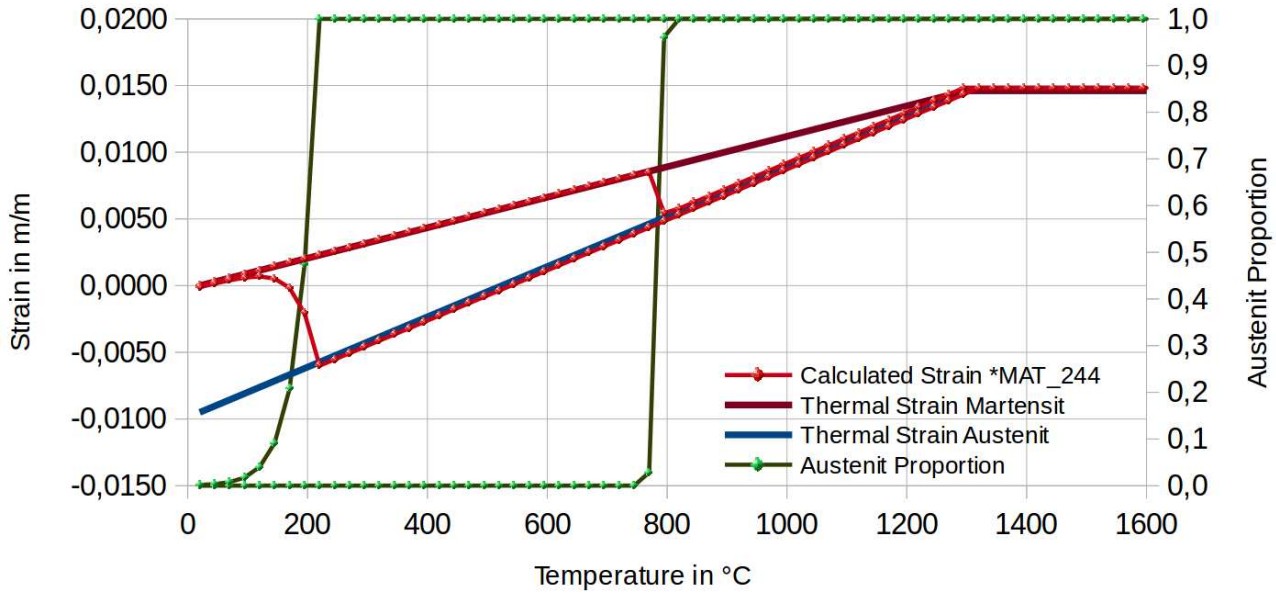
With *INITIAL_STRESS_SOLID



Only one part definition for whole component. Anisotropic coefficients are part of material‘s history field and can therefore be initialized for each integration point individually

*MAT_UHS_STEEL (*MAT_244)

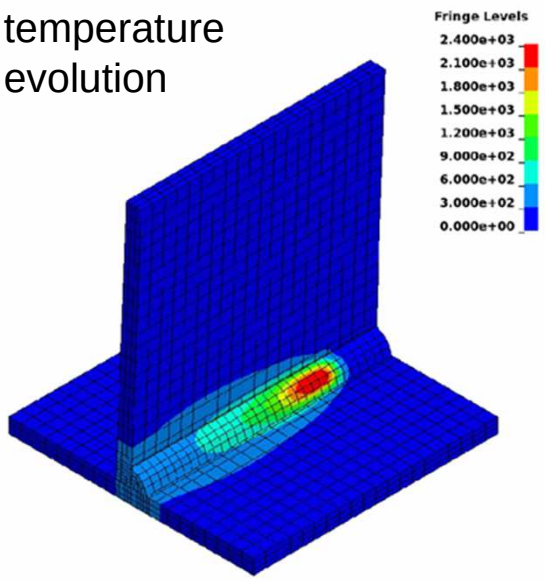
- By default same start temperatures for phase transitions are assumed for heating and cooling
- Now, advanced reaction kinetics input (REACT=1) accepts load curve IDs (input as negative values) for FS, PS, BS, MS
- Temperature dependent thermal expansion for austenite and hard phases
- Added load curve for transformation induced strains



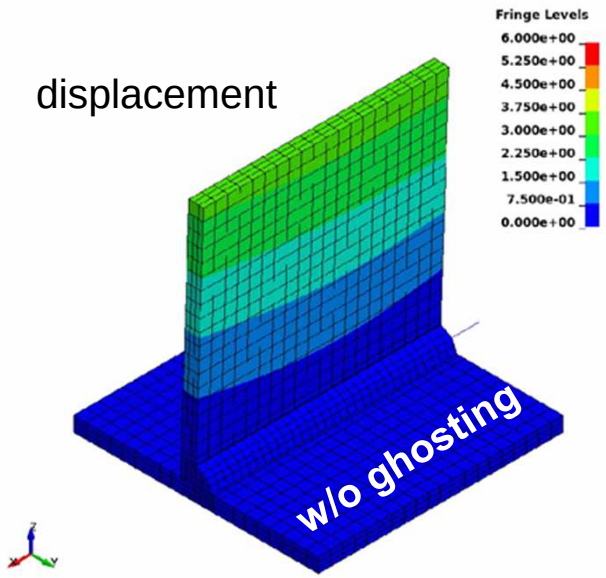
*MAT_UHS_STEEL (*MAT_244)

- New features for welding
- Ghost material approach as in *MAT_270
  - Material is inactive at the beginning, but is activated by temperature
- Annealing is also considered (history variables reset)
- Can be combined with *MAT_THERMAL_CWM

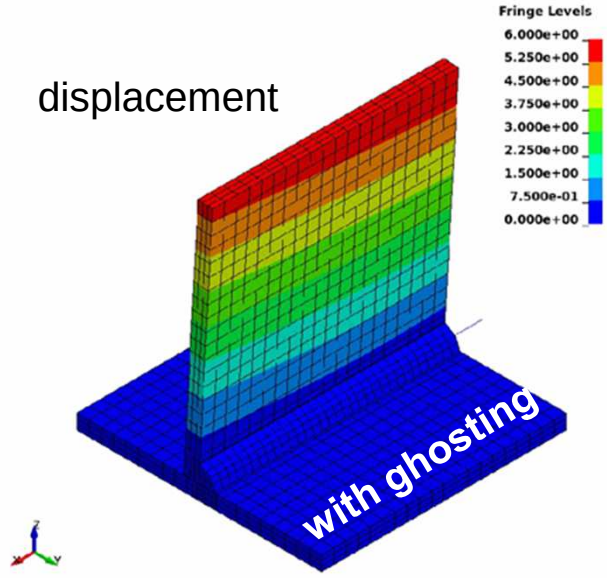
temperature evolution



displacement



displacement



More material model updates

- Add a keyword option called MIDFAIL for *MAT_024
  - failure by plastic strain will only be checked at the mid-plane
- New options for *MAT_224
  - BETA .LT. 0: strain rate dependent amount given by load curve ID = -BETA
  - Implicit stiffness matrix added
- Add enhanced damage model with crack closure effects to *MAT_104
- Some improvements for *MAT_075 (BILKHU/DUBOIS_FOAM)
  - Rate dependence and manual documentation → good for crushable foams
- Increase robustness of *MAT_CORUS_VEGTER (*MAT_136)
- …

Miscellaneous

- Add a new keyword *BOUNDARY_SPC_SYMMETRY_PLANE
  - Define constraints to enforce planar symmetry for nodes on or near a specified plane. These constraints will be enforced even during adaptivity. This command is similar to *CONSTRAINED_LOCAL but allows selectivity via a part ID.
- *DATABASE_SECFORC, *DATABASE_CROSS_SECTION
  - The secforc data for cross-sections through 2D seatbelt elements is recoded to provide more robust and accurate results.
- *DATABASE_EXTENT_BINARY, *MAT , *SECTION
  - For some materials and elements, thermal and plastic strain tensors can be output to d3plot database, see STRFLG.
- *DEFINE_TABLE
  - Add check of table's curves for mismatching origin or end points

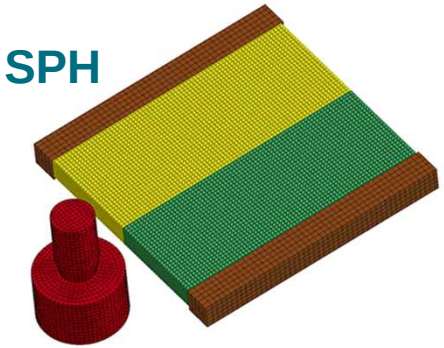
Miscellaneous

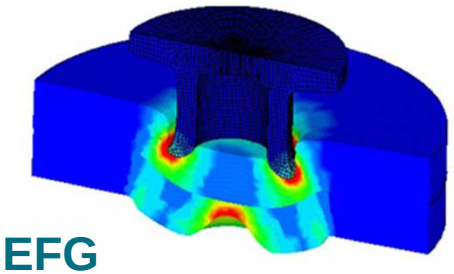
- *CONTROL_OUTPUT and command line option “msg=all” or “msg=10087”
  - Add option for detailed warning/error messages to d3msg, parameter MSGFLG
  - Only a few "long" versions of warnings/errors at this time but that list is expected to grow

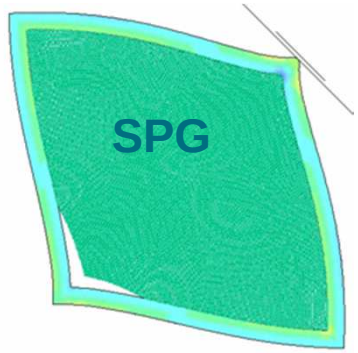
```
 *** Message    10087 (KEY+87)
     in *INCLUDE file name:
     File AAA does not exist
     Keyword read will continue but numerous errors may result

LS-DYNA couldn't read the include file specified in the input deck.

The include file may be improperly specified in the input file.
Please ensure that the path to the include file is specified
correctly. Or, you may use batch queuing system which is allowed to
deal input files only. Please ask your system administrators if the
system considers not only LS-DYNA input file but also include files
you use.
```

Conclusion: LS-DYNA R8.0.0

- Many more developments and enhancements in other areas (ALE, EFG, SPG, SPH, IGA, Thermal, Frequency Domain, …) and the multiphysics solvers (ICFD, CESE, EM, Chemistry)







- Comprehensive list of enhancements and corrections: ftp.lstc.com/ls-dyna/R8.0.0/Release_Notes_LS-DYNA_R8_0_0_rev1.txt
- R8.0 Keyword User‘s Manual can be downloaded from www.lstc.com/downloads/manuals/