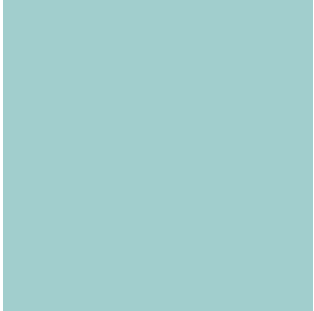




LS-DYNA R12.0.0 - DYNAmore

DYNAmore Nordic AB
DYNAmore GmbH
ANSYS LST LLC
ARUP

Confidential: Only for internal use by DYNAmore Customers and Partners. Any redistribution or publishing to the public, including on paper or web/internet, requires a written permission by DYNAmore Nordic AB.

Other channels for information

■ DYNAmore Nordic

- Online seminars/webinars, see: <https://www.dynamore.se/en/training/seminars>

■ DYNAmore GmbH

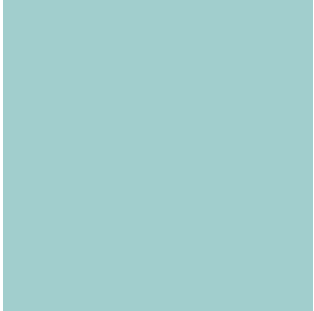
- Online seminars/webinars, see: <https://www.dynamore.de/en/training/seminars>
 - Compact -> new form of online course
 - Video seminar - on-demand seminar
 - DYNAmore Express -> Webinar

■ Support pages:

- www.dynasupport.com
- www.dynaexamples.com

Content

- CPM Airbag
- Belt modelling
- Implicit
- Contact
- Fatigue
- Frequency Domain
- Forming
- Additive Manufacturing
- Thermal
- Materials
- SPH
- SPG
- XFEM
- ALE and S-ALE
- ICFD
- EM
- IGA
- Miscellaneous



CPM Airbag

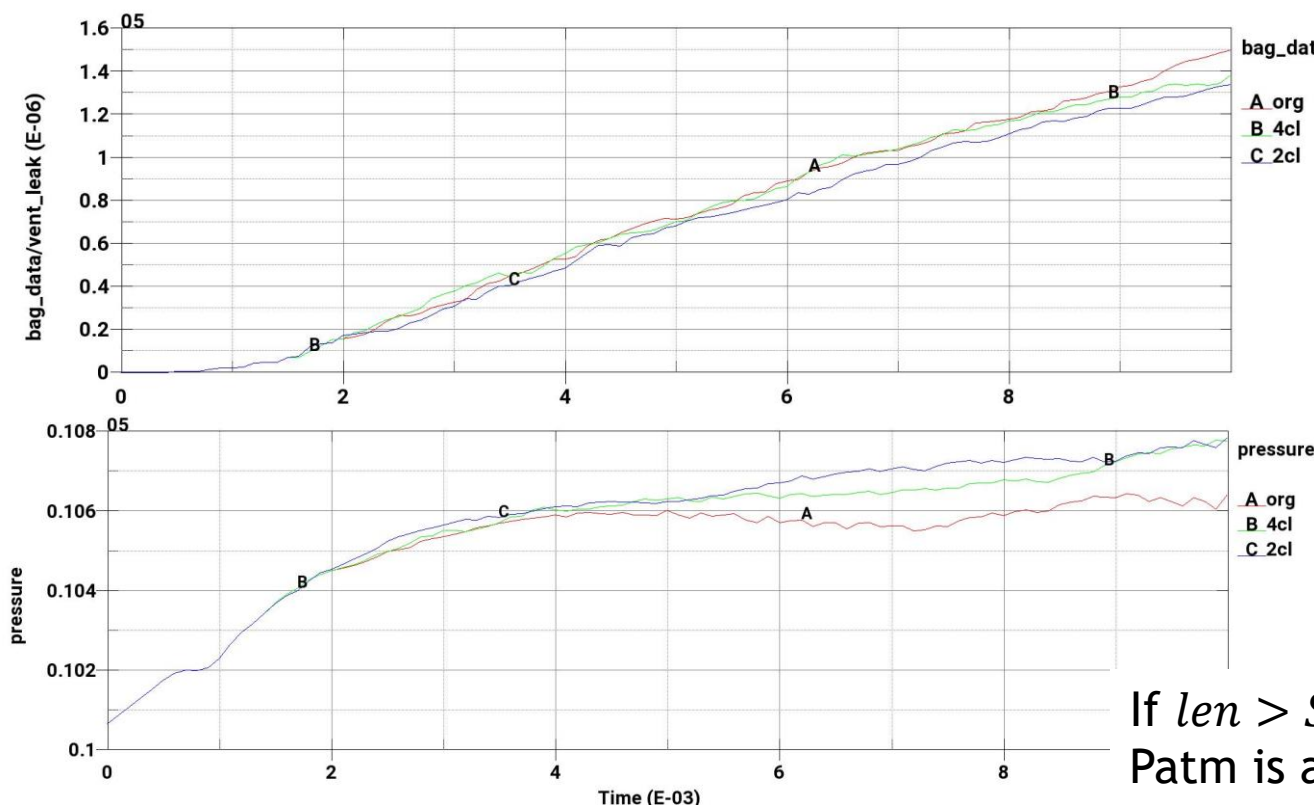
Pushout Vent

Reaction force from choked flow using CPM

CPM - New features for push out vent

■ Pushout vent IOPT=200

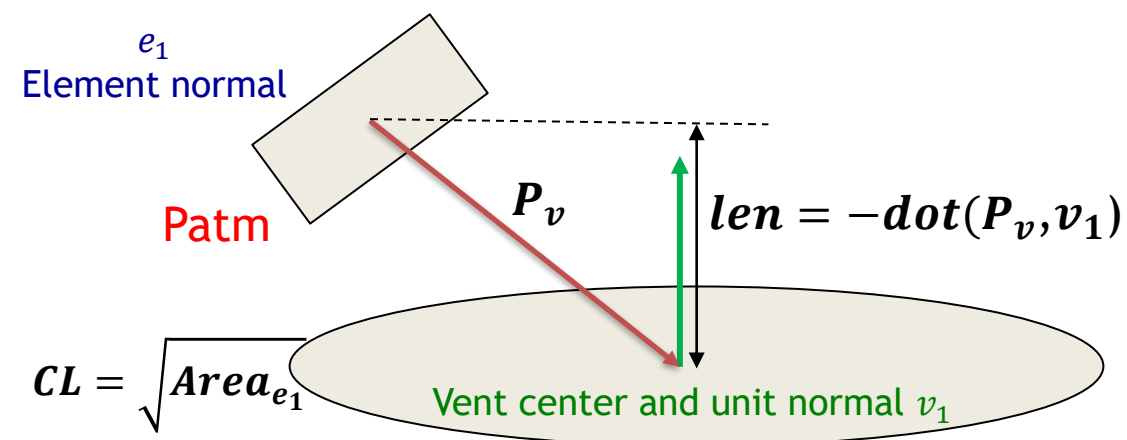
- New option to treat internal material being pushed through vent.



Card 1	1	2	3	4	5	6	7	8
Variable	ID	C23	LCTC23	LCPC23	ENH_V	PPOP	C23UP	IOPT
Type	I	F	I	I	I	F	F	I
Default	None	1.0	None	None	None	None	None	None

Card 2	1	2	3	4	5	6	7	8
Variable	JT	IDS1	IDS2	IOPT1	PID1	PID2	VANG	LCRED
Type	I	I	I	I	I	I	F	I
Default	0	None	None	None	None	None	0.	None

Card 3	1	2	3	4	5	6	7	8
Variable	NID1	NID2	NID3	LCAC23	PSETPV	SFPV		
Type	I	I	I	I	I	I	F	I
Default	0	None	None	None	None	None	0.	None



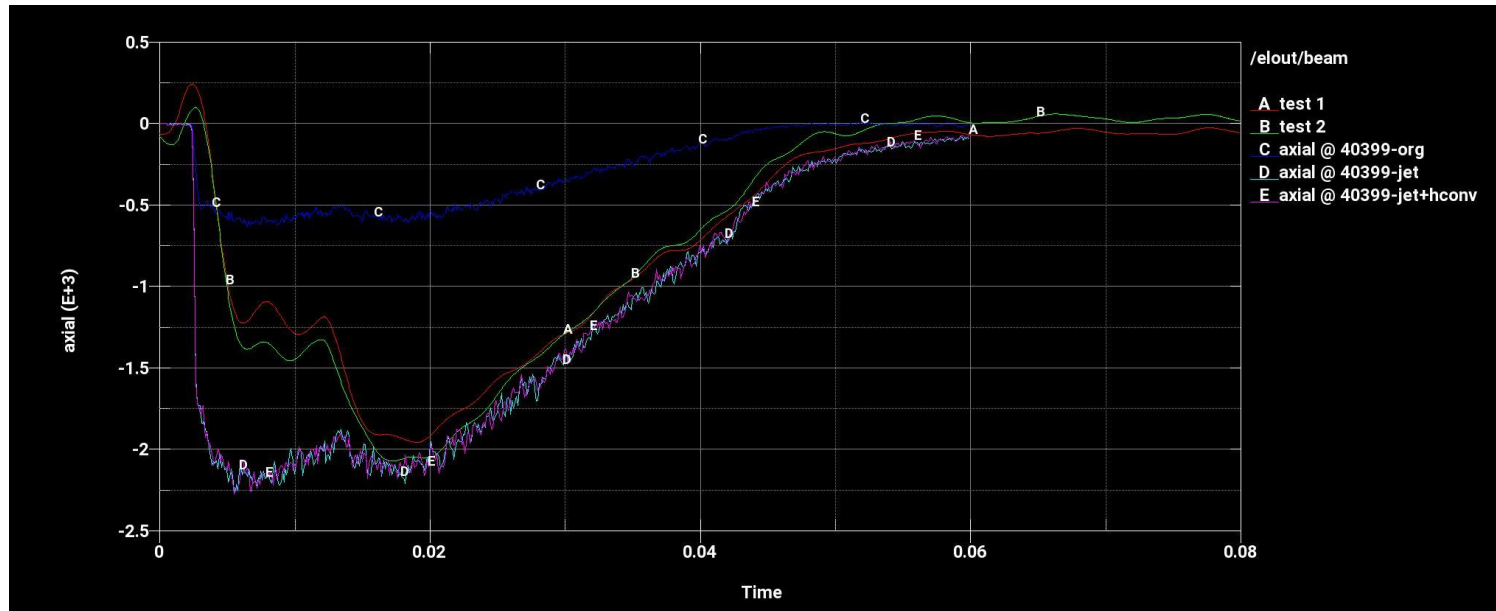
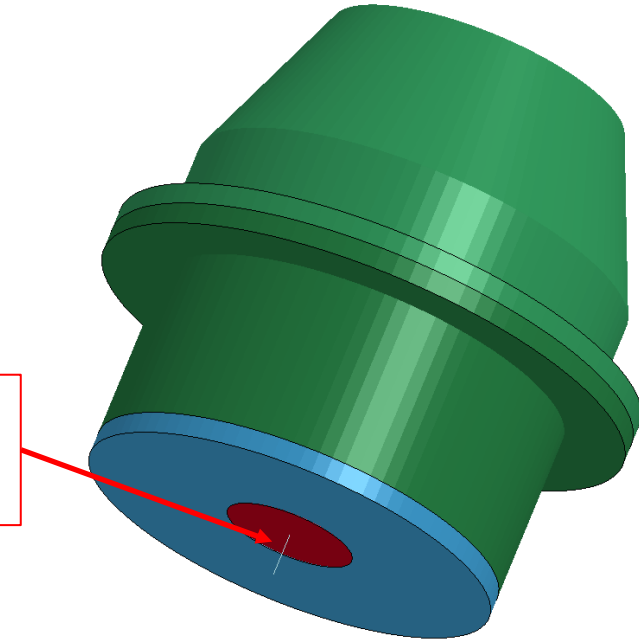
If $len > SFPV * CL$
 P_{atm} is applied to the e_1 surface facing the air.

CPM - Reaction forces

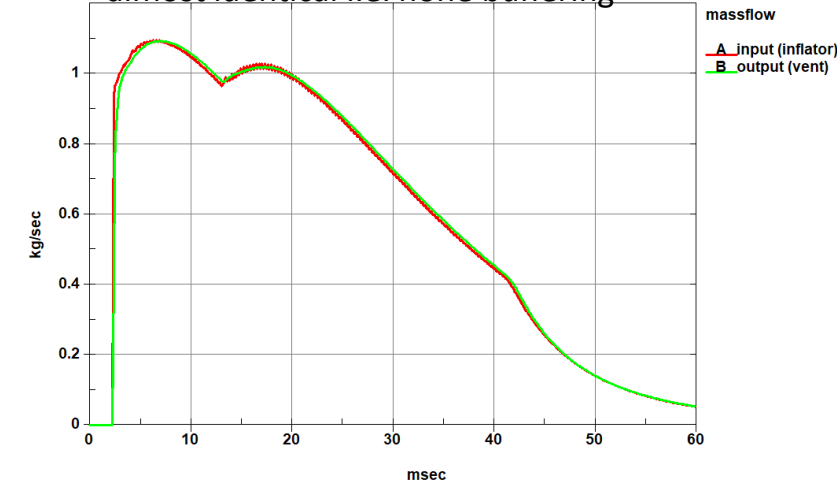
■ Total Reaction Force

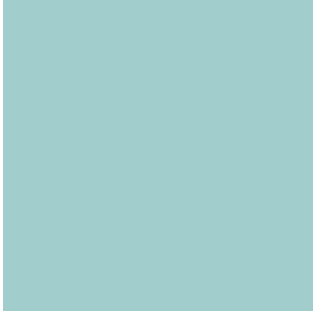
- The part due to choking, CPM needs evaluate as additional term. This feature is implemented in branches of R11, R12, R12.0 and DEV after SVN # 148275 with `_JET` keyword.

Beam element for
total reaction
force registration



Input and vent output mass flow are
almost identical i.e. none buffering





Belt modelling

MAT_SEATBELT_2D updates
Retractor sensor

*MAT_SEATBELT_2D, FORM=-14

■ Strain rate dependency

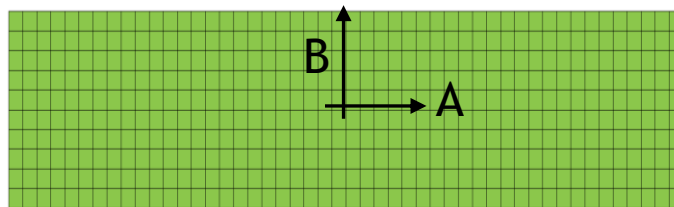
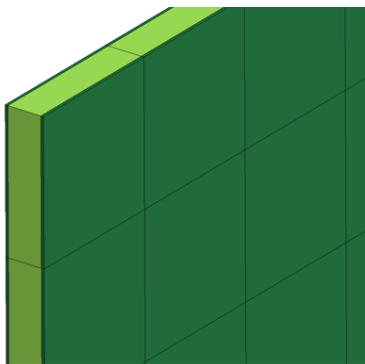
- Table ID for LLCID
- Applied in length direction of belt

■ Orthotropic material behavior

- New parameters to control the orthotropic material behavior: EB, PRAB, PRBA and GAB

■ Coating functionality

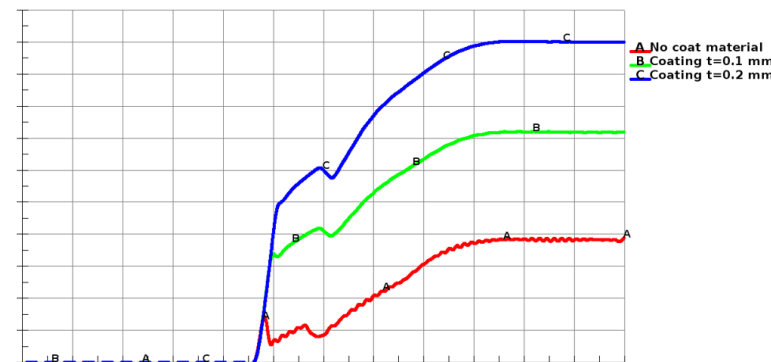
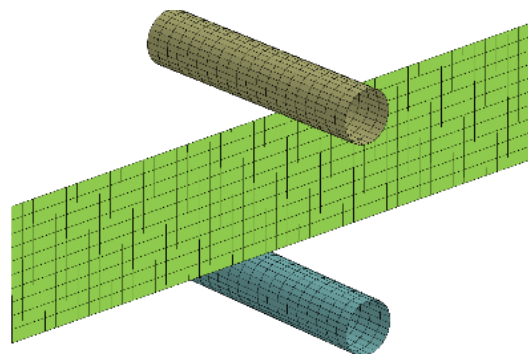
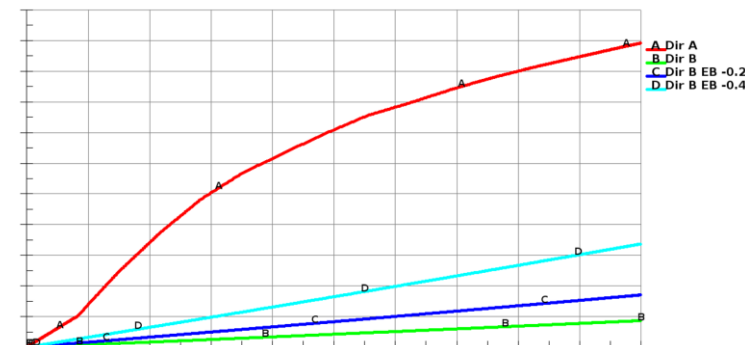
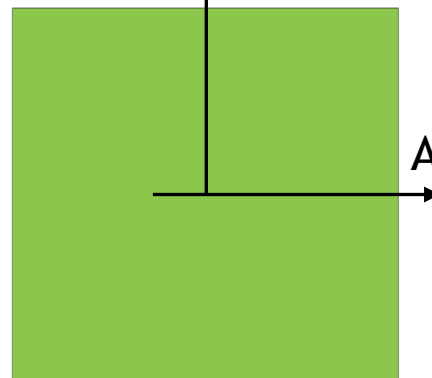
- New parameters ECOAT, TCOAT and SCOAT
- Coating - elastoplastic behavior



Strain rate dependency

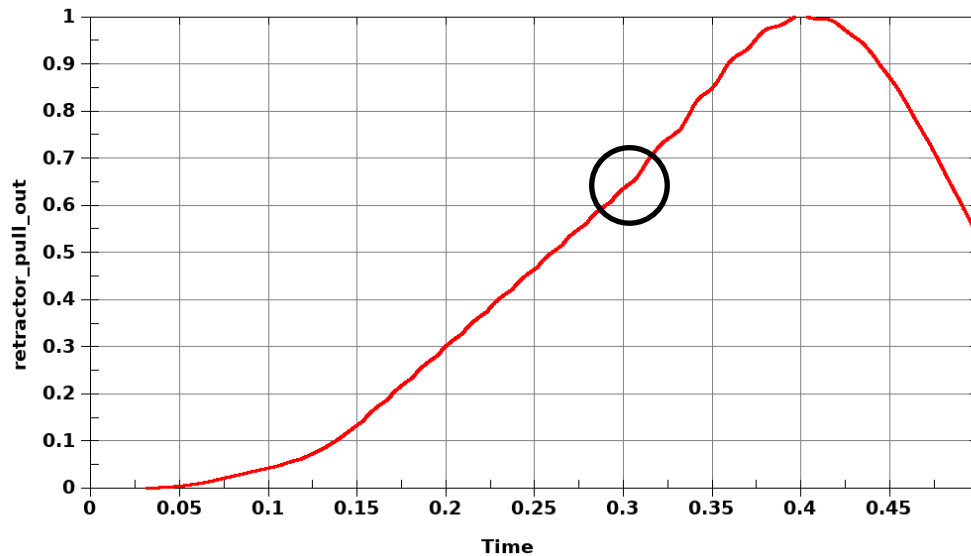


Orthotropic material

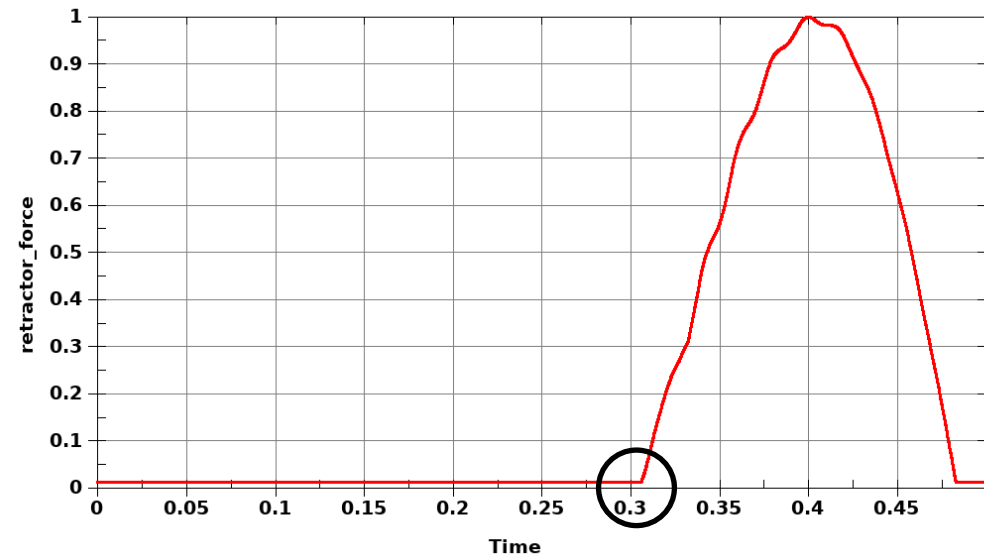


*ELEMENT_SEATBELT_SENSOR

- Retractor locking, and activation of pretensioners, by ELEMENT_SEATBELT_SENSOR has been extended to support also tracing of retractor pullout, i.e. SBSTYP=5
- Below and example with PULMX=0.65, sensor fired at time 0.31



Pull-out



Force



Implicit

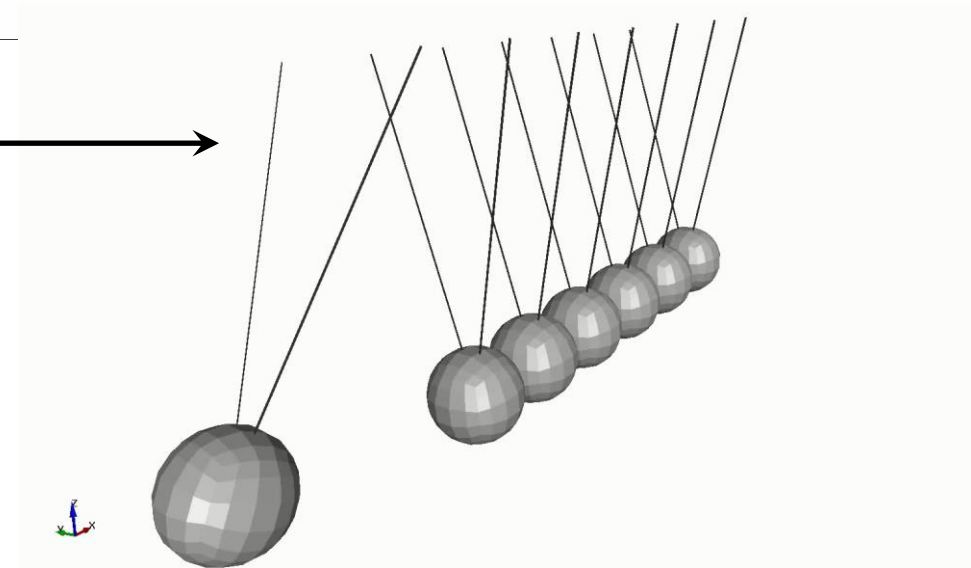
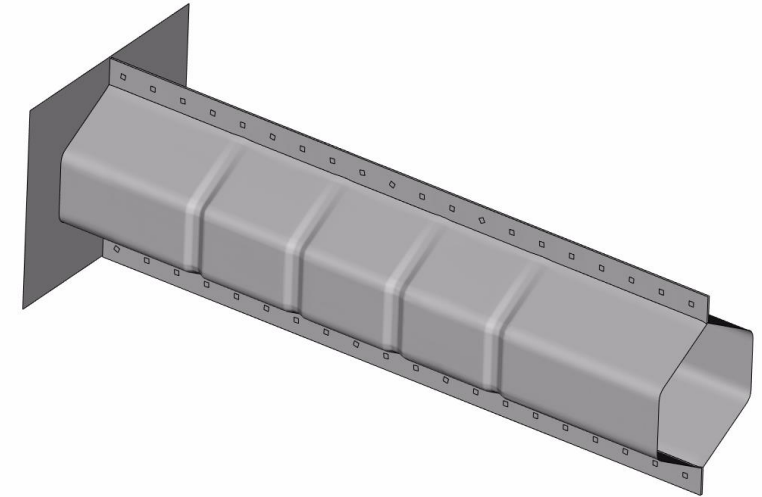
General improvement/curve options/BC

Linear algebra

Rotations

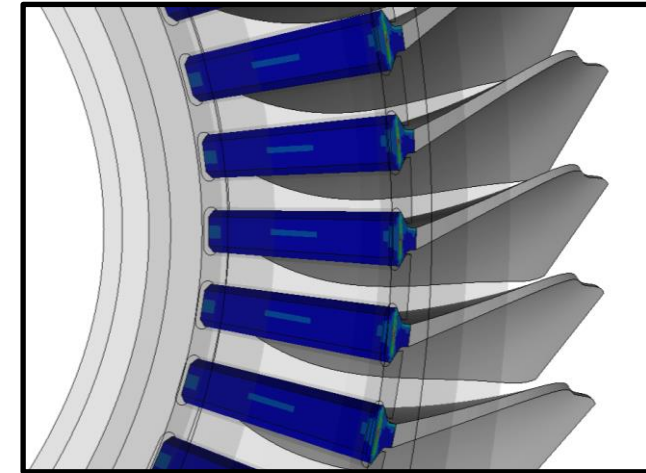
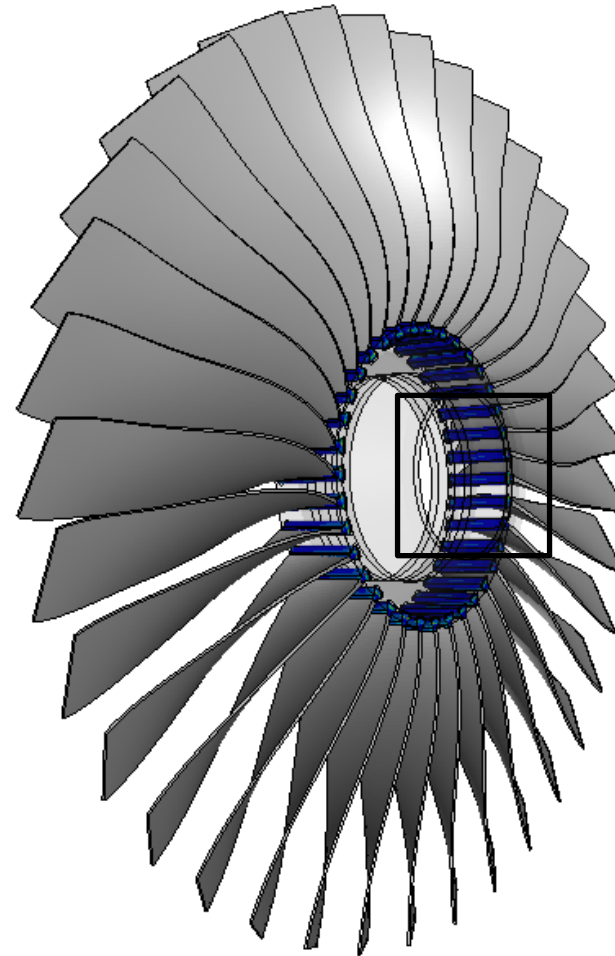
Nonlinear Implicit

- General improvements for numerical accuracy and robustness
 - Contacts, elements, material tangents added or improved as regular maintenance/improvements
- Curve options
 - DTMIN.LT.0 on *CONTROL_IMPLICIT_AUTO - generating keypoints
 - ILIMIT.LT.0 on *CONTROL_IMPLICIT_SOLUTION - switching between BFGS and Full Newton
 - DCTOL/ECTOL/RCTOL.LT.0 on *CONTROL_IMPLICIT_SOLUTION - tolerances as function of time
- Treatment of boundary conditions
 - Prescribed motion and constraints can be applied to rigid body nodes
 - Reaction forces of rigid body and nodal rigid body constraints can be requested
 - SPC2BND=1 on *CONTROL_OUTPUT
 - BNDOUT2DYNAIN on *BOUNDARY_PRESCRIBED_MOTION_RIGID, for porting reaction forces as parameters between simulations



Linear Algebra

- Emphasizing
 - reducing computing resources for solving larger and larger problems
 - performance scalability for higher and higher number of MPI ranks
- Changes should benefit smaller problems
- Example is a 36 blade impeller model with 2.8M nodes and 2.46M solid elements using contact at the root of each fan blade
- 10 Implicit time steps to apply centripetal force loading around the axis of rotation



Result from 36 MPI ranks

R11.1 – 7193 seconds

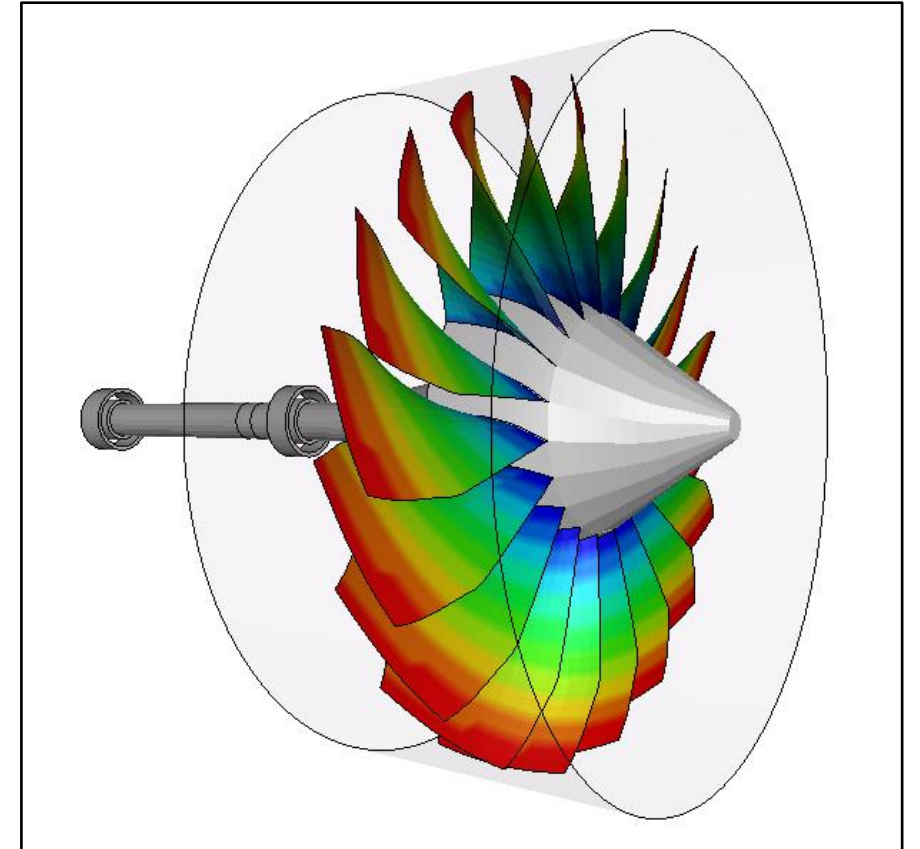
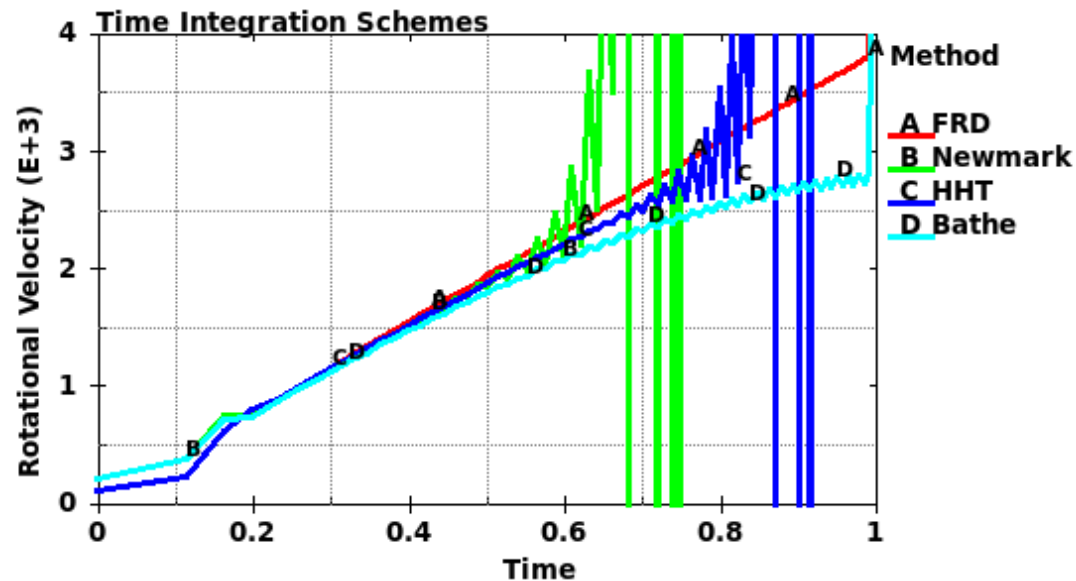
R12.0 – 6913 seconds

Dev – 6541 seconds

Courtesy of Rolls-Royce

Rotations

- Time integration scheme for arbitrarily large rotational increments (FRD)
 - ALPHA on *CONTROL_IMPLICIT_DYNAMICS
 - Generalization of Rotational Dynamics to nonlinear transient
 - Potential for long duration simulation





Contact

Mortar - friction/tied/2D

Mortar - New contact segment due to erosion

Mortar - Output penetration/energy

Mortar - eigenvalue analysis functionality

SOFT=2 edge contact penalty stiffness

Contact updates general

■ User friendliness

- transfer data through binout between simulations
- plots of contact energy (Mortar/Soft=2)
- fringe plot of penetration/gap (Mortar/Soft=2)

■ Functionality consistency between different contact options

- groupable options added for Soft=2 and CAG
- srnde for CAG
- soft=1 for CAG
- part_contact (thickness) for CAG
- ftorque support for Soft=2
- ftorque also for groupable contact

■ Restart functionality

- Soft=2
- Eroding contact

Mortar Contact - General

■ Friction

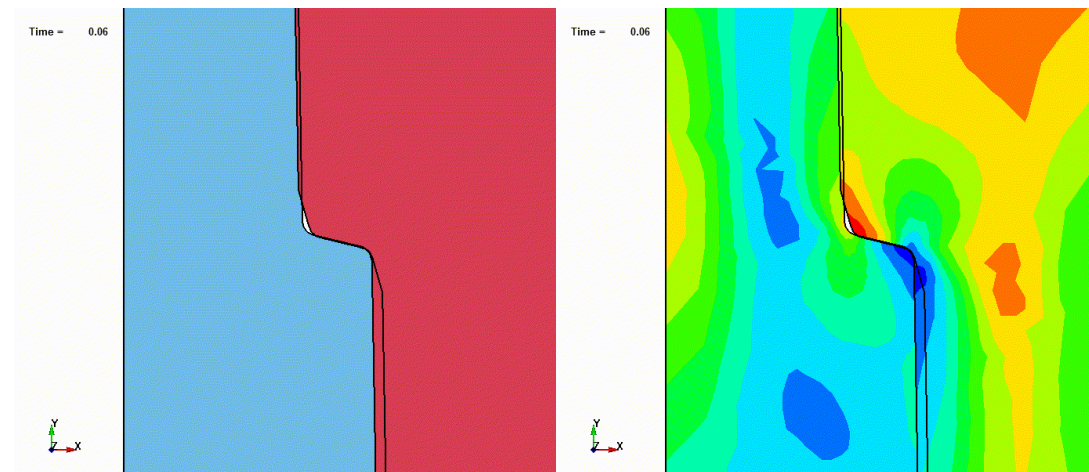
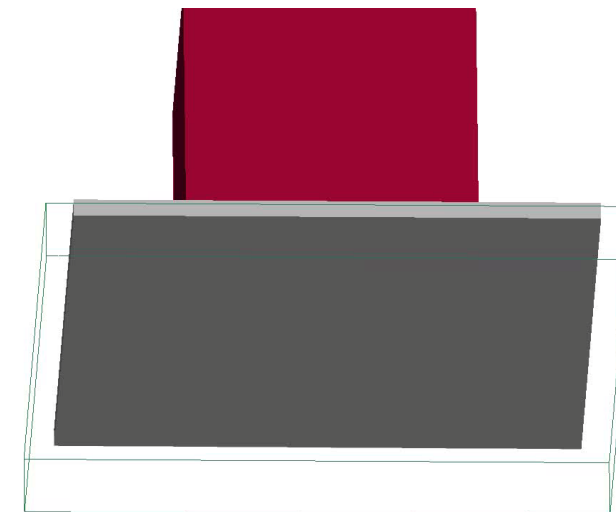
- history variables in user friction can be post processed
- frictional stress limit (VC on *CONTACT) supported

■ Tied weld

- allow general lamination modeling through user interface

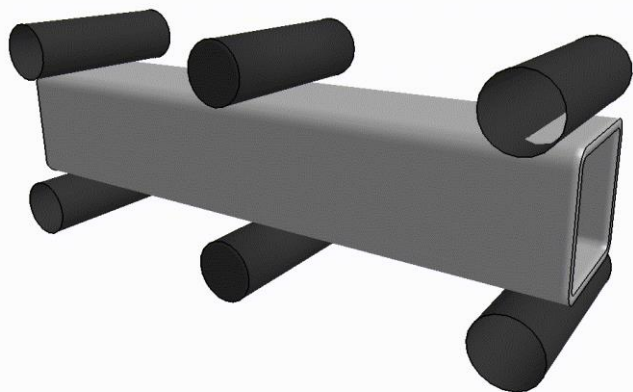
■ 2D mortar contact

- TDPEN introduced, giving the time for depenetration in interference
- This is the analogy to IGNORE=3 for 3D mortar contact

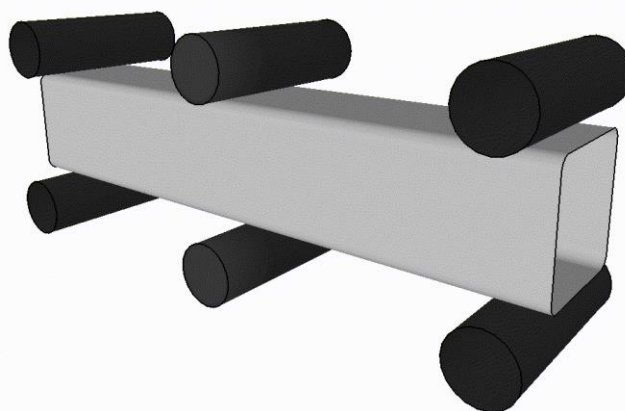


Mortar Contact

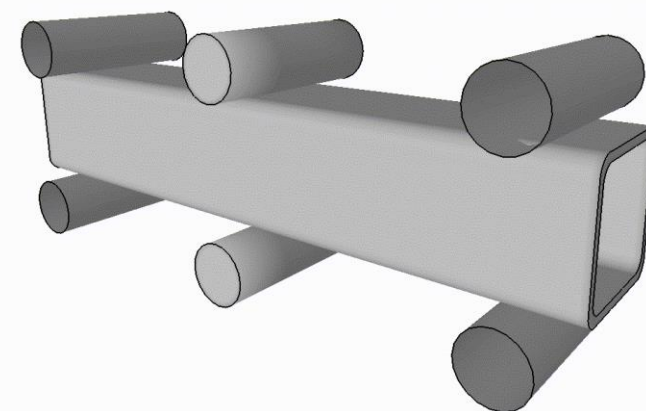
Thick shells



Shells



Solids

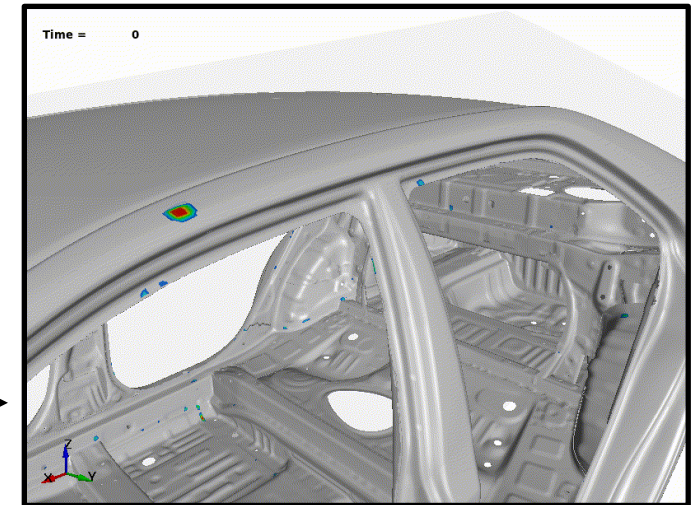
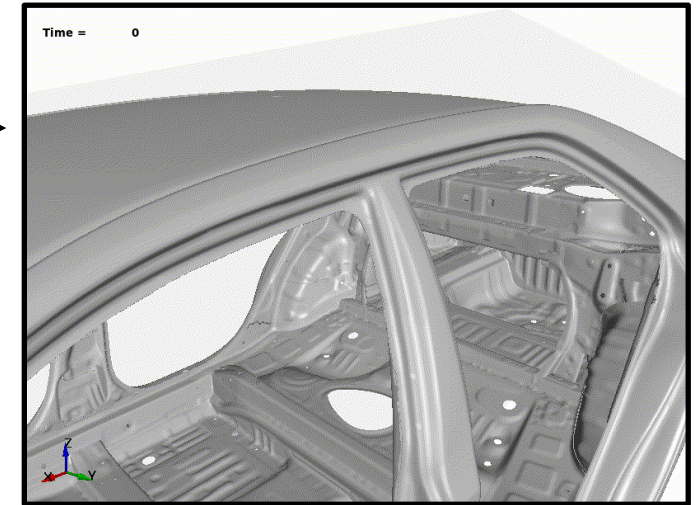
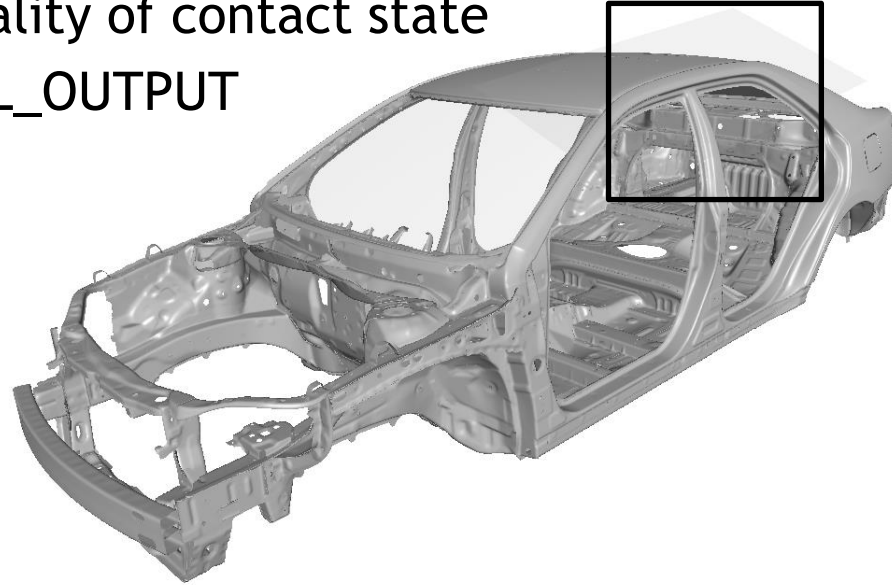


- Exposed segments due to erosion added to the contact
 - Works for solids, shells and thick shells
- For shells, edges of eroded elements are exposed
- Supported for automatic surface to surface and single surface

Mortar Contact

■ Penetrations

- relative and absolute penetrations can be monitored in d3plot
- assessment of the quality of contact state
- PENOUT on *CONTROL_OUTPUT



Results per contact interface is an intfor option

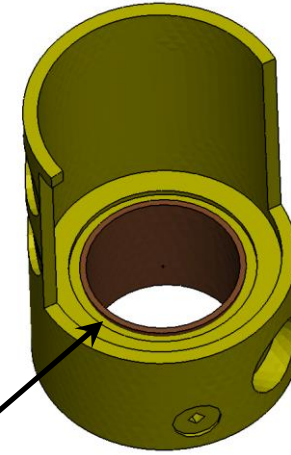
■ Energy

- Contact sliding energy can be monitored in d3plot
- ENGOUT on *CONTROL_OUTPUT

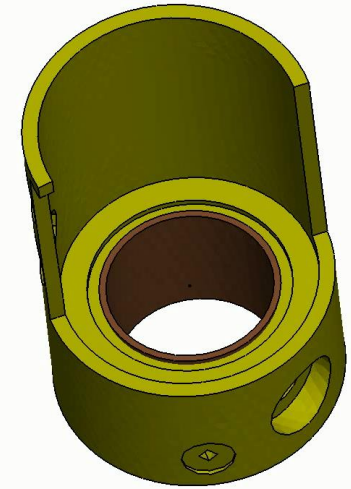
Mortar Contact

- Active contact (non-zero contact pressure) won't affect rigid body modes in an eigenvalue analysis

First deformation mode



Interference in interface

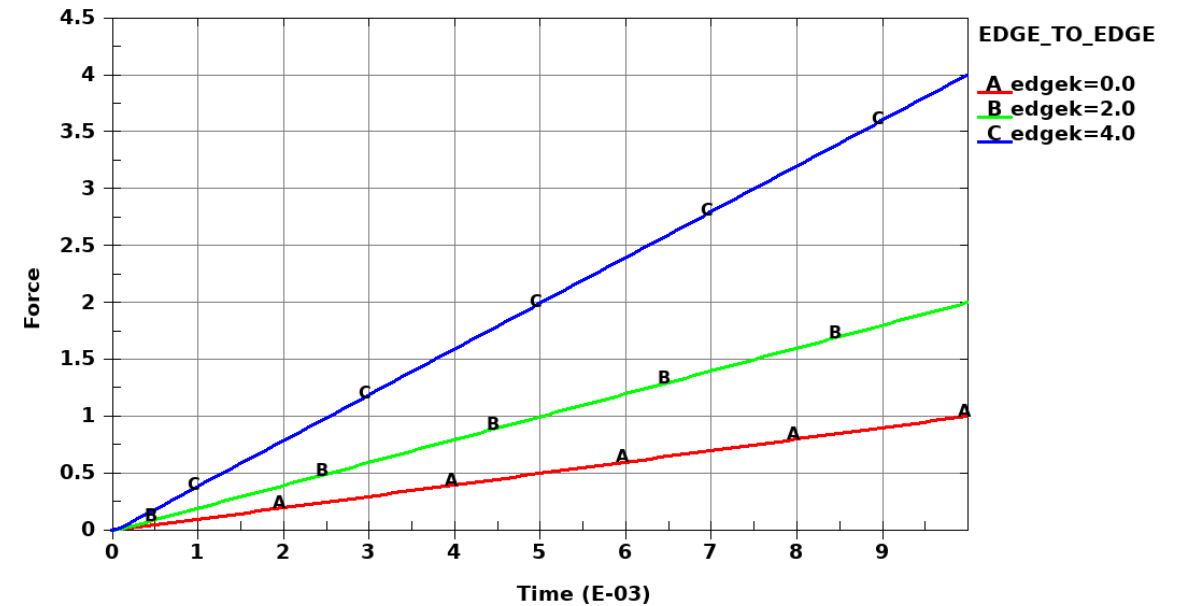
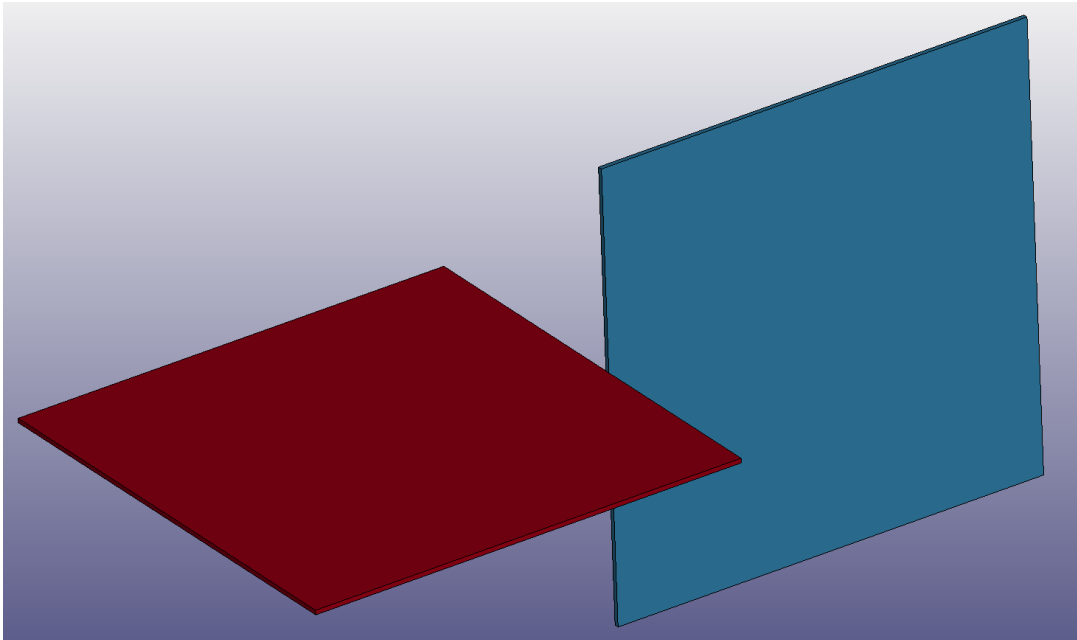


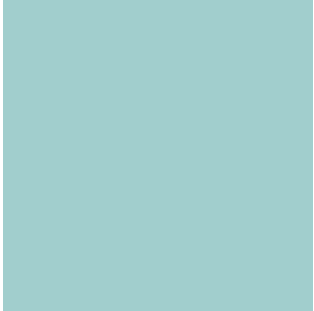
Modes with 0 Hz



*CONTACT, SOFT=2, EDGEK

- *CONTACT, Optional Card C, EDGEK
- Scale factor for edge to edge contact when SOFT=2 and DEPTH=5,15,25 or 35





New features in time domain fatigue

Multiaxial/loadstep

DATABASE_D3FTG - fatigue damage evolution

Time domain fatigue analysis

*FATIGUE_MULTIAXIAL

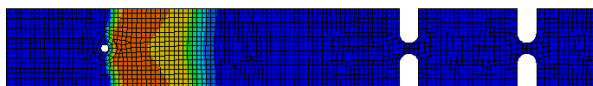
- Critical plane approach

maxial	nplane	Max damage ratio
0		1.26547
1	18	1.30282
1	36	1.30282
1	72	1.30327
1	180	1.30327
2		1.30445

$$F(t) = \sin(22\pi \cdot t)$$

Time = 10000
Contours of Cumulative damage ratio
max IP. value
min=0, at elem# 5462
max=1.26547, at elem# 5291

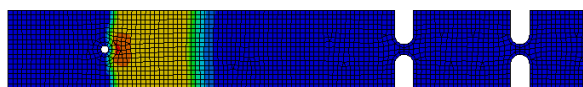
Default



Cumulative damage ratio
1.265e+00
1.139e+00
1.012e+00
8.858e-01
7.593e-01
6.327e-01
5.062e-01
3.796e-01
2.531e-01
1.265e-01
0.000e+00

Time = 10000
Contours of Cumulative damage ratio
max IP. value
min=0, at elem# 5462
max=1.30445, at elem# 5291

Multiaxial



Cumulative damage ratio
1.304e+00
1.174e+00
1.044e+00
9.131e-01
7.827e-01
6.522e-01
5.218e-01
3.913e-01
2.609e-01
1.304e-01
0.000e+00

*FATIGUE_MULTIAXIAL

Card 1	1	2	3	4	5	6	7	8
Variable	MAXIAL	NPLANE						
Type	I	I						
Default	0	18						

VARIABLE

MAXIAL

DESCRIPTION

Multiaxial fatigue analysis criterion:

EQ.0: Fatigue analysis using equivalent stress or strain index (defined by INDEX in *FATIGUE)

EQ.1: Fatigue analysis on multiple planes

EQ.2: Fatigue analysis on critical plane which is determined by the highest 1st principal stress or strain

NPLANE

Number of planes for fatigue analysis (for MAXIAL = 1 only)

*FATIGUE_LOADSTEP

Card 1	1	2	3	4	5	6	7	8
Variable	TSTART	TEND	TEXPOS					
Type	F	F	F					
Default	none	none	0.0					

VARIABLE

TSTART

Start time of current load step

TEND

End time of current load step

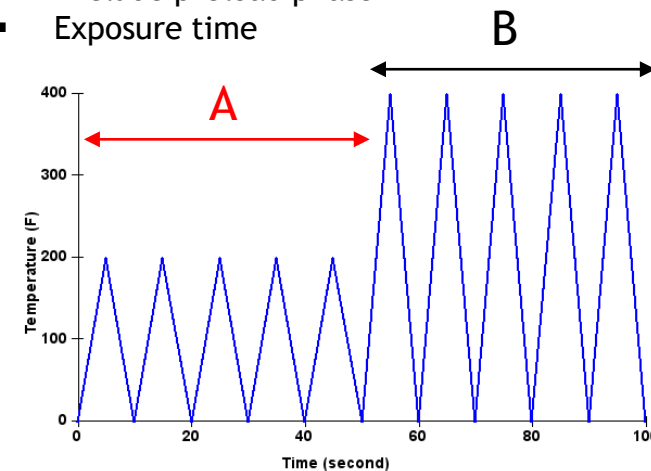
TEXPOS

Exposure time of current load step

EQ.0.0: set to TEND - TSTART (default).

*FATIGUE_LOADSTEP

- Divide load history into fatigue load steps
- Exclude preload phase
- Exposure time



```

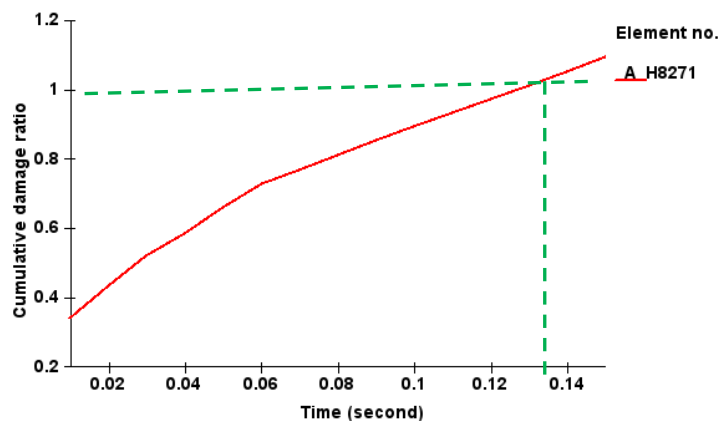
*FATIGUE_LOADSTEP
$#  tstart  tend  texpos
    0.0    50.  10000.
   50.0   100.  20000.

*DEFINE_CURVE
$#  lcid  sldr  sfa  sfo
    888    0   1.0   1.0
$#          al  ol
          0.0  0.0
          5.0  200.0
          10.0  0.0
          15.0  200.0
          20.0  0.0
          25.0  200.0
          30.0  0.0
          35.0  200.0
          40.0  0.0
          45.0  200.0
          50.0  0.0
          55.0  400.0
          60.0  0.0
          65.0  400.0
          70.0  0.0
          75.0  400.0
          80.0  0.0
          85.0  400.0
          90.0  0.0
          95.0  400.0
         100.0  0.0
    
```

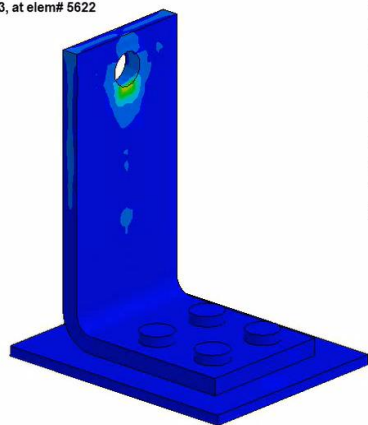
Time domain fatigue analysis

*DATABASE_D3FTG

- Accumulation of fatigue damage, DT>0



Time = 0.020007
Contours of Cumulative damage ratio
min=0, at elem# 475
max=0.735543, at elem# 5622



Cumulative damage ratio
1.000e+00
9.000e-01
8.000e-01
7.000e-01
6.000e-01
5.000e-01
4.000e-01
3.000e-01
2.000e-01
1.000e-01
0.000e+00

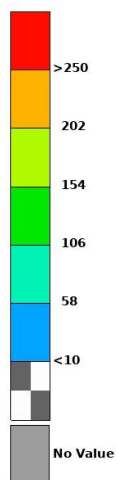


Fatigue analysis of non-linear problems are possible in LS-DYNA using the time domain fatigue solver.

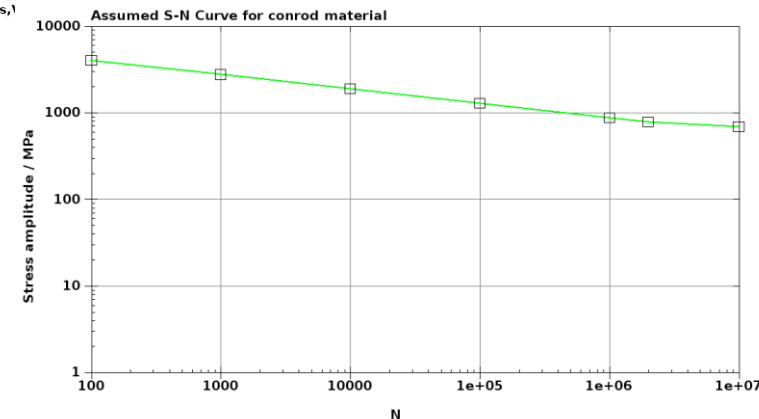
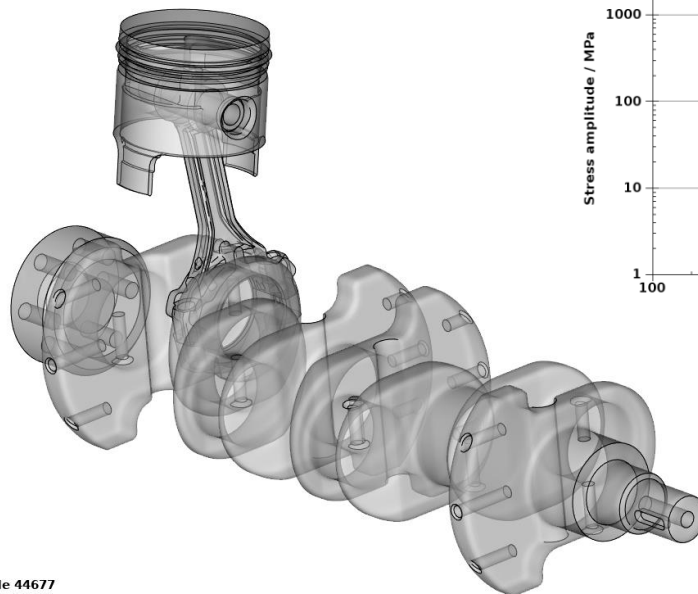
Use stress/strain histories from the complete model, a subset of parts or selected elements.

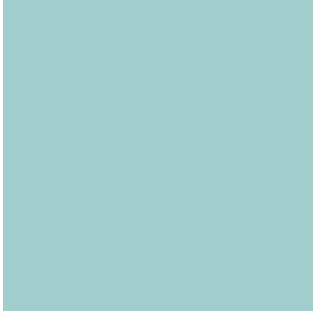
Use D3PLOT or ELOUT as input to rain flow counting and fatigue damage calculation.

0:d3plot : 190121-test of crankshaft, and conrod, rotation, modified contacts : Scalar: Stresses,1
0.00000000E+00



Visible Max 0 at Node 44677
Visible Min 0 at Node 44677





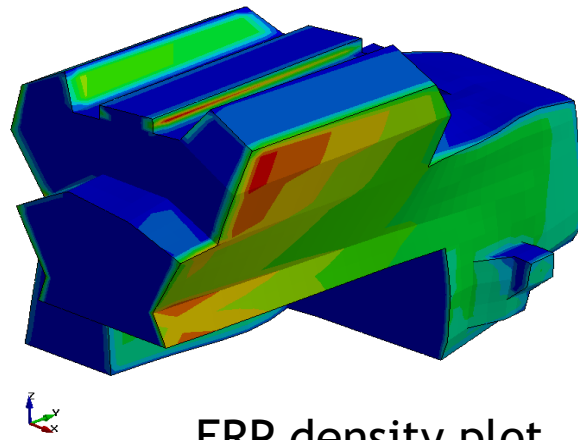
Frequency Domain

ERP - Energy Radiated Power

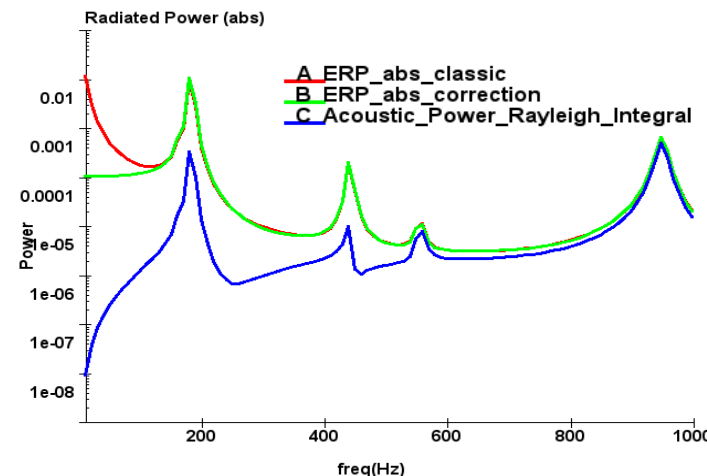
SEA - Statistical Energy Analysis

Frequency domain

- Random vibration analysis (*FREQUENCY_DOMAIN_RANDOM_VIBRATION) of multiple loads of different type by the _SUMMATION option for *DATABASE_FREQUENCY_BINARY_{D3PSD,D3RMS}
- Response spectrum analysis (*FREQUENCY_DOMAIN_RESPONSE_SPECTRUM) : Individual modal responses are now available from a response spectrum analysis in the d3spcm binary database
- More options for *FREQUENCY_DOMAIN_SSD_ERP

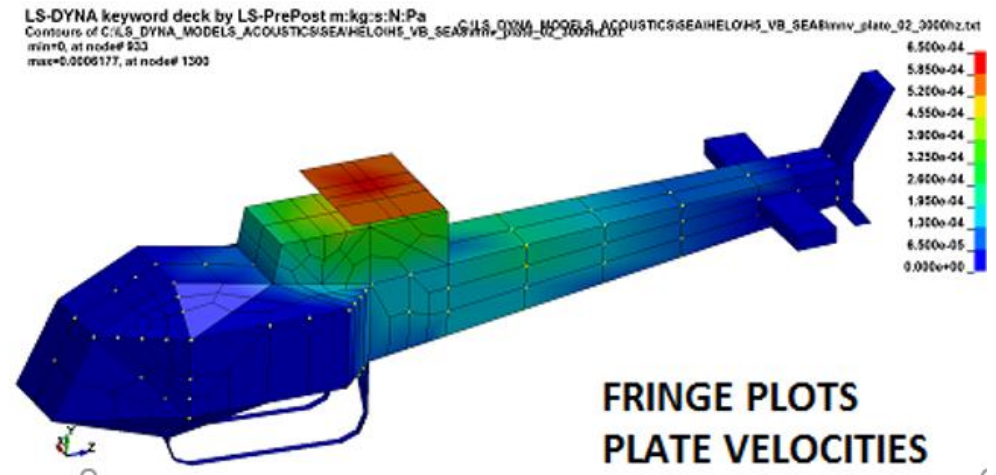
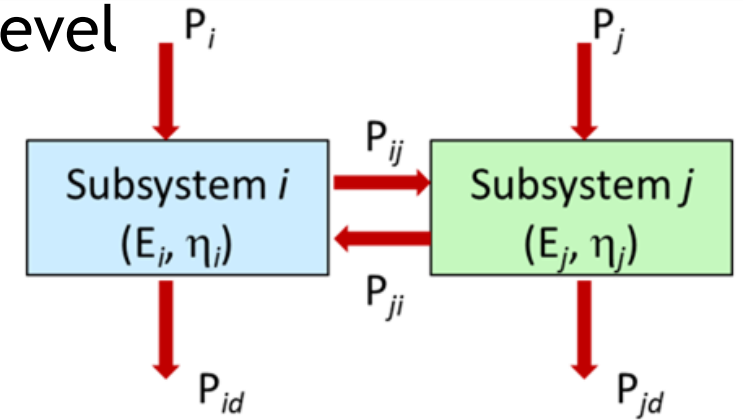


ERP density plot



Frequency domain: Statistical energy analysis (SEA)

- Statistical energy analysis (SEA) is introduced from R12.0
- For analyses of high frequency vibrations on a system level
 - Including acoustics
- Based on energy equilibrium
- New keywords:
 - `*FREQUENCY_DOMAIN_SEA_SUBSYSTEM`
 - `*FREQUENCY_DOMAIN_SEA_CONNECTION`
 - `*FREQUENCY_DOMAIN_SEA_INPUT`





Forming simulations

In-core adaptivity

One step method for carbon fiber reinforced composites

Airbag modification for fluid cell forming

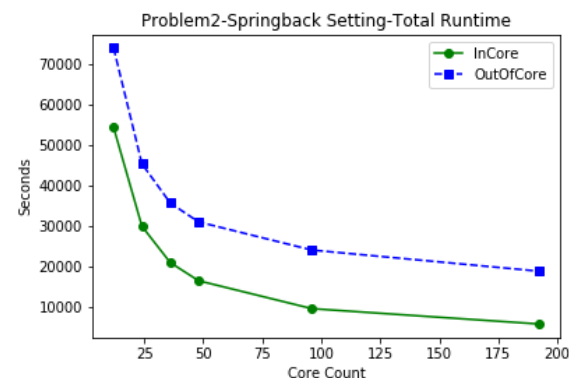
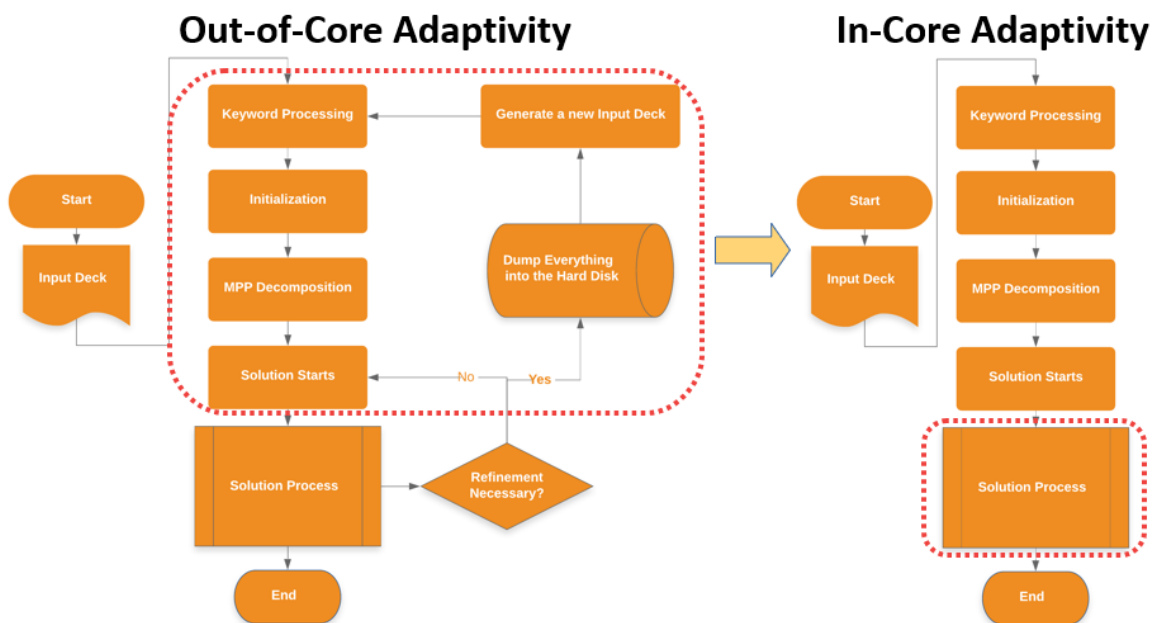
Solid to solid mapping

New Forming Analysis keywords

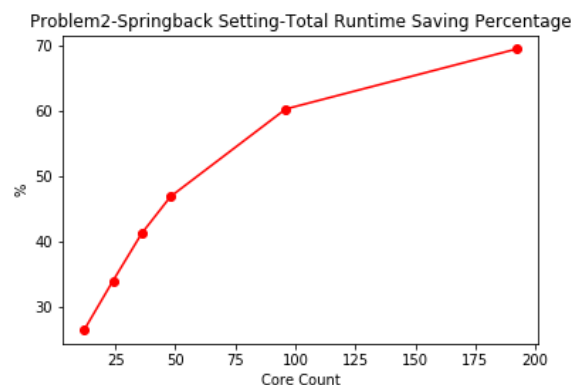
Moving Temperature Boundary Condition

In core adaptivity

- Instead of dumping the simulation to hard drive, remeshing, re- initializing and performing mpp decomposition each adaptive step, the mesh adaptivity is done in- core without shutting down and restarting the simulation.
- Time saving percentages vary for different problems and core counts, but 30 - 70 % increase in speed seems viable.
- Activated by **INMEMORY** flag on ***CONTROL_ADAPTIVE** (currently *mpp, shell h-adaptivity*)



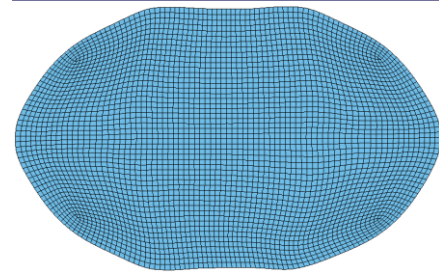
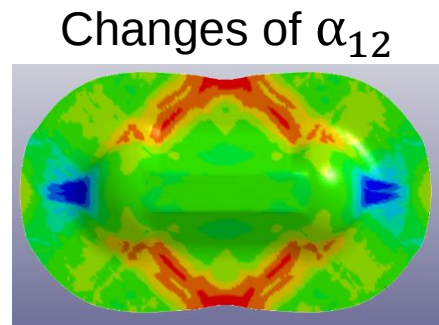
Saving: 27%–70%



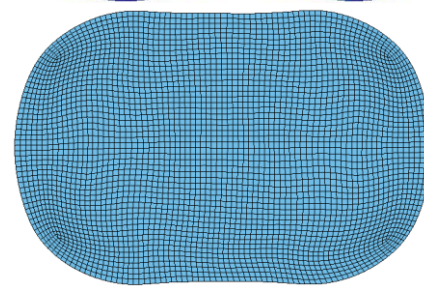
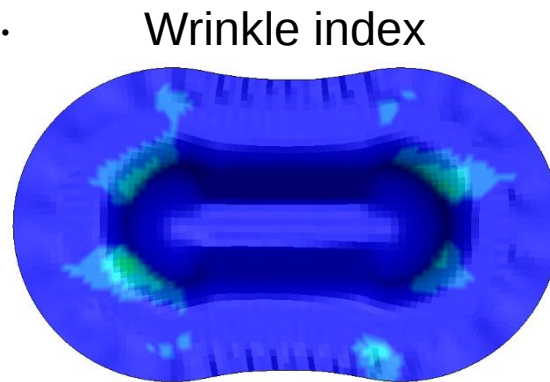
- Initial blank 8186 elements
- Formability setting:
 - 151K cycles and 435 adaptive steps
 - Finished with 1010K elements

One step method for carbon fiber reinforced composites

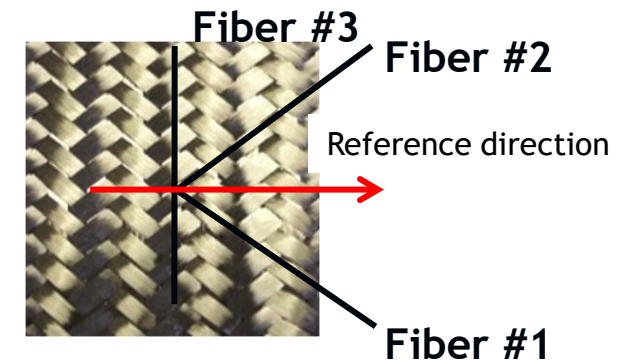
- A one-step method was developed to simulate carbon fiber- reinforced composites with two/three major fiber directions. It is used to inversely predict the initial blank size/shape and fiber angle on the formed parts.
- Matrix and fiber behavior is separated where the matrix can currently be described by *MAT_24 or *MAT_37 and the fiber is modeled as elastic.
- The fiber directions are input together with a normal and shear stiffness through *DEFINE_FIBER keywords
- To better account for the effects of the embedded fibers, the rotation of a local representative “fiber” within a generic element is considered.



Flat blank 45°, 135°



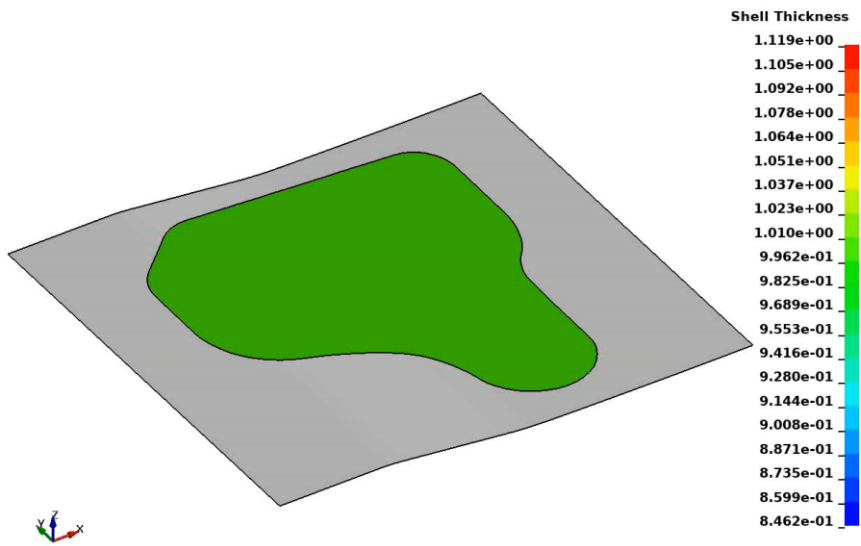
Flat blank 0°, 90°



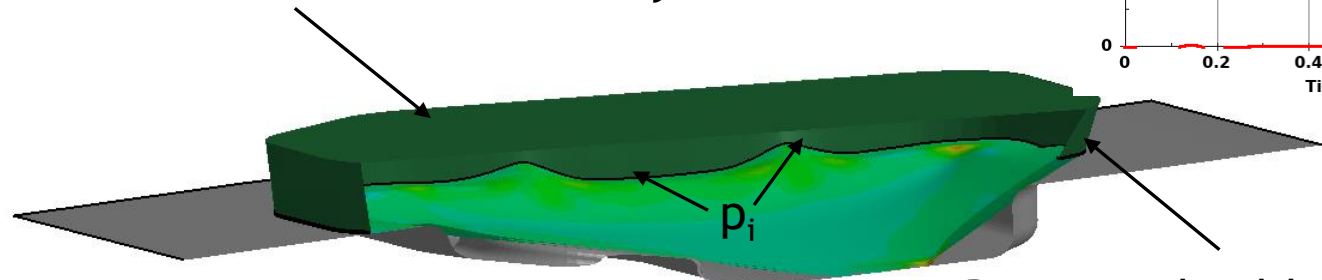
(Courtesy of Dr. D. Zeng from Ford)

Airbag modification for fluid cell forming

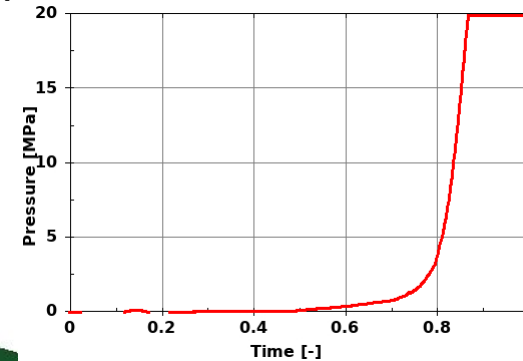
- Fluid cell forming is a process that utilizes a fluid-cell press, or bag press, to form sheet metal on to a die by pressurizing a rubber diaphragm.
- Similar to hydroforming, prescribing a pressure makes this a force controlled process and thus difficult to control.
- By defining a cavity using null elements and using the *AIRBAG_LINEAR_FLUID keyword with a prescribed mass flow into the cavity, the process becomes displacement controlled.
- By using the NONULL option on the *AIRBAG_LINEAR_FLUID keyword, the pressure is only applied on the blank which removes the unphysical stretching of the blank due the pressure load on the null elements.



Null elements define the cavity

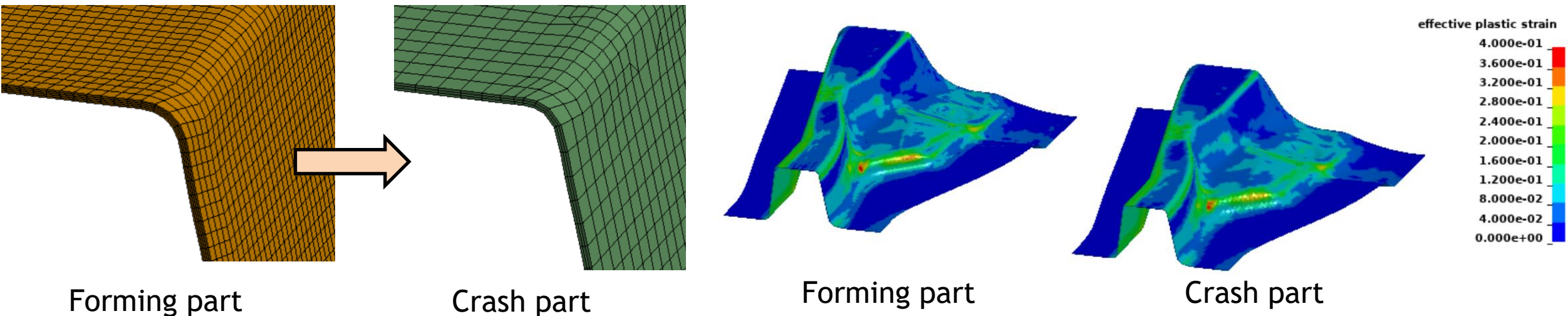


Pressure should not be applied to the null elements since it stretches the blank



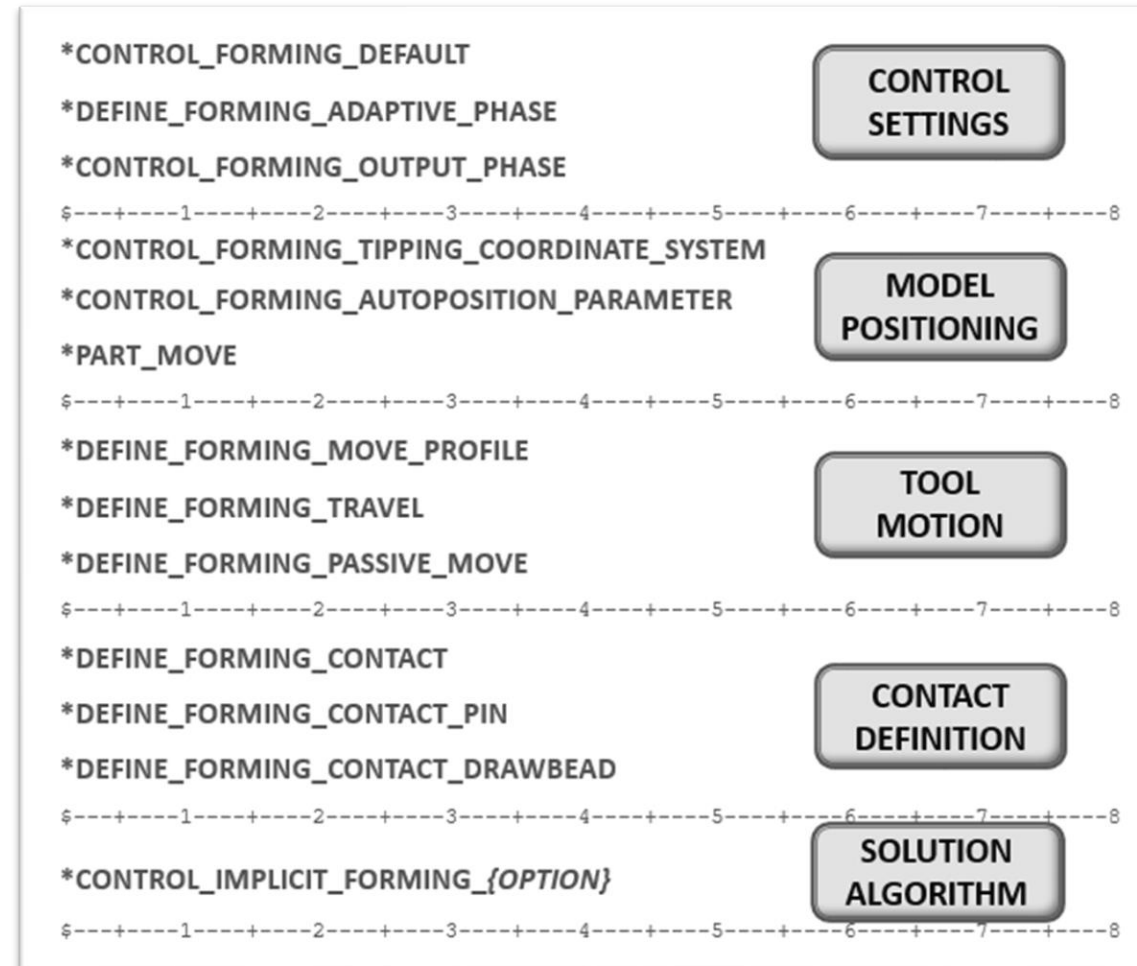
Solid to solid mapping

- The usage of solid elements for forming simulations is increasing
 - Modelling of thick parts where the plane stress assumption is not valid
 - Material failure
 - Utilizing the mass-scaling features and parallelization of LS-DYNA to reduce simulation time
- Constant improvements in e.g. trimming, adaptivity and material models (e.g. *MAT_YLD_2000) closes the gap between forming simulations using shells and solids.
- Novel mapping keyword `*INCLUDE_STAMPED_PART_SOLID_TO_SOLID` maps Stress and strain tensor, history variables and plastic strain from a solid (source) part to a second solid target part (*hex and penta elements*).
- The total thickness of the target part is adjusted to match the thickness of the source part.



New Forming Analysis keywords

- LST is bundling forming input into novel keywords to simplify control settings and input for forming applications
- The forming operation is divided into phases and specific settings and movements can be defined for each phase.
- Mass-scaling, shell defaults, contacts defaults, database output etc. are defined in the simplified control setting keywords.
- For model positioning, the **_AUTOPOSITION** keyword together with ***PART_MOVE** separates the tooling and stores the tool opening movement in a parameter that can be used for the tool motion in ***DEFINE_FORMING_TRAVEL**.
- Simple definition of guide pin and analytical drawbead contacts.
- ***CONTROL_IMPLICIT_FORMING** tailors the implicit settings for gravity loading and binder wrap simulations for fast convergence.
- ***CONTROL_IMPLICIT_FORMING_AUTO_CONSTRAINT** allows the user to do springback analysis without boundary constraints to eliminate rigid body motion.



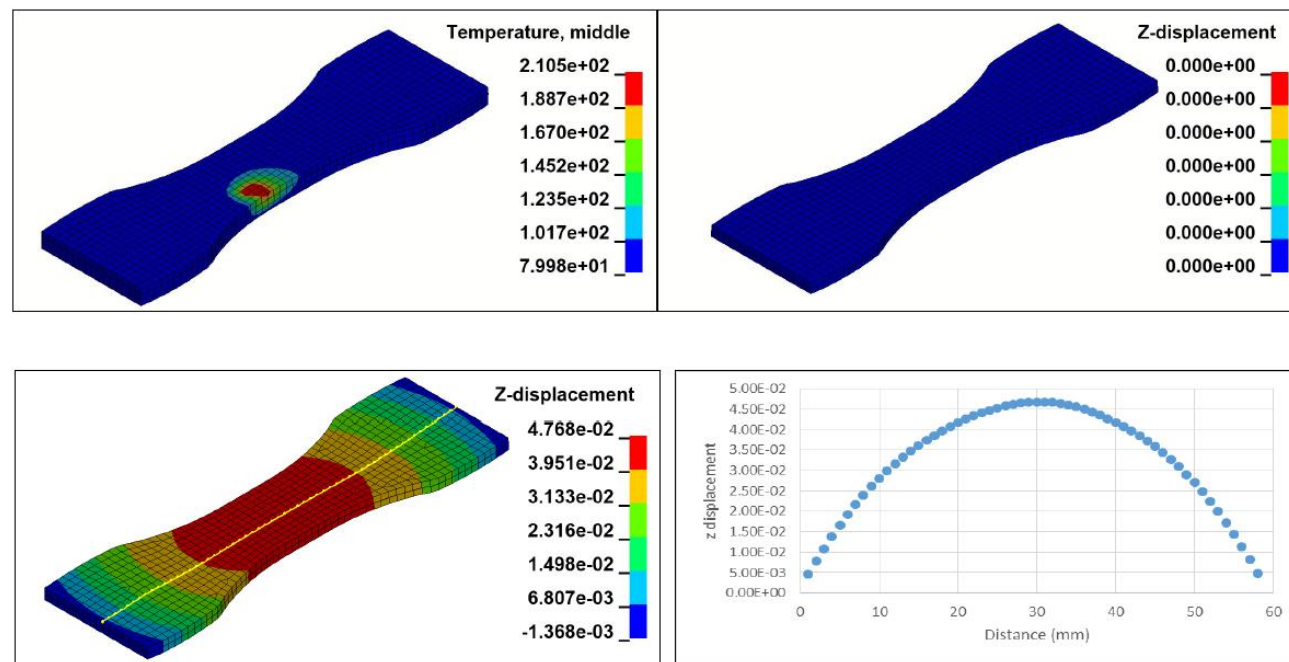


Additive manufacturing

New remeshing algorithm/New remapping scheme
Adaptivity
Thermo-mechanical coupling
Moving temperature boundary condition

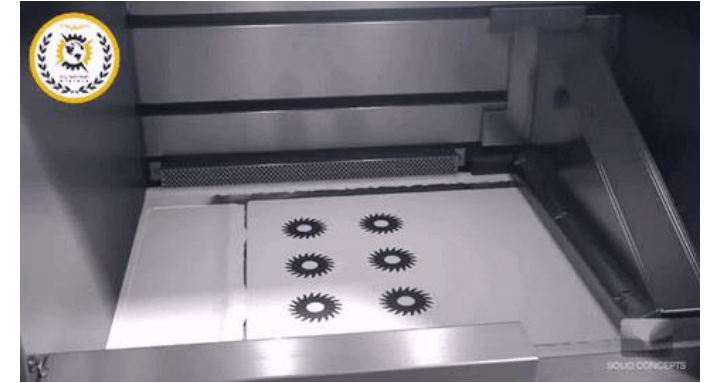
Moving Temperature Boundary Condition

- Novel keyword `*TEMPERATURE_BOUNDARY_TRAJECTORY` to apply temperature boundary condition on a moving volume
 - Fixed or time varying
 - Applied to nodes enclosed in a specified volume (*cylinder, block, etc*)
 - The volume is prescribed to move along a designated nodal path with fixed or time-varying speed.
- Can be used together with e.g. `*MAT_CWM` and `_TIED_WELD` contact option to bond the layers to simulate fused filament fabrication

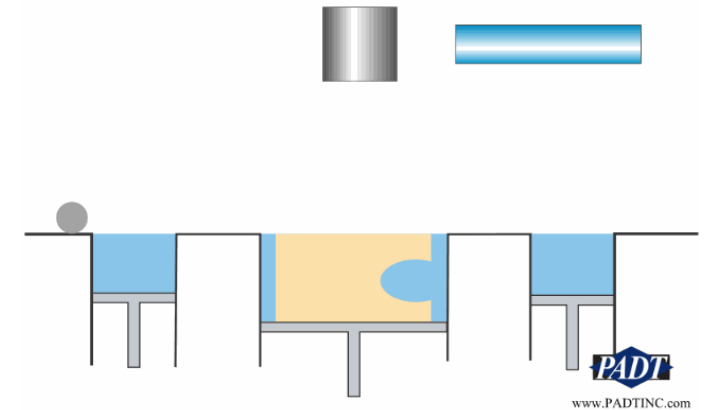


Additive manufacturing

- New remeshing algorithm
 - Dynamic local refinement following heat source
 - Mesh activation through adaptivity
 - Multi-body and multi-part remeshing
- New remapping scheme
 - Mechanical and thermal internal variables
 - Deformation profile
- Multiple heat sources enabled
- Implicit thermo-mechanical couple analysis
- Spring back analysis
- Related keywords involving new development
 - *INCLUDE_AM_BLUEPRINT, *DEFINE_ADAPTIVE_BOX
 - *BOUNDARY_THERMAL_WELD
 - *BOUNDARY_CONVECTION_SET, *BOUNDARY_RADIATION_SET



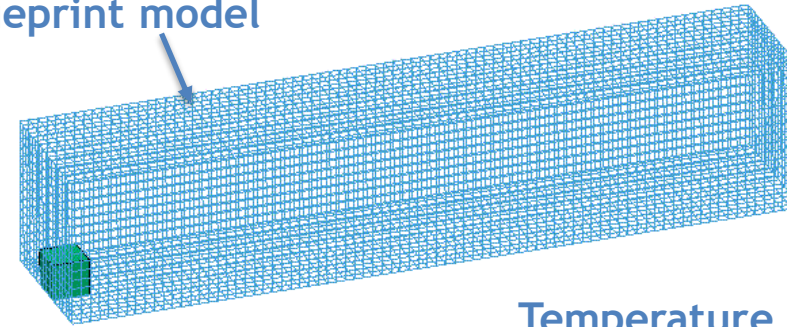
Selective Laser Sintering (SLS)



Additive manufacturing

■ Numerical examples

Blueprint model

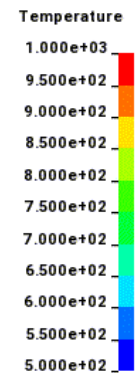
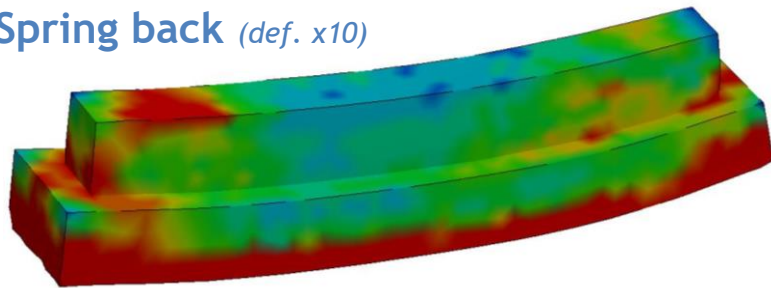


Temperature (20~1600 °C)



Von Mises stress

Spring back (def. x10)



Temperature



Thermal

*BOUNDARY_CONVECTION/RADIATION/FLUX

*BOUNDARY_FLUX_TRAJECTORY

Thermal solver enhancements

*LOAD_THERMAL_RSW

*MAT_GENERALIZED_PHASE_CHANGE (MAT_254)

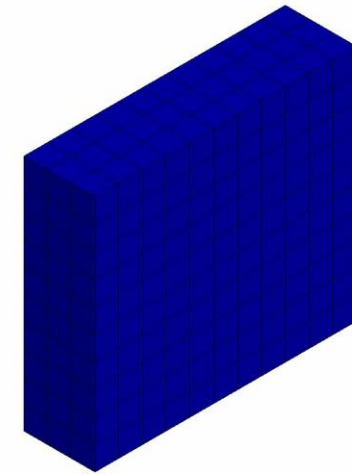
*MAT_THERMAL_ISOTROPIC_TD_LC (*MAT_T10)

Temperature dependent materials

Dealing with solid element erosion in thermal boundary conditions

- New parameter PSEROD for standard thermal boundary conditions (*BOUNDARY_CONVECTION, *BOUNDARY_RADIATION, *BOUNDARY_FLUX)
 - Points to a part set
 - Any new segment attached to an element in this part set, will inherit boundary condition
 - Original input data is used for newly segments
- *BOUNDARY_FLUX is now usable to simulate laser cutting applications
 - Definition of a moving heat source possible but very complicated
 - Rotation of the laser hard to capture

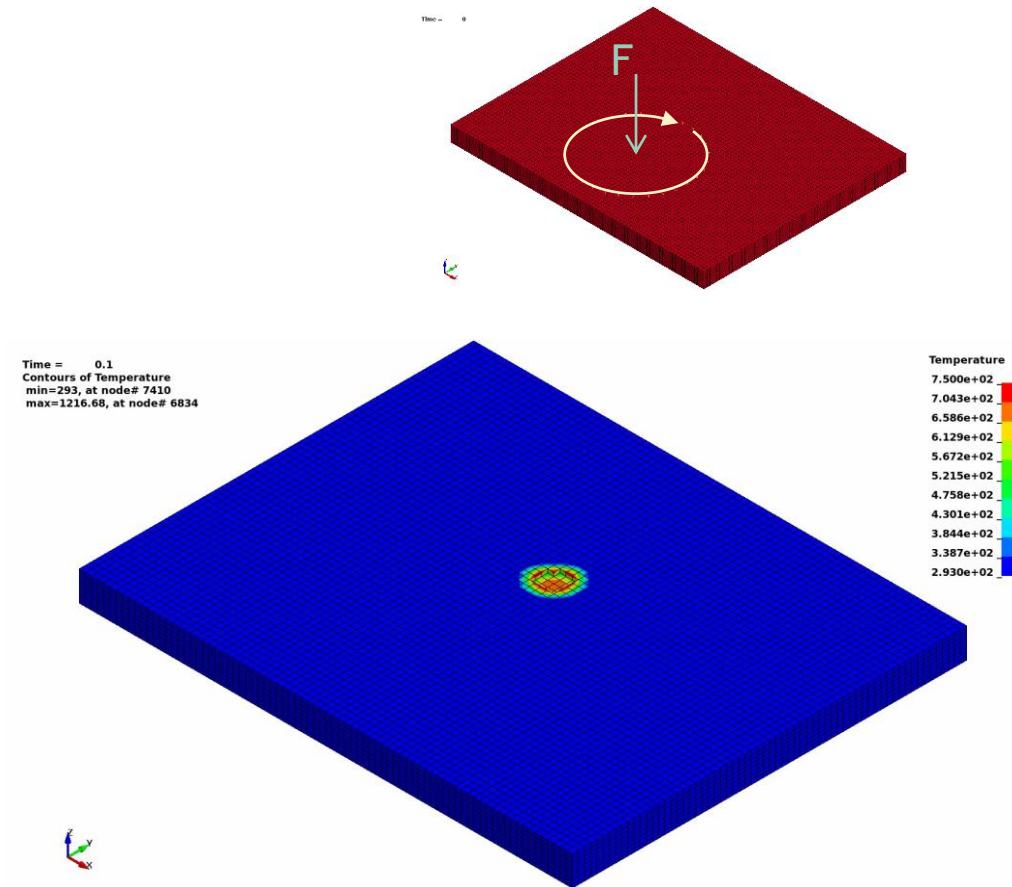
Time = 0
Contours of Temperature
min=200, at node# 67
max=200, at node# 67



Temperature
2.000e+02
2.000e+02
2.000e+02
2.000e+02
2.000e+02
2.000e+02
2.000e+02
2.000e+02
2.000e+02
2.000e+02

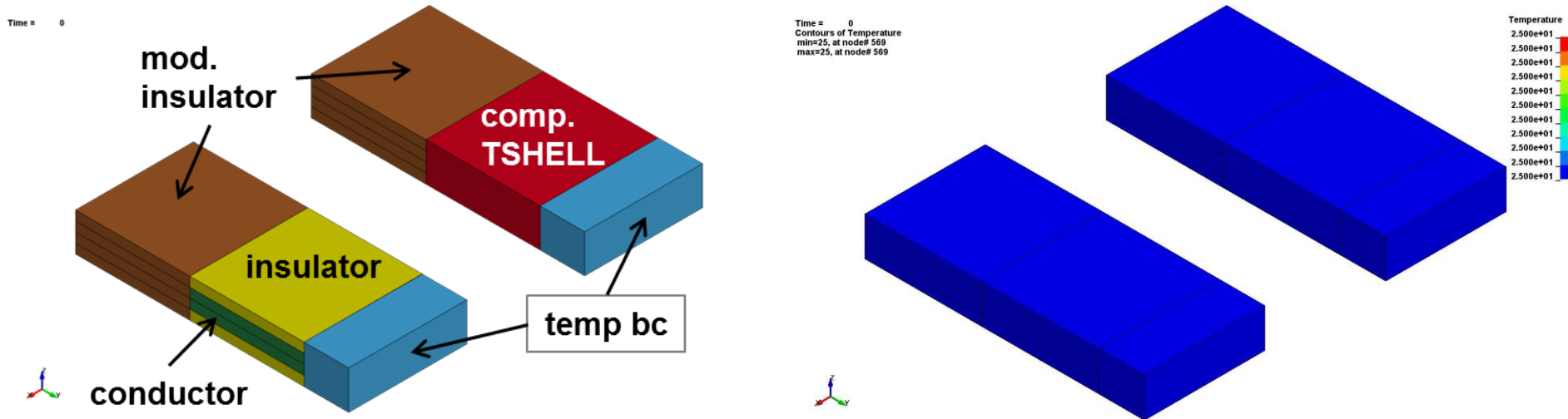
*BOUNDARY_FLUX_TRAJECTORY

- Tailored boundary condition for laser heat treatment and laser cutting
- Surface flux boundary condition that follows prescribed path and orientation
- Propagation to newly exposed segments after element erosion
- Surface heat density
 - Predefined distribution functions
 - User-defined functions
- Tilting of heat source is accounted for
 - Changes projection of beam on surface
 - Heat density can be automatically adapted



Thermal Solver - Miscellaneous

- Contact routines for thermal composite TSHELL elements
 - Composite lay-up internally reconstructed with virtual elements and nodes
 - For “edge”-contact virtual contact surfaces used



- Definition of heat generation function in local coordinates
 - *LOAD_HEAT_GENERATION accepts ID of a reference node in parameter RFNODE
 - Current coordinates of reference node can be referred to in user-defined function

*LOAD_THERMAL_RSW for resistance spot welding simulation

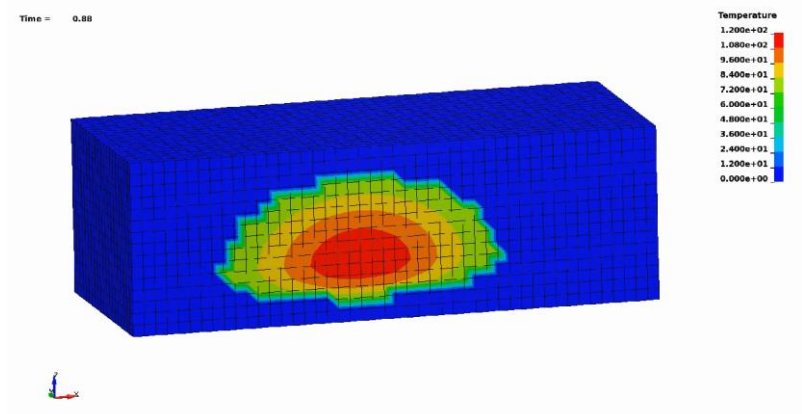
- Simplification of thermal boundary condition *BOUNDARY_TEMPERATURE_RSW

- Direct definition of the temperature profile in the weld nugget as thermal load in structure-only simulation

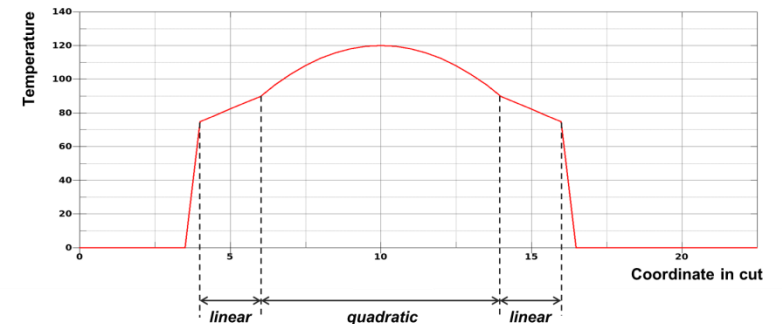
- Prescribed at the center, boundary of nugget, and boundary of HAZ
- Default temperature used outside HAZ
- Default temperature before birth time and after death time of loading condition

- No heat transfer into surroundings

linear temp increase,
IRTH=0.1, DEATH=0.9

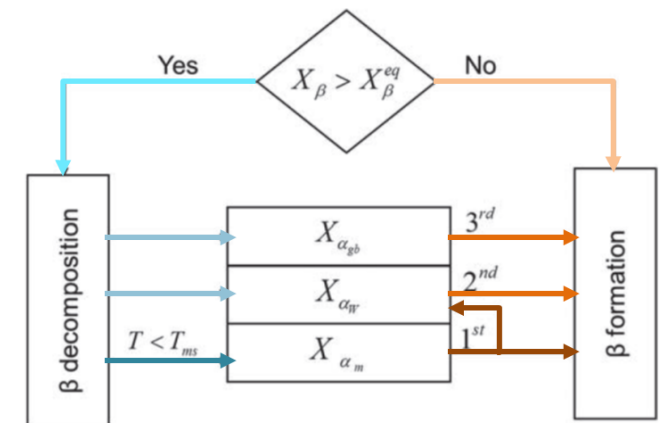


peak temp. profile,
horizontal cut



*MAT_GENERALIZED_PHASE_CHANGE / *MAT_254

- Plastic strain can accelerate/decelerate phase transformation speed
- Parameter ANOPT: define a cut-off temperature for thermal expansion
- Additional history variables for post-processing, e.g. accumulated (thermal) strain data; output controlled by parameter POSTV
- Enhanced annealing option: reset plastic strains based on evolution equation
- New phase transformation laws for titanium Ti-6Al-4V
 - Step-wise dissolution of a group of phases into one target phase
 - Interacting transformations from one common source phase



[C. Charles Murgau,
PhD-thesis, 2016]

*MAT_THERMAL_ISOTROPIC_TD_LC (*MAT_T10)

■ Load curves can now depend on mechanical history variables

This card is included if TGHSV > 0 (see Card 2).

Card 1b	1	2	3	4	5	6	7	8
Variable	TMID	TRO	TGRLC	TGMULT	TLAT	HLAT		
Type	A8	F	F	F	F	F		

Card 2	1	2	3	4	5	6	7	8
Variable	HCLC	TCLC	HCHSV	TCHSV	TGHSV			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
TMID	Thermal material identification. A unique number or label not exceeding 8 characters must be specified.
TRO	Thermal density: EQ.0.0: default to structural density
TGRLC	Thermal generation rate curve number (see *DEFINE_CURVE): NE.0: function of mechanical history variable TGHSV EQ.0: use mechanical history variable TGHSV times constant multiplier value TGMULT.
TGMULT	Thermal generation rate multiplier: EQ.0.0: no heat generation

VARIABLE	DESCRIPTION
HCLC	Load curve ID specifying specific heat as a function of temperature, or, if HCHSV > 0, as a function of a mechanical material history variable HCHSV
TCLC	Load curve ID specifying thermal conductivity as a function of temperature, or if TCHSV > 0, as a function of a mechanical material history variable TCHSV
HCHSV	Optional: mechanical history variable # used by HCLC
TCHSV	Optional: mechanical history variable # used by TCLC
TGHSV	Optional: mechanical history variable # used by TGRLC

Temperature dependent materials

- *MAT_106
 - Define up to eight user-defined history variables referencing to *DEFINE_FUNCTION
- *MAT_270
 - Parameter ANOPT that allows defining a cut-off temperature for thermal expansion
 - Additional history variables for post-processing, output controlled by parameter POSTV
- *MAT_277
 - Arrhenius shift function as alternative to the WLF shift function
 - Curing induced heating
- *MAT_278
 - Curing induced heating
 - Reimplementation of solid formulation

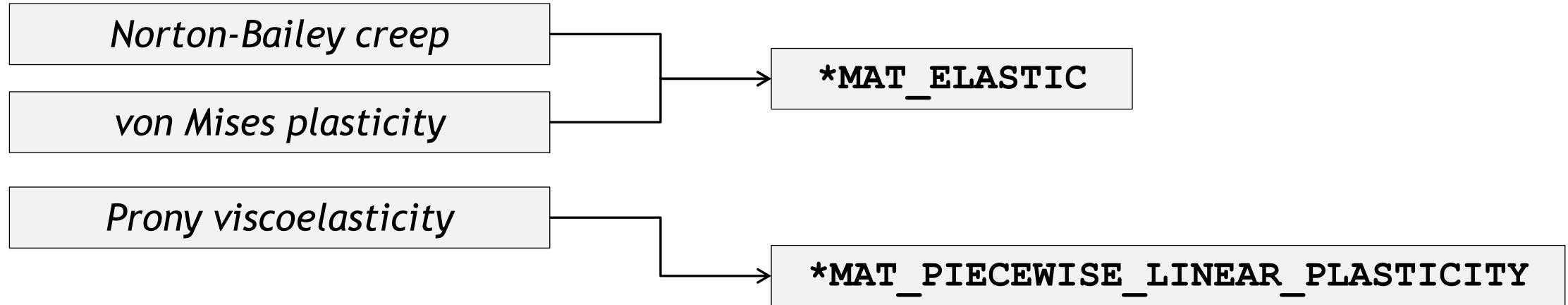
Materials

- *MAT_ADD_INELASTICITY
- *MAT_ADD_DAMAGE_GISSMO
- *DEFINE_ELEMENT_EROSION
- *MAT_SHAPE_MEMORY
- *MAT_SHAPE_MEMORY_ALLOY (*MAT_291)*2
- *MAT_LAMINATED_COMPOSITE_FABRIC_SOLID
- *MAT_ANISOTROPIC_HYPERELASTIC
- *MAT_DISCRETE_BEAM_POINT_CONTACT
- *MAT_HYSTERETIC_BEAM
- *USER_NONLOCAL_SEARCH

Miscellaneous Materials

Materials and Elements

■ *MAT_ADD_INELASTICITY



- Modular concept for introducing inelastic effects in standard material models
- Includes plasticity, creep and viscoelasticity models
- Not intended to replace standard material models but rather complement with missing features
- Models added on request

*MAT_ADD_DAMAGE_GISSMO

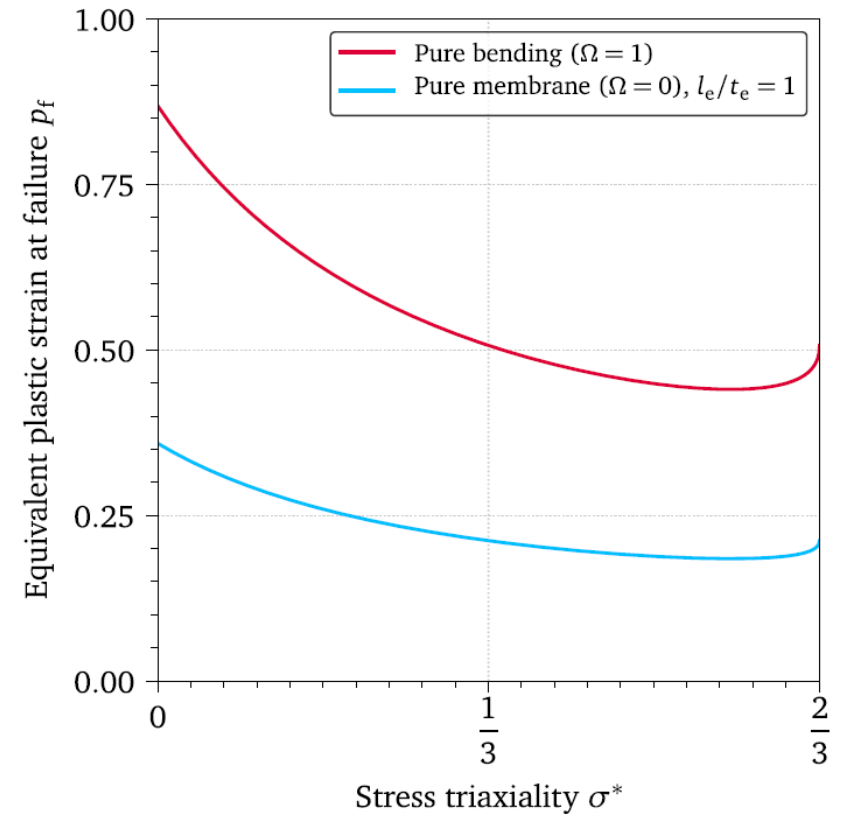
■ New option LP2BI for *MAT_ADD_DAMAGE_GISSMO

- For shell elements (with NUMFIP=1)
- Lode parameter is replaced by bending indicator:

$$\Omega = \frac{1}{2} \frac{|\varepsilon_{p,33}^T - \varepsilon_{p,33}^B|}{\max\{|\varepsilon_{p,33}^T|, |\varepsilon_{p,33}^B|\}}$$

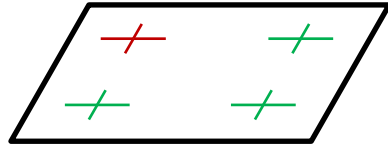
$\Omega = 0$: pure membrane
 $\Omega = 1$: pure bending

- For better failure prediction in (sharp) bending
- Adopted from *MAT_258 (Costas et al. 2018)
- Presentation at IDDRG Conference 2020
by Thornton Tomasetti, Novelis, and DYNAmore



*DEFINE_ELEMENT_EROSION_(SHELL/TSHELL)

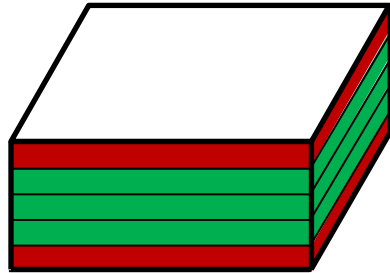
- Define a rule to delete (layered) elements based on:
 - NIFP: Number of in-plane IPs that need to fail to indicate a failed layer



—+— active IP
—+— failed IP

#of failed IPs $\geq$ NIFP: layer marked as failed

- NUMFIP: Number of layers which need to fail prior to element deletion



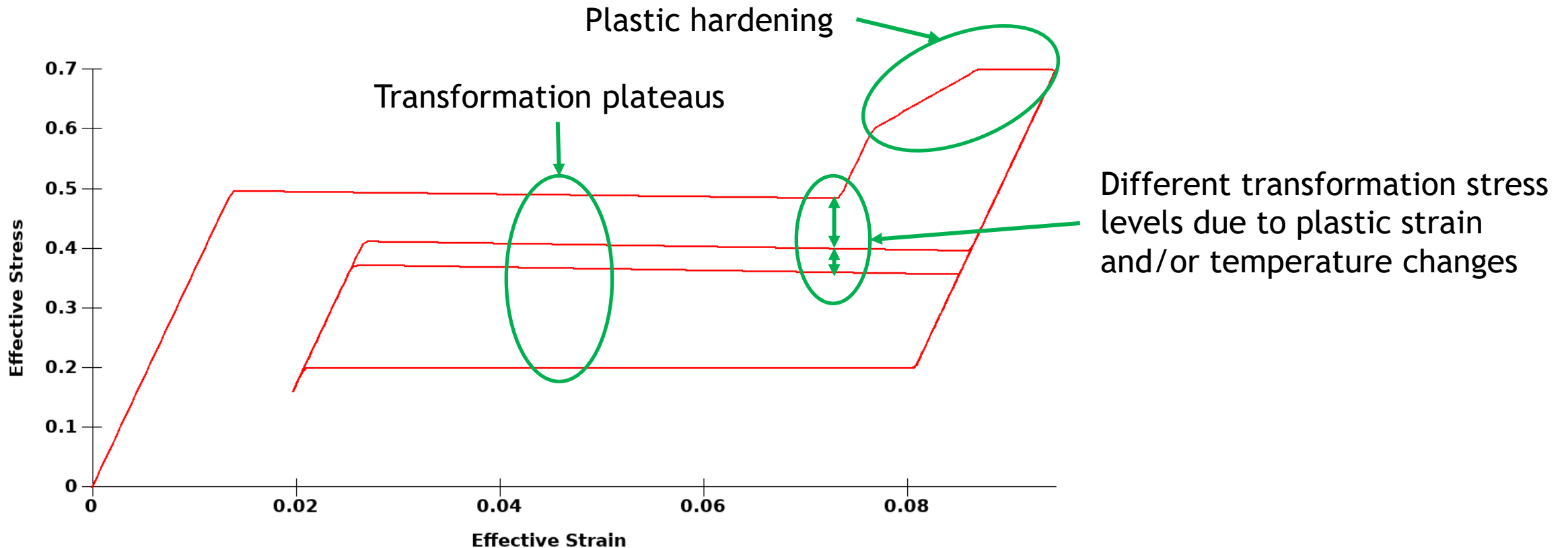
active layer
failed layer

#of failed layers $\geq$ NUMFIP: element will be deleted

- Might be useful in case of composite layered shells using different material models within the layers
- This keyword overwrites similar criteria defined within *MAT_ADD_EROSION or individual *MAT-definitions
- This keyword has to be used in conjunction with material models with failure options

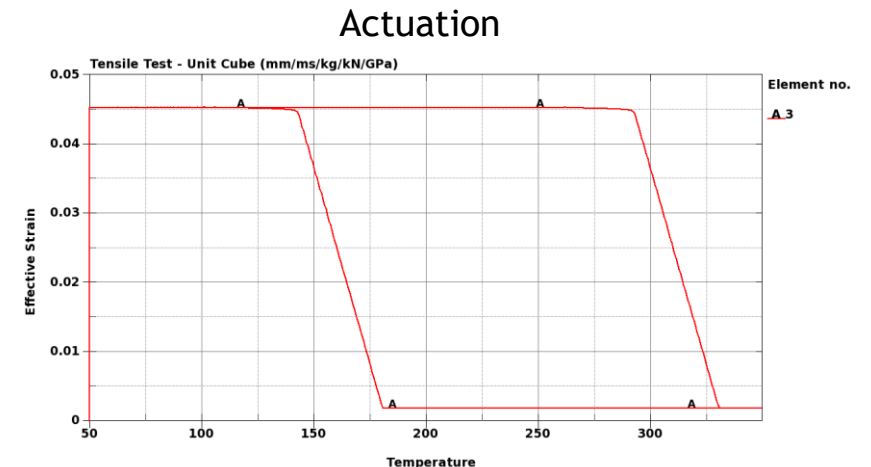
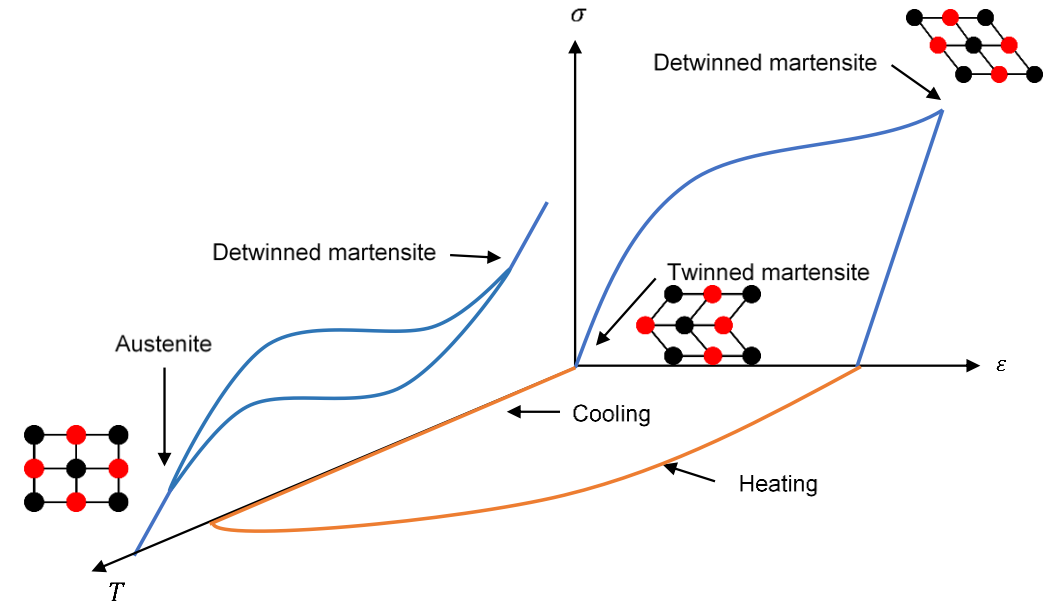
*MAT_SHAPE_MEMORY (*MAT_030)

- Support for isotropic plasticity with load curve hardening law
- Plastic strain (and/or temperature) dependent transformation plateaus



*MAT_SHAPE_MEMORY_ALLOY (*MAT_291)

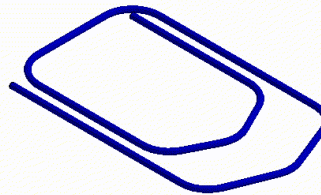
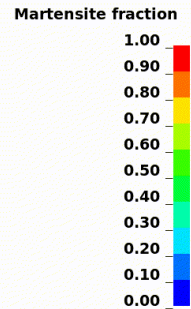
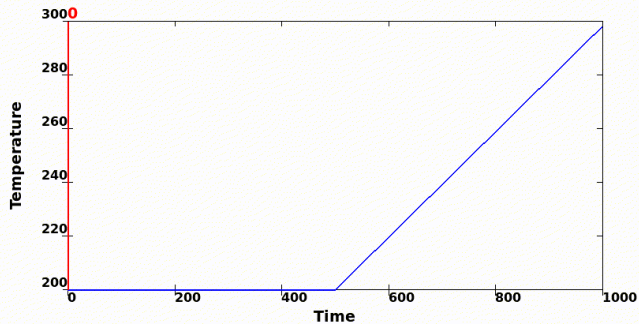
- A new micromechanics-inspired model that models full (ϵ, σ, T) -space
- Explicit/implicit, solids only
- Shape memory effect
 - Recovers original austenite configuration upon heating
- Actuation
 - Heating/cooling under applied load gives thermal hysteresis
- Optional thermal coupling with *MAT_THERMAL_ISOTROPIC_TD_LC



*MAT_SHAPE_MEMORY_ALLOY (*MAT_291), contd.

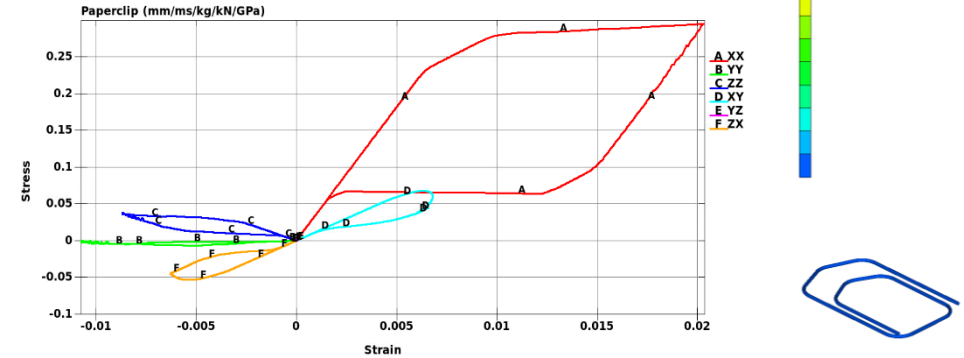
- Shape memory effect
- Permanent deformation at low temperature
- Recovers original configuration when heated

Nitinol Paperclip: tension - relaxation - temperature increase



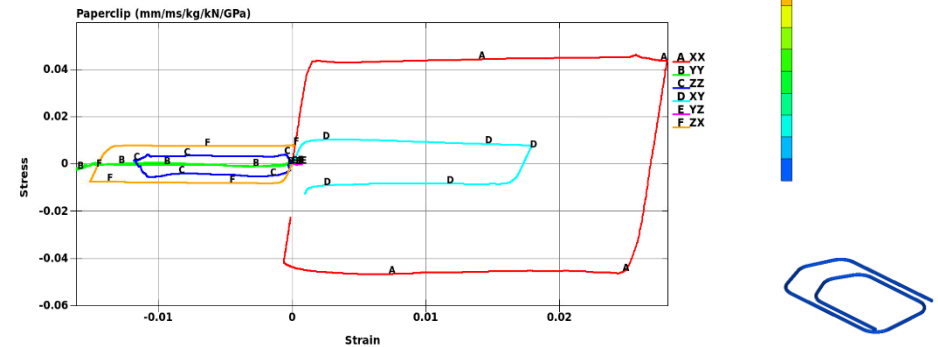
Low temperature

- Martensite phase
- Permanent deformation



High temperature

- Austenite phase
- Superelastic



*MAT_LAMINATED_COMPOSITE_FABRIC{_SOLID} (*MAT_058)

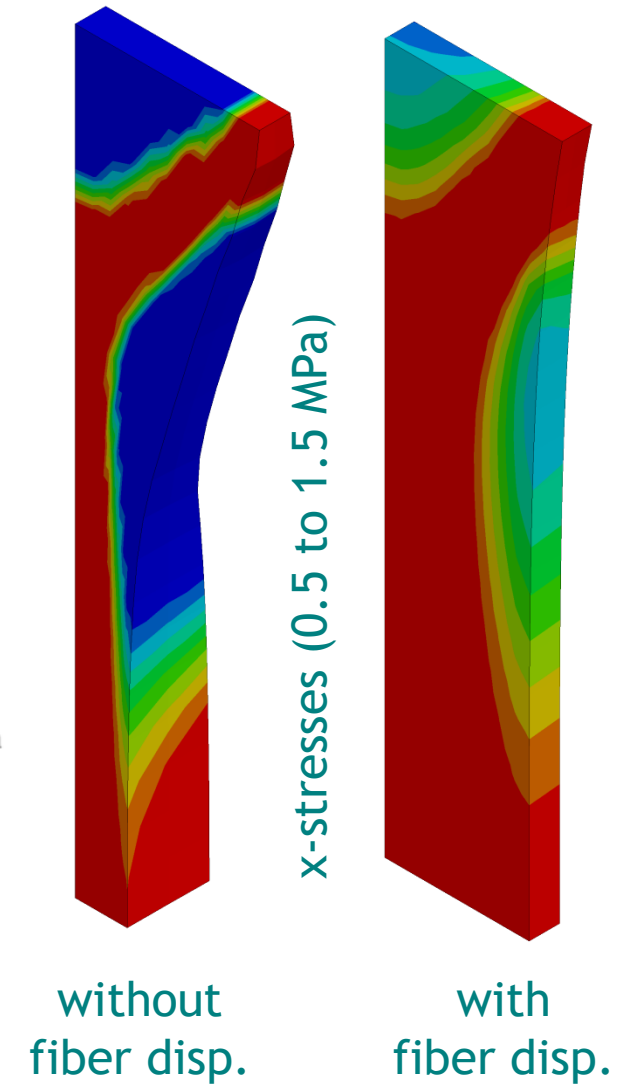
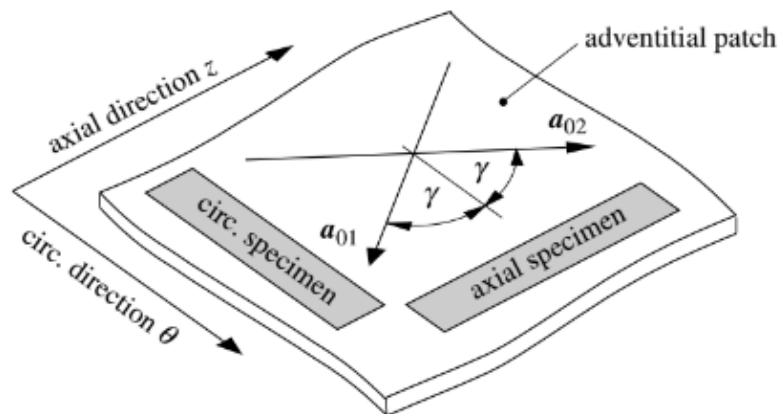
- Now available for solids
 - Requires _SOLID option
 - Three additional keyword cards for _SOLID option
- New parameter LCDFAIL (shells and solids)
 - Allows direction dependent failure strains (defined within a *DEFINE_CURVE)

```
*DEFINE_CURVE
$#      lcid      sidr      sfa      sfo
$#              a1              o1
              1              ef_11T
              2              ef_11C
              3              ef_22T
              4              ef_22C
              5              ef_12
              6              ef_33T
              7              ef_33C
              8              ef_23
              9              ef_31
```

} _SOLID option

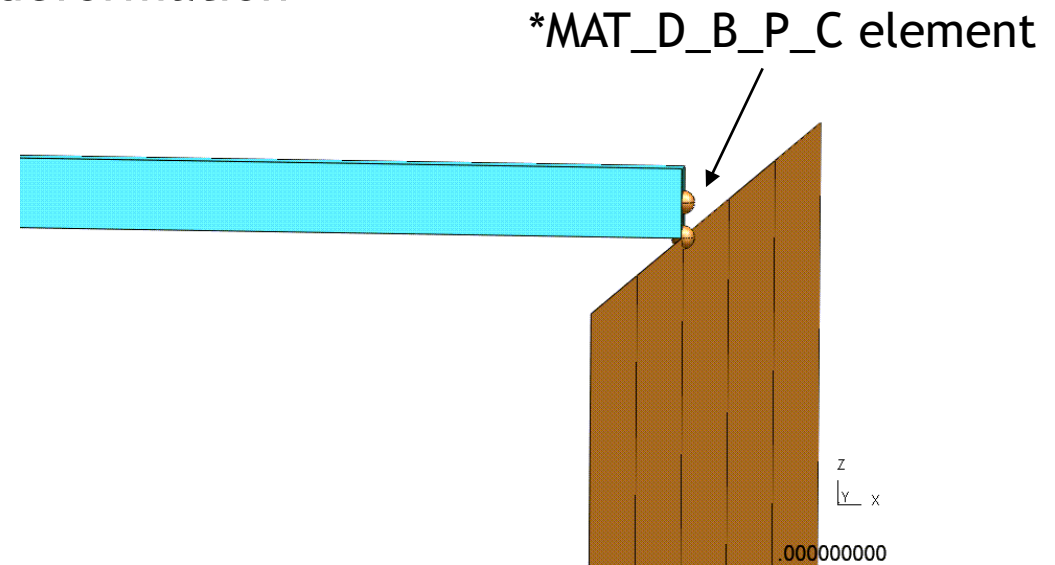
*MAT_ANISOTROPIC_HYPERELASTIC (*MAT_295)

- New modular material model for e.g. biological soft tissues or fiber-reinforced elastomers featuring:
 - Nearly-incompressible and compressible models
 - Rotationally non-symmetric fiber dispersion
 - Electro-mechanical coupling (muscle activation)
- Example problem - Gasser et al. (2006)
 - Uniaxial tension of an iliac adventitial strip (axial case)
 - Nearly-incompressible formulation
 - Two fiber families with and without fiber dispersion



*MAT_DISCRETE_BEAM_POINT_CONTACT (*MAT_205)

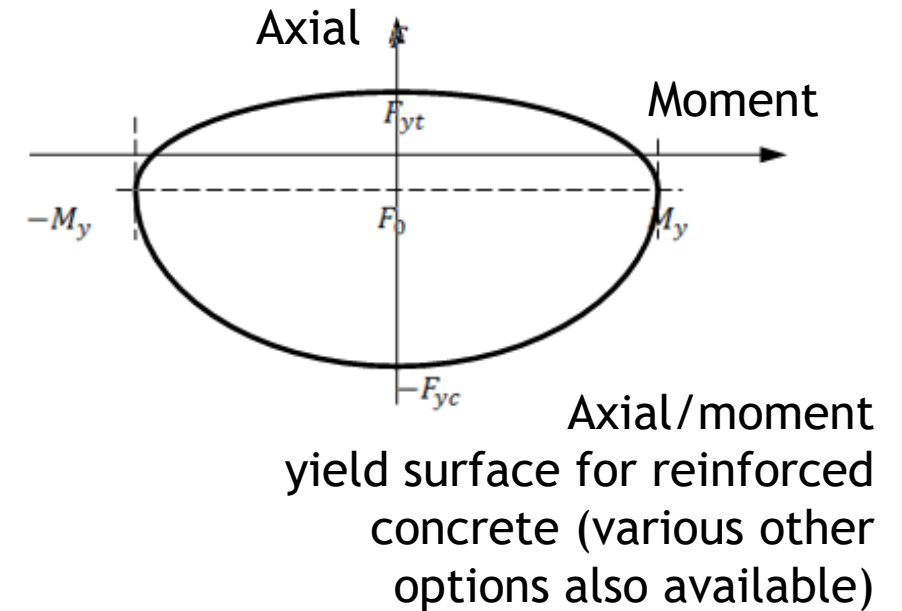
- Discrete beam element representing contact with a flat plane
 - Beam element generates the same forces as if a plane were present
 - Plane is fixed to Node N1
 - Node N2 is a point that can slide on the plane, resisted by friction; uplift is not resisted
 - Dimensions/orientation of plane are specified on *MAT/*SECTION_BEAM cards
 - Options for tiebreak, damping, nonlinear contact deformation
- Example: timber beam element resting on top of a wall made of shell elements



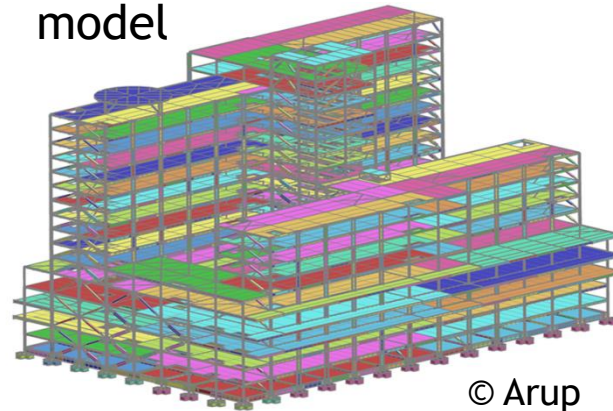
*MAT_HYSTERETIC_BEAM (*MAT_209)

■ Improved version of *MAT_SEISMIC_BEAM

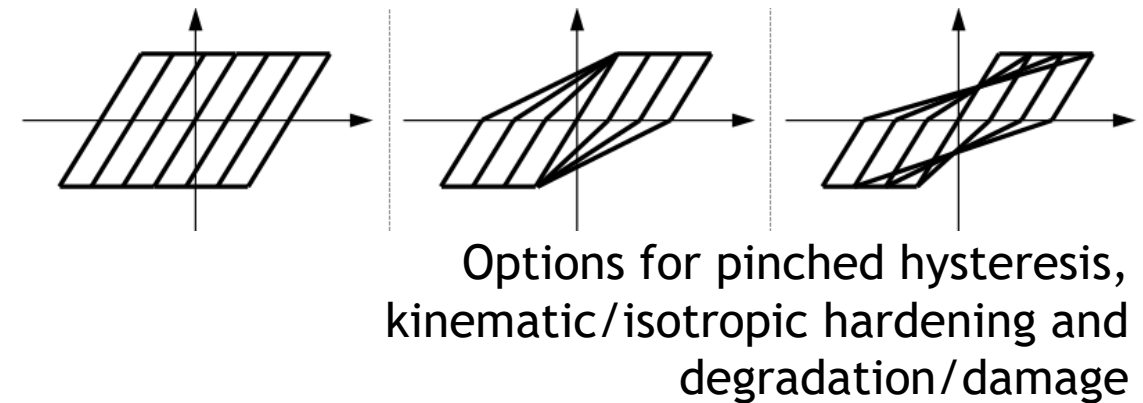
- For seismic analysis of buildings
- Suitable for steel or reinforced concrete
- Plastic hinges at both ends (can be offset from nodes)
- Nonlinear axial and shear behaviour
- Hardening, softening and damage options



LS-DYNA
model



© Arup

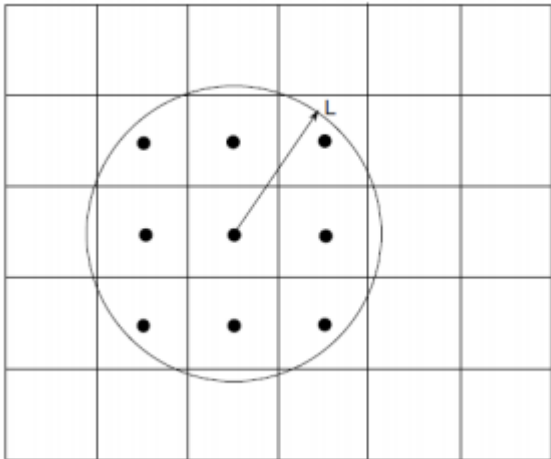


Miscellaneous material model enhancements

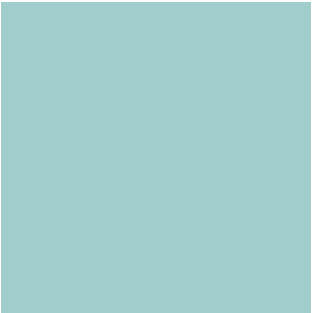
- New options for *MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM (*MAT_068)
 - Nonlinear elastic translational and rotational stiffnesses $TK\{R,S,T\}$ and $RK\{R,S,T\}$
- Make *MAT_BARLAT_YLD2000 (*MAT_133) available for solid elements
 - 3D extension of the Yld2000-2d function based on approach by Dunand et al. [2012]
 - Satisfies growing interest in accurate metal forming with solids
- Make *MAT_TAILORED_PROPERTIES (*MAT_251) available for solid elements
 - Yield stress as a function of strain, rate, and arbitrary history variables
 - For applications such as bake hardening, casting parts, etc.
- New options for *MAT_LAMINATED_FRACTURE_DAIMLER_CAMANHO (*MAT_262)
 - added transverse shear damage (similar to *MAT_054)
 - added flag (DSF) to control integration point failure based on in-plane shear

*USER_NONLOCAL_SEARCH

- Gathers data from neighboring elements
- Transfers user material HSVs and now **user element HSVs** between elements
- Activates the user subroutines `user_nunon1` and `user_nunon1_smooth`
- Enhancement made in conjunction with adaptive user element for FFI2



```
subroutine user_nunon1(nnon,ictl,rctl,auxvec,lochvh,  
  . ixh,nwcon,x,iuhhv,uehh)  
...  
c -----  
c Begin operations of neighboring data  
c Range of elements  
nstr = nrang(ii)  
nend = nrang(ii+1)-1  
do j=nstr,nend  
c Element internal sorted for this group:  
c je sorted ID for this group  
c jje internal sorted ID for this element  
  je=nlist(j)  
  if(ifail(je).eq.0) cycle  
  jje=nsrt(je)  
  do ip=1,nip  
c   Get material history values  
    do k=1,nhisv  
c     Example: sum for average  
      htmp(k,ip) = htmp(k,ip) + histv(k,ip,je)  
    enddo  
  enddo  
c Get user element history values  
  do k=1,nhisvue  
c   Example: sum for average  
    htmpue(k) = htmpue(k)+histvue(k,je)  
  enddo  
enddo  
c End operations of neighboring data  
c -----
```



SPH

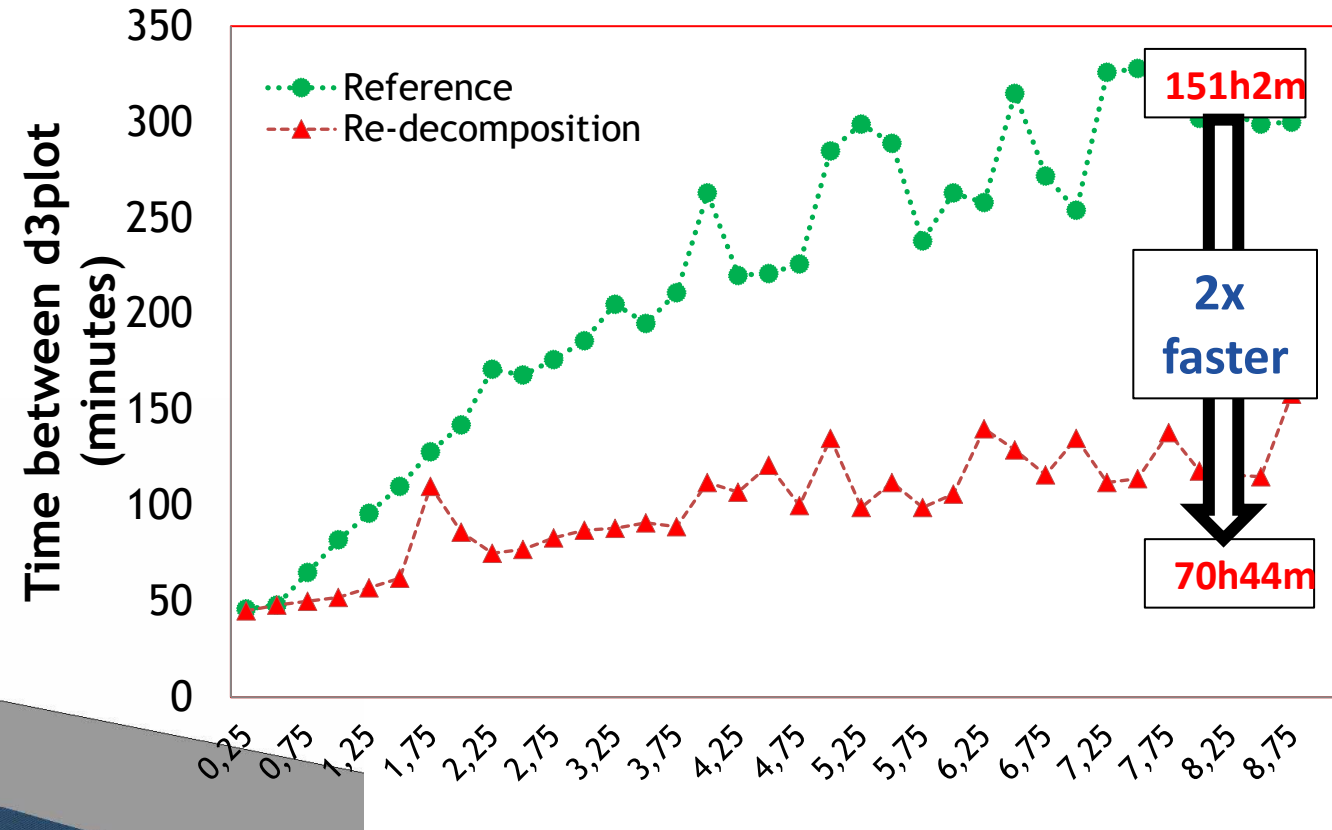
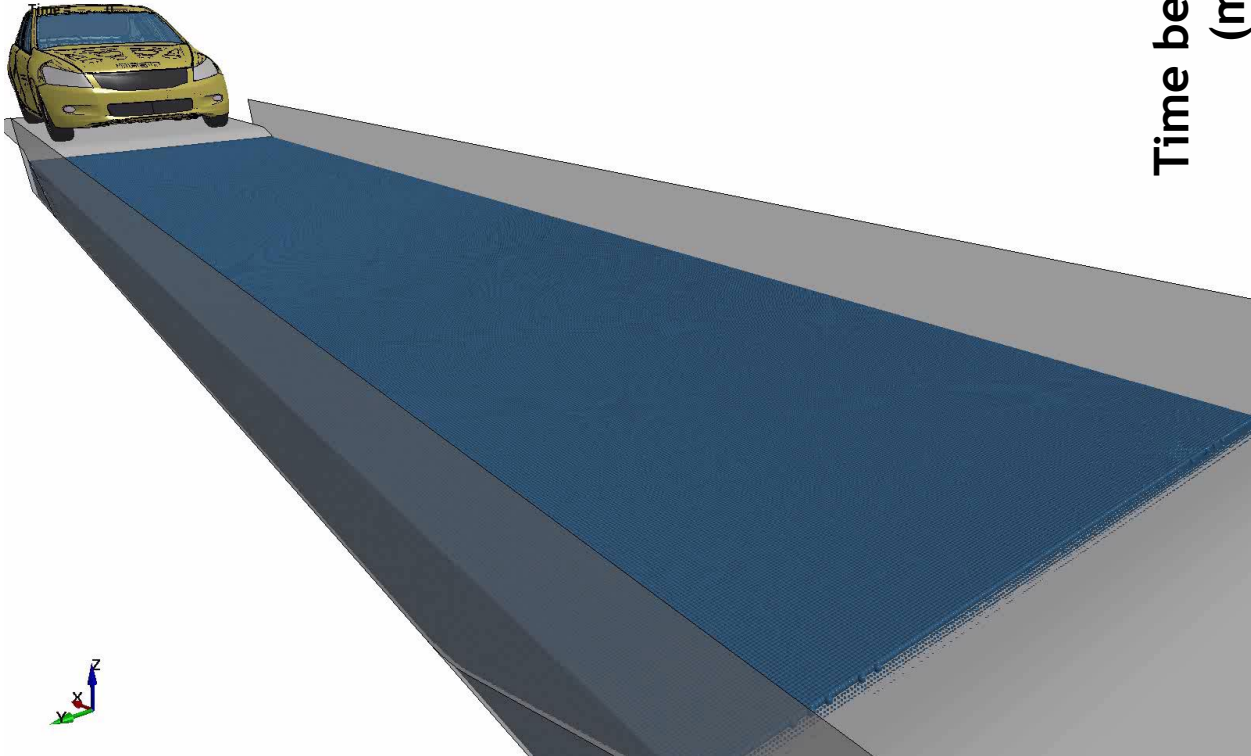
IISPH/Re-Decomposition

SPH updates general

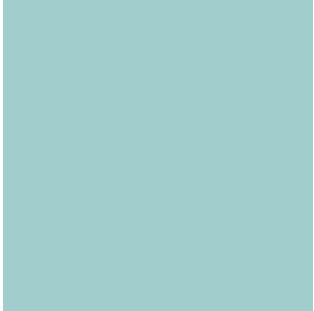
- Add *DATABASE_SPHMASSFLOW and *DEFINE_SPH_MASSFLOW_PLANE to measure SPH mass flow rate across a defined plane.
- Add option to move *DEFINE_SPH_ACTIVE_REGION around by following a moving coordinate system.
- Add option to reactivate particles and add an optional buffer zone in *DEFINE_BOX_SPH.
- *DEFINE_SPH_INJECTION:
 - Speed-up initial smoothing length calculation when *DEFINE_SPH_INJECTION is present. It used to be very slow for models containing a large number of injected particles.
 - Add option to define a variable speed of injection. If the injection speed scale factor is defined as negative, it refers to a *DEFINE_CURVE defining the velocity magnitude vs. time.
- Material models added for SPH:
 - *MAT_193 / *MAT_DRUCKER_PRAGER.
 - *MAT_COMPOSITE_DAMAGE / *MAT_022 for SPH plane strain.
 - *MAT_MODIFIED_CRUSHABLE_FOAM / *MAT_163 for SPH axisymmetric, plane strain, and 3D.
 - *MAT_ENHANCED_COMPOSITE / *MAT_054, *MAT_055 for SPH axisymmetric and 3D

Implicit Incompressible SPH (IISPH)

- Implicit, incompressible SPH formulation (FORM=13) allows larger timestep size
- Well-suited for automotive water wading, gearbox, ...



*CONTROL_MPP_DECOMPOSITION_REDECOMPOSITION
Improve MPP load balancing when a moving box defines the activation region of SPH.
Remove dead SPH particles from the model at each redecomposition step.

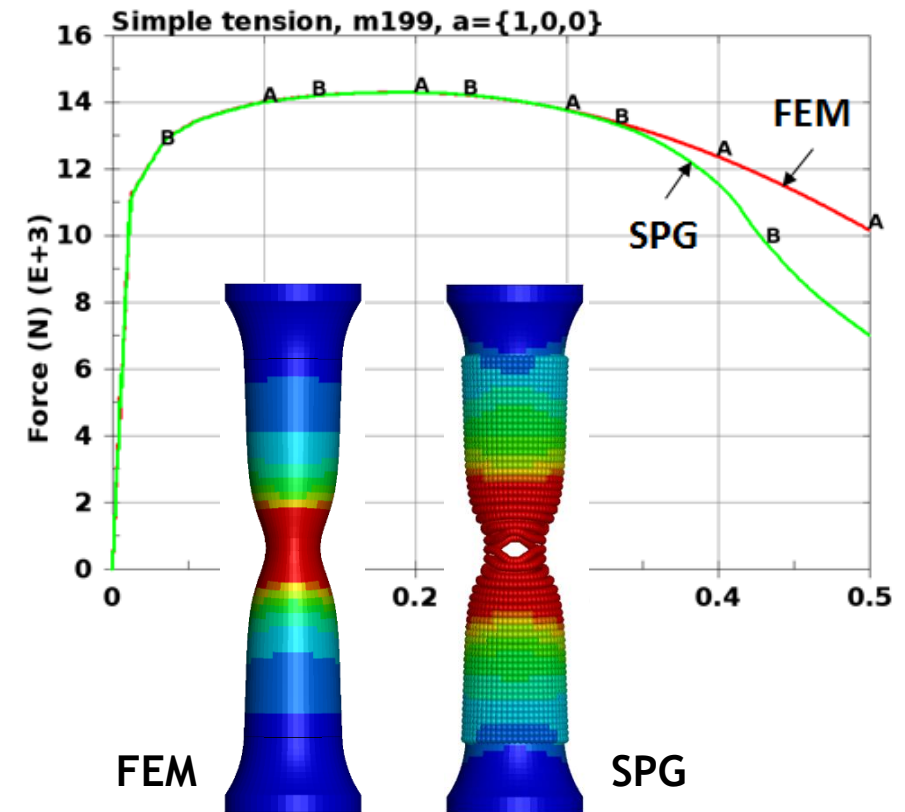


SPG

More material models supported by SPG
Enhanced SPG features

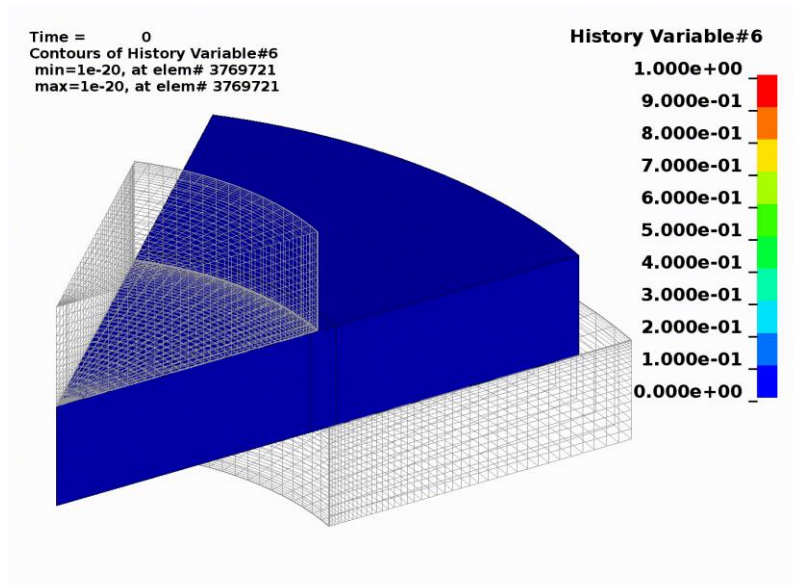
SPG enhancements

- More isotropic material laws supported
 - Mat110: for ceramics, glass and other brittle materials
 - Mat269: for rubber
- More orthotropic material laws supported
 - Mat122/123, **Mat199** and Mat260a: for metals
 - Mat126: for foam
 - Mat143: for wood
- More simplified user input
 - Hardwired ELFORM=47 so user does not need to specify it
 - "FAIL" (if >0) on mat3 and mat24 material cards is used for SPG bond failure

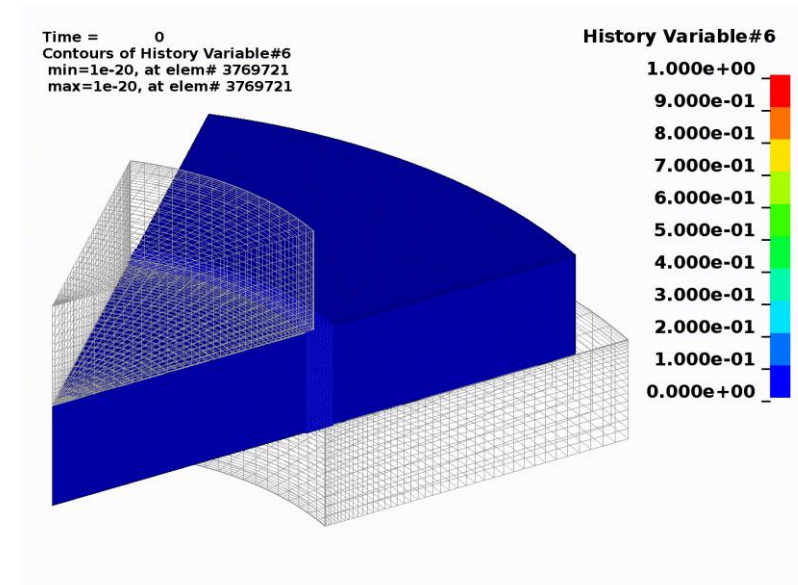


SPG enhancements

- Added SPG bond failure criteria
 - *MAT_ADD_EROSION (1st principal stress, max shear strain, 3rd principal strain)
 - *MAT_ADD_DAMAGE_GISSMO
 - Example: sheet metal punching with GISSMO damage evolution



FEM (GISSMO damage for material failure)



SPG (GISSMO damage for bond failure)

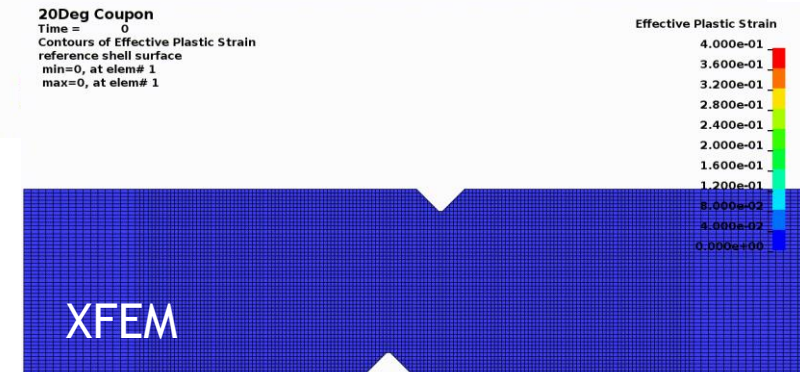
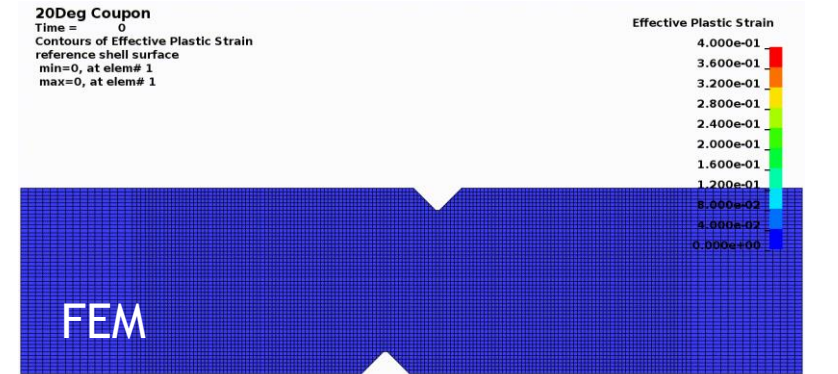
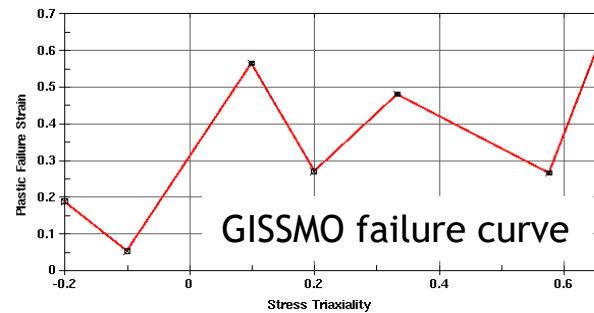
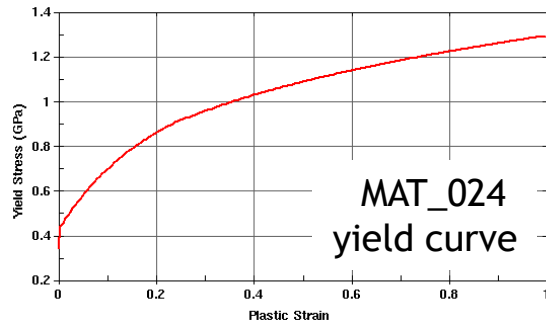


XFEM

Support of GISSMO damage model/Further enhancements

XFEM enhancements

- Added support of GISSMO damage model
 - Activated by FAILCR=0 in *SECTION_SHELL_XFEM
 - Example: Asymmetric V-notched Coupon under Tension



- *BOUNDARY_PRECRACK
 - Adjusted the location of pre-crack to avoid passing through nodal points
- Recoded neighbors list to handle triangular mesh better

ALE and S-ALE

INITIAL_LAG_MAPPING

ALE_MESH_INTERFACE

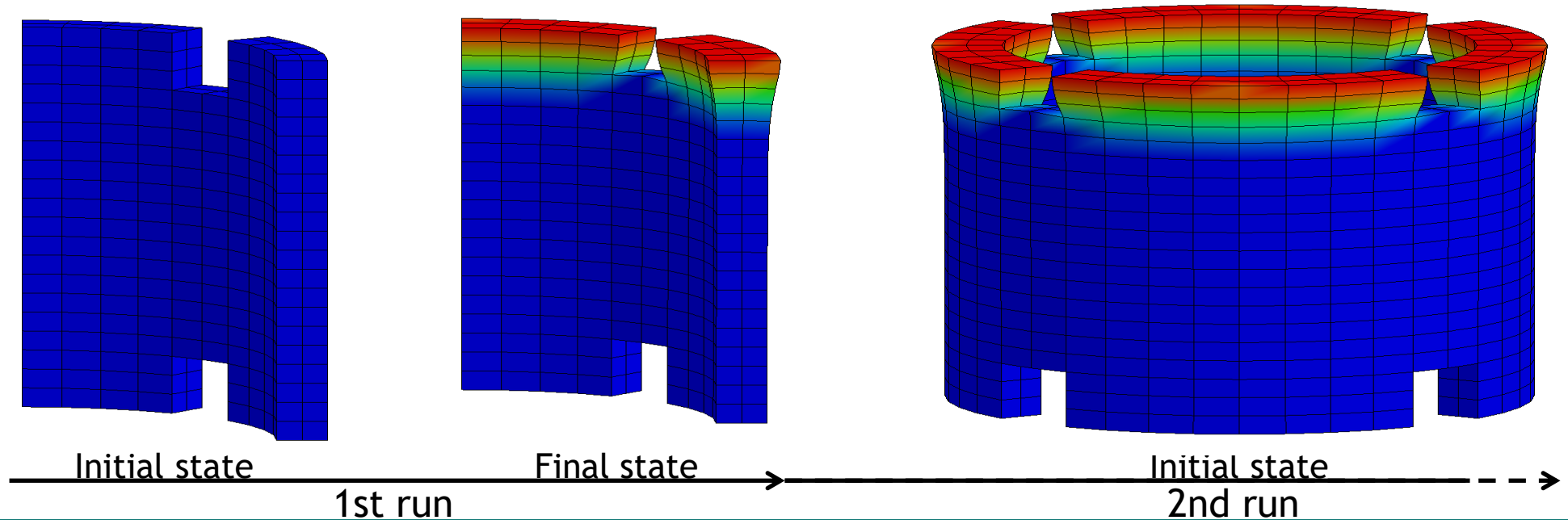
CONTROL_REFINE_(ALE/ALE2D/SOLID)

ALE_STRUCTURED_xxx

ALE_STRUCTURED_xxx/EOS_MURNAGHAN

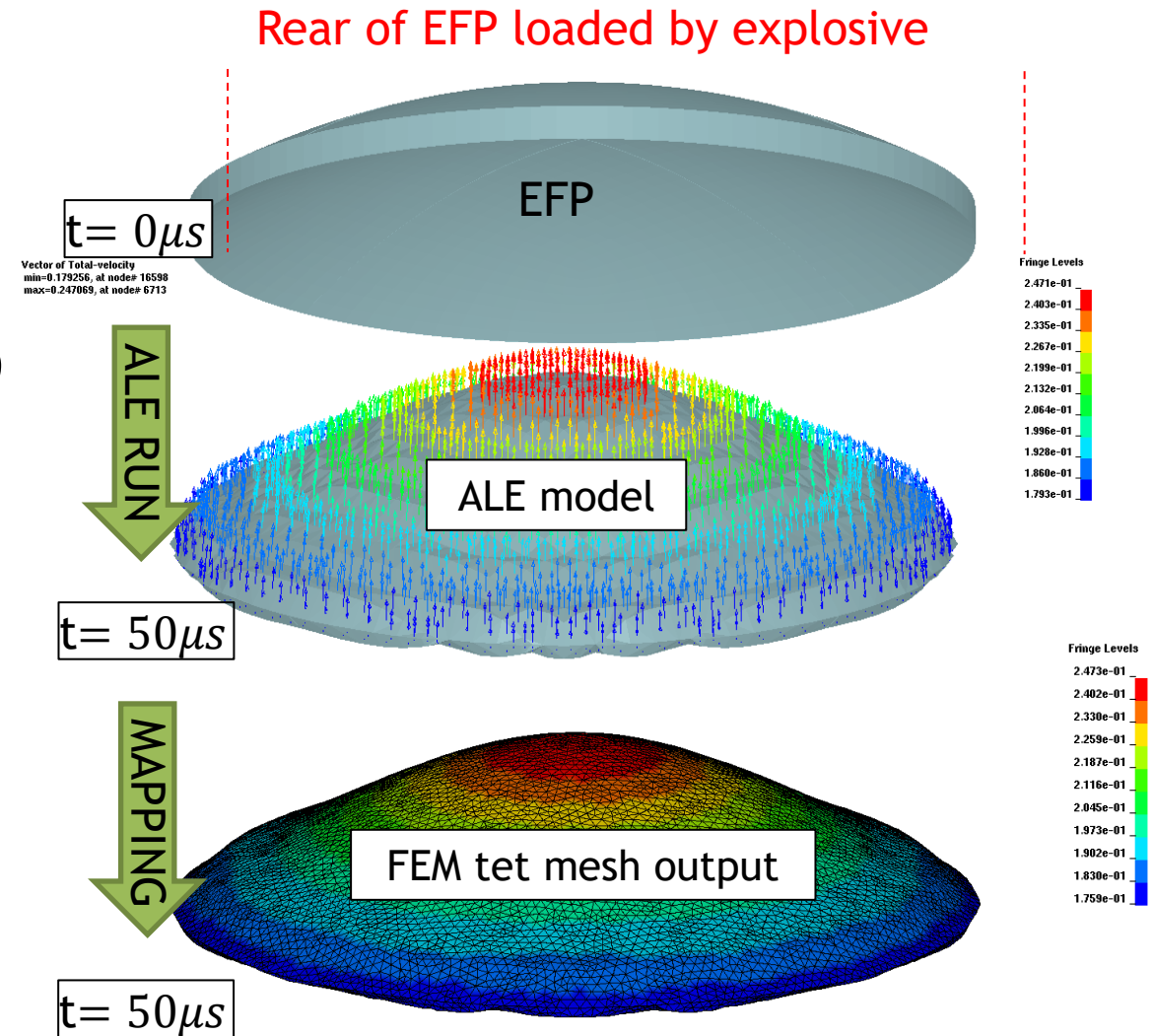
Powerful new mapping function for solid element models

- Keyword: `*INITIAL_LAG_MAPPING`
 - Parallel to the ALE to ALE mapping command `*INITIAL_ALE_MAPPING`
- Powerful mapping of results from one solid model simulation to another: 2D to 2D, 2D to 3D, 3D to 3D, 3D to 2D
 - Optional automatic mesh generation



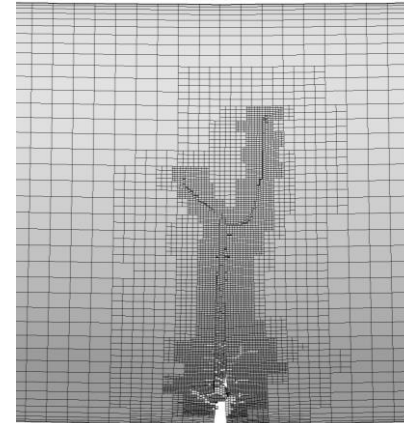
ALE: Create finite element models from simulation results

- Keyword: *ALE_MESH_INTERFACE
- Outputs ALE simulation results as FEM tet-meshed bodies
 - Future development: Also map the results (plastic strain, stress, et c)
- Application: Simulation of EFP - Explosively Formed Penetrator

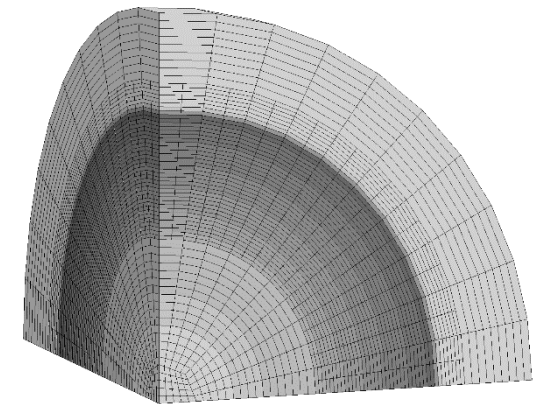
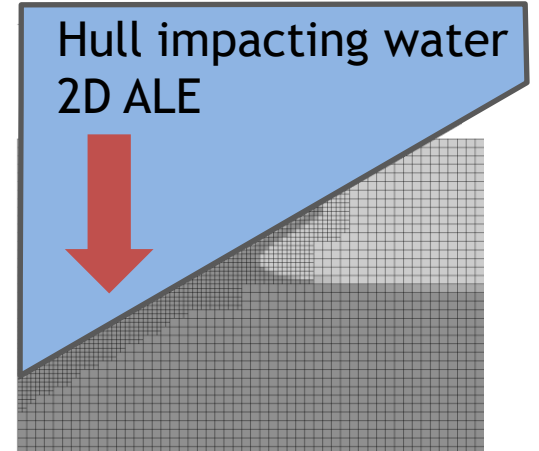


ALE & FEM: Adaptive Mesh Refinement

- Keyword: *CONTROL_REFINE_ALE,
*CONTROL_REFINE_ALE2D,
*CONTROL_REFINE_SOLID
- Automatic mesh refinement and coarsening in areas with large gradients in the solution.
- Applications:
 - ALE Fluid: Boat hull slamming in water
 - ALE Defense: Blast simulation
 - FEM: Crack propagation
- Advantage: Faster simulation run times for a given required accuracy.
- Note: set NTOTREF=-1 to take advantage of LS-DYNAs new dynamic memory management



FEM crack
propagation

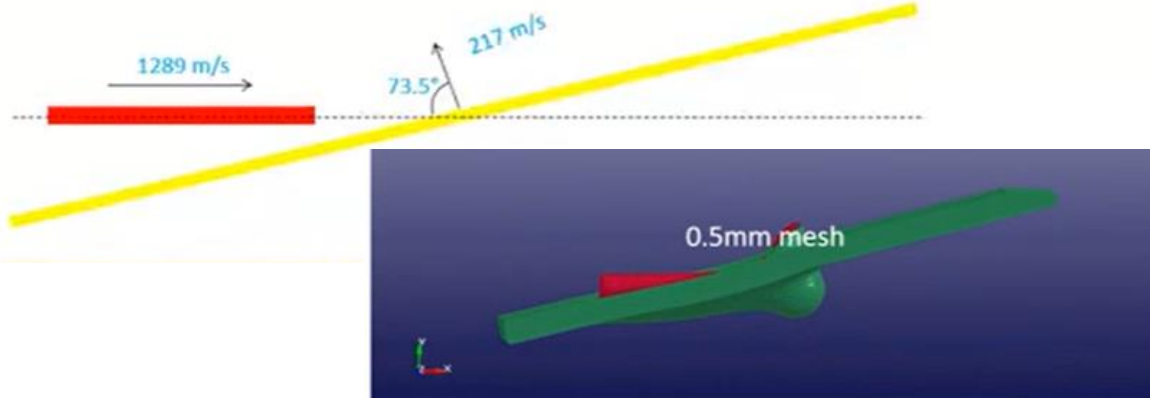


3D ALE blast
simulation

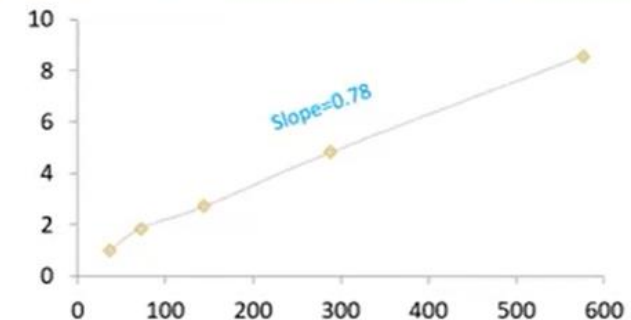
S-ALE - Structured ALE solver

Note! Due a bug in S-ALE in R12.0.0 please wait for the next release our contact DYNAmore for an updated version.

- Keyword family: *ALE_STRUCTURED
- 2015 LS-DYNA introduced the Structured ALE solver based on automatically generated block structured meshes.
- Advantage: faster & ease of use especially for large models (>50M elements), radically smaller input & output files, faster pre & postprocessing
- Example: 200M element long rod penetration simulation, scaling to 576 cores



# of cores	36	72	144	288	576
Total time(s)	84473	45711	32113	17738	9738
T per cycle (s)	10.056	5.4405	3.7014	2.0785	1.1761
Speedup	1.0	1.8484	2.7168	4.8381	8.5503

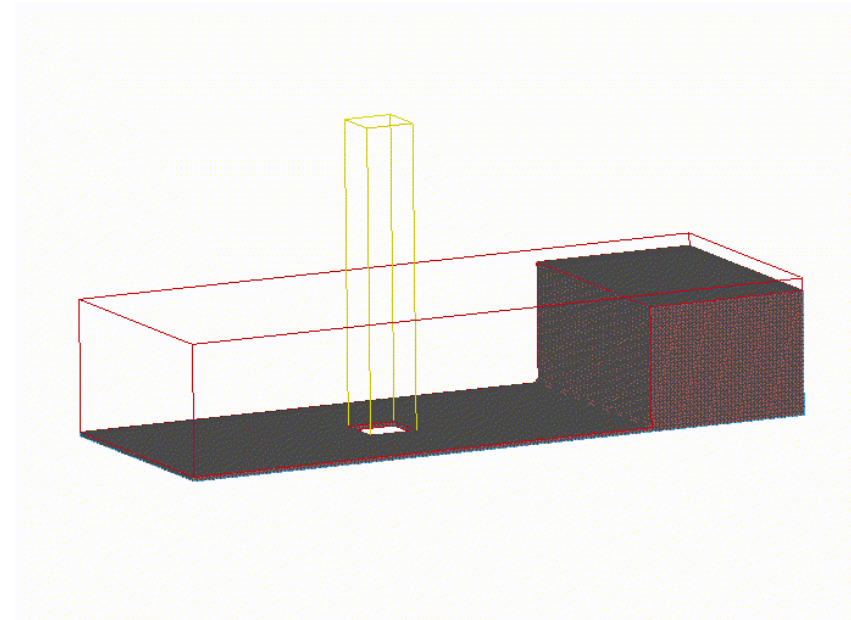


S-ALE - Structured ALE solver - Selected news

- Faster and easier mesh initialization: *ALE_STRUCTURED_MESH_VOLUME_FILLING
 - Replaces *INITIAL_VOLUME_FRACTION_GEOMETRY
- Automatic trimming and follow the motion, e.g. for bird impact simulation
 - *ALE_STRUCTURED_MESH_TRIM
 - *ALE_STRUCTURED_MESH_MOTION



- *EOS_MURNAGHAN - fast simulation of water/fluids
 - Allows use of a larger time step in low-pressure situations, e.g. sloshing, general flow
 - About the EOS: E. Yreux, "Fluid flow modelling with SPH in LS-DYNA", 15th int. LS-DYNA conf., 2018



S-ALE - Structured ALE solver - Selected news

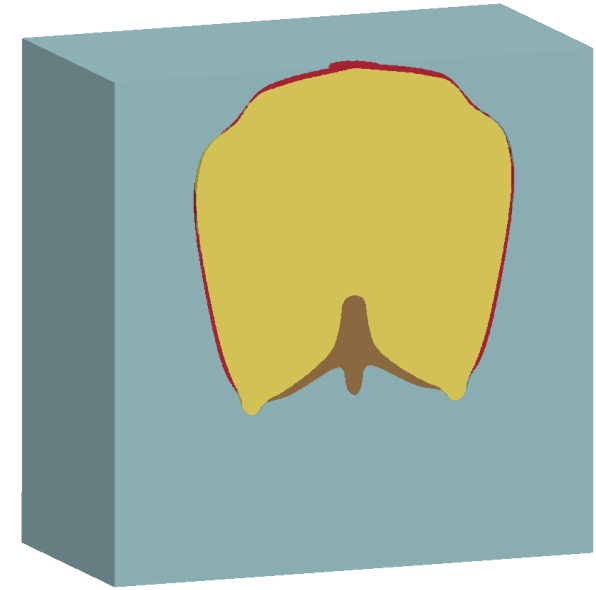
■ Keyword: *DATABASE_SALE (df3sale)

- New output format to speed up the postprocessing of large S-ALE models.
- Note: requires a recent beta version of LS-PrePost 4.9.
- Decreases storage space requirements with same or better accuracy
- Example: 30M element shape charge simulation
 - Old d3plot format: 77 GB output
 - New d3sale format: 13 GB output

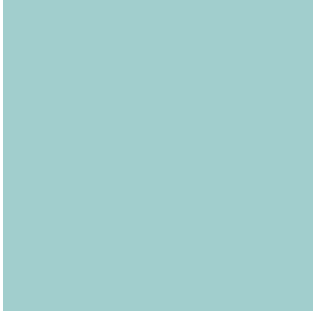
■ Keyword: *ALE_STRUCTURED_FSI

- Faster and easier to set up fluid structure interaction with automatic leakage control
- Replaces *CONSTRAINED_LAGRANGE_IN_SOLID for S-ALE

- For more information see H. Chen, Recent Developments in LS-DYNA S-ALE, 16th International LS-DYNA Users Conference, 2020



30M element shape charge simulation

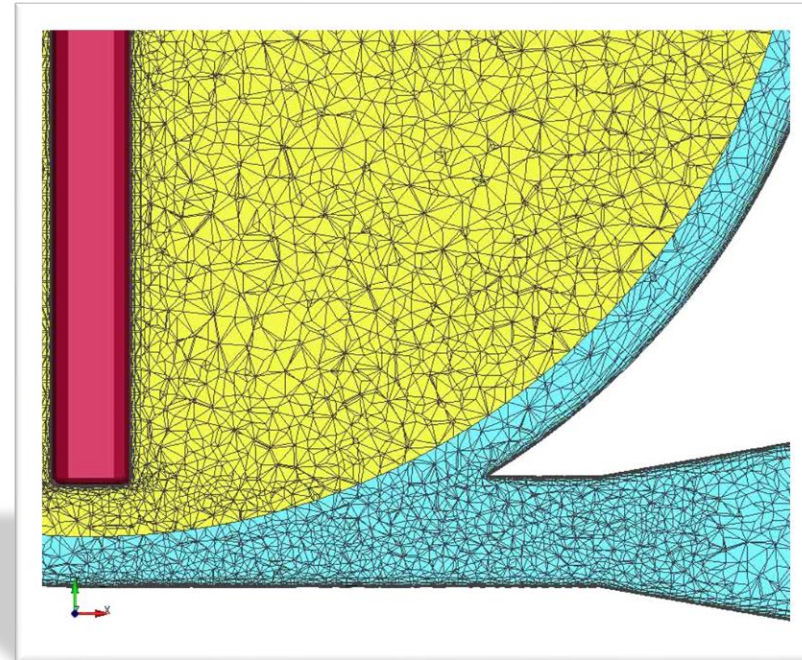
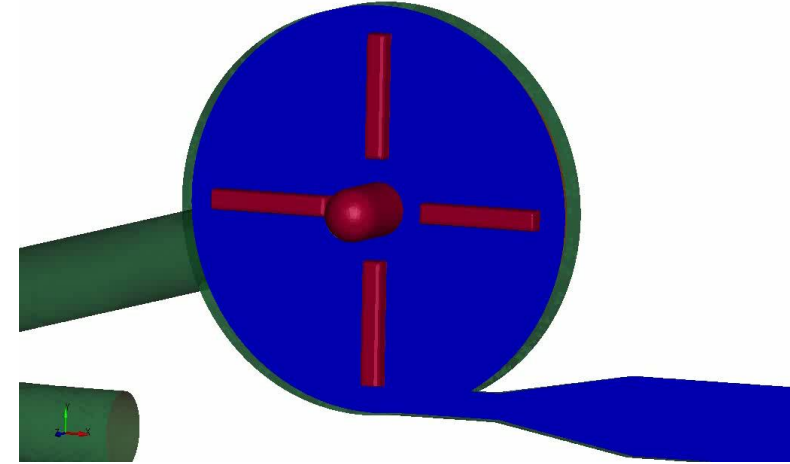
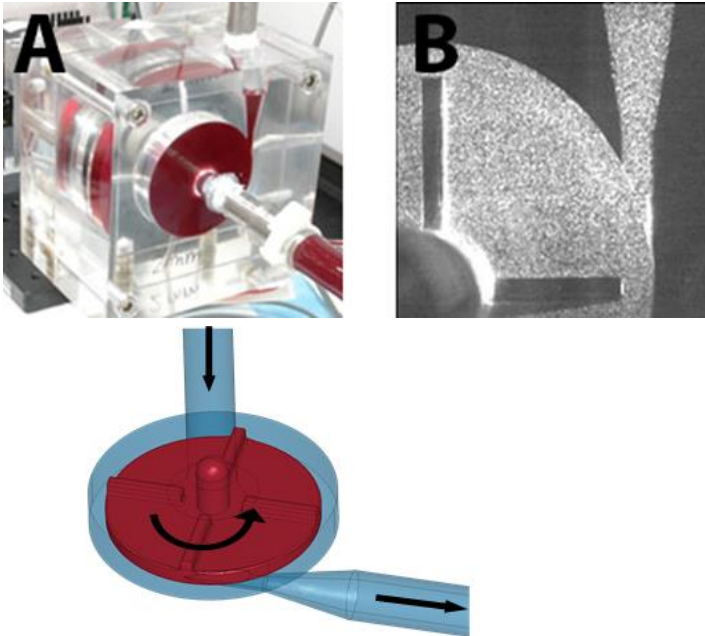


ICFD

- *ICFD_BOUNDARY_PERIODIC - sliding
- *ICFD_CONTROL_PERIODIC - periodic/Wave generation
- *ICFD Immersed
- *ICFD_xxx cards

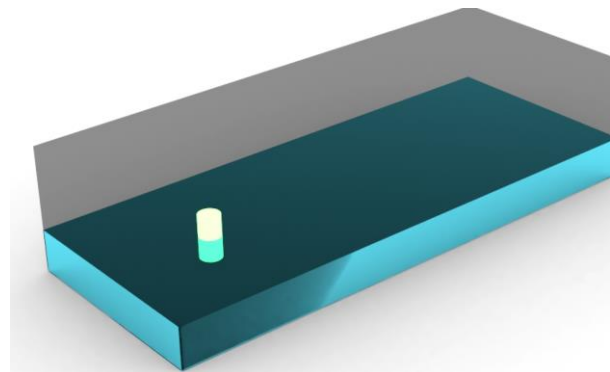
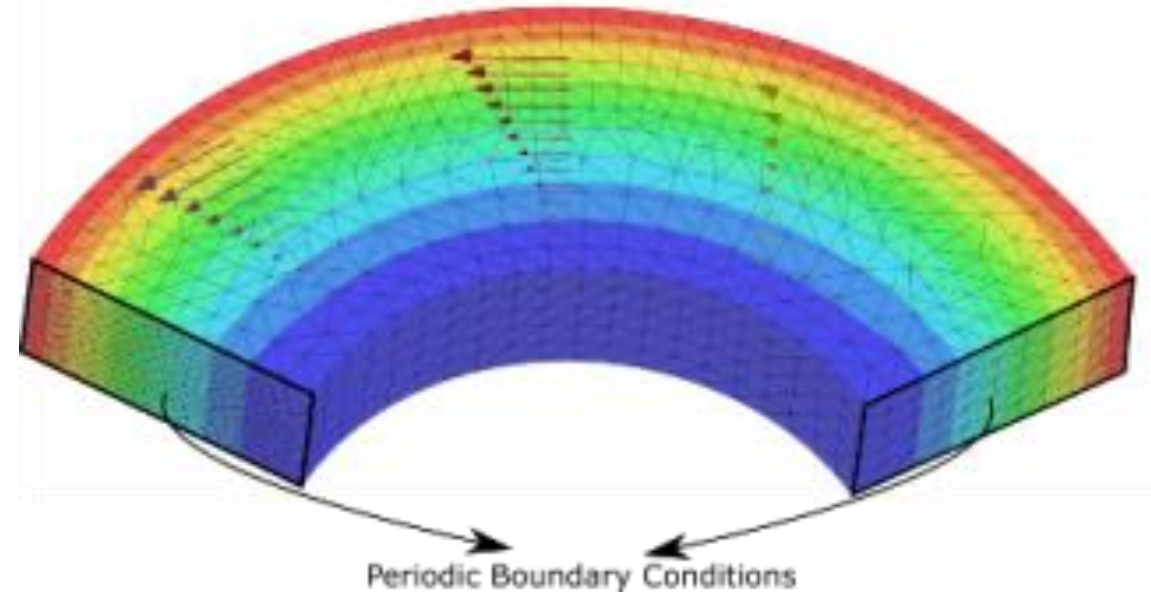
ICFD - Sliding mesh

- *ICFD_BOUNDARY_PERIODIC - sliding
- Avoid mesh distortions when studying rotating machinery. Flow properties for transient or steady state.
- Blood pump example:

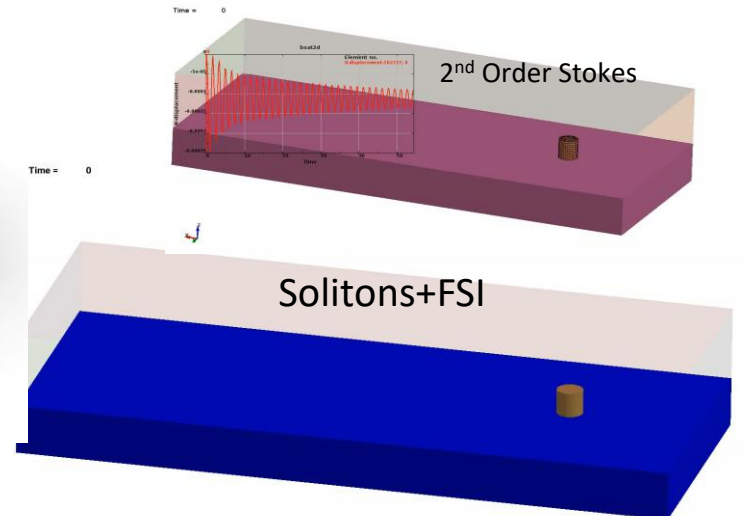


ICFD - periodic boundary conditions

- *ICFD_BOUNDARY_PERIODIC
 - periodic
 - A demo example is available on www.dynaexamples.com/icfd/beta_examples/couetteflow
- New wave generation options: (*ICFD_BOUNDARY_FSWAVE)
 - Fifth order Stokes wave
 - Solitary wave
 - Irregular wave



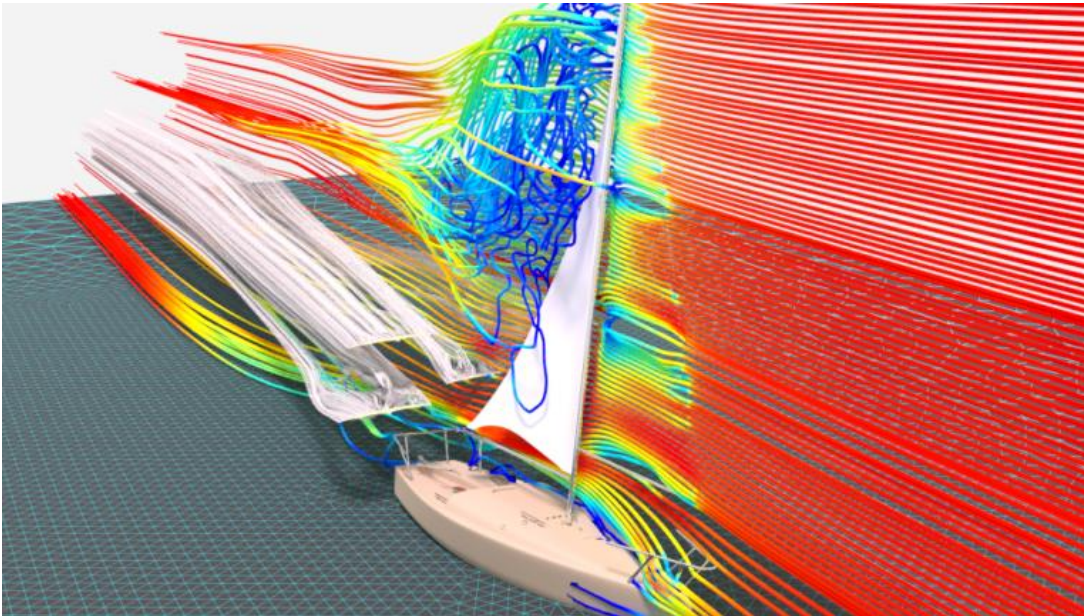
Irregular Ocean waves
+high end rendering



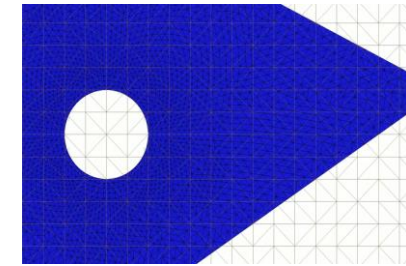
Immersed interface and overset mesh techniques

- Immersed interface
- Overset mesh
- Initial and alpha versions available for SMP

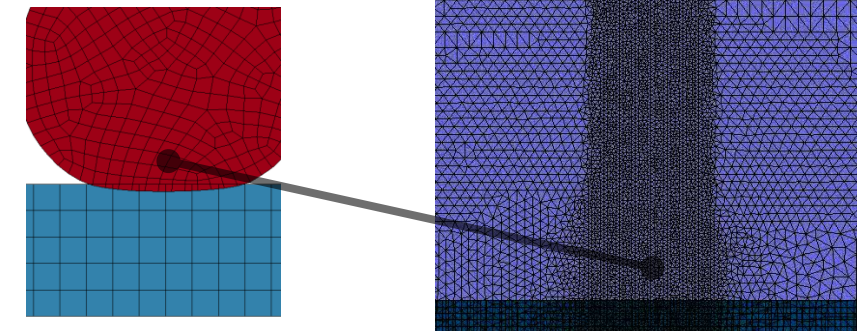
Immersed interface



Overset meshes



Immersed interface



ICFD new functionality

- *ICFD_CONTROL_CONJ
 - Parameter TSF: Thermal Speedup Factor
- *ICFD_BOUNDARY_CONVECTION_TEMP
 - Impose a heat transfer coefficient on the boundary
- *ICFD_CONTROL_DEM_COUPLING
 - Force dependent on velocity
 - Drag force calculation option
- *ICFD_CONTROL_TURBULENCE
 - TWLAW: New option for thermal law of the wall
 - SUBMOD=2: Added LES Dynamic turbulence model originally proposed by DK Lilly (1991) with localization on coefficient CS by Piomelli and Liu (1995).
- *ICFD_DEFINE_TURBSOURCE
 - New keyword which allows addition of turbulent source terms for RANS models
 - LCIDK and LCIDEP: Load curves for source terms of turbulent kinetic energy and dissipation
- *ICFD_DEFINE_SOURCE
 - New keyword which allows addition of a volumetric external force

EM - Electromagnetics and Electrochemistry (Batteries)

Updates for EM solver

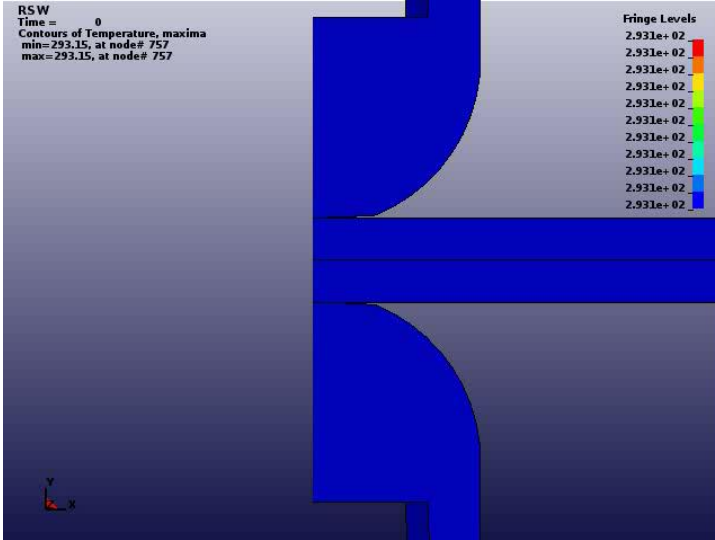
Electrochemistry-thermo-mechanical coupling

Battery module

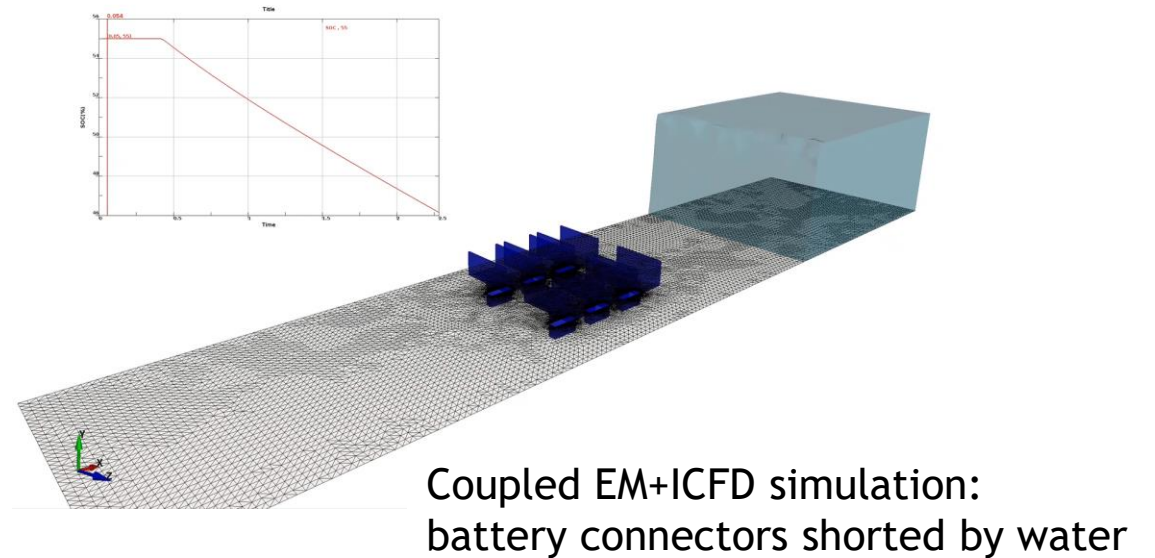
*MAT_ADD_SOC_EXPANSION

Updates for EM solver

- Added EM Mortar types to improve accuracy in RSW and other applications
- Support of eroding conductors
- Added coupling with the ICFD solver
 - ICFD meshes can be defined as conductors. This can be useful for applications such as electrostatics or the study of external shorts caused by water ingress.

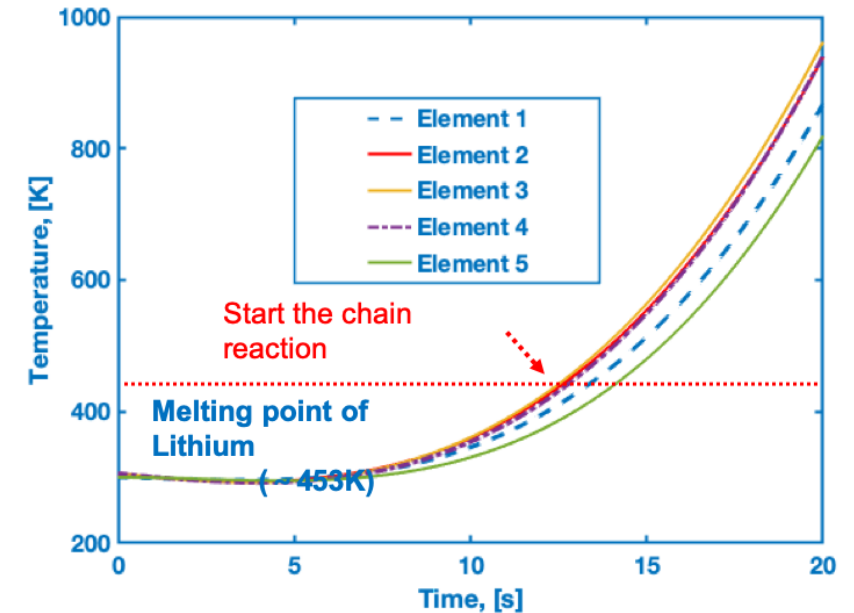
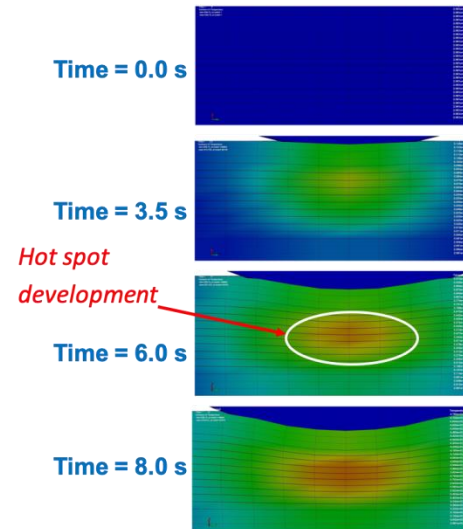
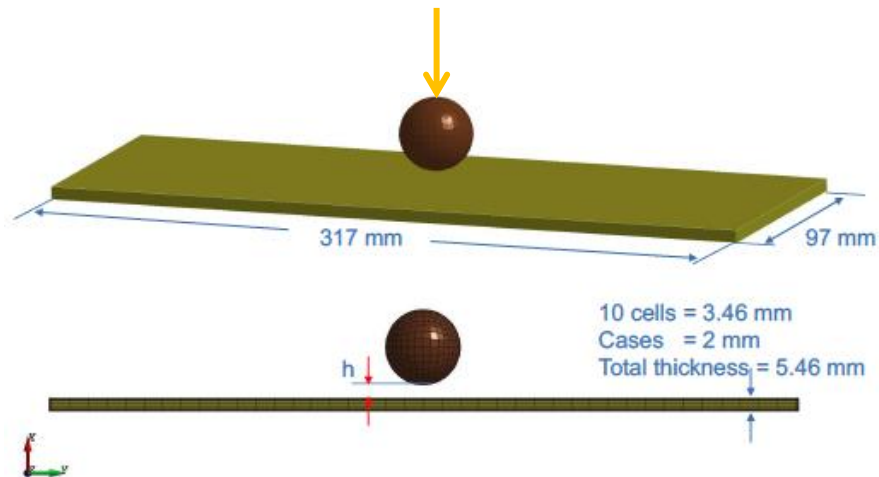


Resistive
Spotwelding
simulation



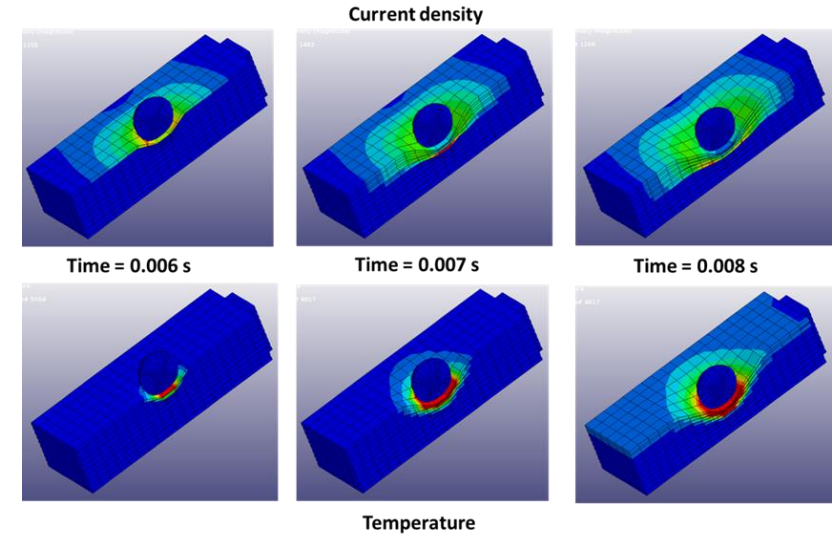
Electrochemistry-thermo-mechanical coupling

- New thermal and mechanical coupling with electrochemical LIB model
 - Models for Lithium Ion batteries (LIB): single and dual insertion
 - Simulation of thermal runaway problems induced by deformation from external forces
- Example: ball impact on 10 LIB cells stack
 - Deformations -> Thermal heating -> thermal runaway



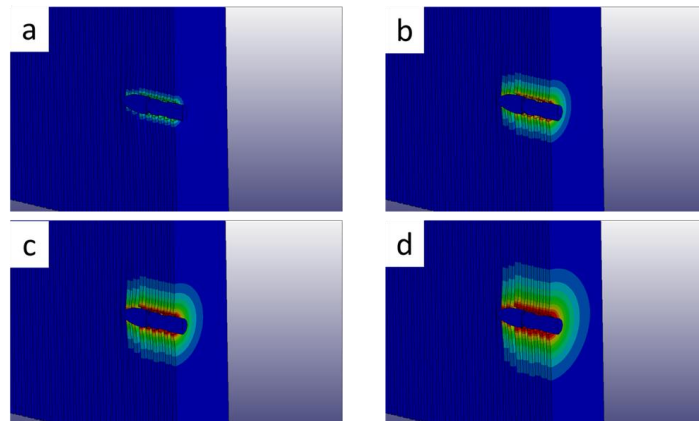
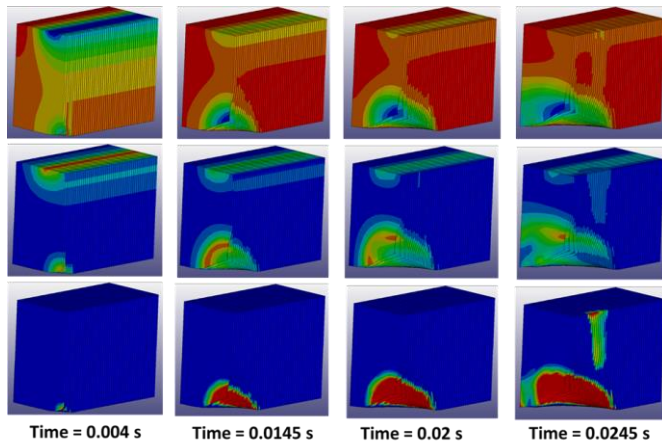
Battery module

- 4 models depending on scale/detail
 - Solid elements: internal/external shorts, cell
 - Composite Tshells: internal/external shorts, cell/module
 - Macro model: internal/external shorts, pack/battery
 - Meshless model: external shorts, module/pack/battery

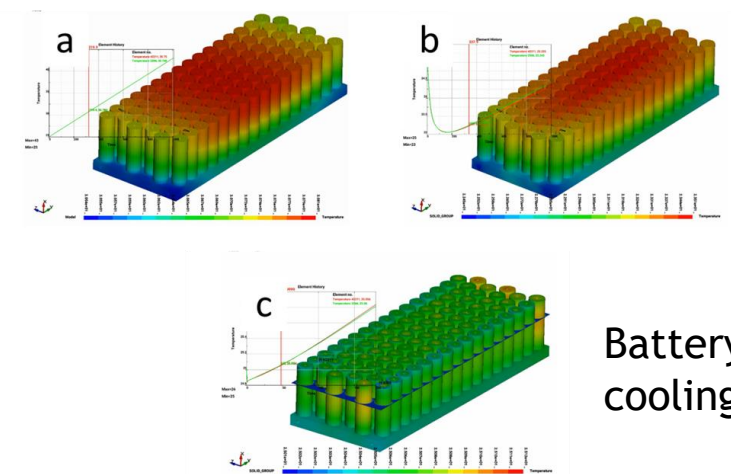


Module crush

Battery crush



Nail penetration



Battery cooling

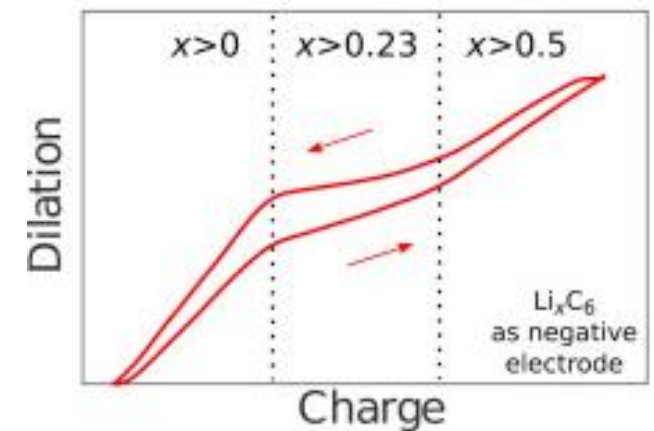
*MAT_SOC_EXPANSION

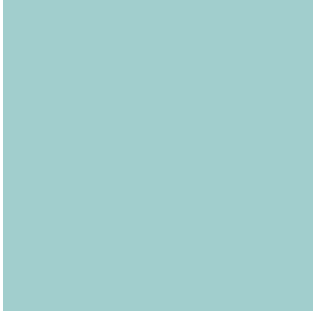
■ New keyword for state of charge expansion

- For EM-mechanical coupled simulations
- State of charge (SOC) as driving quantity for material expansion
- Works as thermal expansion, with SOC replacing temperature
- First implemented for common solid elements and hypoelastic materials

- Application: cell and electrode swelling in Li-ion batteries

“Understanding the dilation and dilation relaxation behavior of graphite-based lithium-ion cells”
by Bauer et al. (2016)





IGA - Isogeometric Analysis

Mechanical coupling of trimmed patches*3
Various other enhancements

Isogeometric Analysis (*ELEMENT_SHELL_NURBS_PATCH_TRIMMED)

■ Mechanical coupling of trimmed patches

- continuity at interface (strong form):

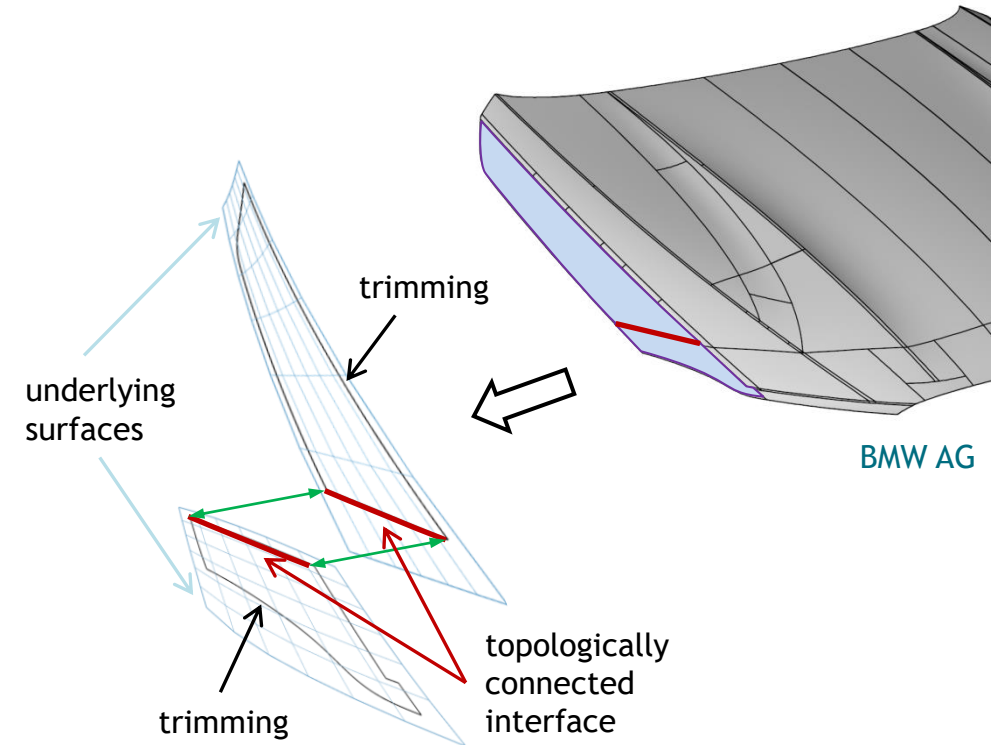
$$\mathbf{u}_1 = \mathbf{u}_2|_{\Gamma} ; \quad \boldsymbol{\theta}_1 = \boldsymbol{\theta}_2|_{\Gamma}$$

- penalty weak form (translations and rotations):

$$\alpha^{disp} \int_{\Gamma} (\mathbf{u}_1 - \mathbf{u}_2) \cdot (\delta \mathbf{u}_1 - \delta \mathbf{u}_2) d\Gamma = 0$$

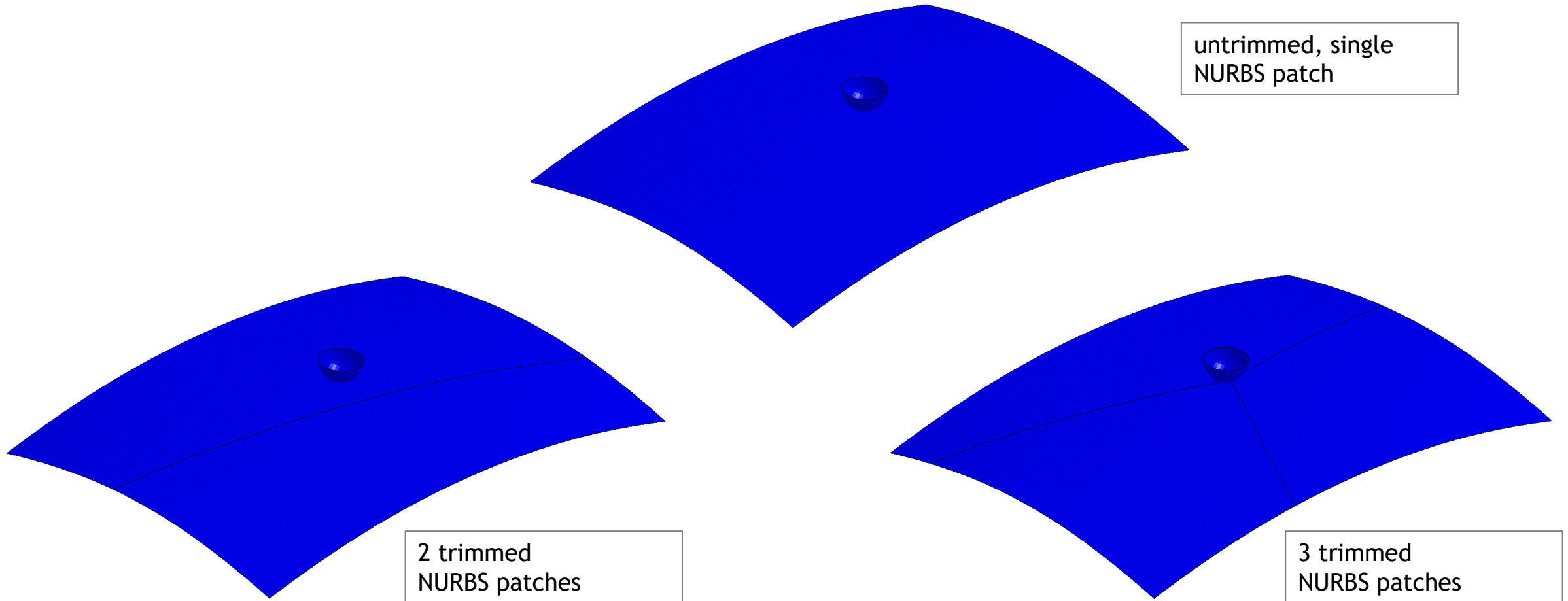
$$\alpha^{rot} \int_{\Gamma} (\boldsymbol{\theta}_1 - \boldsymbol{\theta}_2) \cdot (\delta \boldsymbol{\theta}_1 - \delta \boldsymbol{\theta}_2) d\Gamma = 0$$

- various improvements and bug fixes
- add support for thin (rotation free) shell formulations



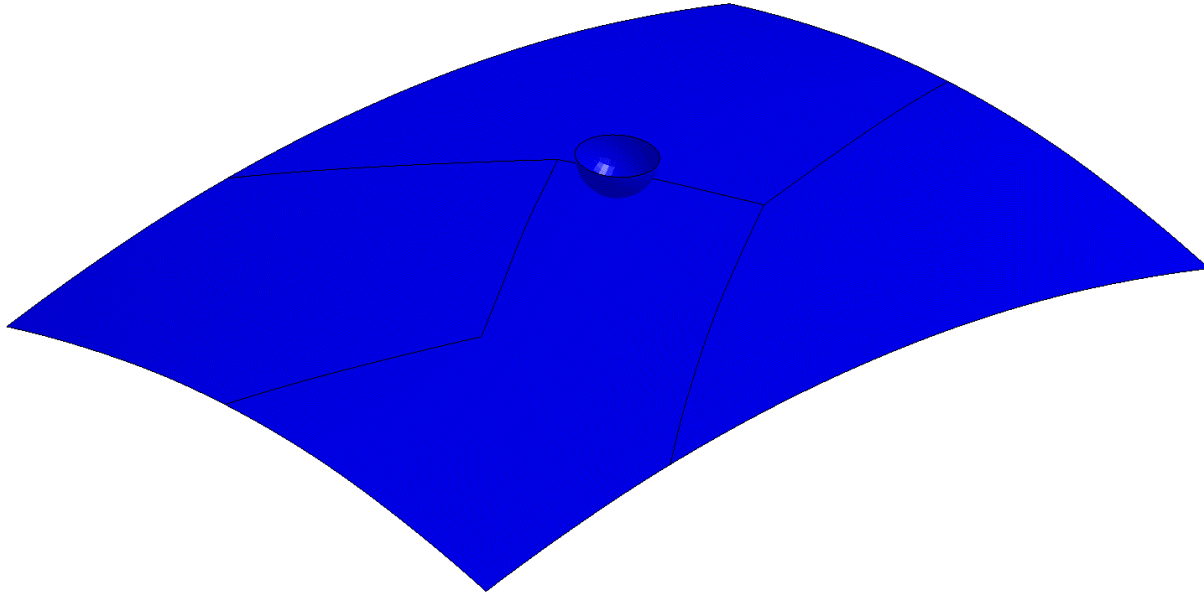
Isogeometric Analysis (*ELEMENT_SHELL_NURBS_PATCH_TRIMMED)

- Mechanical coupling of trimmed patches (shells **with** rotational DOFs)



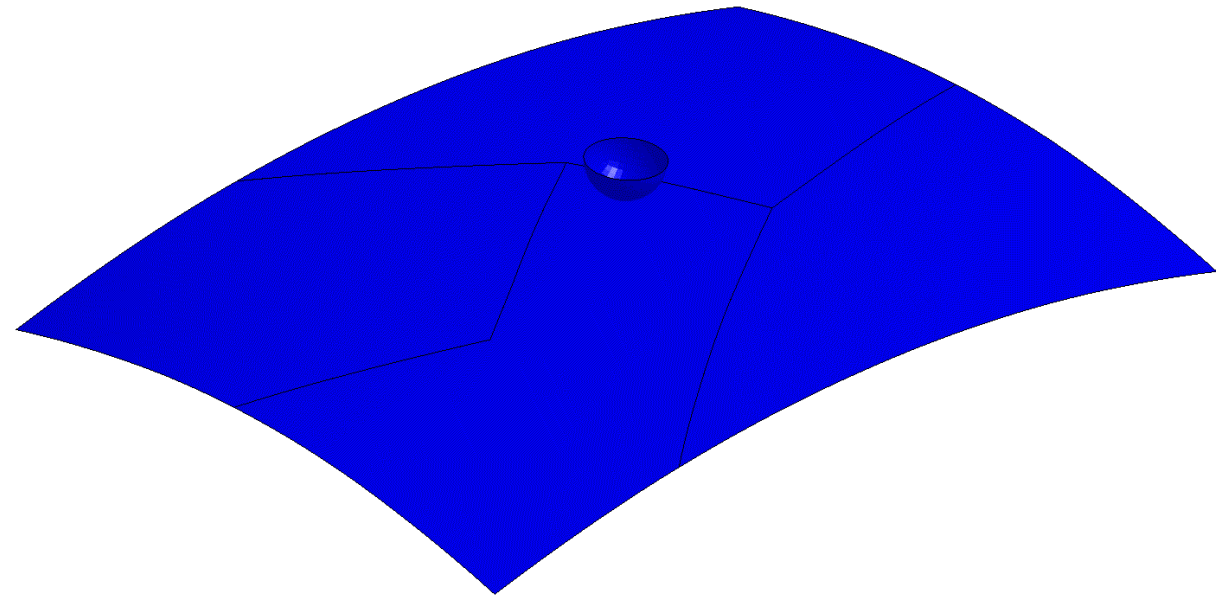
Isogeometric Analysis (*ELEMENT_SHELL_NURBS_PATCH_TRIMMED)

- Mechanical coupling of trimmed patches (shells **without** rotational DOFs)



Without rotational constraints → Kinks!

4 trimmed NURBS patches



With rotational constraints → Continuity!

Isogeometric Analysis (*ELEMENT_SHELL/SOLID_NURBS_PATCH)

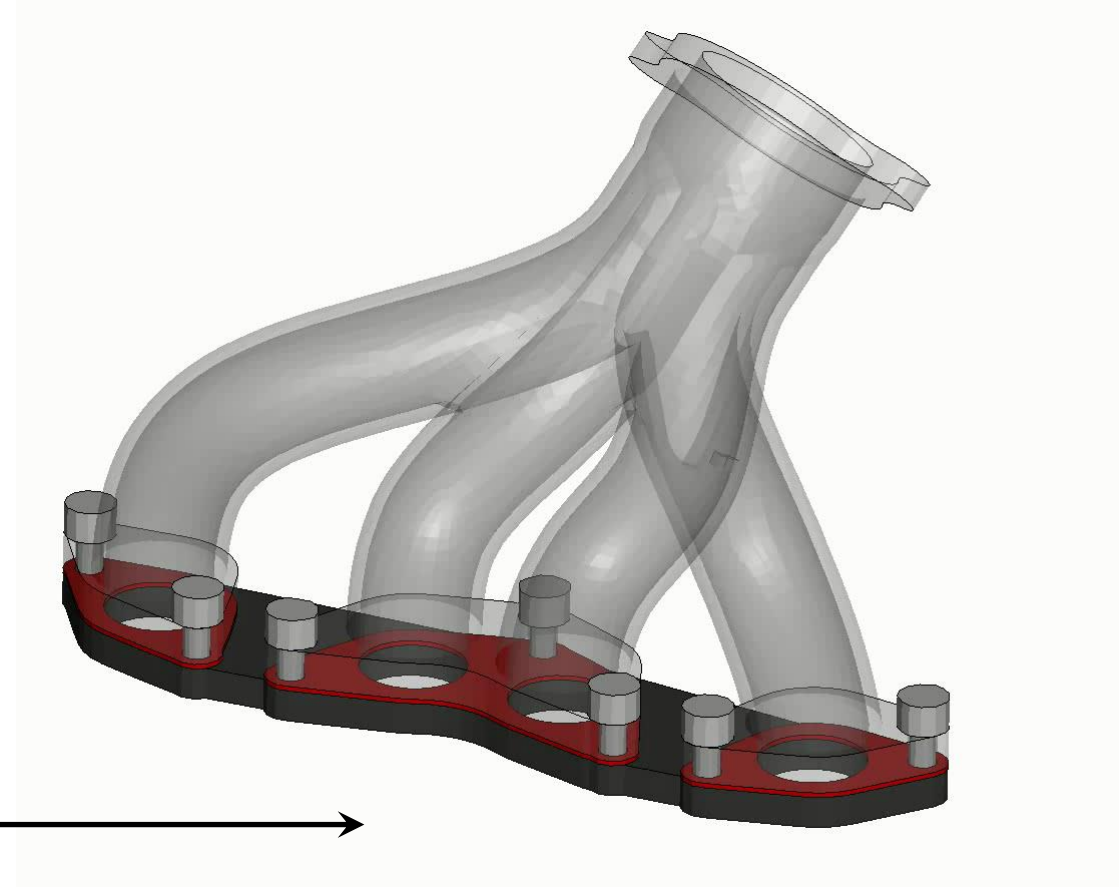
- Allow *DEFINE_SPOTWELD RUPTURE to work with isogeometric shell elements
- Element erosion (deletion) via *MAT_ADD_DAMAGE/EROSION (GISSMO) available for shells and solids
- Implicit contact is now supported via interpolation elements
 - IGA now works for implicit springback
- Thickness change options (ISTUPD in *CONTROL_SHELL) now supported for IFORM=3 IGA shells (rotation free shell formulation)
- Add conventional mass scaling to IGA solids
- Add material models to be supported with IGA shells
 - *MAT_054 (*MAT_ENHANCED_COMPOSITE_DAMAGE)
 - *MAT_224 (*MAT_TABULATED_JOHNSON_COOK)
- Laminated shell theory is now supported for IGA shells
- Remove restriction on “normalized” knot-vectors for NURBS-patch definition
 - “normalization” is not common in CAD files
 - “normalization” may lead to loss of accuracy due to round-off errors

Miscellaneous

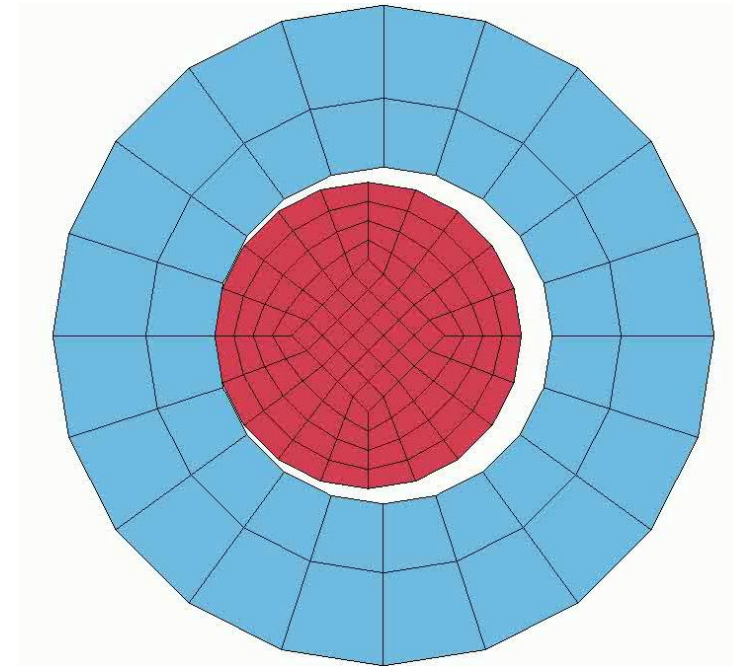
Cohesive elements
Analytical Cylindrical Joint stiffness
Bolt pre-stress ISTIFF/IZSHEAR=2
Define pressure tube updates
SET_PART_TREE
Beam Crush
Erode shells/beams due to low timestep
Case
FMI
QUASAR ROM Coupling
*INITIAL_HISTORY_NODE
Miscellaneous Miscellaneous

Cohesive elements

- Solid cohesive element #20 improvements
 - Drilling stabilization
 - Interface can be arbitrary located between shells - COHOFF on *SECTION_SOLID
- Coming features in R12+, Solid cohesive elements have a gasket option
 - Adds membrane stiffness - GASKET on *SECTION_SOLID
 - Accompanying gasket material - *MAT_COHESIVE_GASKET



- Cylindrical Joint Stiffness for modeling play of axial bearings
 - *CONSTRAINED_JOINT_STIFFNESS_CYLINDRICAL
 - Perfect representation of geometry
 - Friction model and axial limit



Bolt pre-stressing technique - ISTIFF parameter

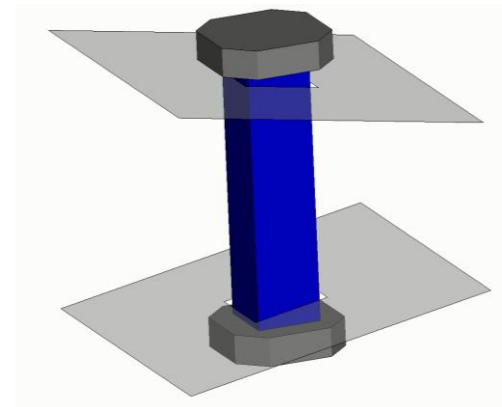
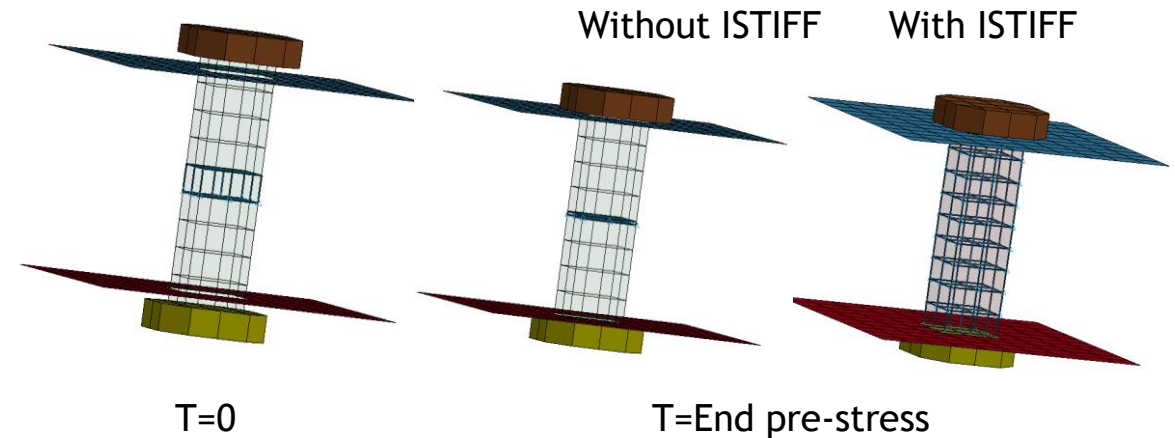
*INITIAL_STRESS_SOLID (IZSHEAR=0/1)

- New option for pre-stress of solid meshed bolts through the ISTIFF parameter. Distribute the pre-stress deformation/distortion along shank of bolt instead of just one row of elements.

IZSHEAR=2

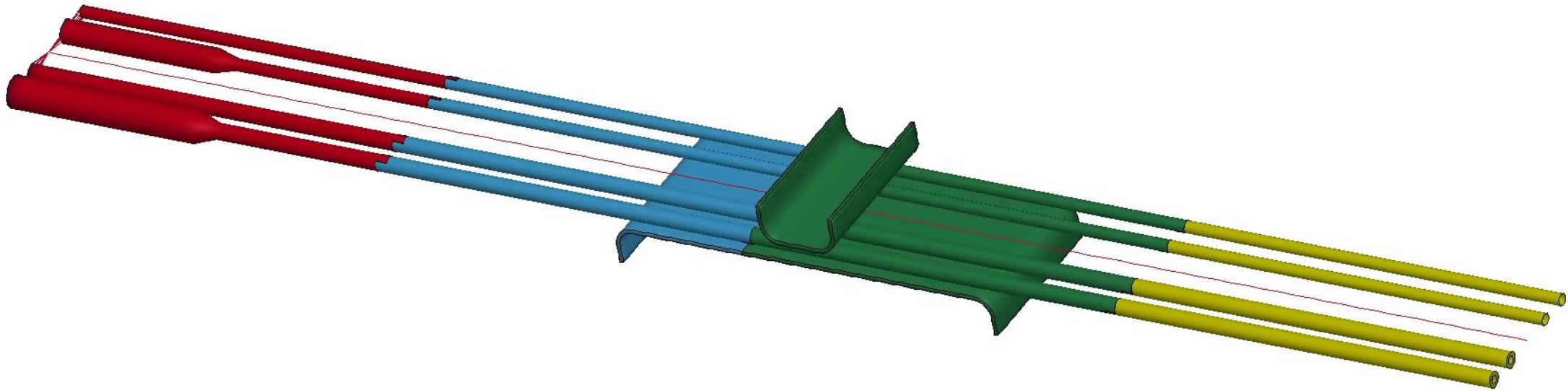
- Similar methodology applied for IZSHEAR=2

Card 1	1	2	3	4	5	6	7	8
Variable	ISSID	CSID	LCID	PSID	VID	IZSHEAR	ISTIFF	
Type	I	I	I	I	I	I	I	
Default	None	None	None	None	None	0	0	



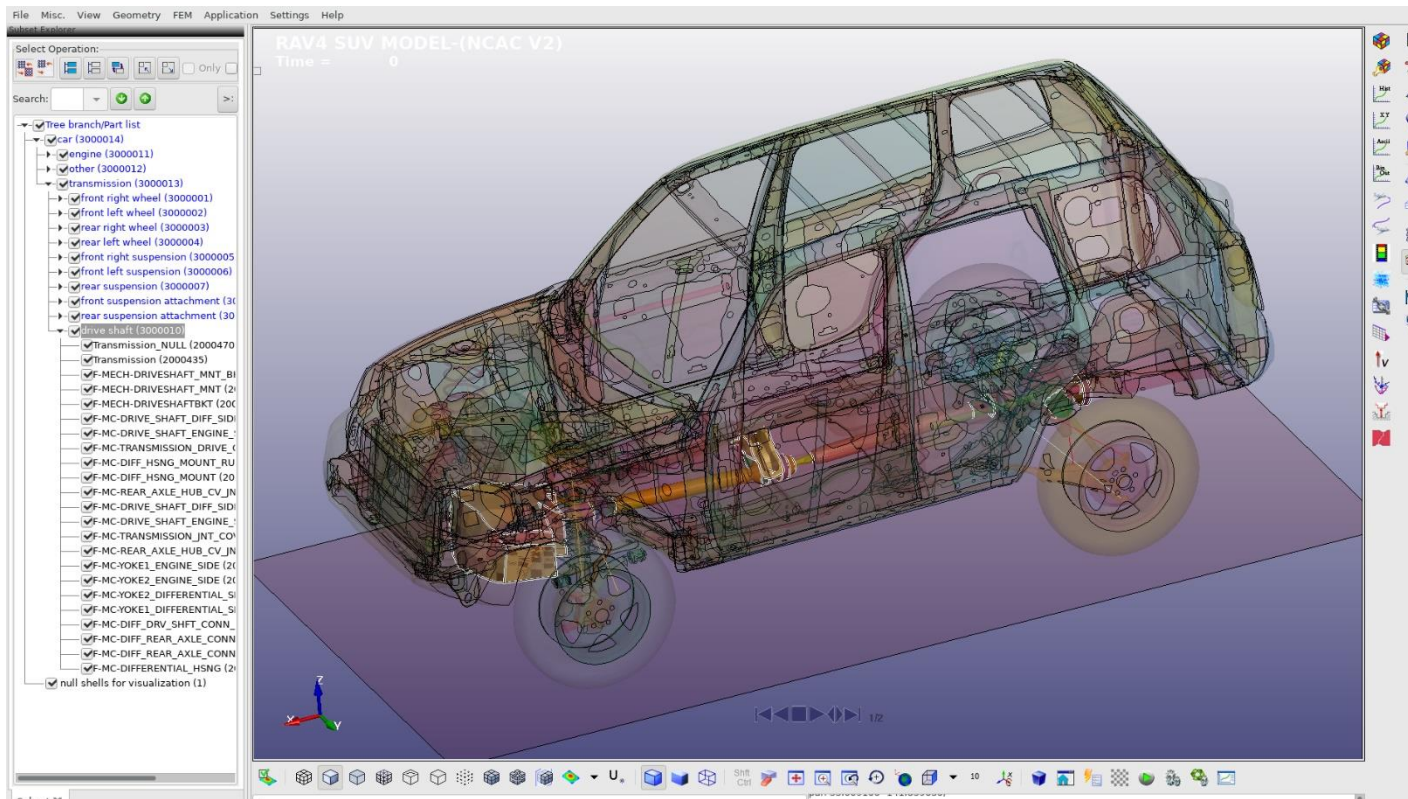
*DEFINE_PRESSURE_TUBE

- Detects collisions by measuring air pressure waves in a front bumper tube
- Now supports decomposition of automatically generated solid/shell tubes in MPP



*SET_PART_TREE

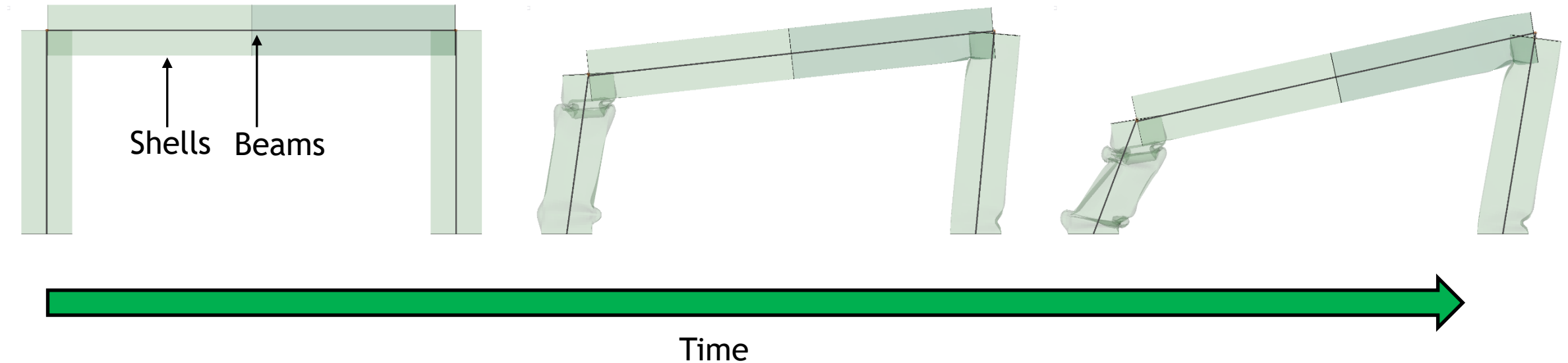
- SET_PART_TREE defines a branch in a tree structure
- With this keyword, the model can be modeled as a hierarchical tree structure
- BRANCH and DBRANCH can be used in *SET_NODE_GENERAL and *SET_SEGMENT_GENERAL



Acknowledgement to George Washington University National Crash Analysis Center

*MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM, IFLAG=2

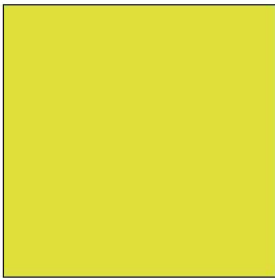
- A displacement formulation to simulate the buckling behavior of crushable frames
- Activate by IFLAG=2 on MAT_119



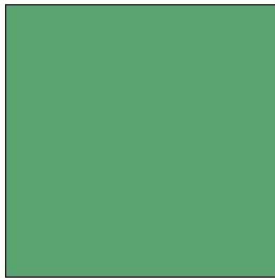
*CONTROL_TIMESTEP, ERODE

- Previously solids and tshells could be eroded based on element timestep, i.e. ERODE=1.
- ERODE has now been extended to also support beams and shells, i.e ERODE=10,11,100,101,110 and 111
- Below is an example with DTMIN = 0.5 and ERODE = 111

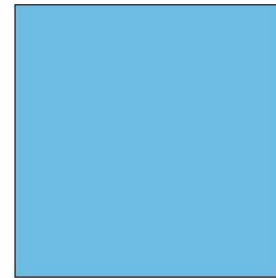
tshell



solid



shell

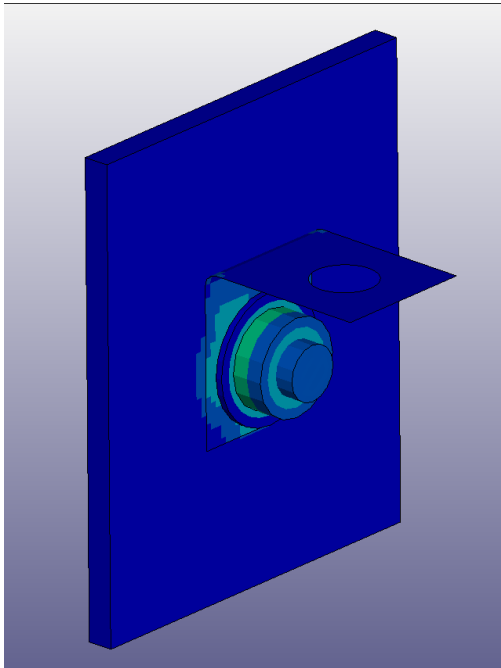


beam

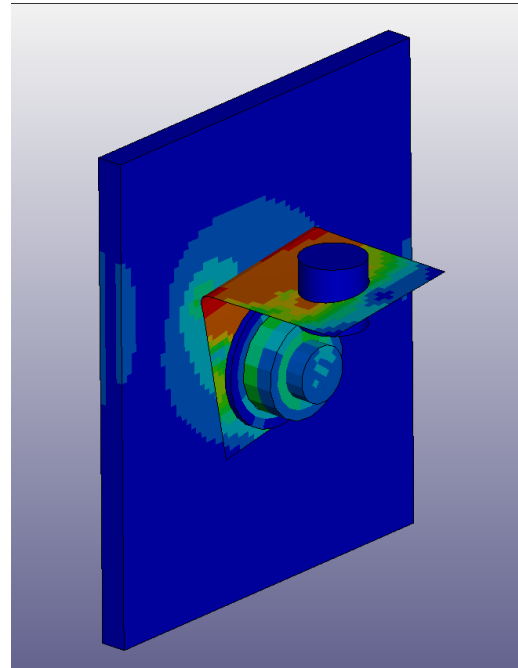


*CASE, run a subset of the cases

- To run a subset of the cases defined in the input deck, specify the case ID number following the word “CASE” on the execution line
- E.g. “CASE=1,3” will run only cases 1 and 3, in sequence



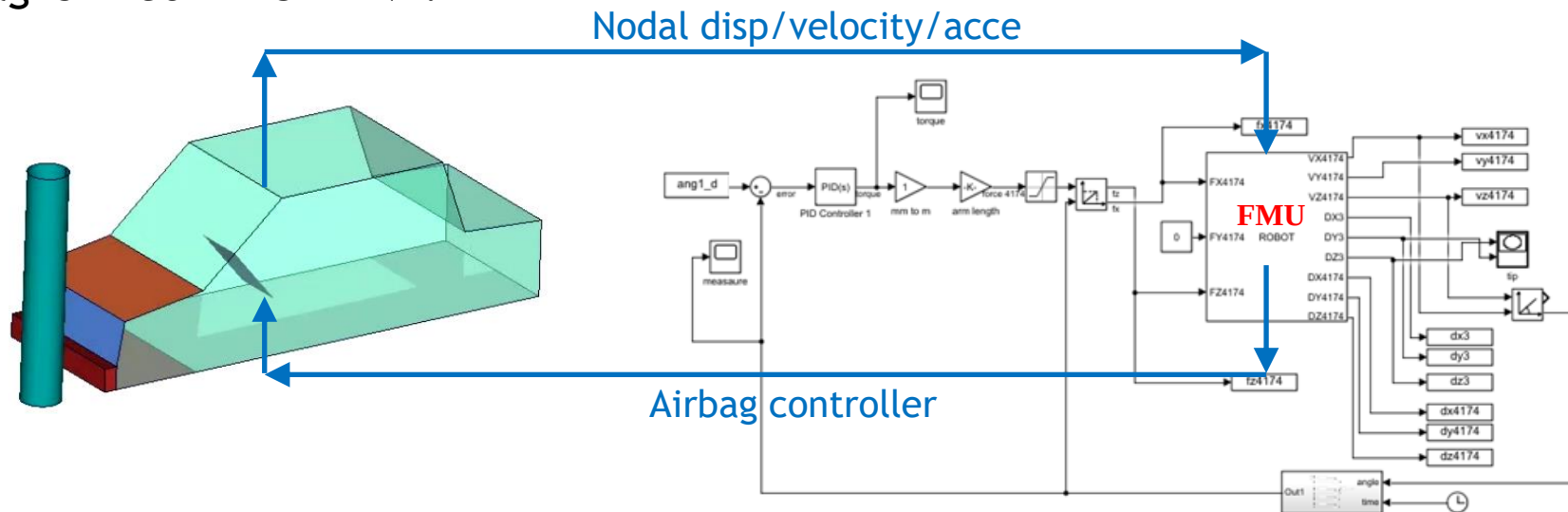
Case 1:Pretension



Case 3:Load in Y-direction

Co-Simulation with LS-DYNA Through FMI (Functional Mockup Interface)

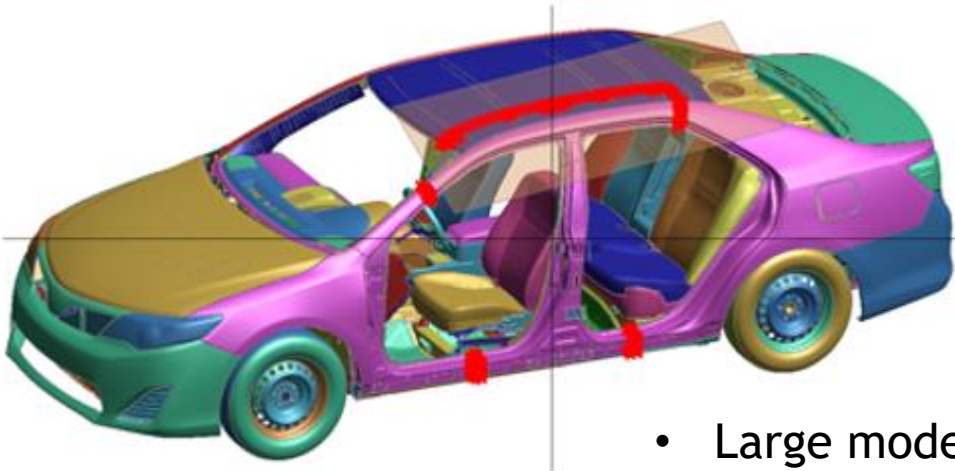
- Co-simulation: Each software contributes their solution results to a coupled, multi-physics problem using specified communication time steps.
- LS-DYNA supports co-simulation with other software through the tool independent FMI standard.
 - TCP/IP communication between solvers
- Example
 - LS-DYNA sends sensor data to airbag controller in another software, that determines when the airbag is fired in LS-DYNA.



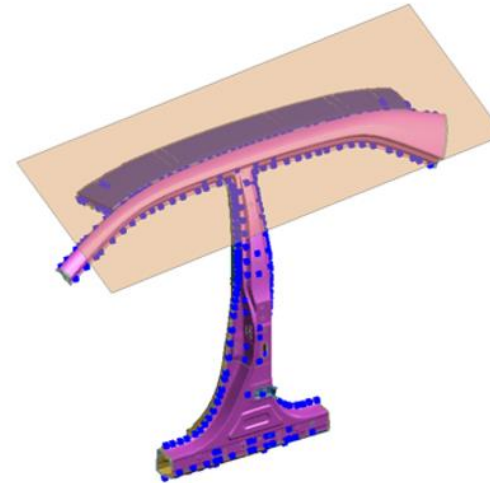
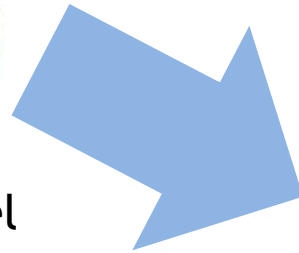
ROM coupling - Reduced Order Modelling

*DEFINE_QUASAR_COUPLING

- Define node/node set that interacts with Cadlm's QUASAR ROM model.



- Large model
- Small model with approximated boundary conditions



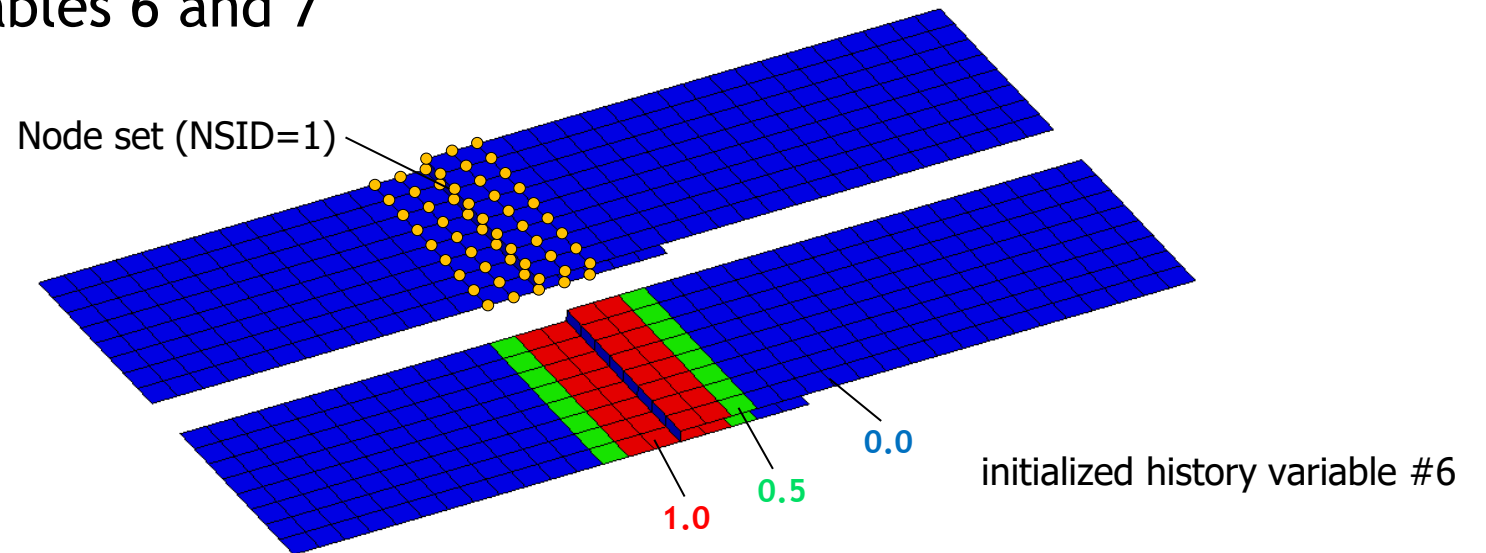
Predict the response at the boundary using ROM and co-simulation with Quasar.

Card 1	1	2	3	4	5	6	7	8
Variable	NODE	TYPE	ROMID	PID	PTYPE			
Type	I	I	I	I	I			
Default	None	1.0	None	None	None	None	None	None
Card 2	1	2	3	4	5	6	7	8
Variable	FILENAME1							
Type	A80							
Default	None							
Card 3	1	2	3	4	5	6	7	8
Variable	FILENAME2							
Type	A80							
Default	None							

*INITIAL_HISTORY_NODE(_SET)

- Initialize certain history variables on a nodal basis
 - Available for: shells, tshells and solids
- The nodal values are interpolated using standard FE shape functions
 - shells: interpolation w.r.t. to in-plane IPs, all IPs through the thickness receive the same value (might not be suitable for layered composite shells)
 - values at uninitialized nodes are assumed to be ZERO
- In contrast to *INITIAL_STRESS_SHELL, individual history variables can be initialized
- Example: initialize history variables 6 and 7

```
*INITIAL_HISTORY_NODE_SET
$   NSID      NHISV
      1         2
$  HINDEX1    VAL1
      6        1.0
$  HINDEX2    VAL2
      7         0.1
```



Miscellaneous

- New option TET13V on *CONTROL_SOLID
 - choose between the efficient or a more accurate version of the tet type 13 implementation (non-default TET13V=1 invokes previous behavior!)
- New options for *DEFINE_TRANSFORMATION
 - TRANSL2ND: translation given by two nodes and a distance
 - ROTATE3NA: rotation given by three nodes and an angle
- New option for *PERTURBATION_NODE
 - DTYPE=1 to allow uniform distribution between $SCL \times [-AMPL, AMPL]$ for random value perturbation (TYPE=8)

Thank you!



Your LS-DYNA distributor and
more

