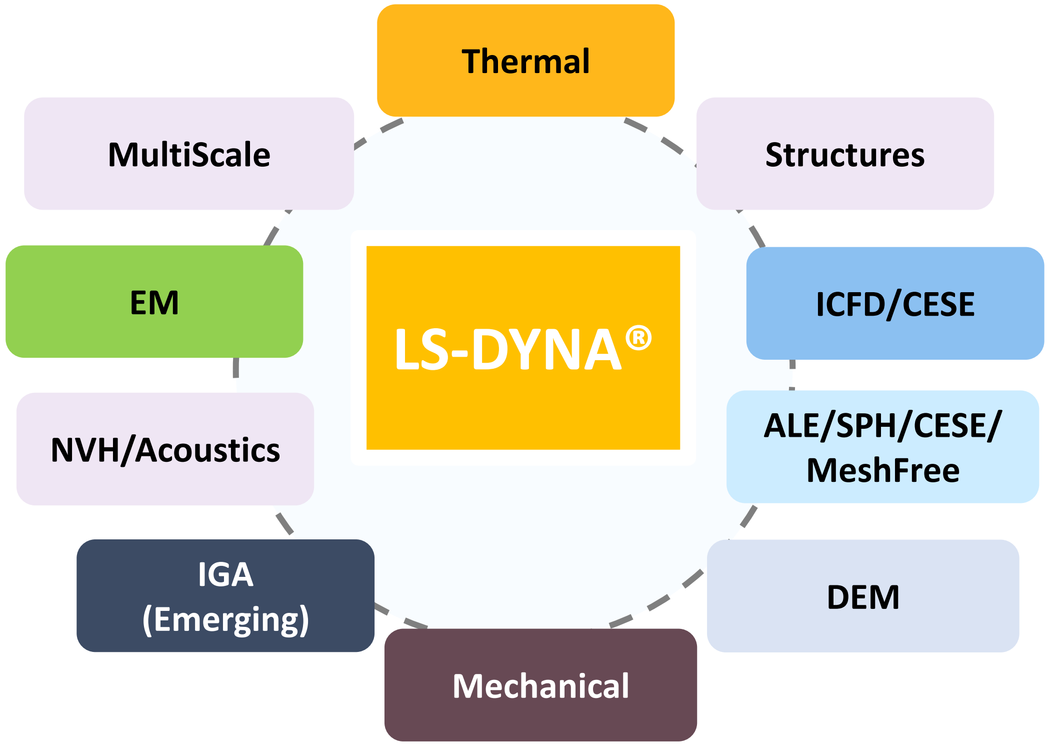
Release 2023 R1 Highlights

Ansys LS-DYNA R14

Ansys

2023R1 (R14) Release features

- Vast amount of new capabilities
- All integrated in One Code strategy
  - Tightly Coupled, Scalable Multi-Physics Solver
- Product available in January 2023
  - Minor release 2023R2 in July 2023
  - Service packs as needed
  - All other tools are released at the same time: ANSYS Forming, LS-OPT Pro, LS-TaSC, …
- Detailed documentation in User's Manuals, release notes



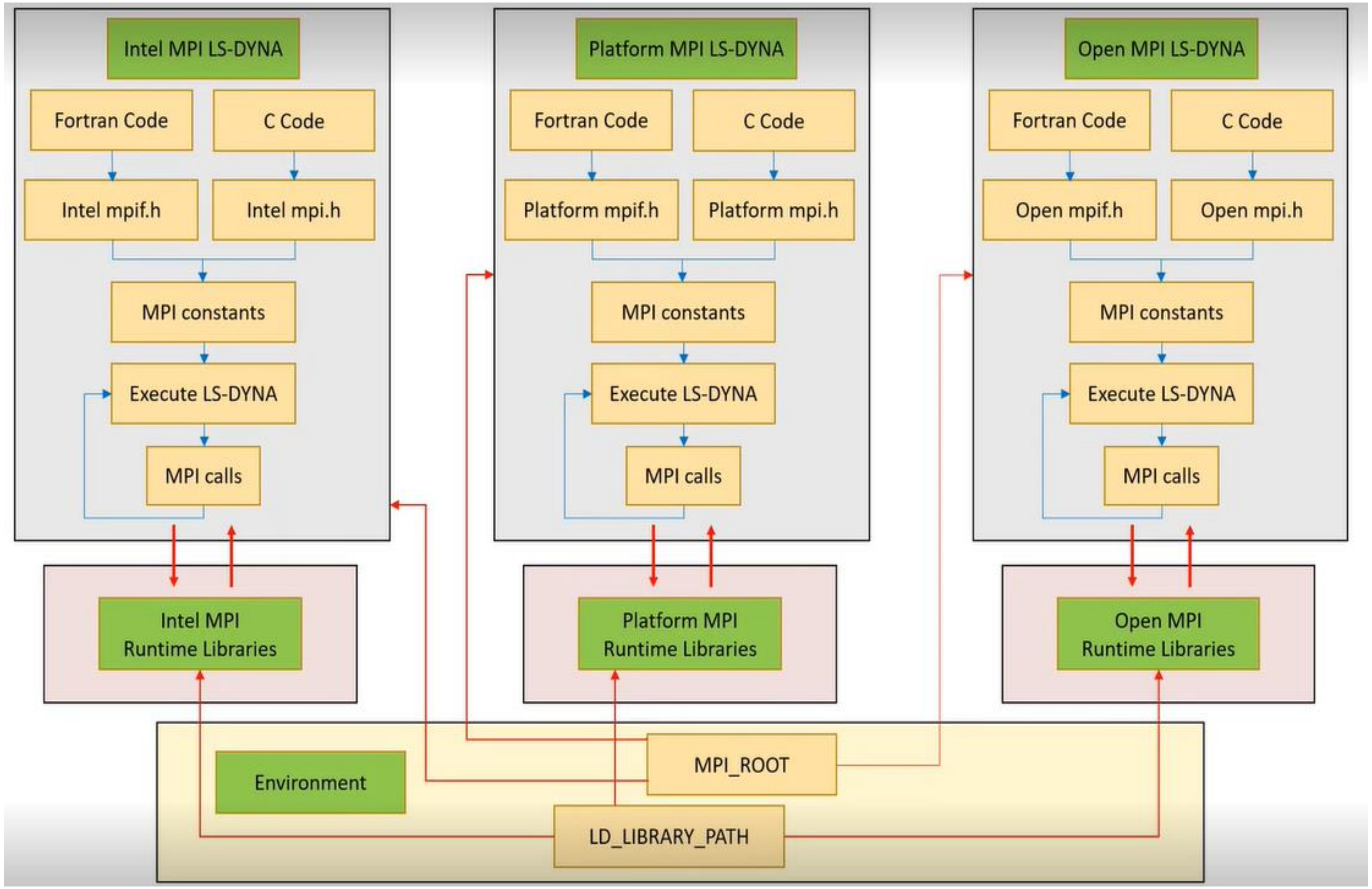
HPC Updates

Ansys

LS-DYNA OneMPI

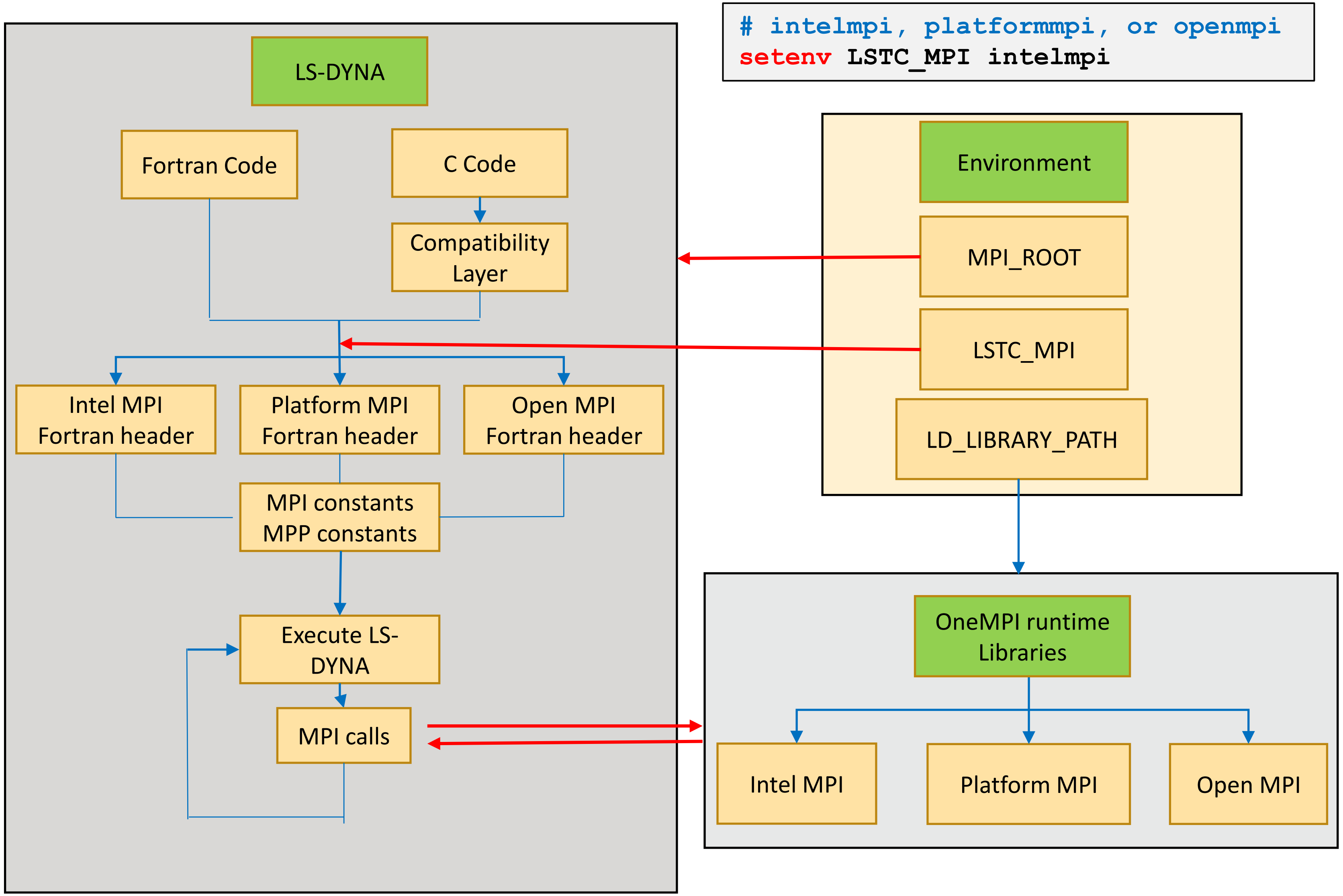
Currently under x86-64 Linux:

1. Three different binaries for Intel MPI, Platform MPI and OpenMPI
2. There is NO more update for Platform MPI (MPI 2.x)
3. Groups may select their own MPI for joint project. (they need use the same revision, same optimization, etc. to get the consistent results between groups.)



LS-DYNA OneMPI

1. **Intel MPI binary** has the functionality of three MPP binaries in one
   Linux: Intel MPI 2018, Open MPI 4.x, Platform MPI
   Windows: Intel MPI 2018 and MSMPI v10
2. Identical solutions between MPI (LSTC_REDUCE on) to make easier between different groups
3. Help to convert from Platform (MPI 2.x) MPI to new default (MPI 3.x and above)
4. Help LS-DYNA align with other Ansys Products by limiting quantity of binaries
5. Accelerate LS-DYNA release process by reducing MPP QA three-fold



LS-DYNA OneMPI

Supports Intel, Platform, and Open MPI in a single binary

- **`LSTC_REDUCE`** provides identical decomposition and solutions among MPI
- Usage
  - Set new environmental variable, **`LSTC_MPI`**
  - Run provided shell script to create OneMPI Libraries

```
# intelmpi, platformmpi, or openmpi
setenv LSTC_MPI intelmpi
```

Benefits

✓Functionality of three MPP binaries in one

✓Identical solutions from all MPI

✓Help to convert from Platform MPI to new default

✓Accelerate LS-DYNA release process by reducing MPP QA three-fold

✓Help LS-DYNA align with other Ansys Products by limiting quantity of binaries

Airbag Enhancements

Summary of CPM airbag features

Fabric blockage evaluation	• Collect nodal contact force (excluding transducers) for each airbag • Set local blockage threshold to avoid inconsistent leakage while changing contacts
IBLOCK=21: External vent control for airbag	• Collected nodal contact force for vent nodes to calculate contact pressure • Enable vent when bag pressure is greater than contact pressure to avoid early leakages
BLOCKV: External vent blockage excluding forces from airbag self contact	• Excluded nodal contact force from airbag single surface contact (soft=2) • Excluded nodal contact force from contact force transducer
POPP for external/internal vent	• Compare up/downstream PART/ambient pressure • Compare up/downstream CHMABER/ambient pressure
CPM/Thermal coupling	• CPM particle receives heat flux from/to airbag structure • Airbag structure receives heat flux from/to CPM as source/sink and perform thermal solution

Blockage evaluation

1) Total nodal contact force from all contacts, F_{total}
2) Define threshold for node in contact based on F_{total}, $F_{threshold}$
3) Airbag segments are triangulated.

$$F_{threshold} = 1.0e\text{-}6 \; \frac{F_{total}}{Number\ of\ nodes\ in\ the\ model}$$

If $F_{node} >= F_{threshold}$ *, one third of the airbag segment is blocked.*

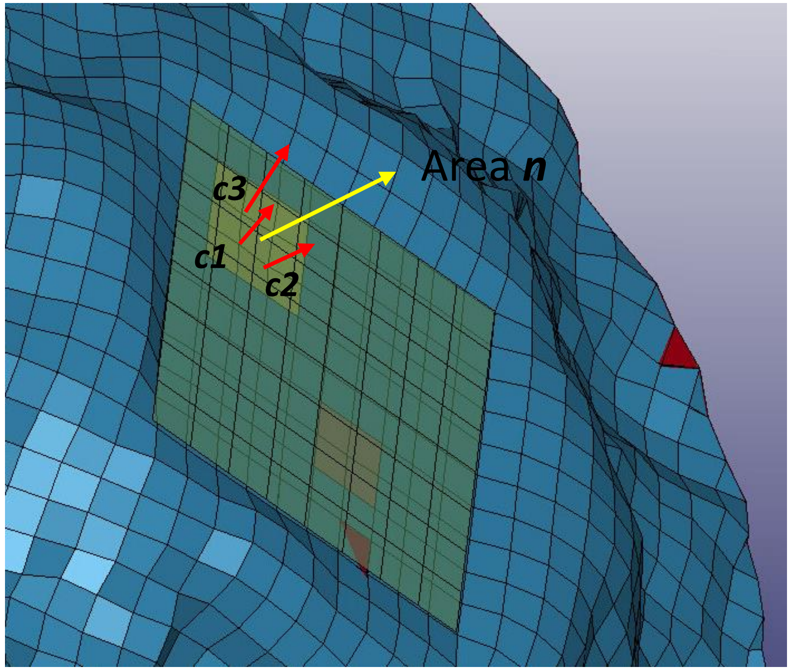
1) Airbag nodal contact force from all contacts except force transducers for each airbag *i*, $F_{i,total\_airbag}$
2) Define threshold *i* for node in contact based on $F_{i,total\_airbag}$, $F_{i,threshold}$
3) Airbag segments are triangulated.

$$F_{i,threshold} = 1.0e\text{-}6 \; \frac{F_{i,total\_airbag}}{Number\ of\ nodes\ in\ current\ airbag}$$

If $F_{node} >= F_{i,threshold}$ *, one third of the airbag segment is blocked.*

IBLOCK=21

1) Only applies to external vents
2) All airbag segments are triangular shape
3) No blockage if bag/chamber pressure > contact pressure
4) Use nodal blockage as usual if bag/chamber pressure < contact pressure



c1, c2, c3 are contact nodal forces collected from all contact definitions

- BLOCK=01 and 11
  If ***c1 or c2 or c3*** > ε, one third of segment area is blocked

- BLOCK=21
  Contact pressure = $\frac{THIRD*(\boldsymbol{c1}+\boldsymbol{c2}+\boldsymbol{c3})\cdot \boldsymbol{n}}{Area}$
  If bag/chamber pressure > Contact pressure, no blockage
  Otherwise, uses BLOCK=01,11 logic

Blockage evaluation for external vent

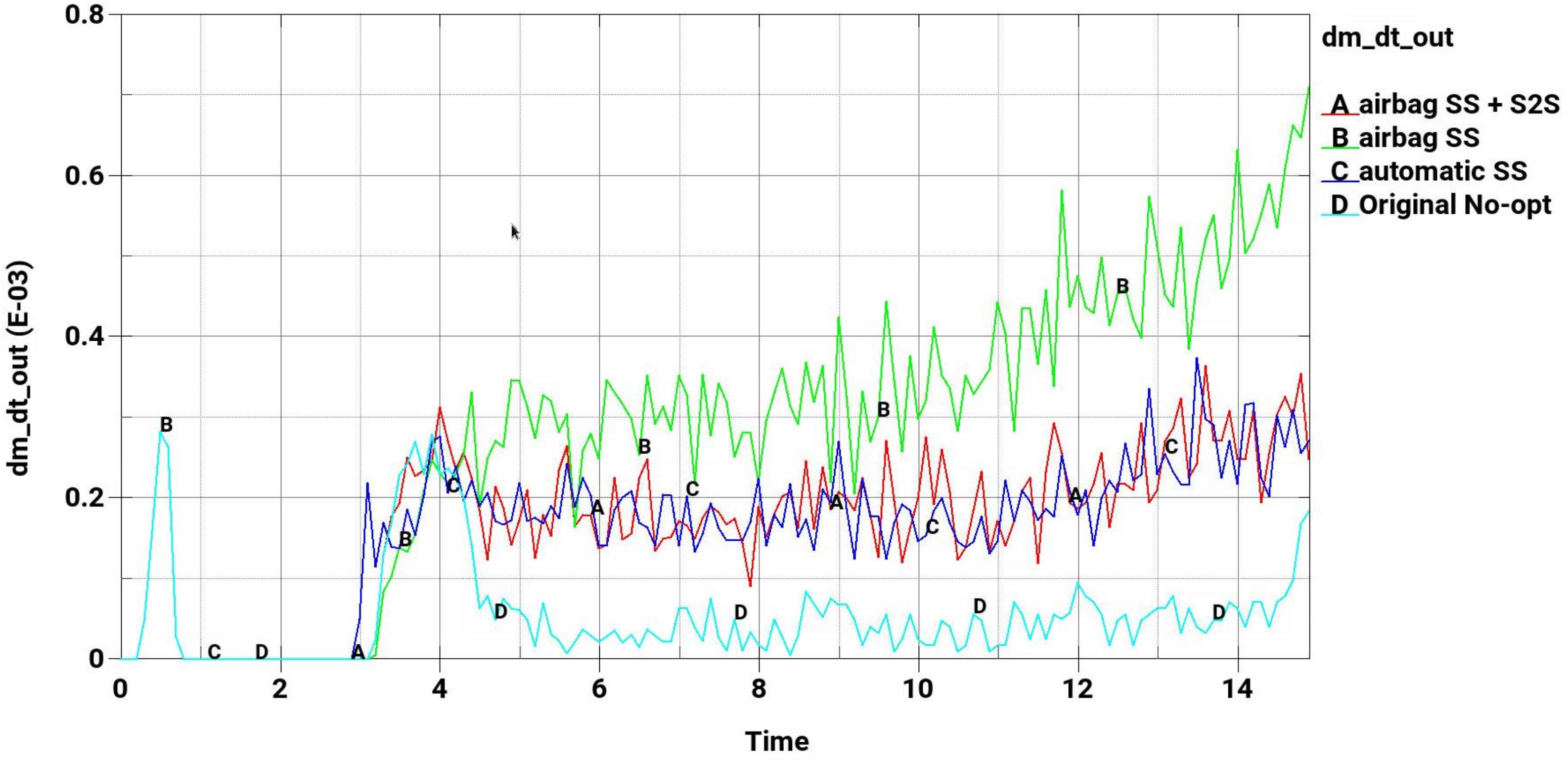
- *CONTROL_CPM, the 6th field BLOCKV
- Additional memory are allocated to store contact nodal force excluding, force transducer, *CONTACT_AIRBAG_SINGLE_SURFACE with soft=2.
- ONLY for external vents

1) Airbag nodal contact force from all contacts except force transducers and AIRBAG SS/soft=2 for each airbag *i*, $F_{i,total\_airbag}$
2) Define threshold *i* for node in contact based on $F_{i,total\_airbag}$, $F_{i,threshold}$
3) Airbag segments are triangulated.

$$F_{i,threshold} = 1.0e\text{-}6 \frac{F_{i\ total\ airbag}}{Number\ of\ nodes\ in\ current\ airbag}$$

If $F_{node} >= F_{i,threshold}$ *, one third of the airbag external vent segment is blocked.*

Blockage evaluation for external vent



1) Model D: Original option with 2 contacts (AIRBAG SS and S2S) – too much blockage
2) Model A: New option (BLKV=1) with 2 contacts – excluding AIRBAG SS effect and less blockage
3) Model C: Replaced 2 contacts into 1 (automatic SS) – there is blockage effect
4) Model B: Replaced 2 contacts into 1 (AIRBAG SS) – no blockage due to new option

POPP for extern/internal vent

Vent definition	External Vent	Internal Vent
Part pressure	$P_{part} - P_{ambient}$	$P_{part1} - P_{part2}$
Chamber	$P_{chamber} - P_{ambient}$	$P_{chamber1} - P_{chamber2}$
None	(3)	(3)

1. The POPP pressure is always the pressure difference between upstream/downstream
2. Priority
   Part pressure > Chamber > none
3. If part pressure and chamber are not defined, up/down stream pressure are evaluated by collecting translation kinetic energy of near by particles therefore the signal is very unpredictable.
4. UP(Switching from CPM) need more development

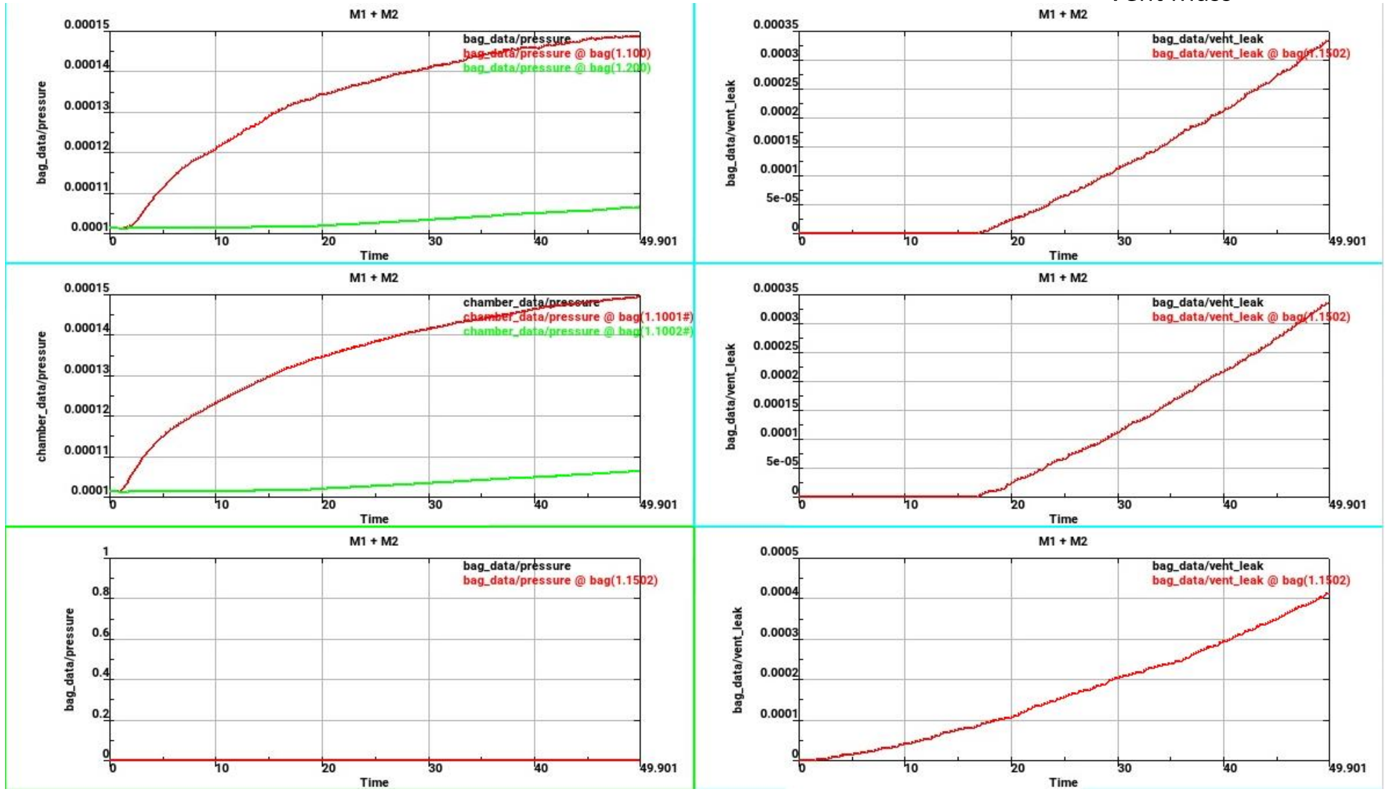
LCPC23 for external vent

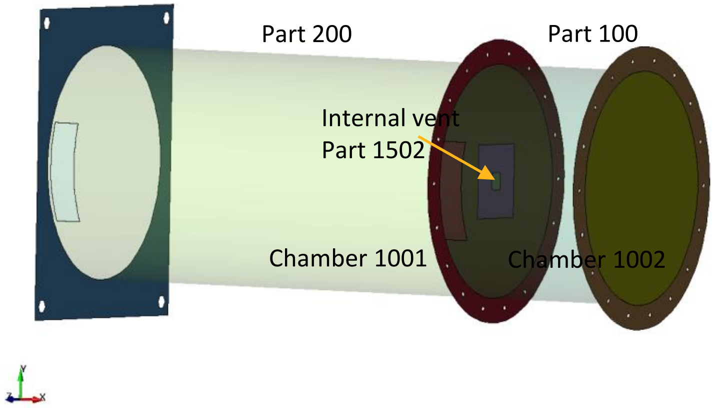
Pressure | Vent Mass

Part pressure

Chamber pressure

None



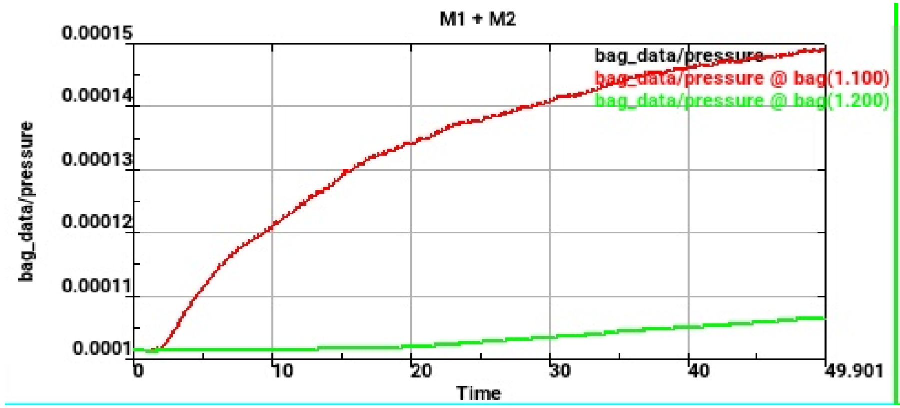


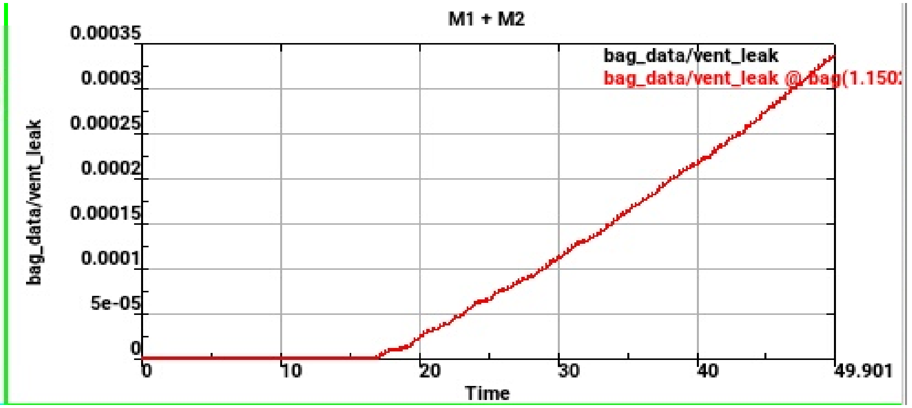
Pressure for lcpc23:

- Absolute pressure
- 1st priority: Part pressure from part define in *DEFINE_CPM_VENT PID1
- 2nd priority: Chamber pressure associated with this vent
- 3rd priority: Pressure measure near vent using SENSOR algorithm (unpredictable)

LCPC23 for internal vent

```
*AIRBAG_PARTICLE_ID
         1
      1507         1      1509         1         1         1         0         1
     10000         0         1     296.0  1.013E-4         1       0.0       0.0
         1         1         1         0         0         0      1000       0.0
      1507         1    1.0E-7         0       0.0       0.0         0       0.0
      1502         0       0.0         0       313         0       0.0
  1.013E-4     296.0   0.02897     26.71  0.007466 -1.323E-6         0         0
       311       312  0.004003     20.77       0.0       0.0         0
     11083       1.0      1001      30.0         0         0         1      1001
….
*DEFINE_CURVE
       313         0       0.0       0.0       0.0       0.0         0         0
                 0.0          9.9999997E-6
          0.29999E-4          9.9999997E-6
             0.30E-4                   1.0
                 2.0                   1.0
```





Pressure for lcpc23:

- Pressure different between up/down stream of the vent
- 1st priority: Part pressure difference from part define in *DEFINE_CPM_VENT PID1/PID2
- 2nd priority: Chamber pressure difference between this vent

Ansys

CPM/ Thermal solver coupling

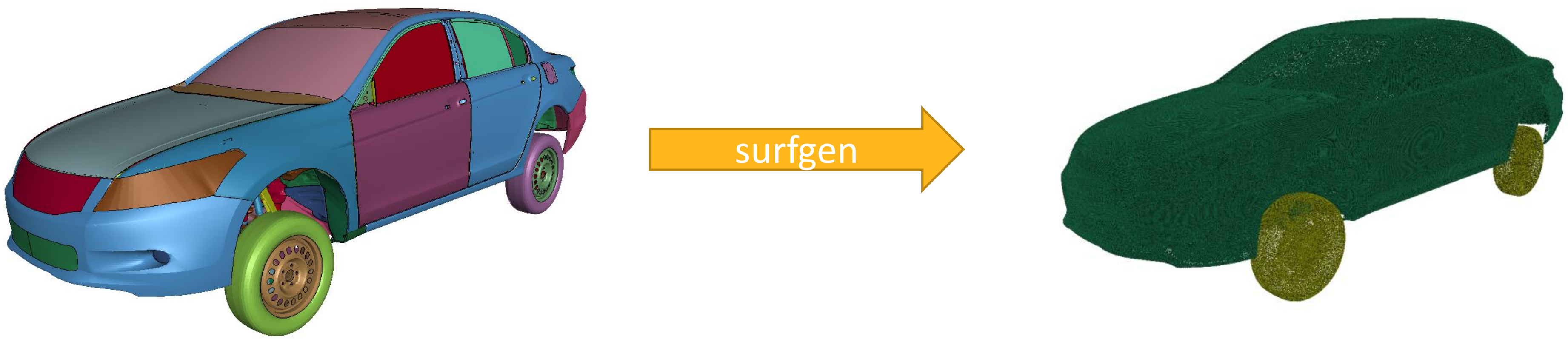
1) *AIRBAG_PARTICLE
2) *CONTROL_THERMAL_SOLVER
3) CPM particle receives heat flux from/to airbag structure
4) Airbag structure receives heat flux from/to CPM as source/sink and perform thermal solution
5) Extra data defined in *DEFINE_CPM_NPDATA

SPH

Summary of new features in R14

New workflow for ISPH	• Created utility tool to process SpaceClaim data and create an LS-DYNA ISPH input file. • Added OBJ geometries in D3PLOT and interface files for better visualization. • Free-surface SPH rendering with Ensight.
ISPH: Downstream Structural Analysis	• Added capability to record time history of FSI forces on designated parts in LSDA *INTERFACE file. • FSI forces can then be interpolated and played back on implicit deformable analysis of components.
ISPH: Volume Constrained Particles	• In some scenarios like gearbox analysis, some fluid particles end up trapped in narrowing gaps. • State-of-the-art was to simply deactivate and remove such particles. • Created an innovative mapped displacement solution to maintain stability and conserve volume.
ISPH: Heat Transfer Coefficient Calculation	• Dittus-Boelter type correlation laws are employed to compute HTC (no thermal analysis). • Time-averaged, spatially resolved HTC maps can be output as point cloud data. • Resulting profile file can be imported in Mechanical or Fluent for thermal analysis.
Misc. SPH	• Added new fragmented SPH box, coupled with MPP Redecomposition, to add particles to input file as they gain active status. • No-slip boundary condition for explicit SPH to impose velocities near boundaries.

New Workflow for ISPH Analysis



CAD geometry from SpaceClaim

Geometry sampled with particles.

Provide complete workflow to perform ISPH analysis directly from CAD geometry, without the need for a full vehicle FEA mesh:

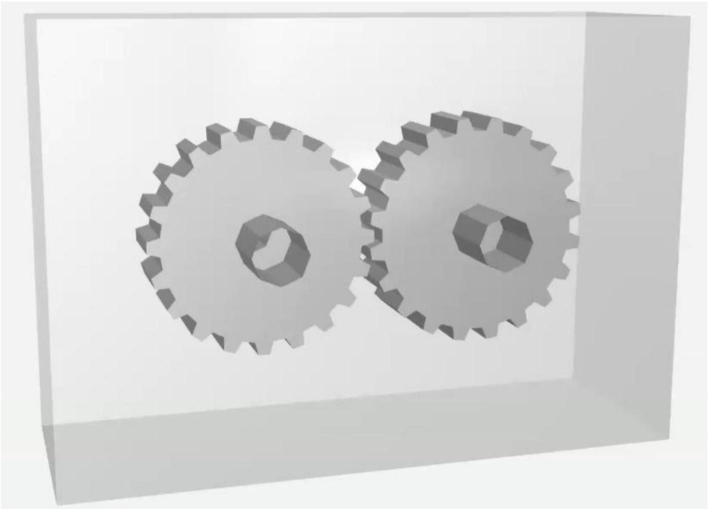
- Generate LS-DYNA input file from SpaceClaim data (*surfgen* utility).
- Import OBJ geometry for visualization in D3PLOT results.

```
*DEFINE_SPH_MESH_OBJ
$#                                                                      filename
component.obj
$#     ppid         pid      sphpid
        101           0           0
```

New SPH visualization in Ensight

SPH particles rendered
as spheres

SPH particles rendered
as smooth surface



Example on demo gearbox

Ensight's new rendering scheme for SPH particles enables realistic visualization of fluid. The animation looks realistic and the FSI interaction is a lot more evident. Setup in Ensight is trivial.

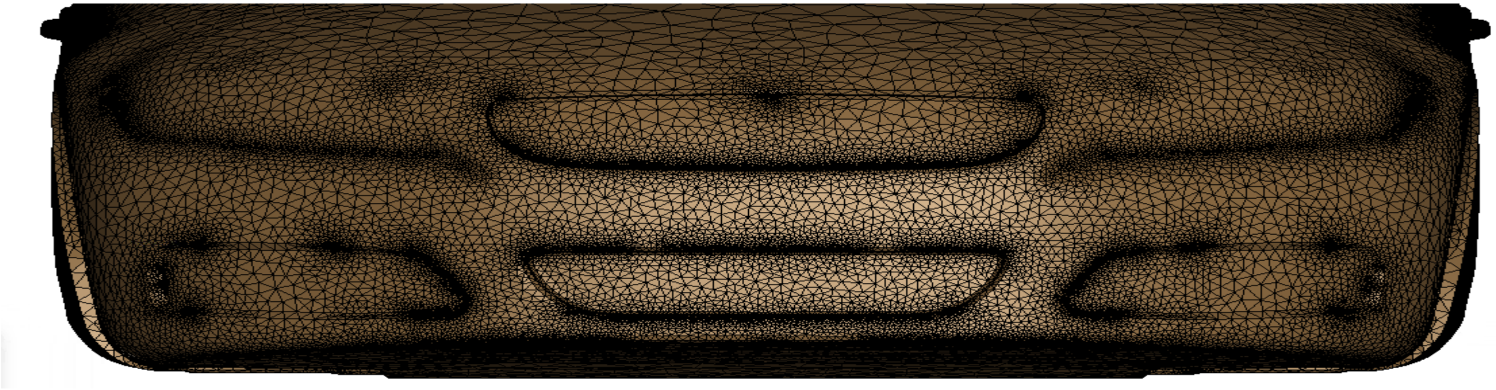
ISPH: Downstream Structural Analysis

OBJ geometry used in wading analysis. The whole structure is considered rigid.

Transfer of forces vs. time

FEA mesh used for component analysis.

OBJ Geometry

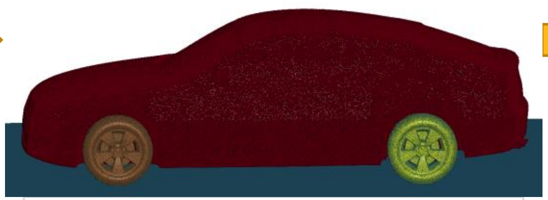


FEM mesh

ISPH: Downstream Structural Analysis

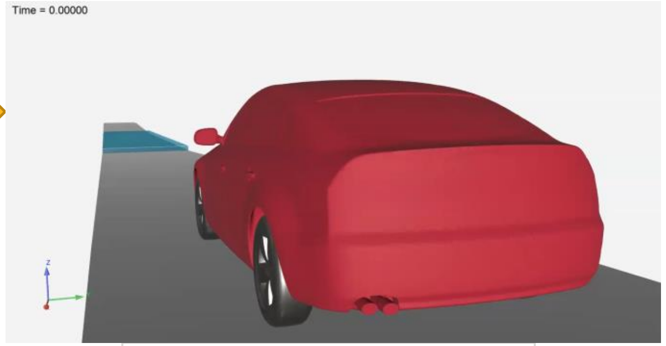
Typical CAD geometry

surfgen



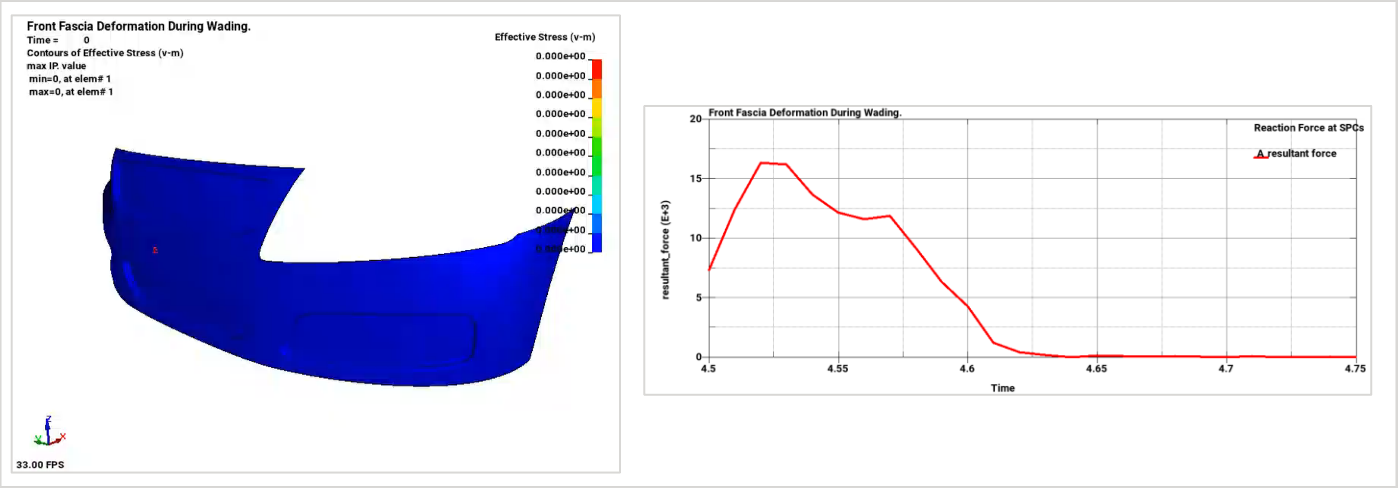
Geometry sampled with particles.

LS-DYNA



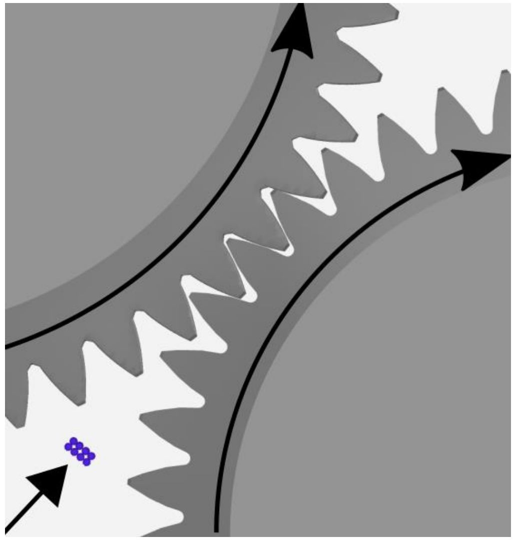
Wading Analysis, ISPH solver

Interface forces

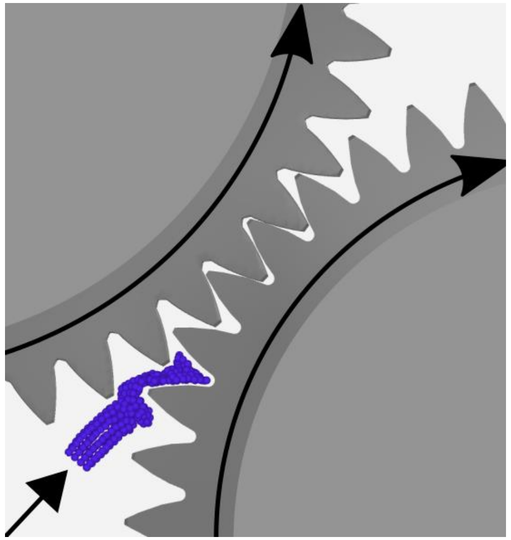


Structural Analysis, LS-DYNA implicit solver

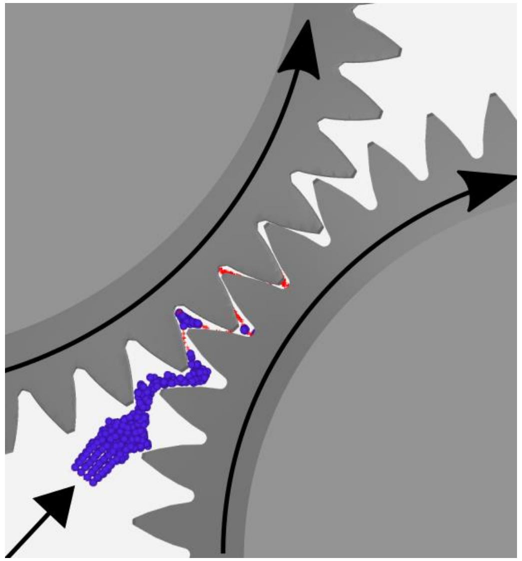
ISPH: Volume-Constrained Particle Treatment



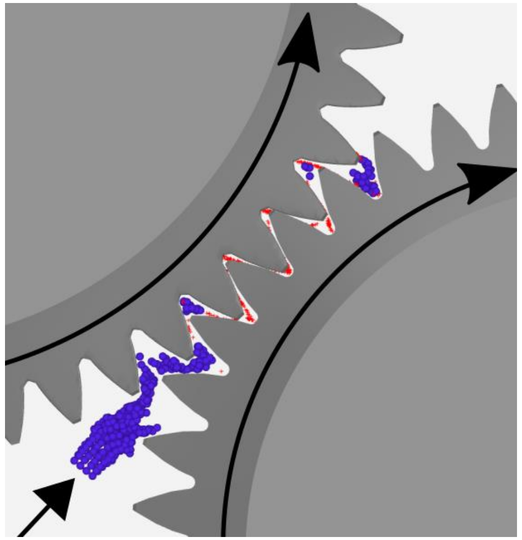
1
Fluid particles (blue) are approaching two converging structures.



2
The fluid gets pushed in a narrowing gap between gear teeth.



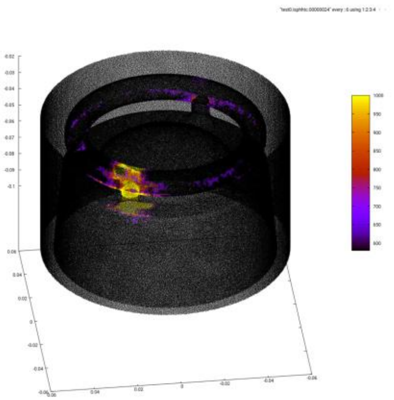
3
Some fluid particles are detected as Volume-Constrained, and get flagged accordingly (red crosses).



4
As the VC particles get carried to the other side through mapped displacements, they exit their Volume-Constrained condition and become regular fluid particles again.

ISPH: Heat Transfer Coefficient Calculation

- The classical approach for calculating HTCs on structural components involves calculating heat flux at the wall, which would require fine boundary layer discretization. -> Impractical in ISPH.
- Empirical laws correlating Nusselt number and Prandtl number are used instead. The resulting HTCs can be imported in Ansys Mechanical or Fluent for thermal analysis.
- Custom correlations can be employed through *DEFINE_FUNCTION.



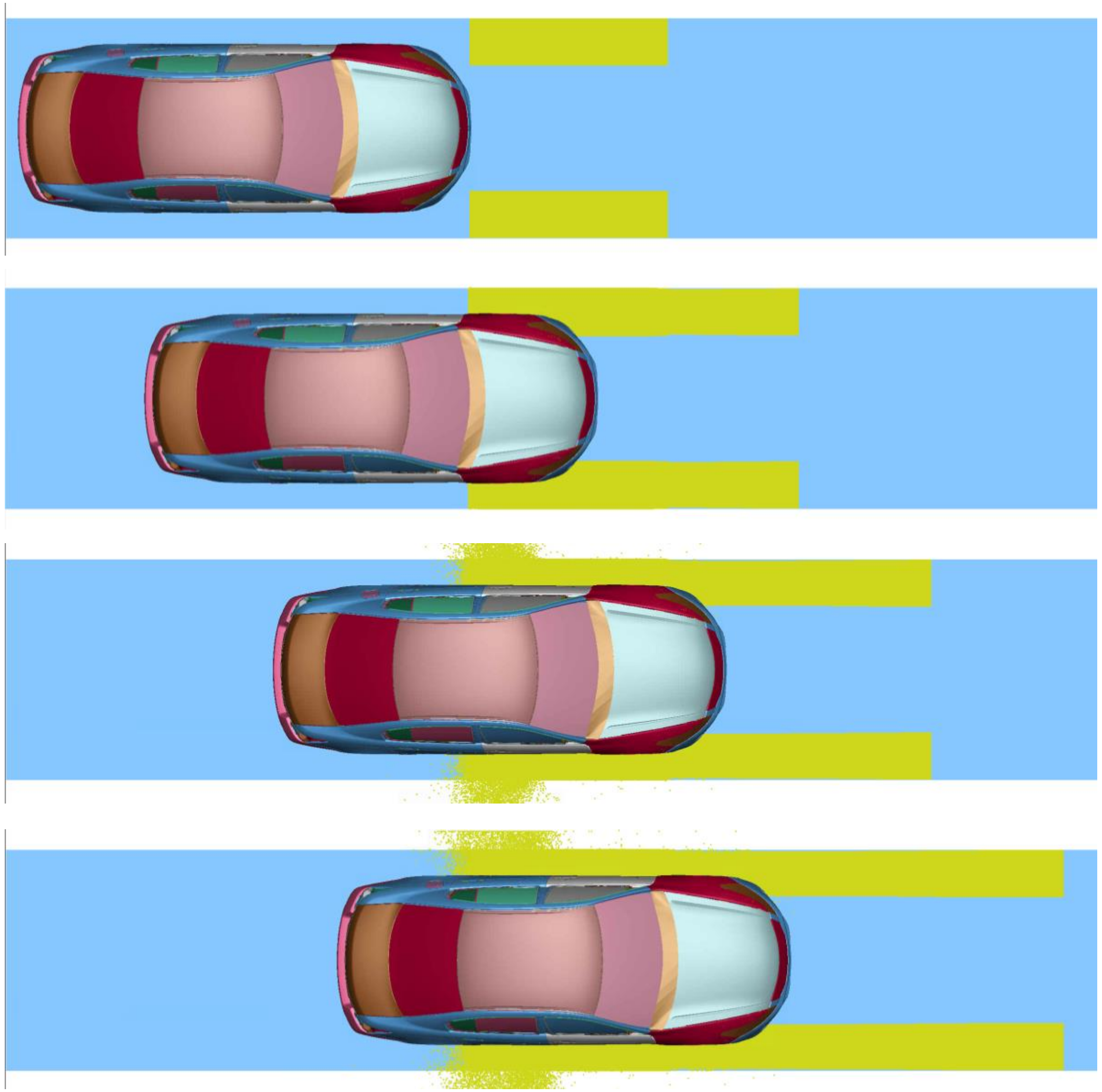
Heat Transfer Coefficient Mapping

```
*CONTROL_SPH_INCOMPRESSIBLE
$#    ithk      istab         ql         rol         htc     newmat
         1          0.011.00000E201.00000E20        -100          0
*DEFINE_FUNCTION
100, HTC
float htc(float cp, float lambda, float dynvisc, float rho, float vtan,
           float dxi, float rdis) {
  double pr, reynolds, nux;
  pr       = dynvisc*cp/lambda;
  reynolds = vtan*dxi/dynvisc*rho;
  if(rdis>2.00) return -1.0;
  if(reynolds<1.e5) {
    nux = 0.3332*pow(pr,0.33333)*sqrt(reynolds);
  } else {
    nux = 0.0293 *pow(pr,0.33333)*pow(reynolds,0.8);
  }
  return lambda*nux/dxi;
}
```

Custom *DEFINE_FUNCTION

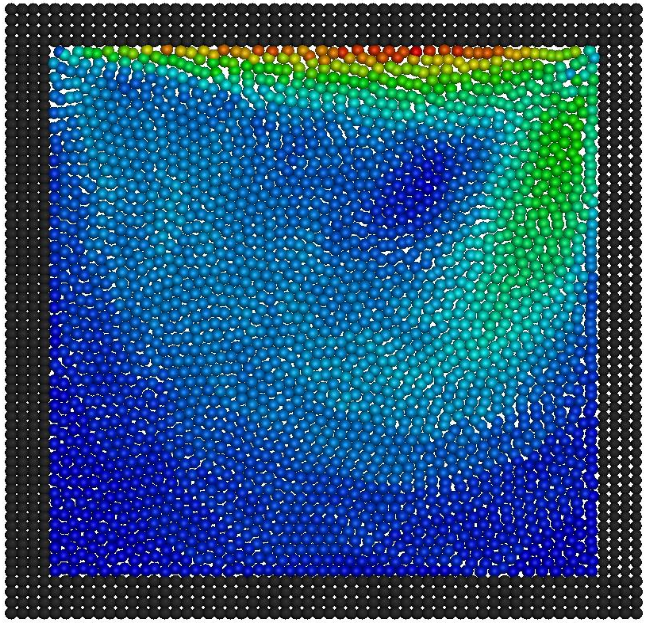
SPH – Fragmented Box

- At each MPP redecomposition, a new input file gets created with additional SPH particles.
- Deactivated particles behind the car get removed from the new input file.
- Allows simulations that would contain over 500M particles if the whole domain were to be simulated at once.

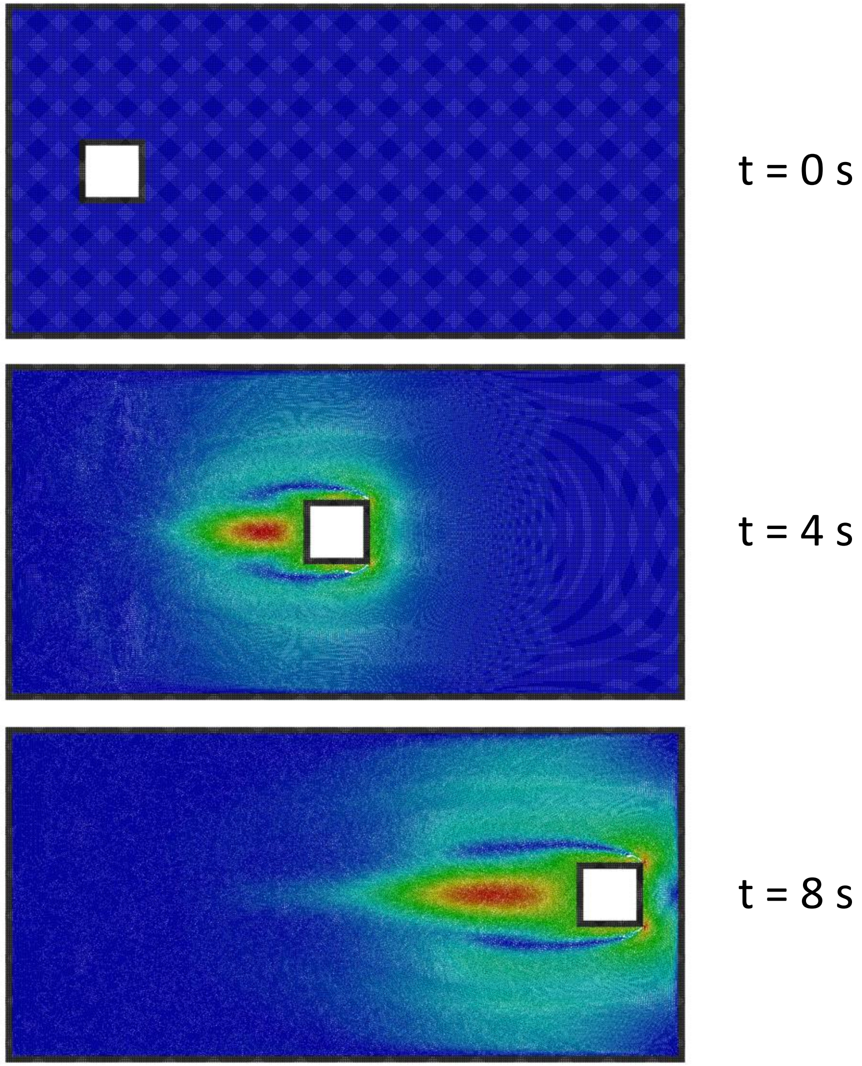


SPH – No-Slip Boundary

- Use SPH particles (dark gray) to impose velocities near boundaries.
- Fictitious velocity is assigned to selected SPH particles, and pressure is interpolated to prevent leakage.



Lid-Driven Cavity Flow



Moving Rigid Square

EM

Ansys

Summary of new features in R14

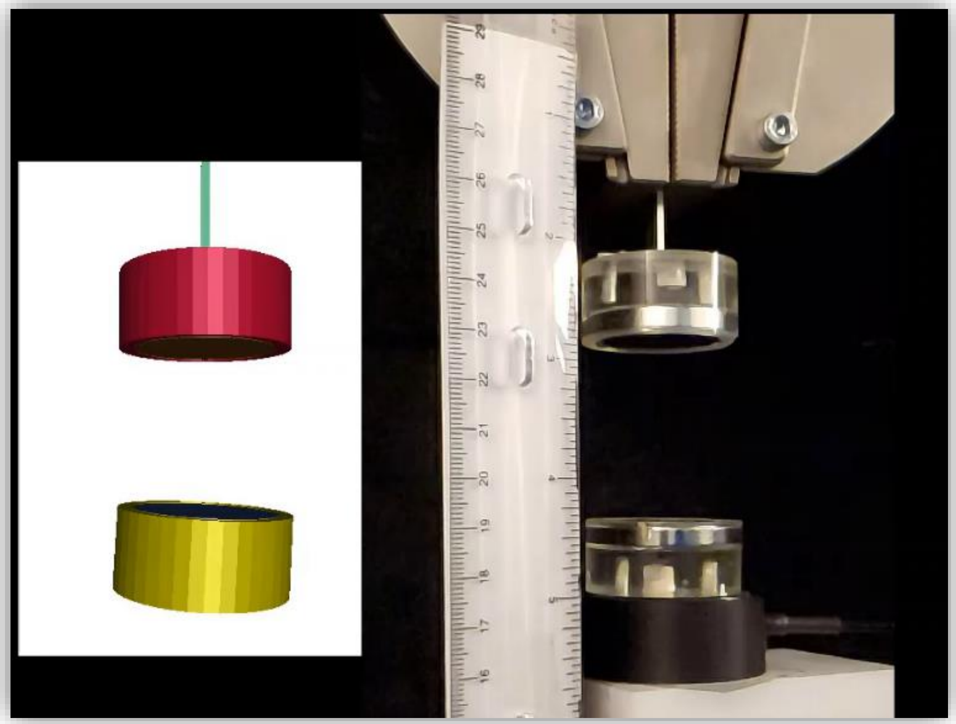
Magnets	• Introduction of magnetostatic capabilities : Possibility to define magnets as non conductor regions (sigma=0.) • Extension of magnet definition capabilities (for eg special definition of magnets in magnetic gear applications)
Inductive heating	• Added support of magnetostatic capabilities to include flux concentrators (ferrite) in simulations. • Added support of conducting shells with monolithic solver.
Actuators	• Introduction of Voltage driven Source Circuits for actuator and similar applications (loudspeakers). • Introduction of LLT Preconditioner for speed up in non linear (B-H) applications.
Battery modelling	• Erosion of elements can be combined with solid model for separator melting simulations
Electrophysiology	• Automatic build of Purkinje network coupled with monodomain. • Addition of computation of EKG from transmembrane (TM) potential. • Addition of several cells models and user defined cell models. • Computation of external potential function of TM potential in monodomain.

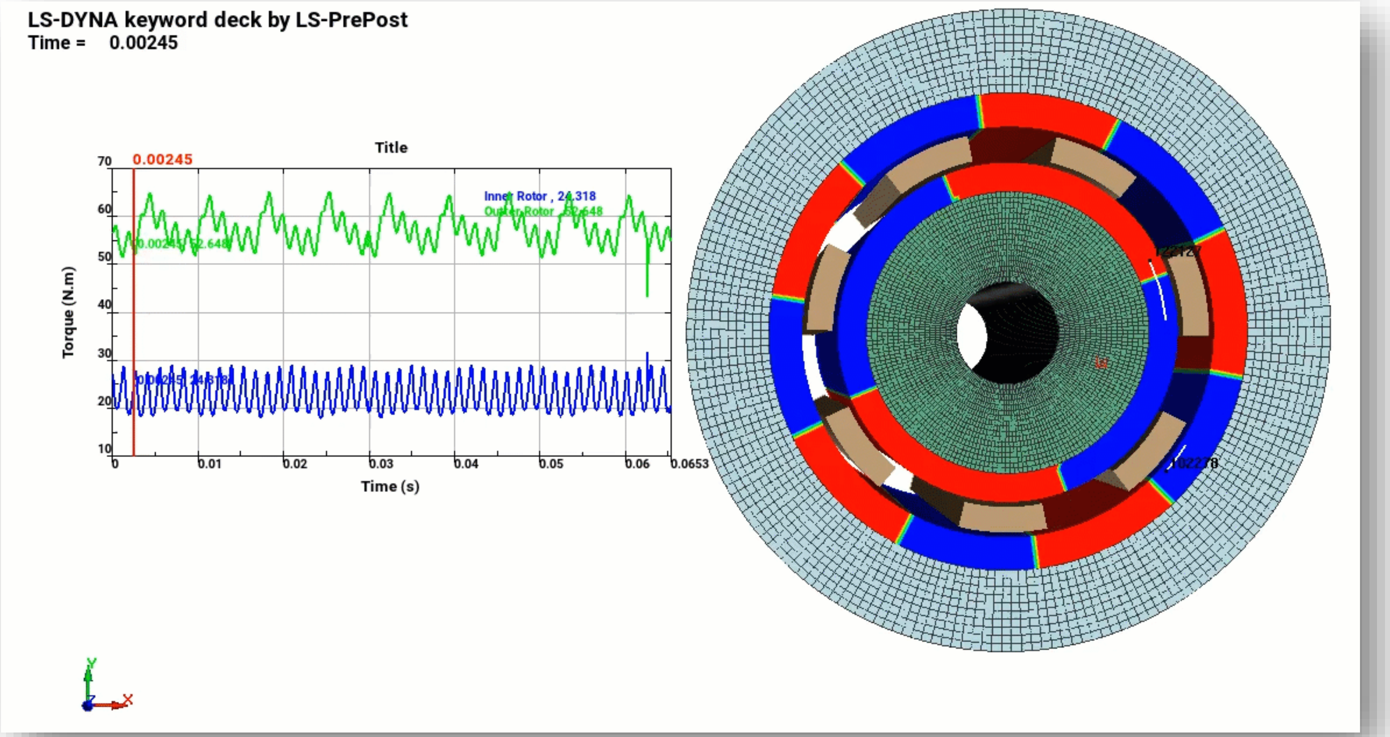
Ansys

Magnet simulations

- Introduction of magnetostatic capabilities : Possibility to define magnets as non conductor regions (sigma=0.) for speed up.

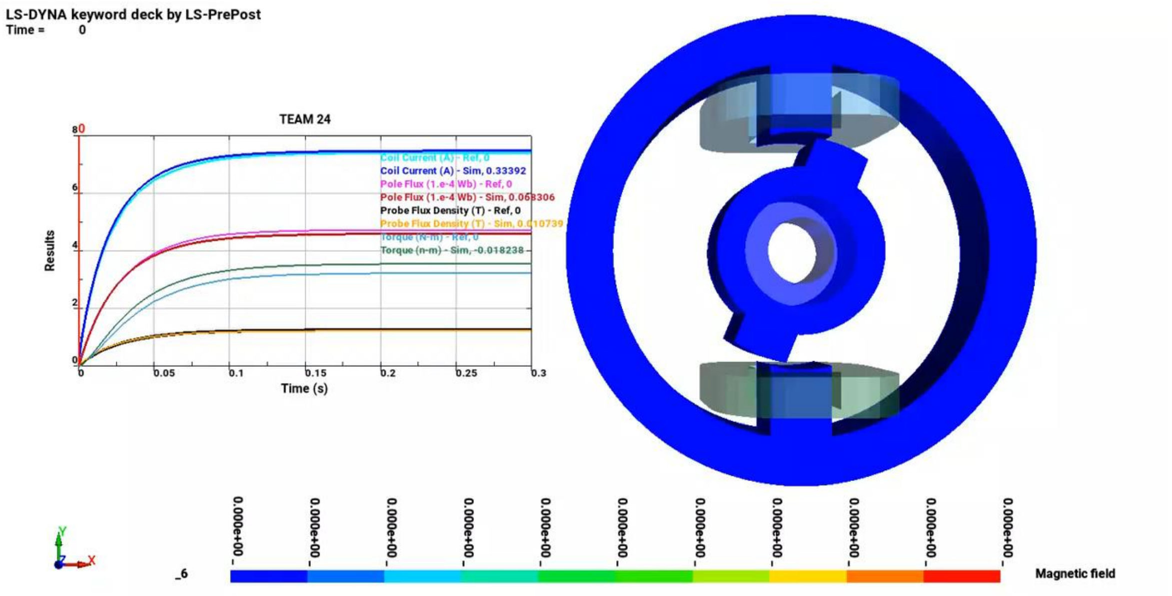
- Extension of magnet definition capabilities (for eg special definition of magnets in magnetic gear applications)

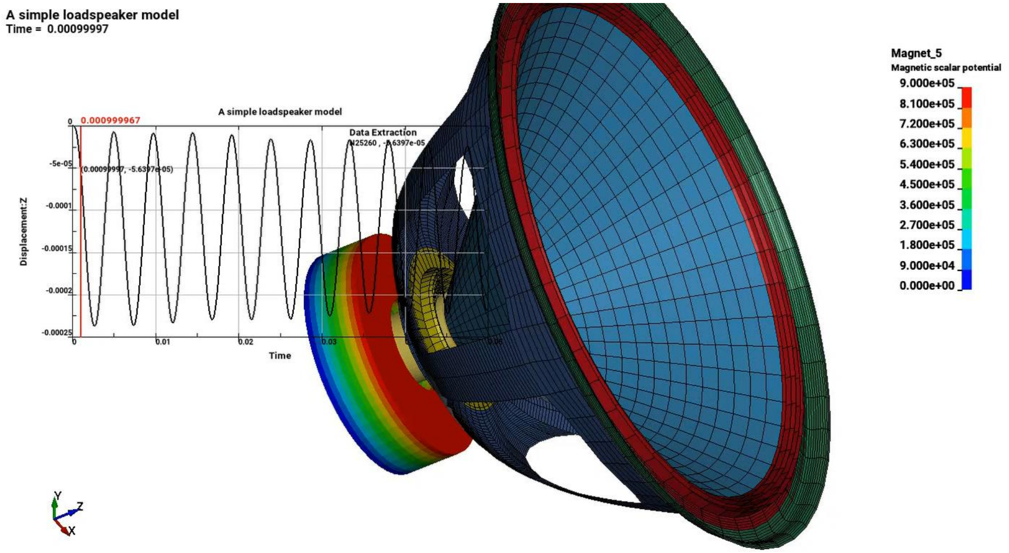




Actuators

- Introduction of Voltage driven Source Circuits for actuator and loudspeaker simulations.
- Introduction of BEM LLT Preconditioner for speed up in non linear (B-H) applications

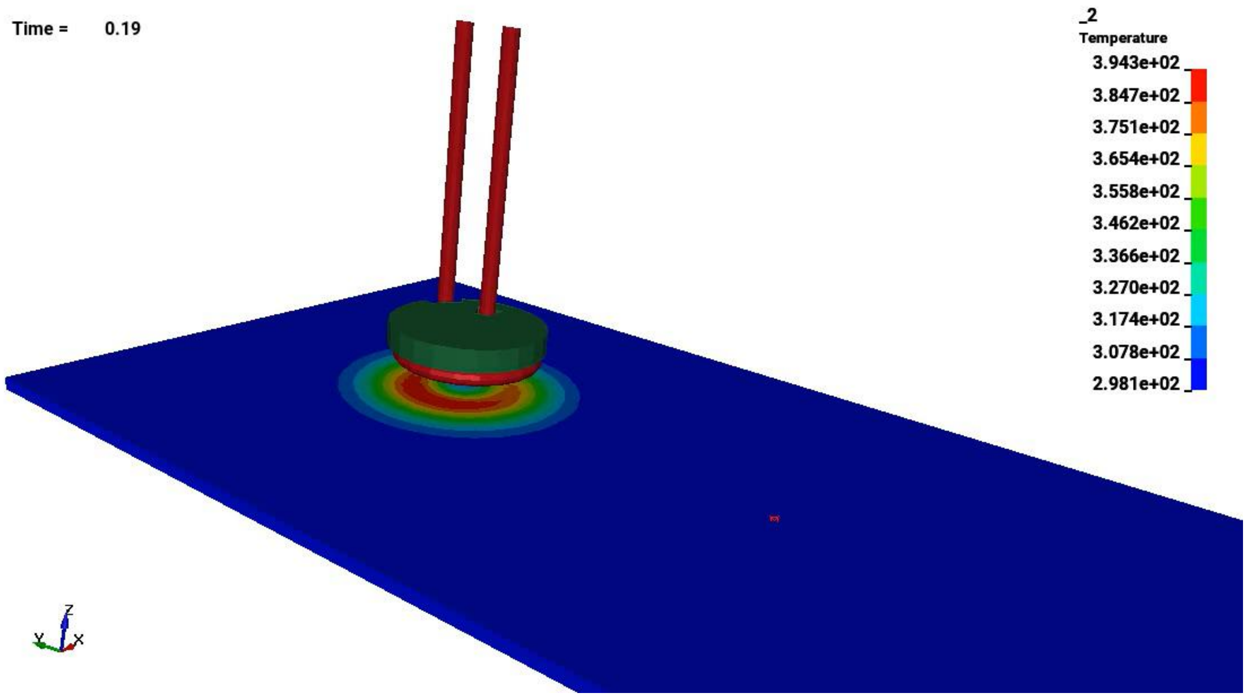




Inductive heating

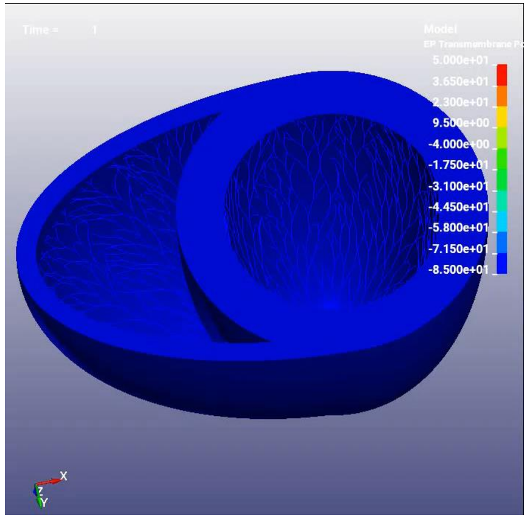
- Support of flux concentrators (zero conductivity regions) for more accurate inductive heating simulations thanks to a robust AMS preconditioner.

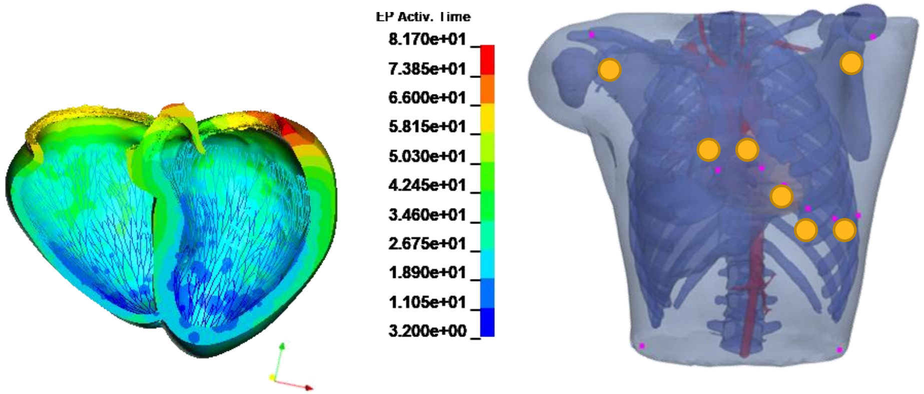
- Support of features such as temperature dependent permeability of conductive shells with monolithic solver.

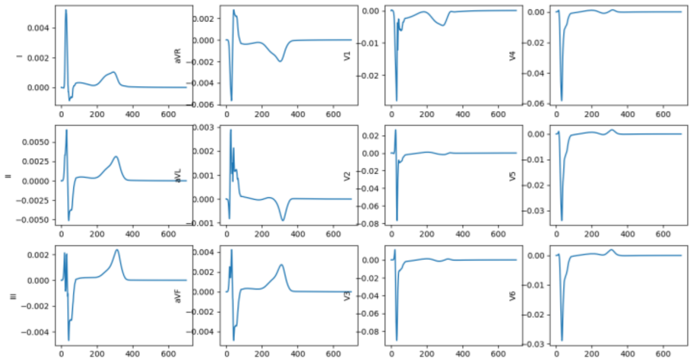


Introduction of Electrophysiology (EP) Models

- Automatic generation of Purkinje network which can be coupled to mono/bi domain models
- Ionic cell models: FitzHugh-Nagumo, Fenton-Karma, ten-Tussher, TO-ORD, user defined
- Computation of EKG from trans-membrane potential
- Computation of external potential function of TM potential in monodomain.







ECG simulation

Multiscale

Summary of new features in R14 from CMMG

Machine Learning-based Multiscale Analysis of Composites	• A data-driven multiscale material model • Main keyword *MAT_DMN_COMPOSITE_FRC • Unique feature: multiscale, anisotropic, nonlinear
RVE package for Multiscale Material Modeling	• Image RVE in RVE analysis, *RVE_ANALYSIS_FEM. • Partially periodic boundary conditions in RVE analysis.
Two-scale Co-simulation for Assembly Analysis	• New coupling interface in *INCLUDE_MULTISCALE for automatic generation of solid models • Setup Co-Sim analysis using standard command line • Enhancement on tie-contact based coupling • Improvement on numerical stability
SPG	• MPP thermal-mechanical coupled analysis • Bond-based damage model (IDAM=11, 13) • Particle to particle damping
ISPG	• MPP for large scale reflow soldering simulation (> 1000 solder balls). • Particle shifting technique for even distribution of ISPG nodes • Smoothed fluid-to-solid coupling technique
Peridynamic	• Apply initial strain and displacement fields
3D r-adaptivity	• Monotonic Remeshing • Metal Flow Line

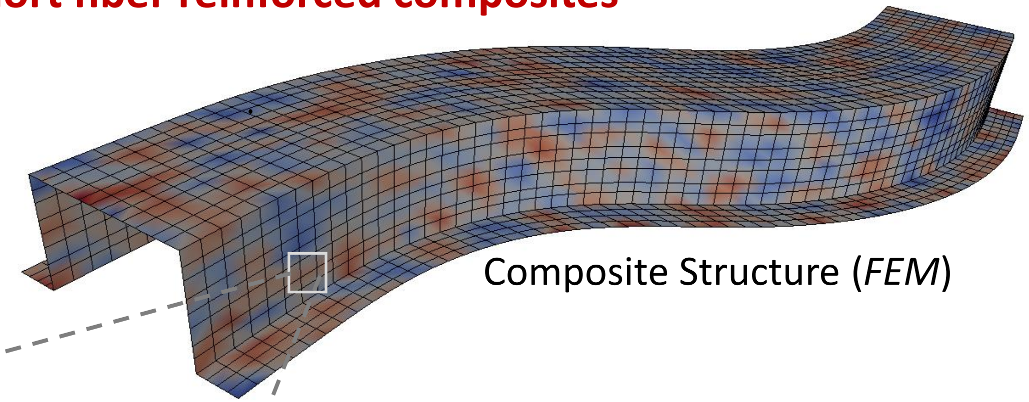
Ansys

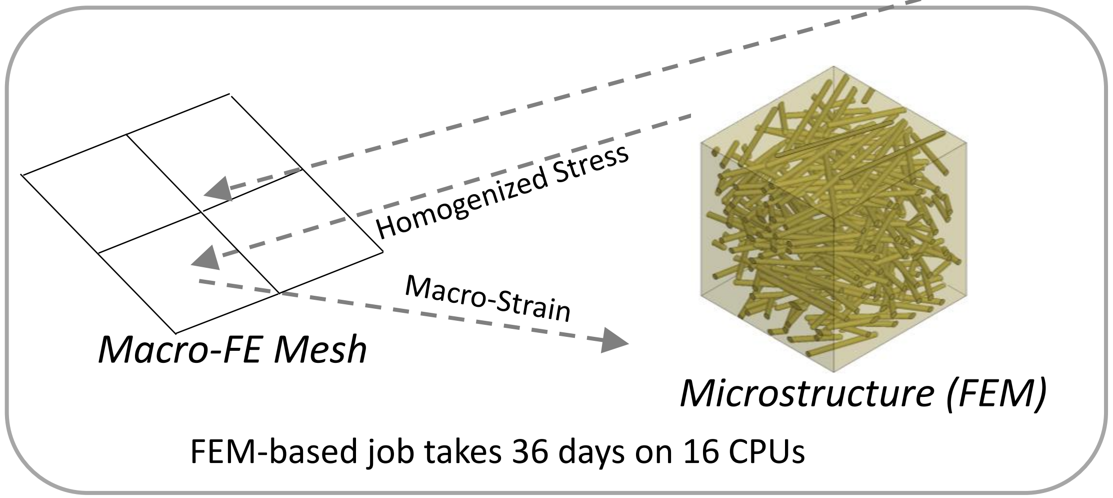
Machine Learning-based Multiscale Analysis of Composites

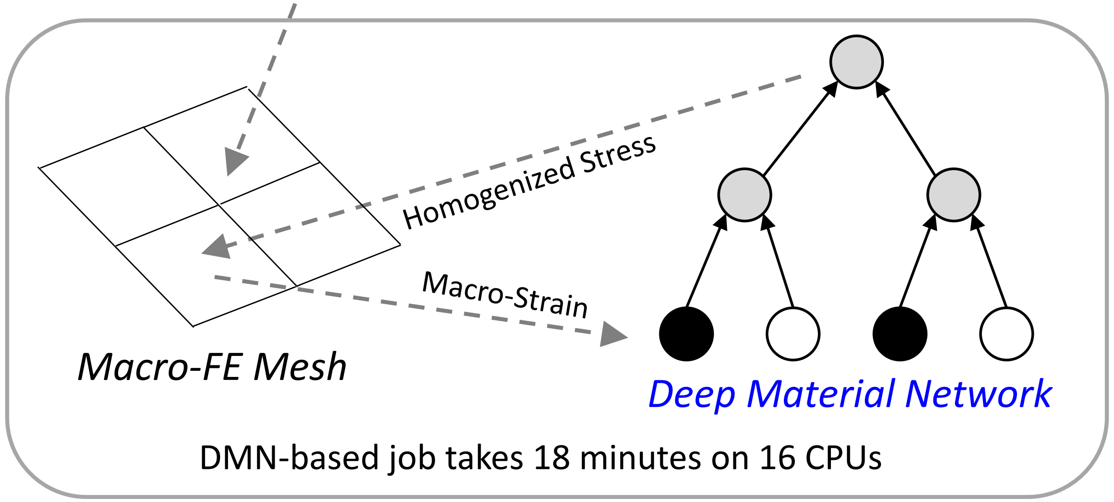
✓ **First time release in LS-DYNA**

✓ **A data-driven material model for multiscale analysis of short fiber reinforced composites**

- **Multiscale** – *predicts macroscale composite responses based on heterogeneous microstructures*
- **Anisotropic** – *captures effects of fiber orientations, volume fractions, aspect ratios*
- **Nonlinear** – *models tension-compression asymmetric elastoplastic material responses*





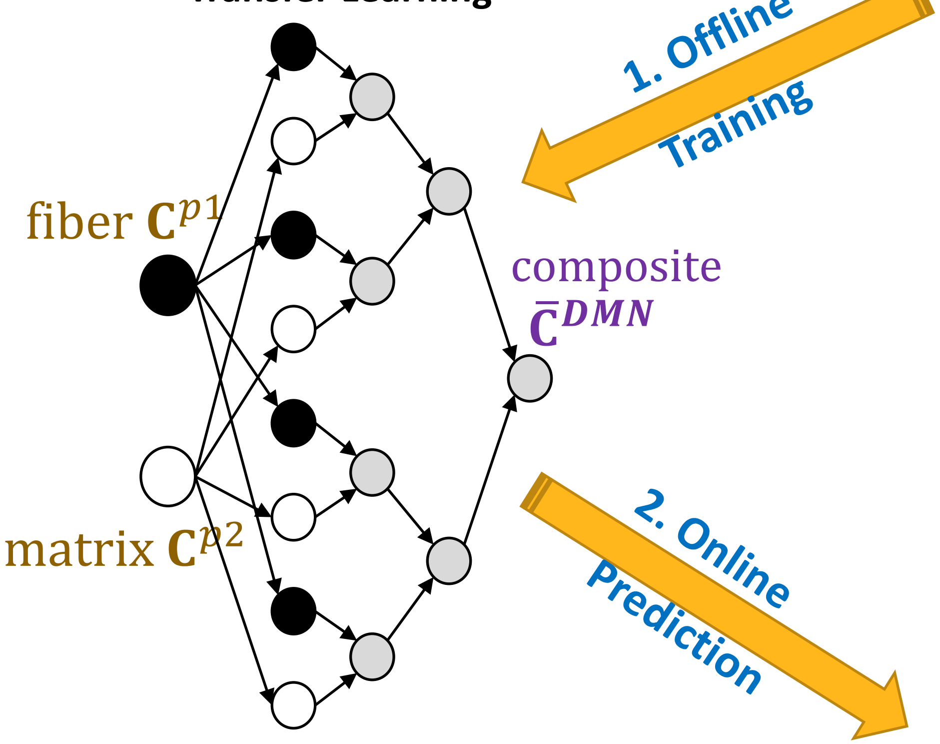


Reference: Liu, Z., Wu. C.T. (2019). Exploring the 3D architectures of deep material network in data-driven multiscale mechanics. Journal of the Mechanics and Physics of Solids, 127, 20-46.

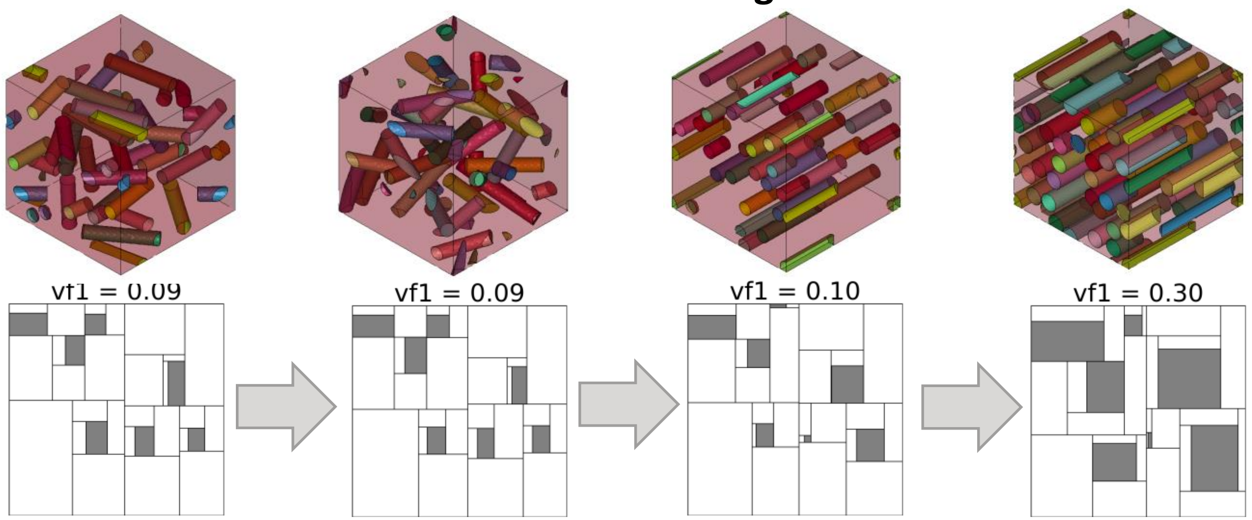
Machine Learning-based Multiscale Analysis of Composites (cont.)

✓ **Key Technology**

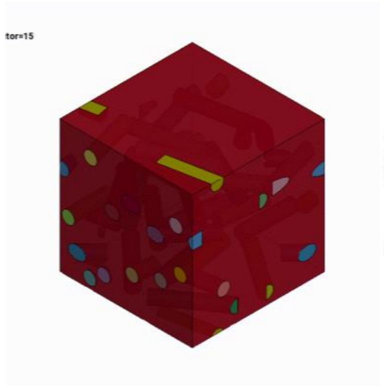
- **DMN (Deep Material Network)**
- **Transfer Learning**

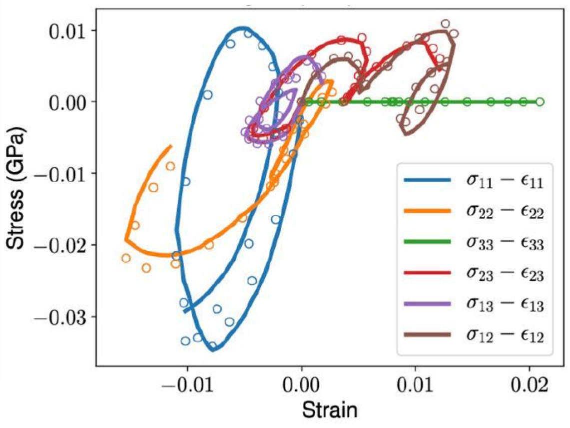


Transfer Learning



DMN vs FEM





Computational time

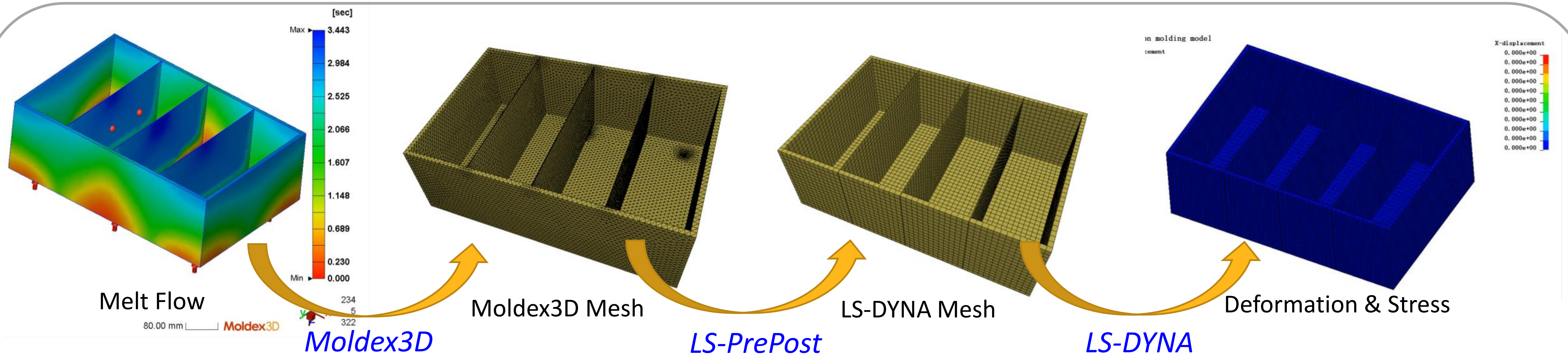
FEM	1100s on 8 CPUs
DMN	3s on 1 CPU

Reference: Wei, H., Wu, C. T., Lyu, D., Hu, W., Rouet, F. H., Zhang, K., Ho, P., Oura, H., Nishi, M.,Naito, T. Shen, L. (2021). Multiscale simulation of short fiber reinforced composites: from computational homogenization to mechanistic machine learning in LS DYNA. 13th European LS DYNA Conference, Ulm, Germany.

Machine Learning-based Multiscale Analysis of Composites (cont.)

✓ Customized and Seamless Simulation Workflow for Injection-Molded Short-Fiber-Reinforced Composites

- ***Moldex3D*** *– predicts injection-molding-induced microstructural distributions*
- ***LS-PrePost*** *– maps the heterogeneous microstructural data to LS-DYNA models (solid-to-solid, solid-to-shell)*
- ***LS-DYNA*** *– performs nonlinear explicit structural simulation using the machine learning-based data-driven material model*
- Collaboration with Honda, Moldex3D and JSOL for experimental validation



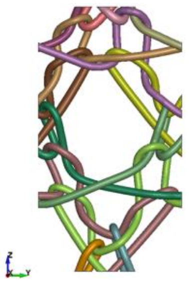
- *molding design (gate location, ...)*
- *material selection (glass fiber, carbon fiber, PP, PC, PA6, ...)*
- *structural optimization (partition shape and thickness, ...)*

Reference: Wei, H., Wu, C. T., Lyu, D., Hu, W., Rouet, F. H., Zhang, K., Ho, P., Oura, H., Nishi, M.,Naito, T. Shen, L. (2021). Multiscale simulation of short fiber reinforced composites: from computational homogenization to mechanistic machine learning in LS DYNA. 13th European LS DYNA Conference, Germany.

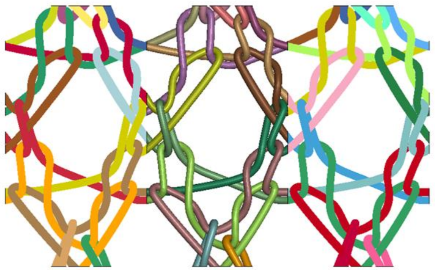
RVE Package for Multiscale Material Modeling

✓ **New feature to model textile material in RVE analysis (*RVE_ANALYSIS_FEM)**

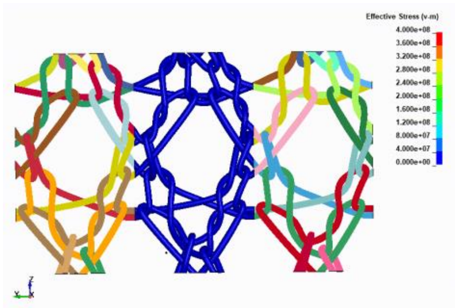
- Automatically creates image RVE in RVE solver to impose the periodicity of contact used in health care industry, e.g. ***Medtronic***
- The motion of the image RVE follows that of the true RVE with an offset to preserve structural continuity, which is achieved by enhancing the current capability of *CONSTRAINED_NODE_INTERPOLATION
- New flag name *IMAGE* in Card 1
- This feature enables users to perform RVE analysis and predict homogenized property of textile RVEs
  - ➢ Partially periodic boundary conditions in RVE analysis
  - ➢ Support the feature to define the periodic boundary conditions in only one or two directions of textile RVE
  - ➢ Flag name *BC* in Card 1
  - ➢ This feature enables users to impose partially periodic boundary conditions to textile RVEs



RVE model



Two image RVEs and one true RVE



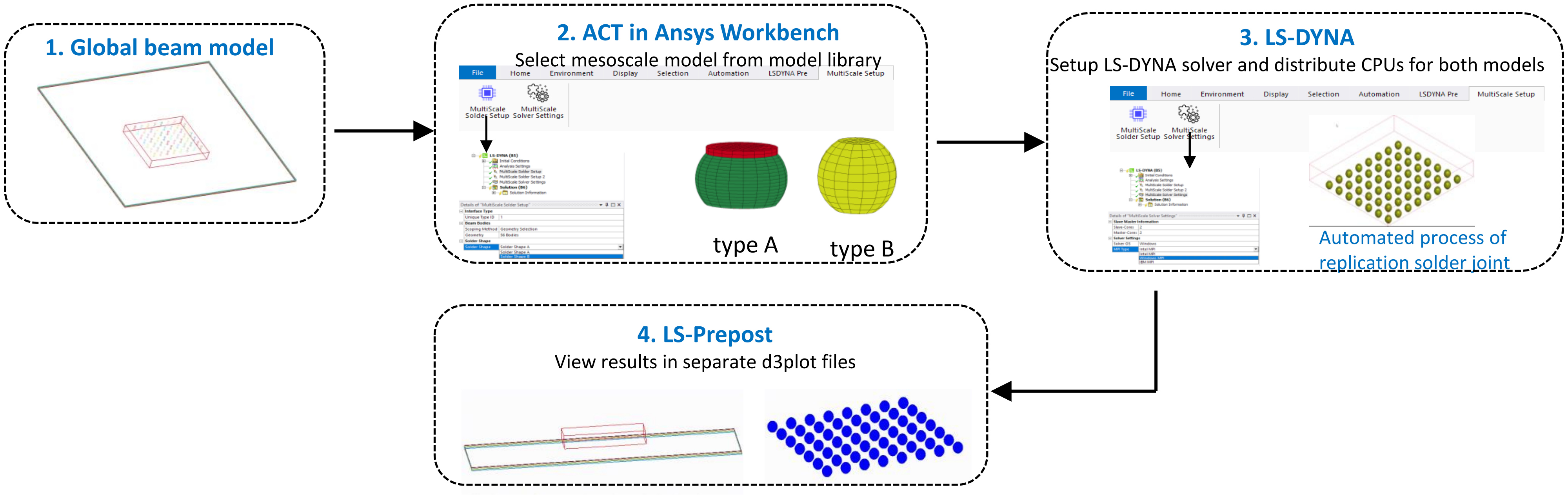
Simulation movie (Von Mises stress)

Courtesy of Medtronic

Two-scale Co-simulation

✓ **New coupling interface (*INCLUDE_MULTISCALE) for automatic generation of solder ball models**

- The new coupling interface can automatically generate/replicate meso-scale solid models from macro-scale beam elements. (New flag name *CTYPE* in Card 1)
- This feature replaces the previous two-scale **one-way** co-simulation and allows users to perform **two-scale two-way co-simulation** (*INCLUDE_COSIM) using the global beam model to obtain high fidelity results effectively.



Ansys

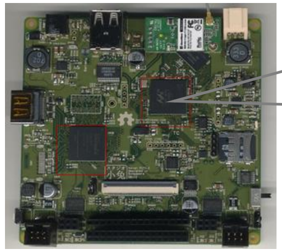
Two-scale Co-simulation (cont.)

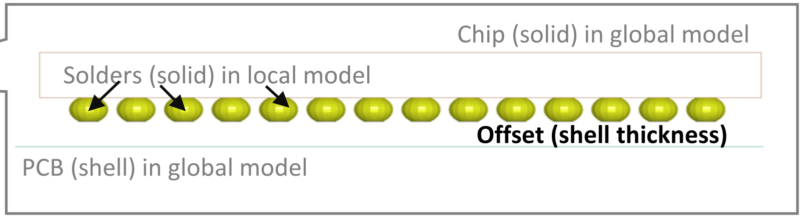
✓ Highlights in new version

- Easy setup co-simulation analysis using standard command line
  - New command line flag *ncsp*
    - ➢ To specify number of MPI processes for local model

      mpirun -np 96 mppdyna i=input.key *ncsp=32*
  - This enhancement allows users to run two-scale co-sim job in the very similar way of running single LS-DYNA MPP job

- Enhancement on tie-contact based coupling
  - OFFSET can be defined to take the global shell thickness into account
  - This enhancement enables more accurate interface coupling when shells are used in global model

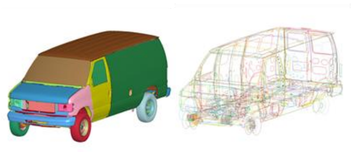




- Improvement on numerical stability
  - Improve the numerical stability of tie-contact coupling by redistributing interface nodal mass from local to global

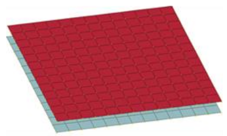
- Can be applied to other assembly problems

input.key

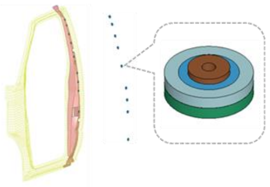


- *CONTROL_TERMINATION
- *DATABASE_BINARY_D3PLOT
- *CONTROL_TIMESTEP
- …
- ➢ *INCLUDE_COSIM

  mcosim.key
- *SET_SEGMENT

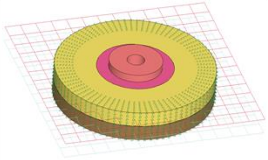


cs_input.key



- *PART
- *SECTION
- *MAT
- *CONTACT
- ➢ *INCLUDE_COSIM

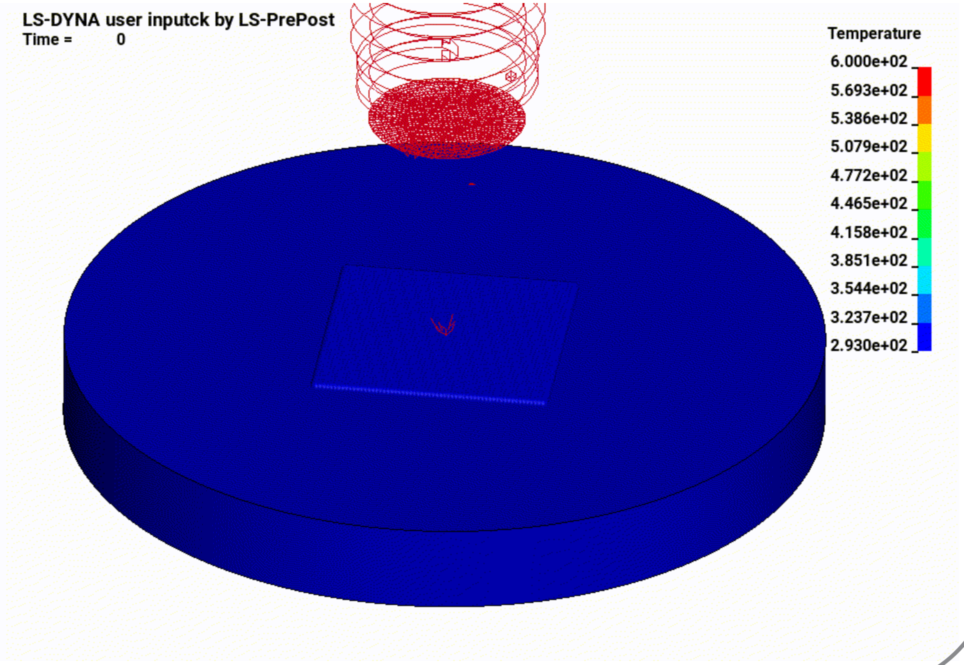
  scosim.key
- *SET_NODE



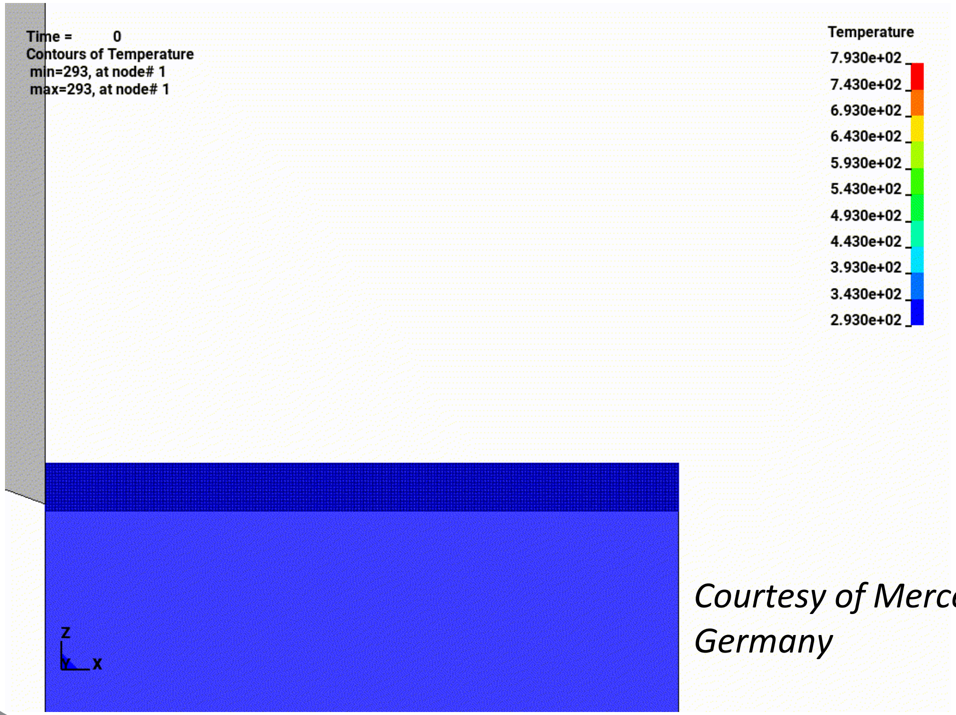
SPG: Thermal-mechanical Coupling Analysis (SMP and MPP)

✓ **Thermal effects in SPG parts can now be effectively modeled in metal fabrication simulations**

- Temperature dependent material properties
- Thermal expansion, thermal conductivity, heat generation due to friction and plastic material work
- Collaboration with major OEMs for experimental validation



<u>Temperature distribution</u>

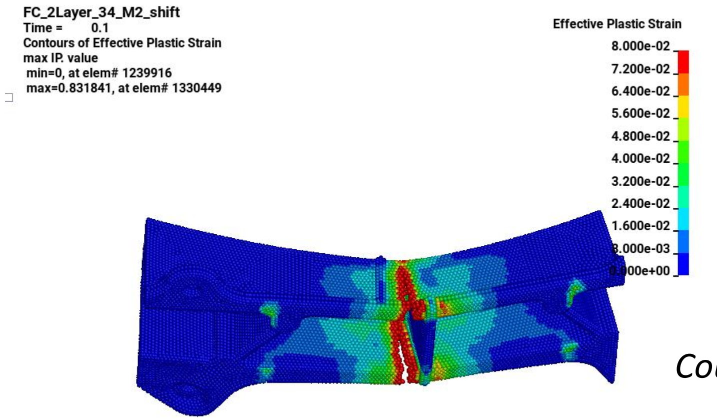


Courtesy of Mercedes, Germany

SPG: bond-based damage Model (IDAM=11,13)

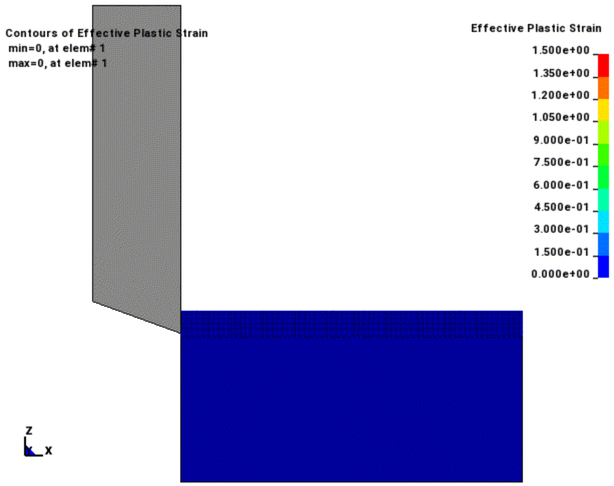
✓ **SPG provides two new mechanisms for material failure analysis**

- With bond-based pre-damage model (IDAM=11), MC-SPG can predict the fast and sharp crack propagation in the brittle material, which agrees well with that observed in the experimental test
- SPG bond-based pre-damage failure model (IDAM=13) can capture the shear band in metal cutting analysis by considering both the tension failure and tension-compression damage



Courtesy of Toyota

Final deformation for 2mm coarse mesh size with bond-based Pre-damage(IDAM=11) α_B =3.0 in MC-SPG

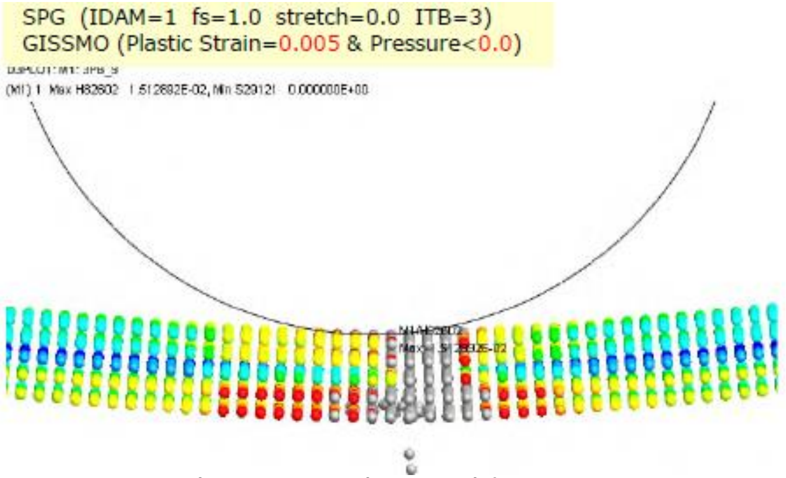


Cutting simulation of Ti-Al-4V with MC-SPG and IDAM=13

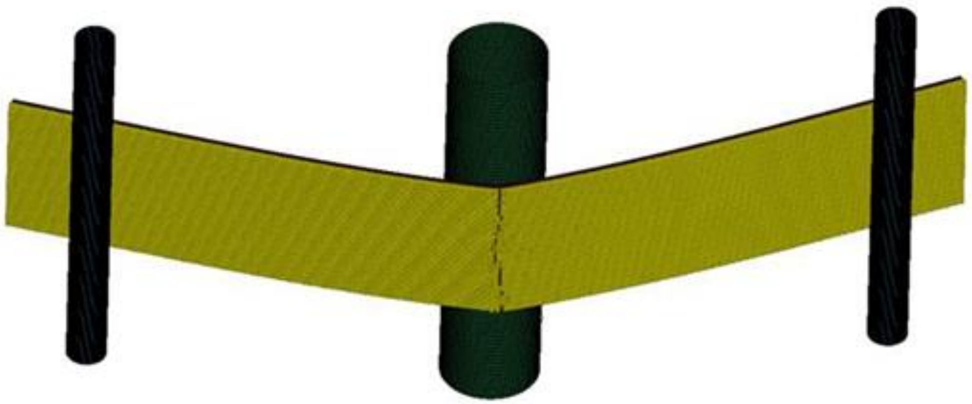
SPG: Particle-to-Particle Damping for MC-SPG

✓ **A particle damping algorithm was developed to replace FEM damping algorithm for SPG modeling**

- Particle-to-particle damping was developed to stabilize the MC-SPG solution in severe bond-breakage of particles without non-physical flying particles
- It preserves the desired conservation of linear momentum and angular momentum properties



Flying particles in old version

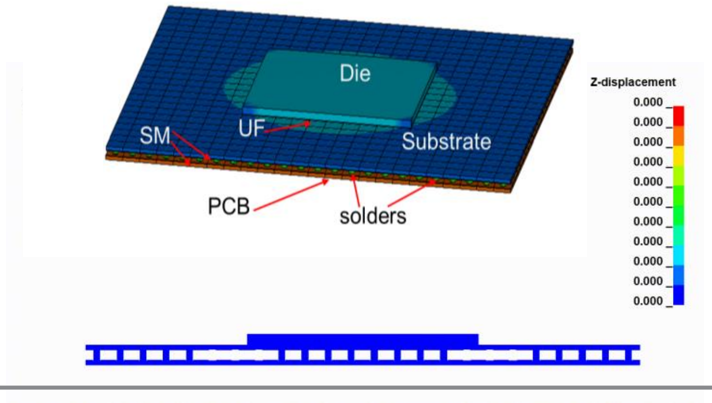


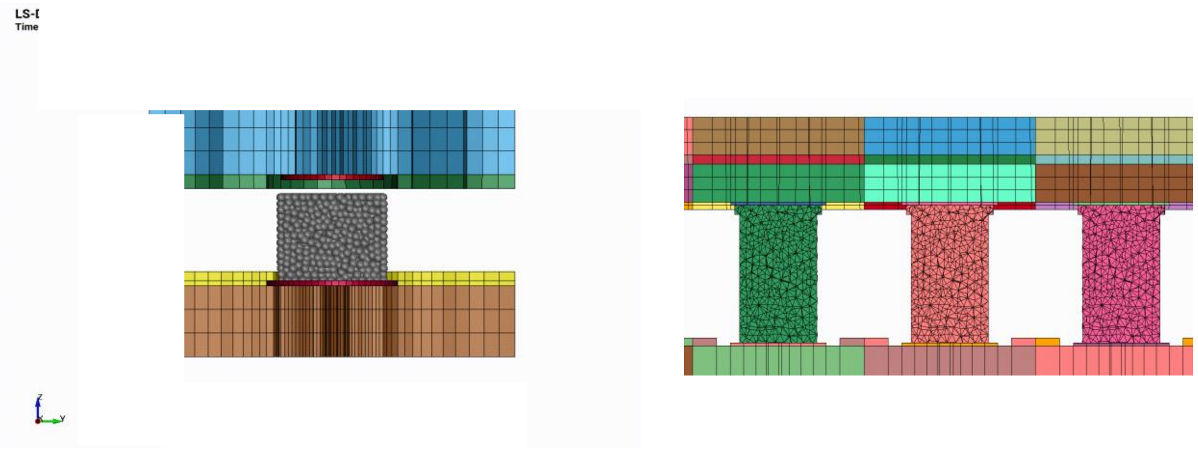
Stable result with new damping algorithm

ISPG: Fully implicit ISPG method for large-scale fluid modeling

- ✓ ISPG was enhanced with MPP for large scale reflow soldering simulation (> 1000 solders as requested by TSMC and Intel)
- ✓ Particle shifting technique was developed for ISPG to guarantee the evenly distributed ISPG particles in severe free-surface viscous incompressible fluid flow
- ✓ Smoothed fluid-to-solid coupling technique was developed for ISPG for smooth transition of the fluid particles between sharp edges or corners of the structure

Reflow soldering simulation using fully implicit ISPG + implicit structure and thermal solver





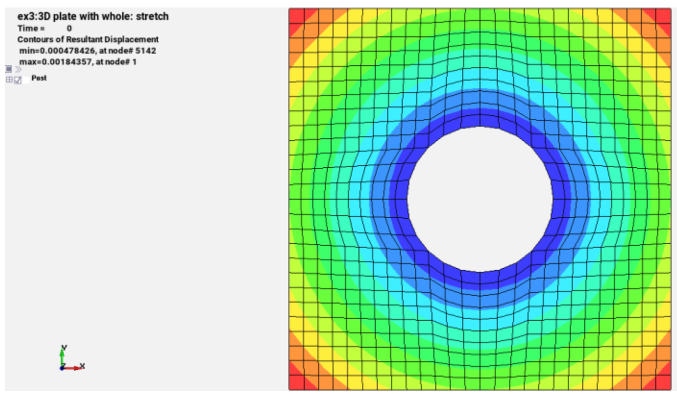
Gravitational Reflowing of Solder Balls with Masks

Peridynamic: Applying initial strains and displacements

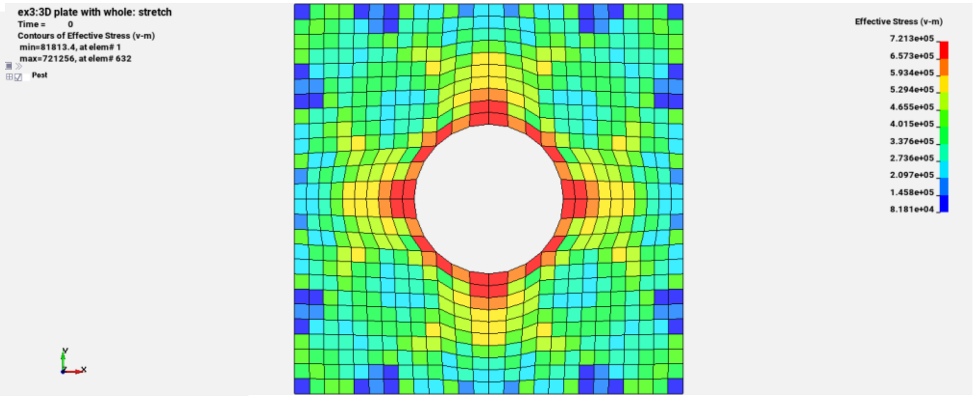
✓ **Initial strain and displacement fields can now be applied to Peridynamics for dynamics crack propagation analysis**

- Initial strain and displacement fields strongly affect the dynamic crack propagation results
- New version introduces a preload approach with dynamic relaxation process to impose the initial fields

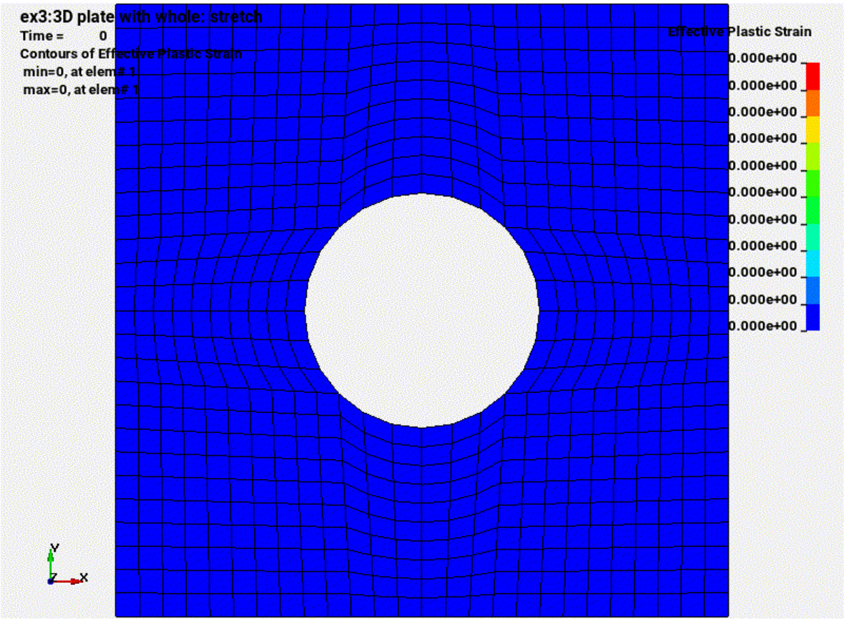
```
*MAT_ELASTIC_PERI
$#    mid        ro          E        G      GS     PSX     PSY     PSZ
        3  8.000E+3  190E+09   8.5E+5          0.0001  0.0001  0.0001
```



Initial displacement



Initial stress

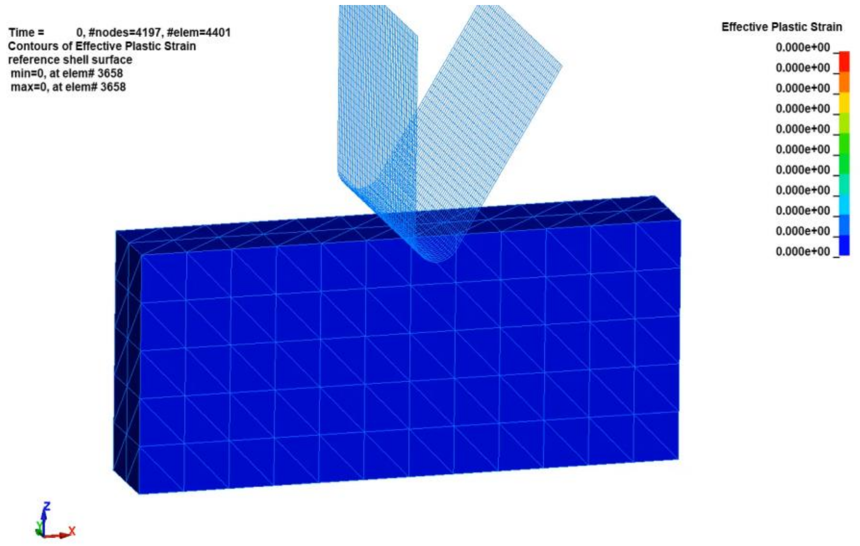


Normal simulation.

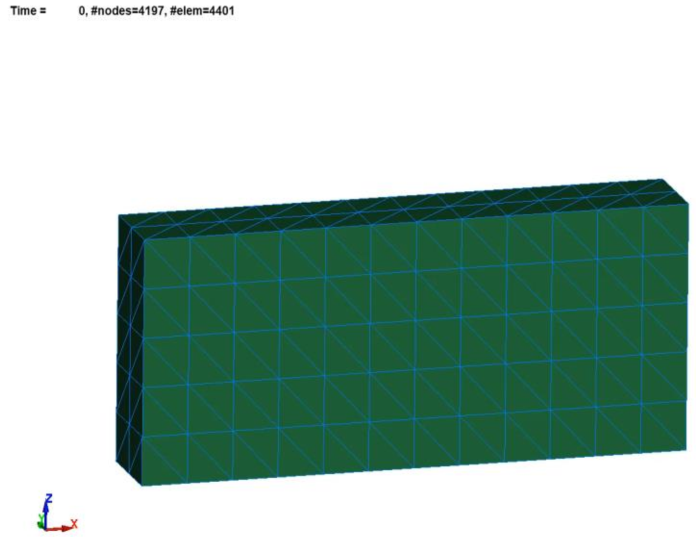
3D r-Adaptive EFG/FEM: Monotonic Remeshing

✓ **Monotonic remeshing algorithm was developed to improve accuracy in forging analysis**

- New feature to maintain the refined mesh during subsequent adaptive steps
- This feature enables the adaptive remesher to keep the stress/strain data in the refined area and improve the solution accuracy and convergence performance



Forging Simulation Movie (Effective Plastic Strain)

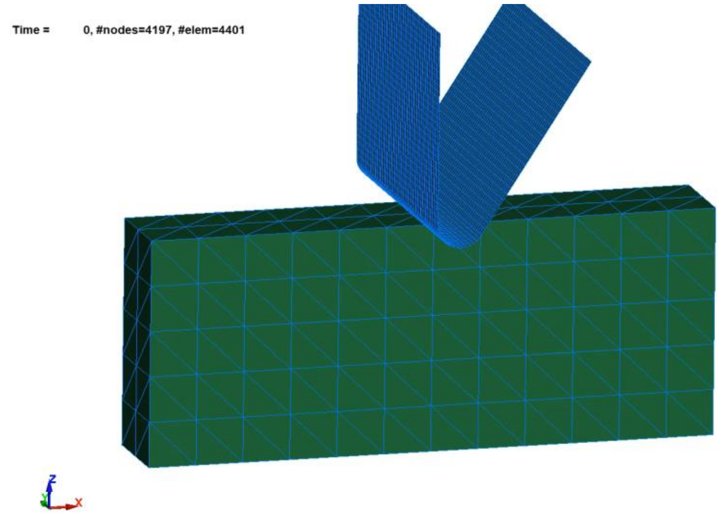


Mesh Refinement Movie

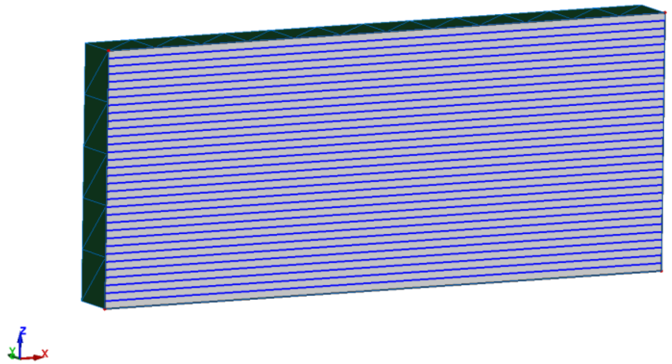
3D r-Adaptivity: Metal Flow Line

✓ **New feature added to display the material flow line of metal forging results in LS-PrePost**

- The feature was developed with LST LS-PrePost team
- Available in LS-PrePost 4.9 or later version
  - ➢ Display metal flow line with user selected lines
  - ➢ Display metal flow line on a cut plane



Forging Simulation Movie



Metal Flow Line Movie (on a Plane)

Ansys

Incompressible CFD

Summary of new features in R14

Solvers	• New Block Low-Rank factorization for both continuity and momentum equations. • Scales very well in parallel.
Gap Closure	• Allows complete closure using a user defined threshold, that prevents flow through a gap.
Residence Time	• A new metric for flow stagnation. Useful in healthcare hemodynamics applications to predict regions of thrombosis.
Species Transport	• Provides a tool to track drug delivery using intradermal injections.
Boundary Layer Meshing	• Improved quality close to surfaces with poor quality. • Adaptive velocity of layer inflation.
Steady State	• Added a new SIMPLEC steady state algorithm adapted for FEM.
DEM coupling	• New FSI interface representation using DEM particles on beams and shells. • New coupling formulation that uses the fluid pressure gradient instead of the velocity drag. • Added buoyancy effects.

Velocity and pressure solver preconditioners

- Preconditioners (PRECOND field of *ICFD_SOLVER_{MOM,TOL}_PRE):
  - Diagonal preconditioner: default for velocity solve;
  - Zero-fill incomplete factorization;
  - Threshold-based incomplete factorization: default for pressure solve, effective for most problems;
  - Block Low-Rank factorization: new in R14;
    - Expensive but very robust, good for tough problems like external aero;
    - Number of iterations doesn't change with number of MPI ranks.
    - Example, DrivAer model, 3.5M dofs, 10 time steps, pressure solve:

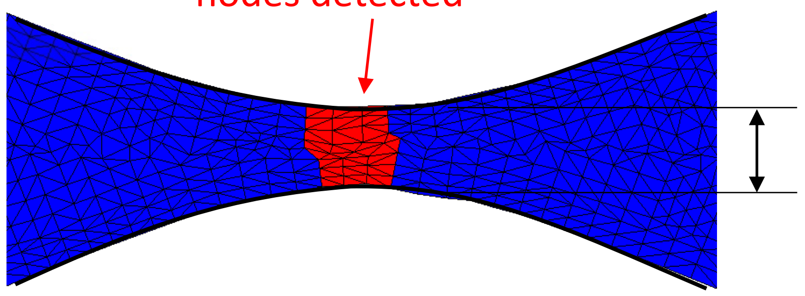
Precond	Setup time(s)	Solve time(s)	Total time(s)	Iterations per solve
Default	13.7	210.0	223.7	351
MUMPS	30.0	26.5	56.5	3

Ansys

Gap Closure

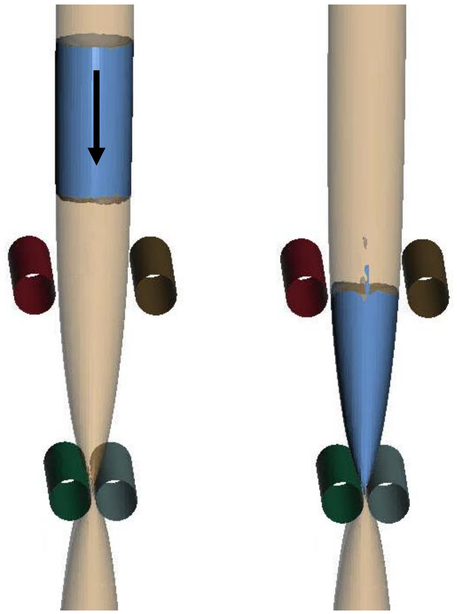
- If the distance between boundaries is less than a user-defined threshold value, the flow is blocked at this region.
- Use new keyword *ICFD_CONTROL_GAP to specify the threshold value and the surface IDs.

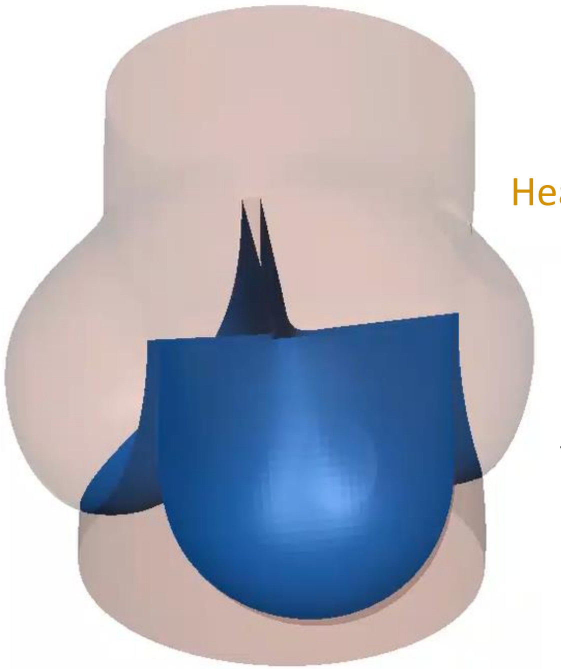
Gap elements and nodes detected



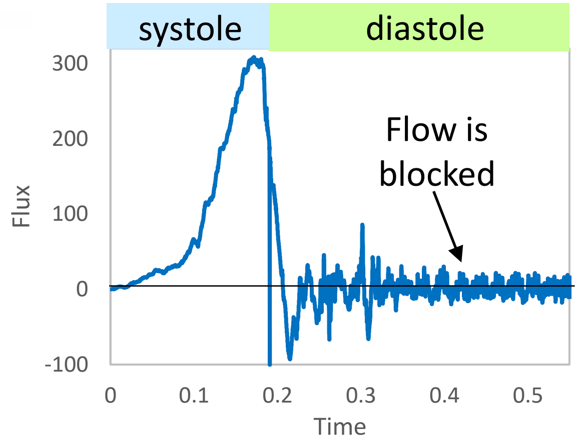
Applications

Filling Simulation





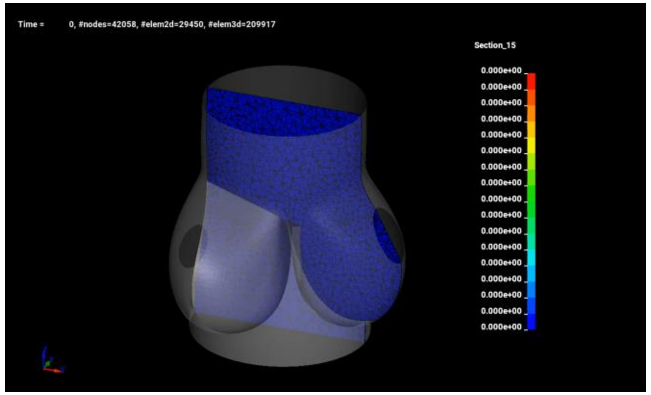
Heart valves

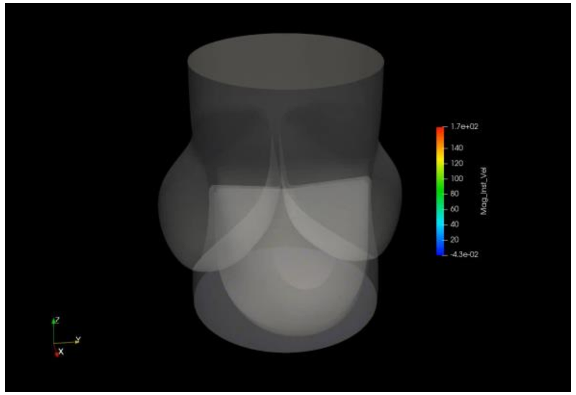


New development supporting healthcare in ICFD/FSI and Species Transport Phenomena.

Eulerian Residence Time Solver for Thrombogenesis and Stagnation Region studies

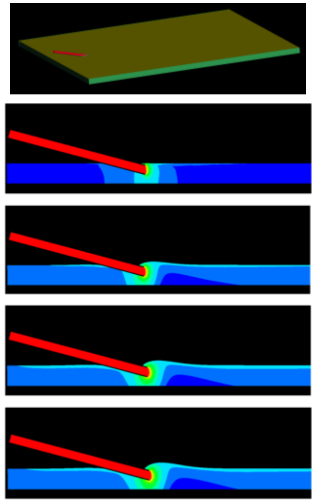
- Industry and academia look for residence time as a metric of where there is blood stagnation regions which could point to possible thrombogenesis risk.
- Particles and Species residency evolution.

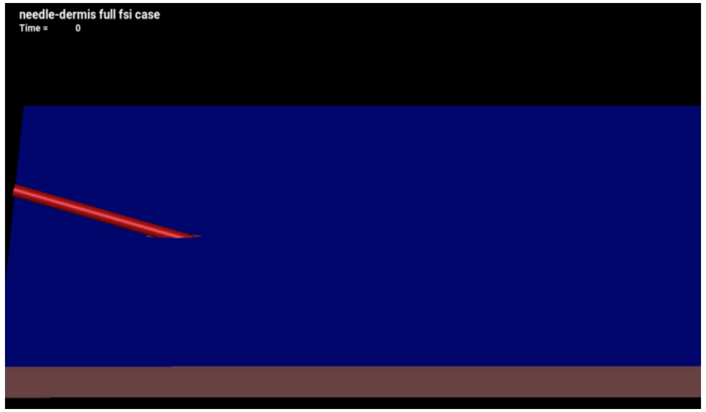




Intradermal injections and drug delivery

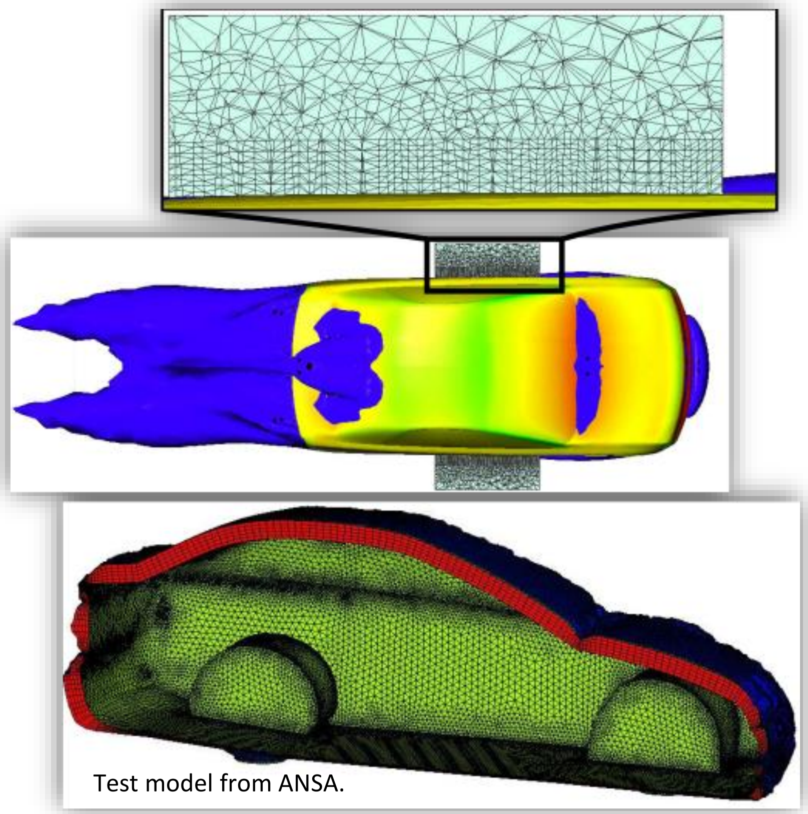
With the increase in vaccination campaigns during the COVID-19 outbreak, the med Industry started using numerical models for coupled physics to study the drug delivery through the human tissue as well as for the delivery on other devices like the one used by patients with other conditions like diabetes.





Improved Boundary Layer Meshing and Steady State Simulations

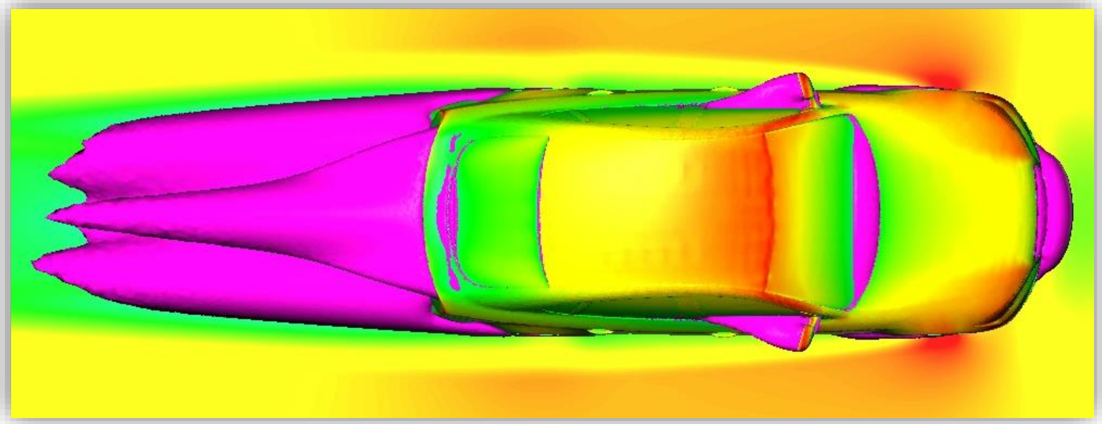
Boundary layer meshing is a key feature of a viscous flow solver. LS-DYNA automatic meshing tool generates an anisotropic boundary layer mesh at run time. In the current release the algorithm has been improved to provide better quality specially in cases with poor surface mesh resolution.



Test model from ANSA.

Added a **SIMPLEC steady formulation** adapted for FEM. The main changes include:

- Implicit relaxation for the momentum equation.
- Accounting for the implicit relaxation in the continuity equation.
- Relaxation for RANS solvers.



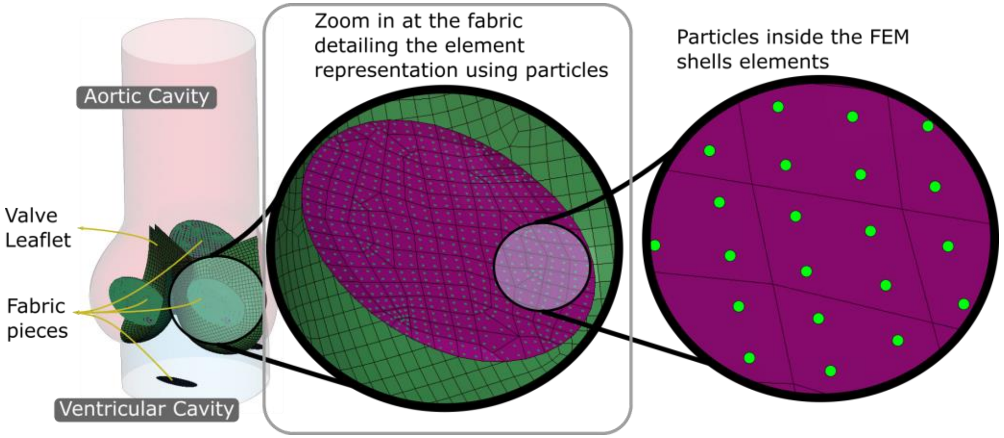
DrivAer model.

ICFD + DEM for Implicit FSI

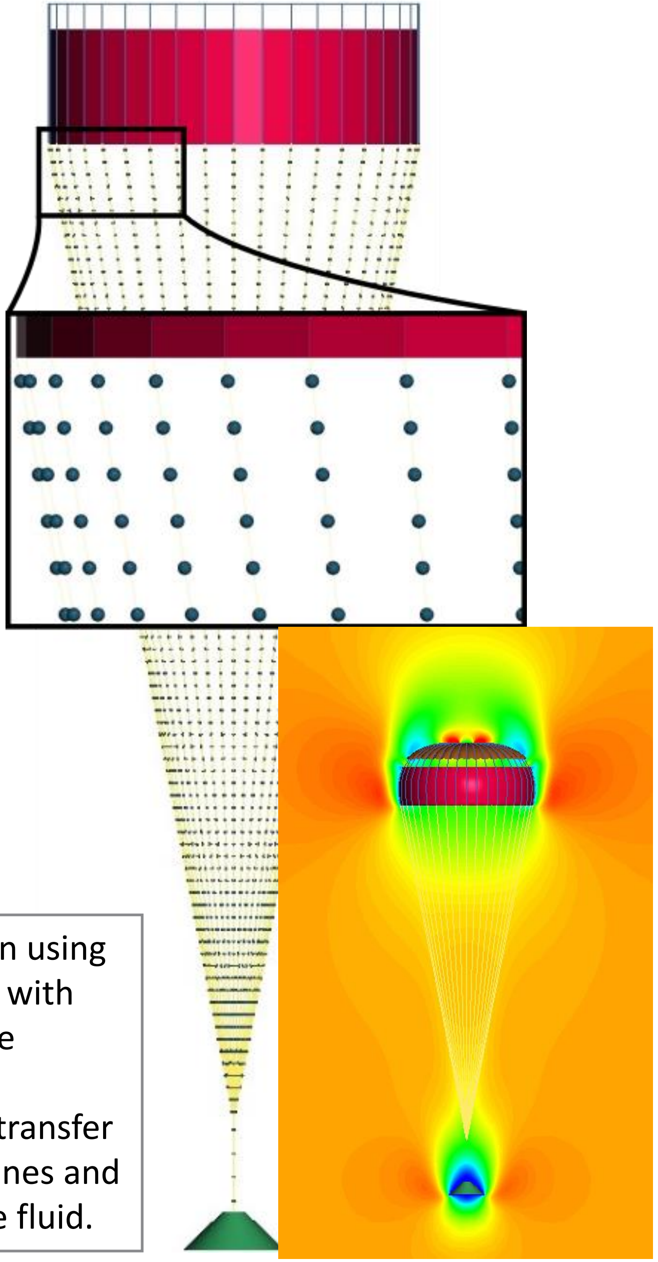
Patent pending

- DEM particles in shells and beams provide an alternative FSI interface in some highly complex models.
- The FSI simulations can have a mixture of both body fitted and DEM interfaces even in implicit mechanics.
- A new coupling formulation has been added that uses the fluid pressure force instead of the drag force computed from the velocity field.
- The particle density is now considered for problems that involve buoyancy.

Prosthetic heart valve solved using body fitted implicit FSI is interacting with foreign bodies represented by shells and DEM particles.



Parachute simulation using implicit FSI coupling with DEM particles on the beams representing suspension lines to transfer momentum to the lines and from the lines to the fluid.



Compressible CFD:
Dual-CESE

Ansys

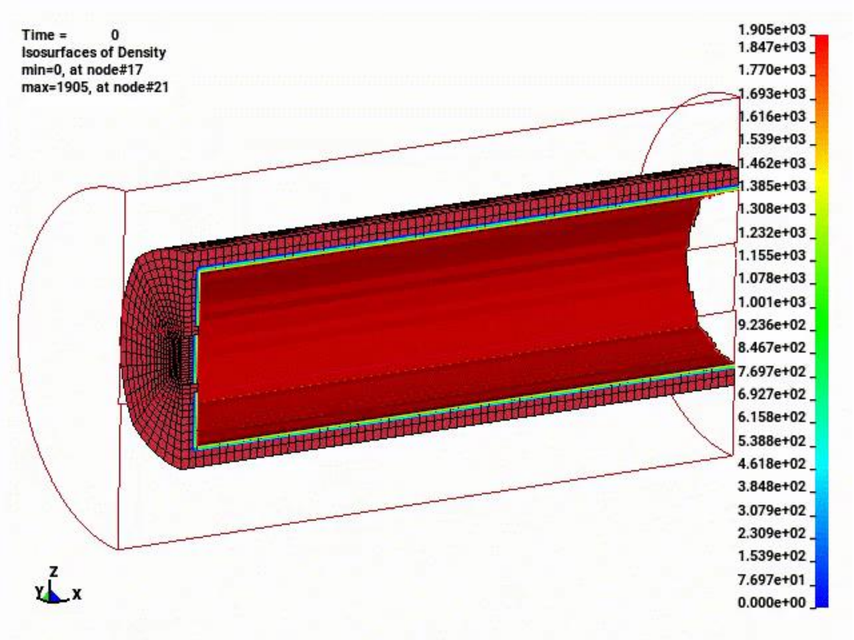
Summary of new features in R14 dual CESE solvers

Multiphase FSI capabilities have been added	• Available with:
	  • Hybrid multiphase solver
	  • Two-phase multiphase solver

Cavitation solver with FSI capabilities	• Models cavitation using Schmidt’s homogeneous equilibrium model (HEM)
2D axisymmetric solvers with FSI capabilities	• Added for each type of dual CESE solver.
Improved keywords with better organization	• This updated way to specify dual CESE problems will assist training new users.
‘binout’ style time history output	• While not yet supported in LSPP, users can use LSDA tools to access the binary data in the ‘binout’ files created with new *DUALCESE_DATABASE_HISTORY_... cards. • Allows solution data to be sampled on subsets of nodes, elements, and segment sets.

Ansys

FSI capabilities have been added to the hybrid multiphase flow solver



Density contours for a 3D rate-stick interacting with cylinder container

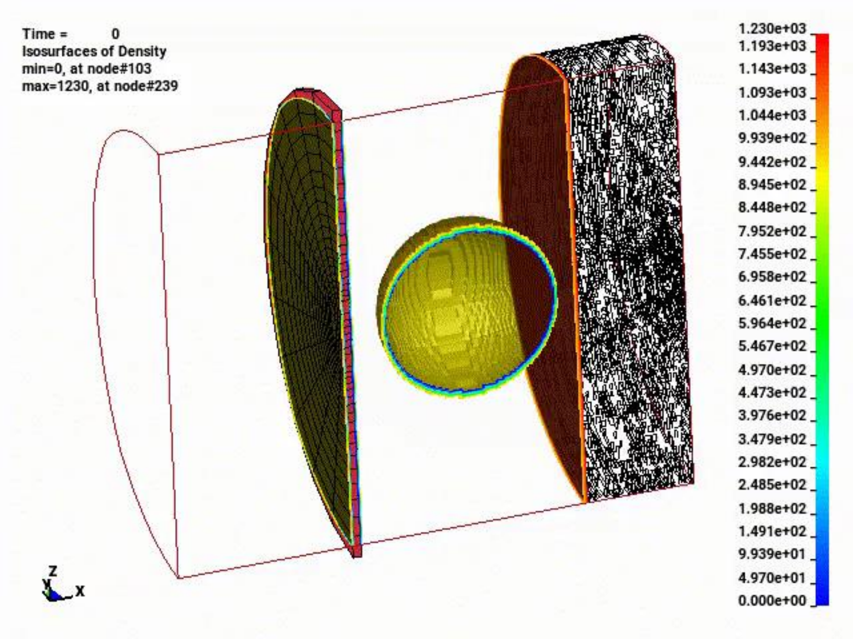
Features:

- This hybrid model can handle 3 materials, including the two of the mixture reactants (reactants & products)
- A one-step chemical reaction is assumed
- A general Mie-Grunisen type of EOS can be used for each material

Main applications:

- Simulation of **combustion** and transition to **detonation** of condensed-phase explosives
- **propagation** of detonations in compliantly-confined charges (rate-stick-type problem)
- **Sensitization** simulation of commercial explosives by means of collapsing micro-balloons
- shock-induced cavity **collapse** in liquid explosives

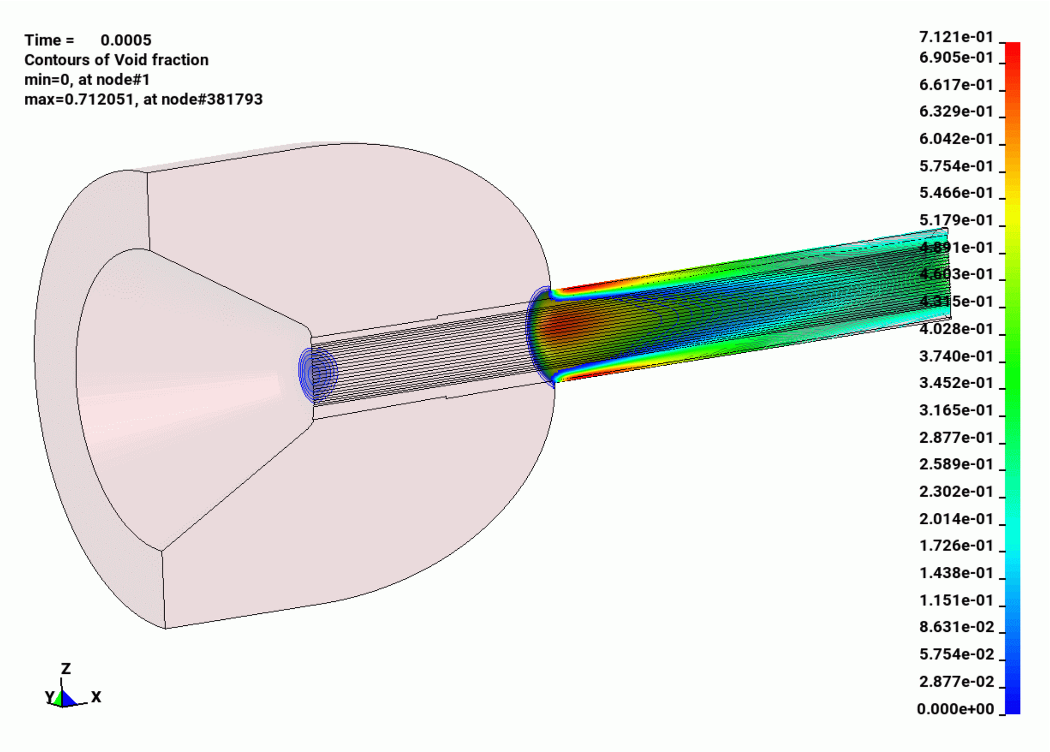
FSI capabilities have been added to the two-phase multiphase flow solver



Density contours in a 3D Shock-bubble-plate interaction problem

- This two-phase model is a reduced model of the hybrid multiphase model
- This model can handle two immiscible flow materials
- When additional structural parts are added, an FSI solver can be used

Cavitating flow and it's FSI solvers are ported to dual-CESE solver



Void fractions of cavitating flows in a 3D non-hydro nozzle

The cavitation model:

- Schmidt's homogeneous equilibrium model (HEM)

Applications:

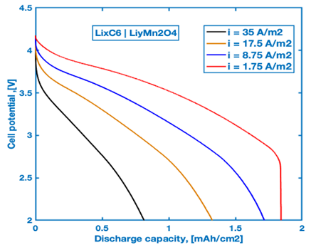
- suitable for high-speed flows in a small geometry, like diesel injection systems

Battery Plus

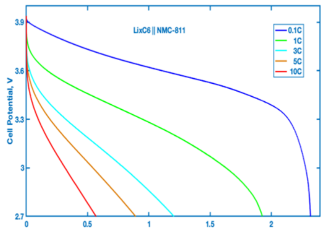
Summary of new features in R14 battery electrochemistry

Updated 3 different electrochemical LIB models	• 6-equation model (Newman) • 10-equation model (Thermal) • 14-equation model (Multiphysics)
New modeling capabilities	• New battery aging model. • Thermal mechanism for SEI formation and decomposition. • Battery swelling model. • Gas generation model based on Ethylene oxidation and Lithium hydration reaction mechanisms
Thermo-mechanical model coupling	• Available with each of the new/updated battery electrochemistry models to support battery abuse simulations.
User-requested feature	• The shutdown key is installed based on Minimum ignition energy criterion
'binout' style time history output	• While not yet supported in LSPP, users can use LSDA tools to access the binary data in the 'binout' files created with new *BATTERY_DATABASE_HISTORY_... cards. • Allows solution data to be sampled on subsets of battery mesh nodes.

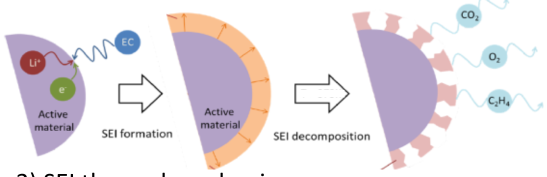
Electrochemical Battery Plus Models



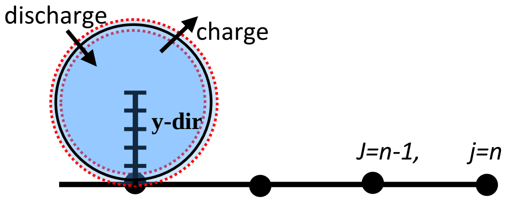
1) Li_xC_6 | $Li_yMn_2O_4$



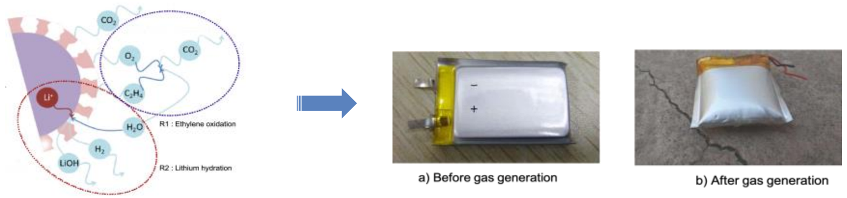
2) Li_xC_6 | NMC811



3) SEI thermal mechanism



4) Electrode swelling mechanism



5) Gas generation

Features:

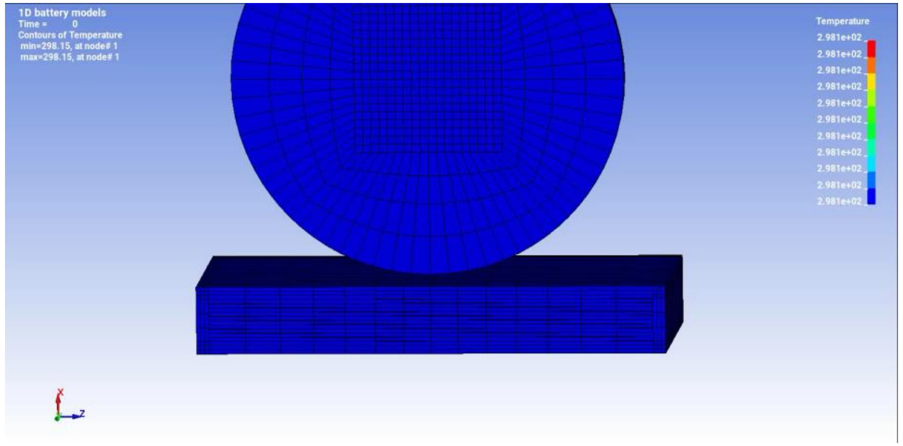
- Three different electrochemical LIB models are updated: Newman (6 equations), Thermal (10 equations), and Multiphysics (14 equations) model.
  - Updated Modified B-V kinetics.
  - Aging model.
  - SEI formation and decomposition thermal mechanism.
  - Battery Swelling model.
  - Gas generation model based on Ethylene oxidation and Lithium hydration reaction mechanisms

Main applications:

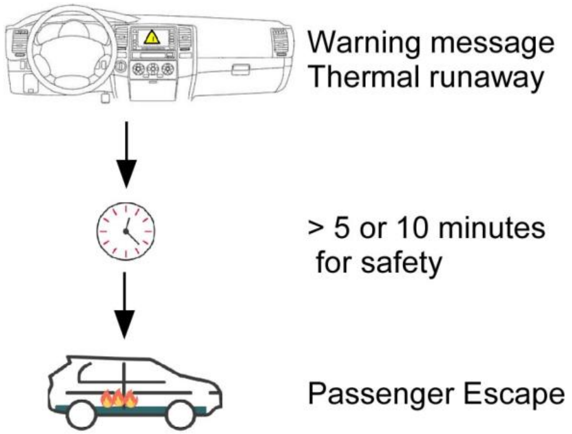
- Electrochemical LIB battery performance tests: Anode, Separator, and Cathode.
- Battery overcharge tests
- New LIB battery design in battery manufactures.

Ansys

Electrochemical Battery-Themo-Mechanical Coupled Models



1) Temperature profile of Themo-Mechanical coupled model.



2) Warning scenario based on Minimum ignition energy model.

Features:

- Newman(6 equations)-thermo-mechanical model
- Thermal (10 equations)-thermo-mechanical model
- Multiphysics(14 equations)-thermo-mechanical model.
- The shutdown key is installed based on Minimum ignition energy criterion.
- SMP and MPP capabilities are implemented.

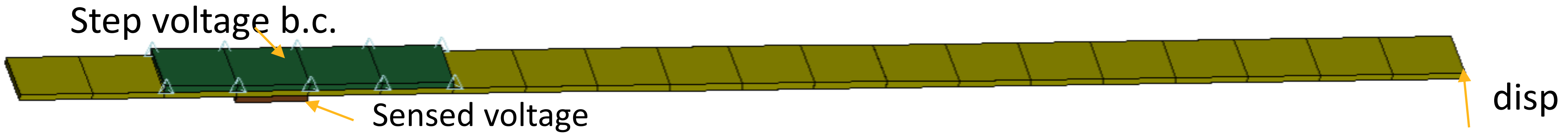
Main applications:

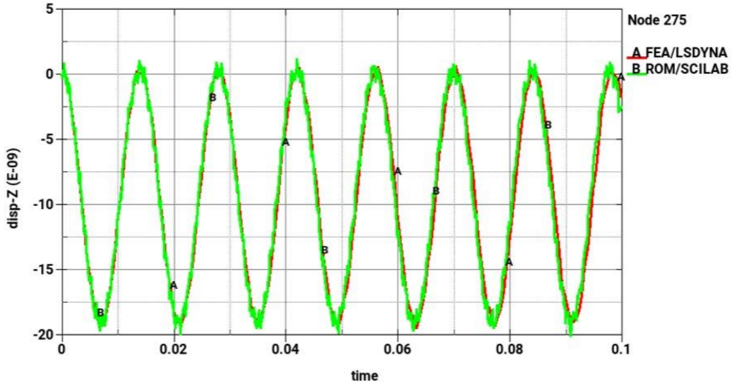
- Battery abuse test by external forces.
- Electrochemical battery plus applications can be tested with thermal and mechanical coupled cases.

Reduced Order Model (ROM): State Space Equation

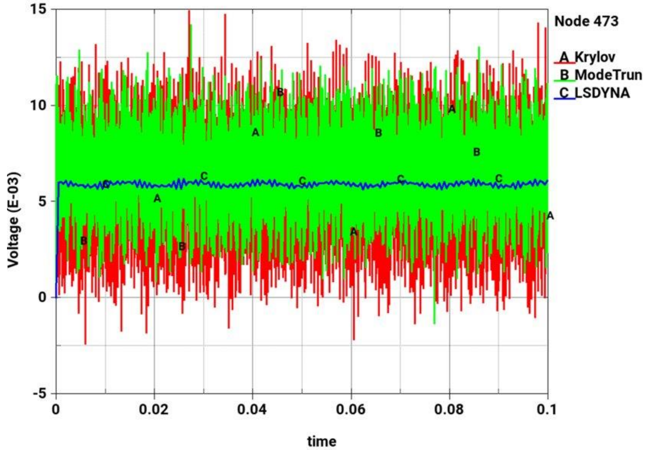
*CONTROLLER_PLANT

- CONTROLLER_PLANT offers two methods to derive ROM for controller plant design: mode-truncation method and Krylov's method
- Mode-truncation method works fine, SMP and MPP. However, Krylov sometime is preferred for its simplicity of input and sometimes offers better solution.

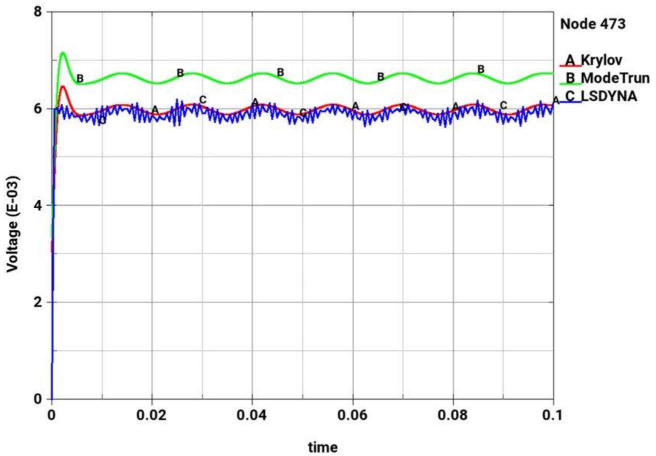




Disp



Sensed voltage



filtered sensed voltage

- Prior implementation of Krylov's method has difficulty handling large models. The improved implementation, currently SMP only, enables Krylov's method to be used for models of millions DOFs.

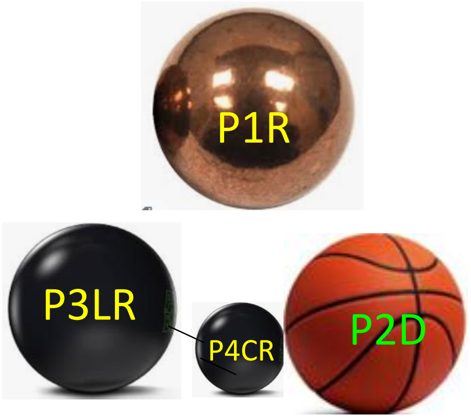
Rigid Body

Ansys

*CONTROL_RIGID

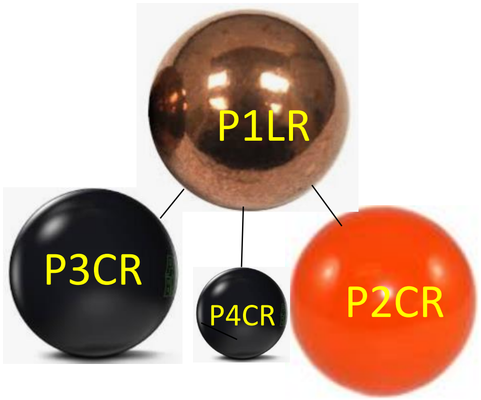
P2CR

- add option of RCVLR2D to *CONTROL_RIGID to recover the lead rigid body of constrained rigid bodies, which was changed due to *DEFORMABLE_TO_RIGID_AUTOMATIC.



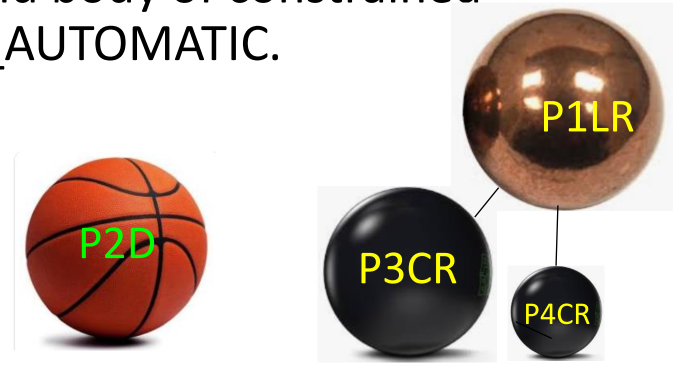
Initial status:
Deformable: P2
Independent Rigid: P1, P3,
Dependent Rigid: P4
P2 and P4 share common nodes

*DEF2RIG
PID=P2
LRB=P1



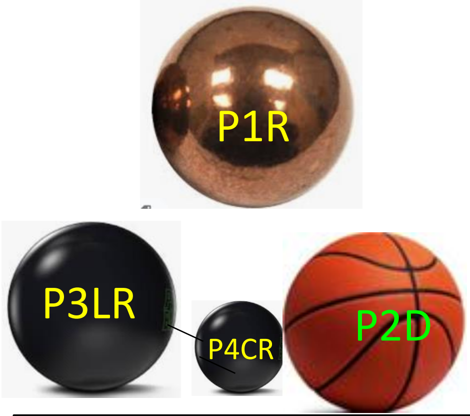
After DEF2RIG w. lead RB=P1:
Independent Rigid: P1
Dependent Rigid: P2, P3 & P4
P2 and P4 share common nodes

*RIG2DEF



RCVLR2D=0

After DEF2RIG w. RCVLR2D=0 :
P3 & P4 remains constrained to P1



RCVLR2D=1

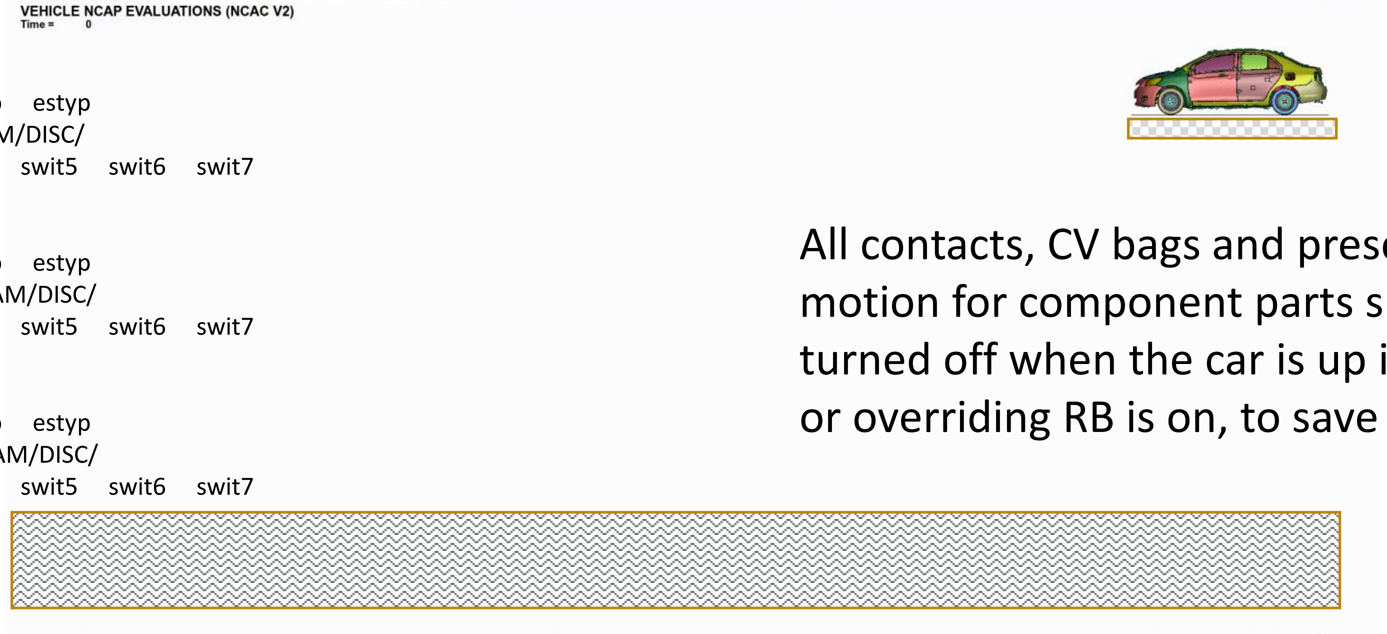
After DEF2RIG w. RCVLR2D=1:
P3 becomes independent, P4 is now constrained to P3

** CR: constrained RB
LR: Lead RB

Ansys

*CONSTRAINED_NODAL_RIGID_BODY_OVERRIDE (or MASTER)

- Some crash scenarios involve long pre-crash simulation, during which the whole vehicle moves as a rigid body, e.g., dynamic rollover testing and autonomous driving.
- Mostly, modes for such applications are adapted from a passive model which had existing airbag, contact, prescribed motion (ACP) definitions. Those ACP can be turned off to speed up the simulation when the overriding (master) rigid body is active.




```
*SENSOR_CONTROL
$#  cntlid      type   typeid   timeoff      nrep     estyp
         2AIRBAG          1        0       0BEAM/DISC/
$# initstt    swit1    swit2    swit3    swit4    swit5    swit6    swit7
     OFF,7
*SENSOR_CONTROL
$#  cntlid      type   typeid   timeoff      nrep     estyp
       11CONTACT          1        0       0BEAM/DISC/
$# initstt    swit1    swit2    swit3    swit4    swit5    swit6    swit7
     OFF,7
*SENSOR_CONTROL
$#  cntlid      type   typeid   timeoff      nrep     estyp
     103PRESC-MOT         2        0       0BEAM/DISC/
$# initstt    swit1    swit2    swit3    swit4    swit5    swit6    swit7
     ON,7
```

All contacts, CV bags and prescribed motion for component parts should be turned off when the car is up in the air, or overriding RB is on, to save CPU time

*CONSTRAINED_NODAL_RIGID_BODY_OVERRIDE (or MASTER)

- A safety model can have up to hundred of airbag, contact and prescribed motion (ACP) cards. Manually setting up senor cards to control them could be challenging.
- An optional card to automatically turn off/on overriding-rigid-body related ACP is added.

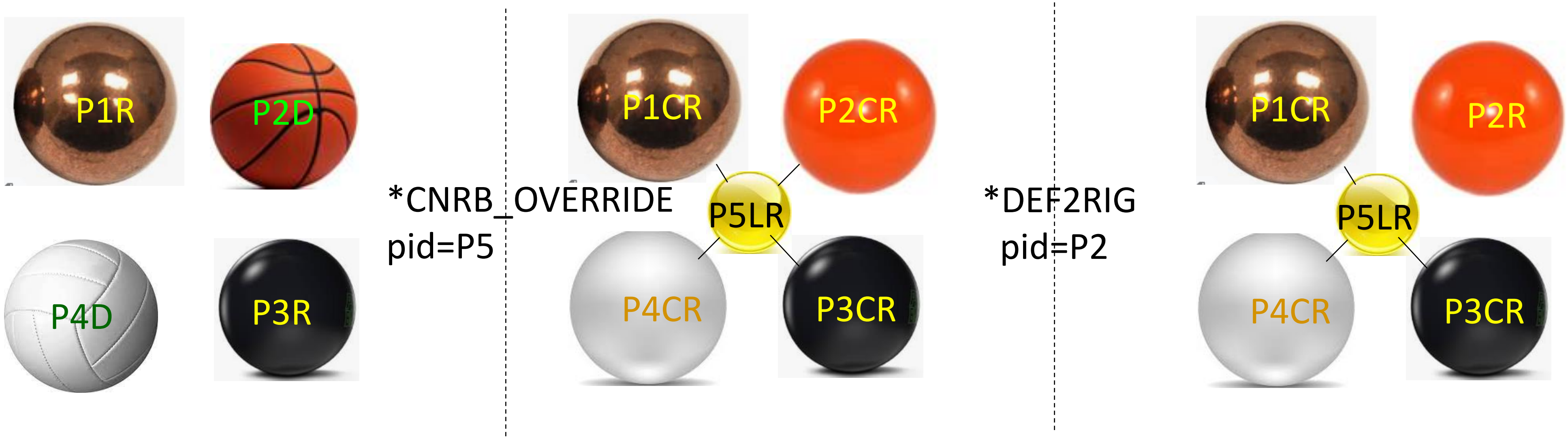
Card 7. This card is read when the OVERRIDE keyword option is used. It is optional.

ICNT	IBAG	IPSM					

- ICTC: LS-DYNA will check all contact cards, if the involved parts are part of overriding rigid body and no *SENSOR_CONTROL is defined for a contact, the contact will be automatically turned off when the overriding rigid body is active; and turned back on when the overriding rigid body is inactive
- IBAG: LS-DYNA will check all CV bag cards, if the involved parts are part of overriding rigid body and no *SENSOR_CONTROL is defined for a CV bag, the CV bag will be automatically turned off when the overriding rigid body is active; and turned back on when the overriding rigid body is inactive with all time-related curves offset automatically.
- IPSM: LS-DYNA will check all prescribed motion cards. If the involved parts are part of overriding rigid body, the prescribed motion will be automatically turned off when the overriding rigid body is active; and turned back on when the overriding rigid body is inactive. Setting IPSM=2 will offset the time history curves of prescribed motion.

How OVERRIDE_CNRB and DEF2RIG_AUTOMATIC work together

- After overriding CNRB is disabled, automatic DEF2RIG can be applied to parts where were part of overriding CNRB.
- Automatic DEF2RIG can also be applied to the parts which are part of an active overriding CNRB. The involving parts in DEF2RIG will be separated from overriding CNRB.



All parts are constrained to lead RB P5

P2 is separated from P5LR and becomes am independent RB

FMI-based co-simulation

*COSIM_FMI for FMI_based co-simulation

- add ModelStructure for all export variables, with causality=”outputs”, in model description file, some compliance check needs it for all output variables.

```
<ModelVariables>
  <ScalarVariable name="FX621" valueReference="0" description=" "
                causality="output" variability="continuous" initial="exact">
    <Real start="0"/>
  </ScalarVariable>
```

```
<fmiModelDescription
  fmiVersion="2.0"
  generationTool="LS-DYNA DEV-91399-gaa3c5712c2"
  modelName="INV_PEND3"
  guid="{er24s8ur-zpzx-4da1-oje8-60xlsls7e9th}"
  generationDateAndTime="2022-02-22T09:40:51Z"
  numberOfEventIndicators="0">
```

- output more details to FMU's xml description file: git-build-id, creation time and date.
- fix compliance issue of FMU 1.0 in Windows.
- allow co-simulations involving more than one FMU.


```
*COSIMULATION_FMI_INTERFACE
           PID1
$#  impexp    regtyp     regid     field      init     ratio      coor       ref
       EXP      NODE       608        VX         0         1         0         0
$#  impexp    regtyp     regid     field      init     ratio      coor       ref
       IMP      NODE       608        FX         0         1         0         0
$
*COSIMULATION_FMI_INTERFACE
          PEND2
$#  impexp    regtyp     regid     field      init     ratio      coor       ref
       EXP      PART         2        WY         0         1         0         0
$#  impexp    regtyp     regid     field      init     ratio      coor       ref
       IMP      PART         2        TY         0         1         0         0
```

- allow co-simulation with FMUs of different version, FMI1.0 and FMI2.0, Linux only.
- add airbag pressure, volume and temperature for EXPORT
- make the size of array for exchange variables flexible to accommodate huge models

Ansys

Piezoelectric Material

Piezoelectric material

- Bug fixes
  - happened when *CONTACT_TIED ties two elements, one is piezoelectric, while the other one is not.
  - for eigenvalue calculation
  - for 20-node hexahedron element.

```
calculation with mass scaling for mi
   added mass    =    0.0000E+00
   physical mass=    2.4437E-03
   ratio         =    0.0000E+00

*** Error 40148 (SOL+148)
    error return from kslvr_init()
```

- *INCLUDE_TRANSFORM
  - apply *INCLUDE_TRANSFORM to *MAT_ADD_PZELECTRIC
  - add electric charge transformation factor, FCTCHG, currently applied to MAT_ADD_PZELECTRIC only
- BOUNDARY_NON_REFLECTION for piezoelectric material
  - enabled 20-node hexahedron element
  - fix a bug for TET10 elements, in which some segments failed to locate its related elements
  - fix a MPP bug that happened when a processor is not involved in any piezoelectric boundary condition

Piezoelectric material

- Bug fixes
  - happened when *CONTACT_TIED ties two elements, one is piezoelectric, while the other one is not.
  - for eigenvalue calculation
  - for 20-node hexahedron element.

```
calculation with mass scaling for mi
   added mass    =   0.0000E+00
   physical mass=   2.4437E-03
   ratio         =   0.0000E+00

*** Error 40148 (SOL+148)
    error return from kslvr_init()
```

- *INCLUDE_TRANSFORM
  - apply *INCLUDE_TRANSFORM to *MAT_ADD_PZELECTRIC
  - add electric charge transformation factor, FCTCHG, currently applied to MAT_ADD_PZELECTRIC only
- BOUNDARY_NON_REFLECTION for piezoelectric material
  - enabled 20-node hexahedron element
  - fix a bug for TET10 elements, in which some segments failed to locate its related elements
  - fix a MPP bug that happened when a processor is not involved in any piezoelectric boundary condition

Ansys

Piezoelectric material

- Bug fixes
  - happened when *CONTACT_TIED ties two elements, one is piezoelectric, while the other one is not.
  - for eigenvalue calculation
  - for 20-node hexahedron element.

```
calculation with mass scaling for mi
   added mass    =   0.0000E+00
   physical mass=    2.4437E-03
   ratio         =   0.0000E+00

*** Error 40148 (SOL+148)
    error return from kslvr_init()
```

- *INCLUDE_TRANSFORM
  - apply *INCLUDE_TRANSFORM to *MAT_ADD_PZELECTRIC
  - add electric charge transformation factor, FCTCHG, currently applied to MAT_ADD_PZELECTRIC only
- BOUNDARY_NON_REFLECTION for piezoelectric material
  - enabled 20-node hexahedron element
  - fix a bug for TET10 elements, in which some segments failed to locate its related elements
  - fix a MPP bug that happened when a processor is not involved in any piezoelectric boundary condition

Ansys

Model Transformation

*NODE_TRANSFORMATION

- add IMMED option to *NODE_TRANSFORM for the processing sequence of *NODE_TRANSFORM

Card 1	1	2	3	4	5	6	7	8
Variable	TRSID	NSID	IMMED					

EQ.0: Node transformation is performed after all input cards are read through. It is more efficient, and the definition sequence of *NODE_TRANSFORM and its NSID is irrelevant, meaning the referred NSID doesn't have to be defined prior to *NODE_TRANSFORM. However, if nodes in NSID are used in, for instance, POS6N of *DEFINE_TRANSFORMATION, the original coordinates, not the transformed coordinates, will be used to define the transformation matrix.

EQ.1: Node transformation is performed immediately after *NODE_TRANSFORM is read. The referred NSID and its nodes must be defined prior to *NODE_TRANSFORM.

PART 20, POS6N IMMED=0

```
*NODE
21        0.00000      30.000000           0.0
22      -50.00000      30.000000           0.0
23        0.00000      30.000000         -30.0.
*DEFINE_TRANSFORMATION
20
POS6N   21     22     23     11     12     13
*NODE_TRANSFORM
   20      20
```

PART 1, target positions

```
*NODE
   1    -100.00000    -100.000000         0.0
   2    -100.00000    -150.000000         0.0
   3    -100.00000    -100.000000       -30.0
```

PART 10, rotation + translation

```
*NODE
     11        0.00000       0.000000          0.0
     12      -50.00000       0.000000          0.0
     13        0.00000       0.000000        -30.0
*DEFINE_TRANSFORMATION
     10,
          ROTATE, 0.0,  0., -30. ,0., 0. ,-30.,-90
          TRANSL , -100.0 ,-100.0
 *NODE_TRANSFORM
     10       10
```

PART 20, POS6N IMMED=1

```
*NODE
21        0.00000      30.000000           0.0
22      -50.00000      30.000000           0.0
23        0.00000      30.000000         -30.0.
*DEFINE_TRANSFORMATION
20
POS6N   21     22     23     11     12     13
*NODE_TRANSFORM
    20      20      1
```

Ansys

S-ALE General

*ALE_STRUCTURED_MULTI-MATERIAL_GROUP

*ALE_STRUCTURED_MULTI-MATERIAL_GROUP(_PLNEPS/_AXISYM)							
AMMGNM	MID	EOSID					PREF
TNT	101	101					0.0
insideAir	102	102					101325.
water	103	103					0.0
outsideAir	102	102					101325.

The new keyword *ALE_STRUCTURED_MULTI-MATERIAL_GROUP is added for sake of user-friendliness

1. No need to define “material parts” and “sections”
2. Each fluid could have its own reference pressure definition (PREF) in a cleaner way.
3. Use of strings to refer to fluids rather than the order of appearance, much less error-prone.
4. Simpler 2D setup: add _AXISYM (axisymmetric) or _PLNEPS (plane strain)

*All ALE keywords containing fields with “AMMGID” now supports string-type “AMMGNM” (ammg name). Before, once users change order of AMMGs, all occurrences of AMMGIDs need to be changed too. This caused too many input deck mistakes. Now not needed at all.

Ansys

*BOUNDARY_SALE_MESH_FACE

*BOUNDARY_SALE_MESH_FACE							
BCTYPE	MSHID	NEGX	POSX	NEGY	POSY	NEGZ	POSZ
NoFlow	1			1		1	
NonRefl	1	1	1		1		1
Fixed	2			1	1	1	1
Sym	2	1					

BCTYPE Available boundary conditions:

- EQ.FIXED: All nodes at the face are fixed in all directions.
- EQ.NOFLOW: No flow allowed through the face.
- EQ.SYM: The face is a symmetric plane (same as NOFLOW).
- EQ.NONREFL: Non-reflective boundary condition

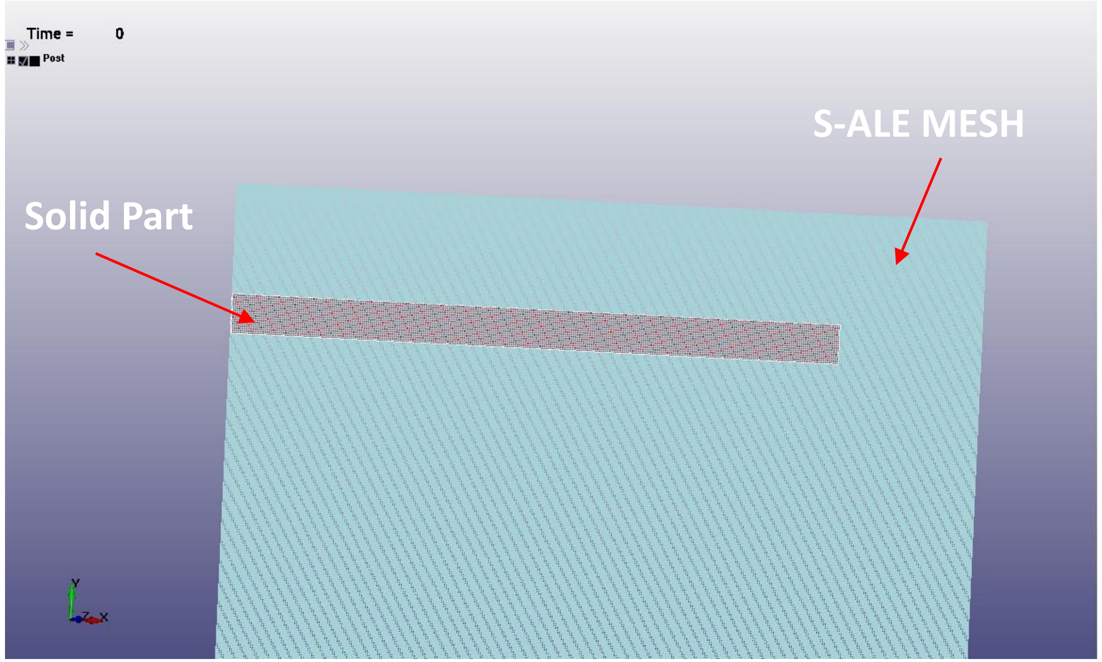
MSH ID: S-ALE mesh ID
NEGX,POSX: 0/1; off/on flag on –x/+x S-ALE mesh face

Added for user-friendliness. A macro-like keyword internally converted during the keyword reader phase. This keyword translates to a combination of several different keywords. Those keywords include *SET_NODE_GENERAL (with SALEFAC option), *SET_SEGMENT_GENERAL (with SALEFAC option), *BOUNDARY_SPC_- SET, and *BOUNDARY_NON_REFLECTING.

Volume Filling using LAG Solid Parts

```
*ALE_STRUCTURED_MESH_VOLUME_FILLING
$     MSHID                AMMGTO
         21                air_in
$      GEOM     IN/OUT     PSETID
    PARTSET                    11
```

PARTSET: Lagrange structure. In 3D, this structure is defined using shell elements. And those shell elements form a container. We fill the inside of that container with certain ALE fluid.



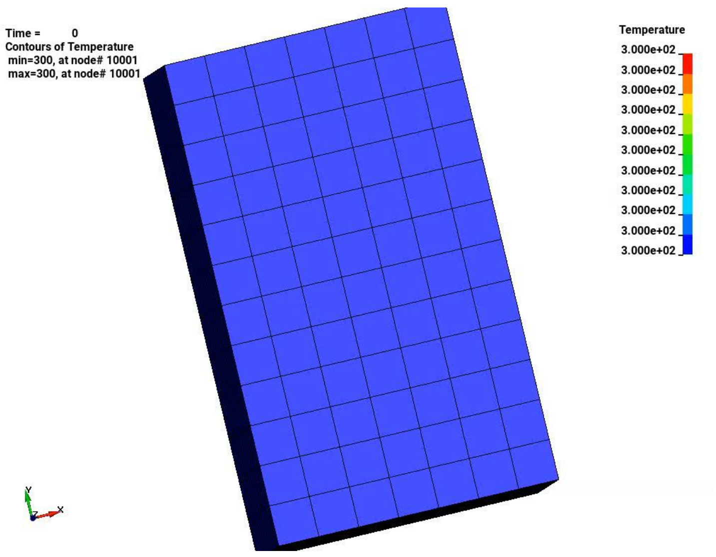
NEW DEVELOPMENT:
Now users want to fill the inside of a solid part, i.e. Fill the space occupied by that solid part.

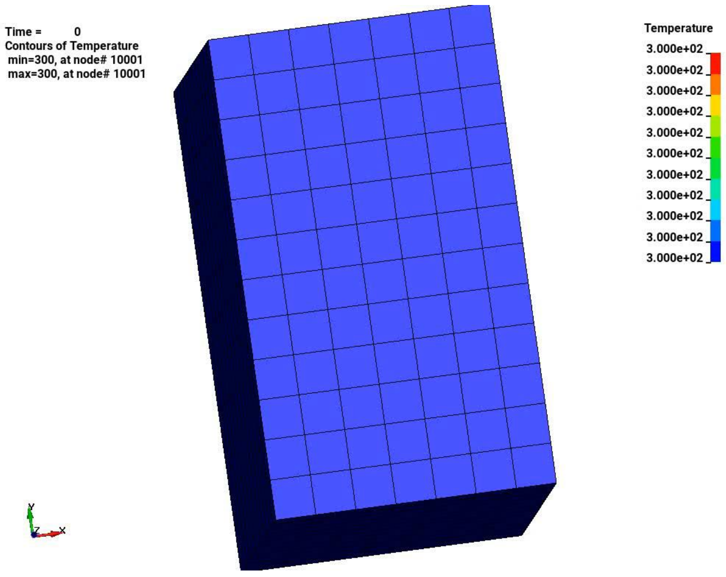
Left figure shows a LAG structure, modeled by solid elements. It is a target which is subject to blast loading. We want to fill the space taken by the solid part by "air inside target".

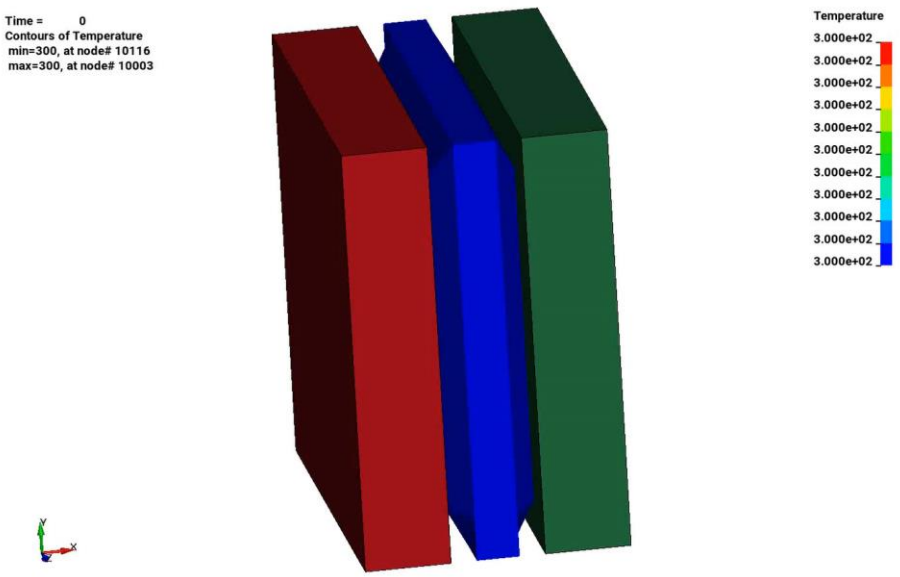
Previously, users need to construct a fake shell part, at the surface of the solid part; and reverse its normal.

Now, we do that automatically inside the code.

S-ALE Thermal Enhancements



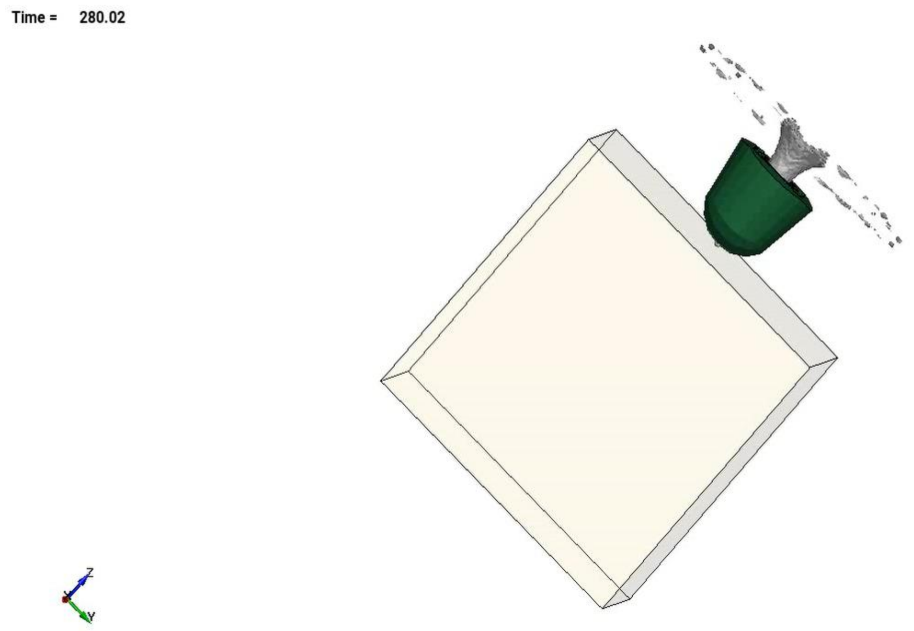


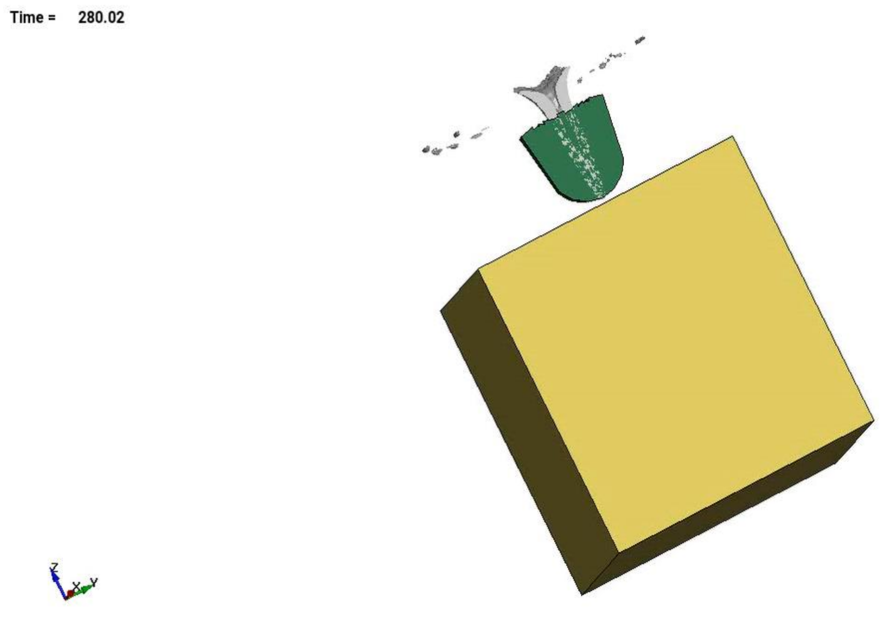


Instability and wrong temperature in partially filled S-ALE elements with vacuum. New algorithms implemented in thermal advection to solve the problem.

Full deck restart, limited supported added

Full deck restart was not never supported well; As a first step, we map S-ALE element history variables, nodal properties. Now we could have a pure S-ALE 1st run and add LAG parts in the second one. (FSI history variables remapping not supported yet.)



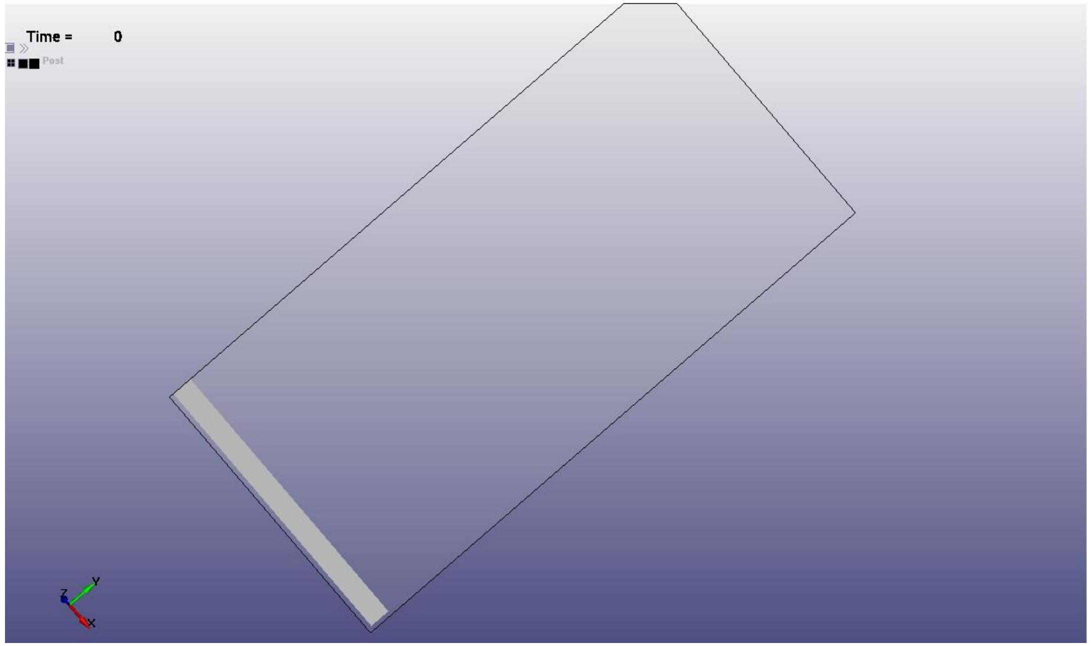


Double impact: EFP(explosive formed projectile) hits first, then the kinetic head follows in the same path

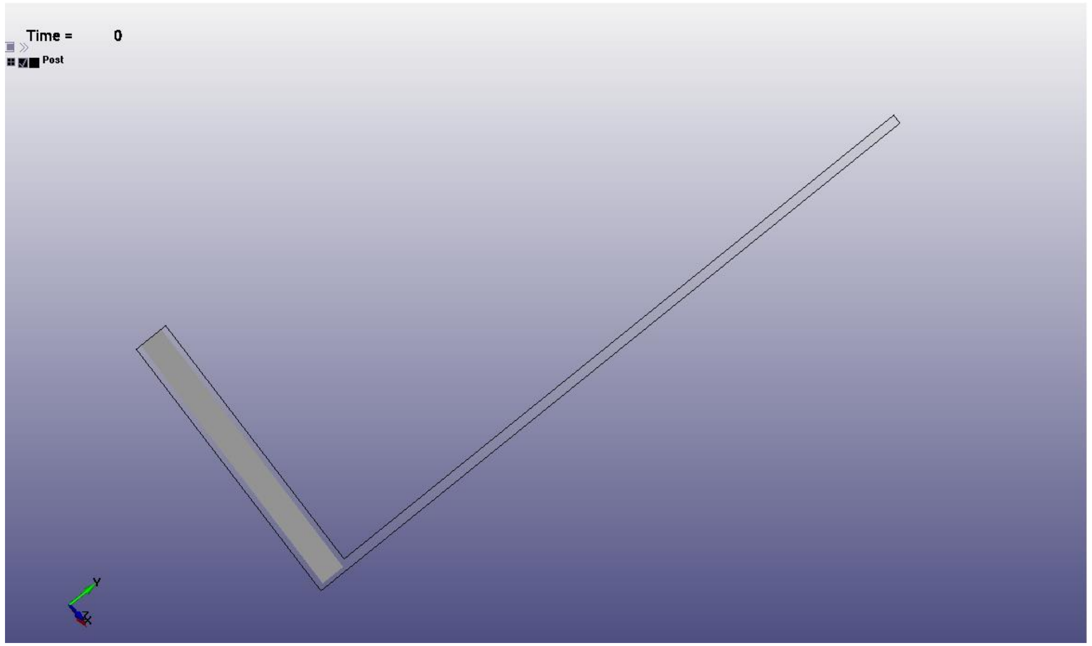
2D S-ALE

2D S-ALE: Mesh Trimming to Reduce # of Elements

Mesh Trimming is to trim S-ALE elements out of the domain of interest. With less number of elements, simulation time is reduced proportionally. For this model, before trimming # of elements=233245, after trimming # of elements=43045, reduced to 1/6. This, in turn, saves 5/6 of running time.



Before



After

*ALE_STRUCTURED_FSI for 2D (with Eroding Supported)

*ALE_STRUCTURED_FSI							
LSTRSID	ALEID	SSTYP	MSTYP				MCOUP
		PFAC			IFLIP		

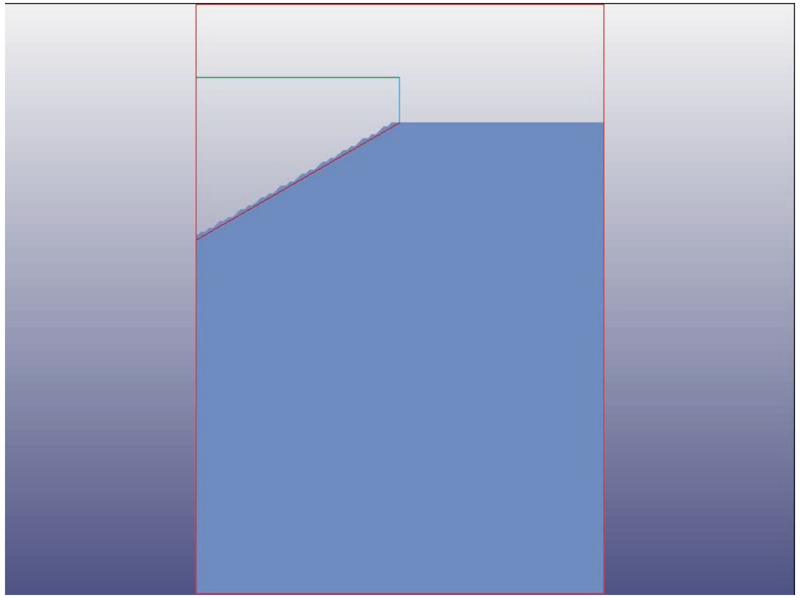
LSTRSID: Lagrange Structure ID
ALEID: S-ALE mesh PART ID
SSTYP: PARTSET/PART/SEGSET (0/1/2)
MSTYP: PARTSET/PART (0/1)

MCOUP: ALE fluids to be coupled
FLIP: Flip structure normal or not
PFAC: Penalty Stiffness
=-N: Load Curve (recommended)

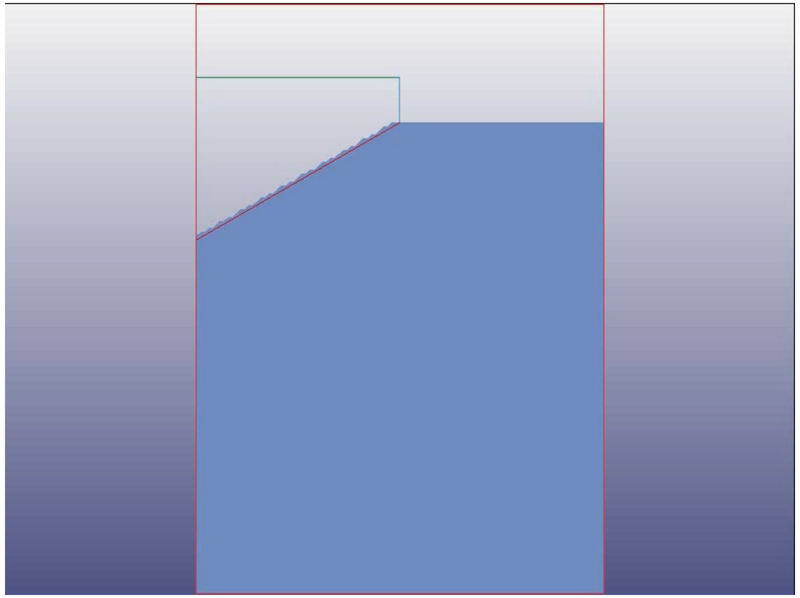
In 2D, Lagrange structure definitions could be either from BEAMs or SHELLs.

- Element Formulation
  - 2D Plane strain: Beams eleform 7; Shells eleform 13
  - 2D Axisymetric: Beams eleform 8; Shells eleform 14/15
- Coupling Surface Segments Generation
  - Beams: Segments generated directly from beam elements. NOTE: beam element normal must be aligned so it points to the fluid(s) in coupling
  - Non-erodible Shells: Surface segments are picked to generate coupling segments
  - Erodible Shells: Interior segments also generated and become active when exposed.

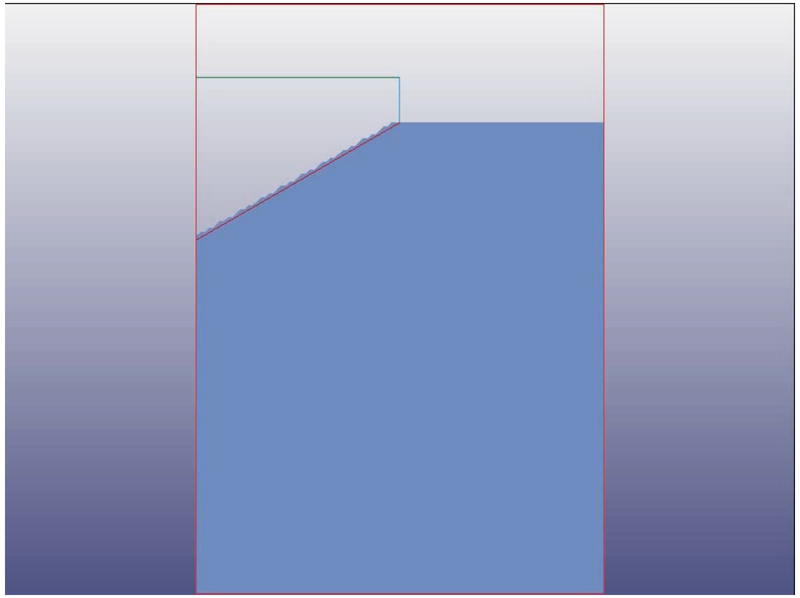
*ALE_STRUCTURED_FSI for 2D



3D



2D Plane Strain



2D Axisymmetric

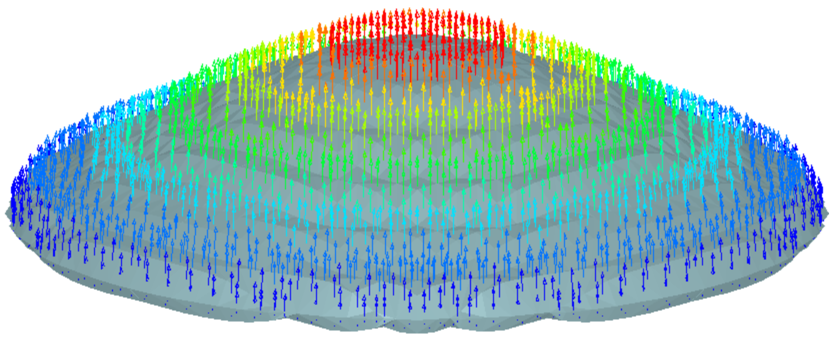
Test Model: Wedge Slam

ALE General

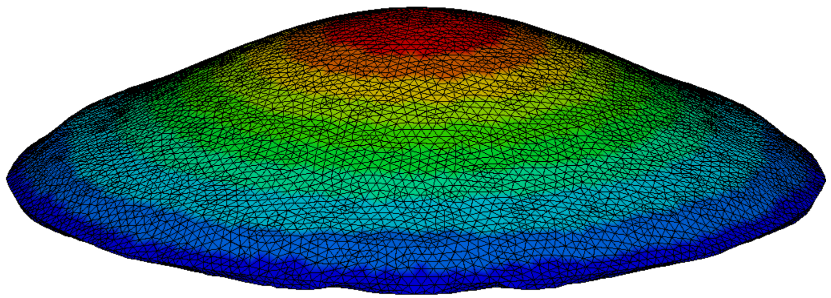
*ALE_MESH_INTERFACE

*ALE_MESH_INTERFACE and *INITIAL_LAG_MAPPING can map the last cycle data from an ALE run to a tetrahedral mesh for a Lagrangian run (a classical finite element model). The tetrahedra mesh a selection of ALE materials in their deformed configuration after the ALE run. The data from the ALE deformed materials initialize the tetrahedral mesh.

Test case : Explosively Formed Projectile



MAPPING



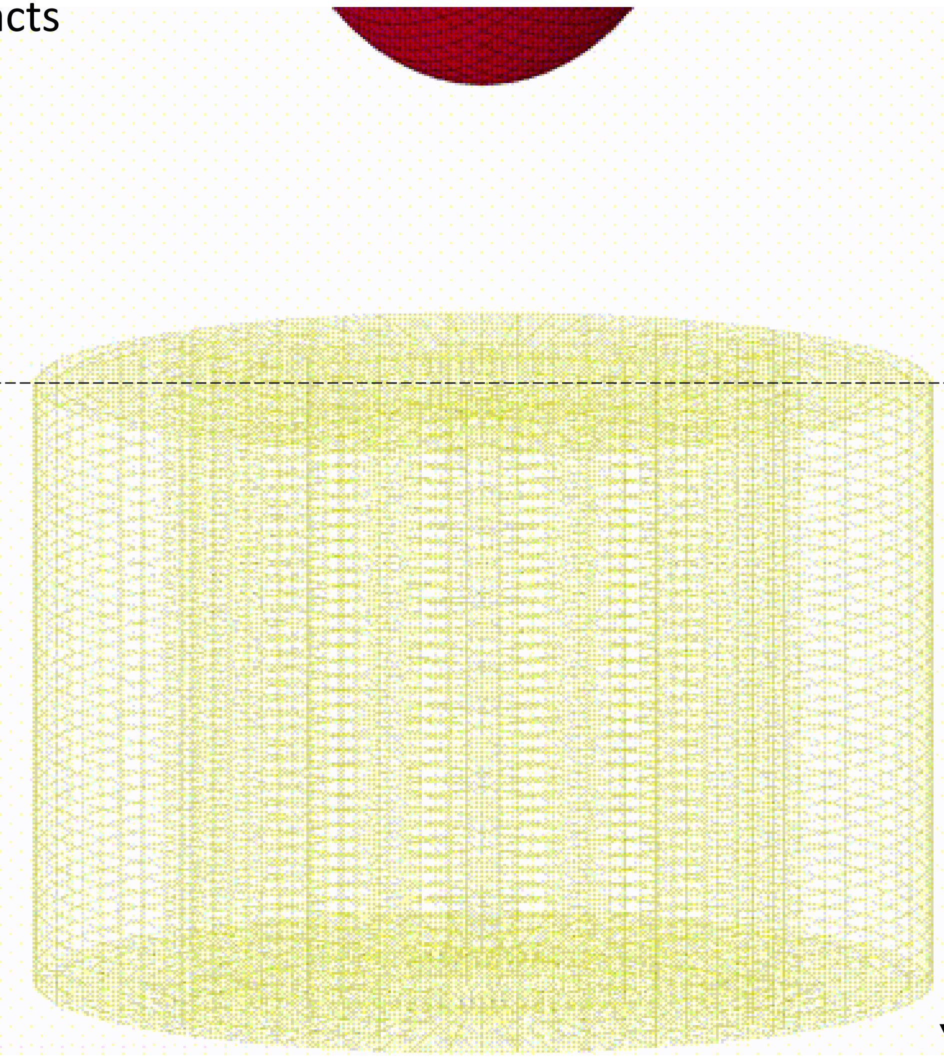
ALE material after the 1st run

Tetrahedral mesh at the beginning of the 2nd run

*ALE_MAPPING_FROM_LAGRANGIAN

*ALE_MAPPING_FROM_LAGRANGIAN transforms Lagrangian parts into ALE groups

Test case : a full metal jacket bullet impacts
a solid cylinder of steel



The last cycle of the 1st Lagrangian run creates a mapping file that can be read by the 2nd ALE model for its initialization. Lagrangian data are mapped to ALE elements and nodes

T=0ms

1st run : A full metal jacket bullet (2 parts) meshed with Lagrangian hexahedra (*SECTION_SOLID elform=1) travels at 300m/s towards a cylinder

T=23microsec : right before the impact, the 2 bullet parts become 2 ALE groups

2nd run : The full metal jacket bullet made of 2 ALE groups fills an ALE mesh (*SECTION_SOLID elform=11) and deforms on impact with the cylinder

IGA

Summary of new features in R14

Spatially Varying Baseline Orientation	• Generic scheme that supersedes conventional AOPT definitions.
Feature-based BC and Constraint Application	• Definition on the geometry directly.
Extended Element Erosion Scheme	• Corrects earlier deficiencies related to basis functions spanning across cracks. • Supports (un)structured shells and solids. • SMP only for now.
Support for Unstructured Splines	• `*IGA_2D_BEZIER_XYZ` and `*IGA_3D_BEZIER_XYZ` keywords. • Same pipeline as for their structured counterparts. • Allows for feature-based BCs and constraints in the future.
Improved Time Step Estimates	• `*CONTROL_TIMESTEP` IGADO • Accounting for higher-order interelement continuity
New Memory Management Scheme	• Decreased memory use. • Faster analysis.
Hybrid Models	• Various enhancements to support linear FE meshes as well as structured and unstructured meshes in the same assembly. • Welds with HAZ, CNRB, residual strain/stress mapping, etc.

Spatially Varying Baseline Orientation

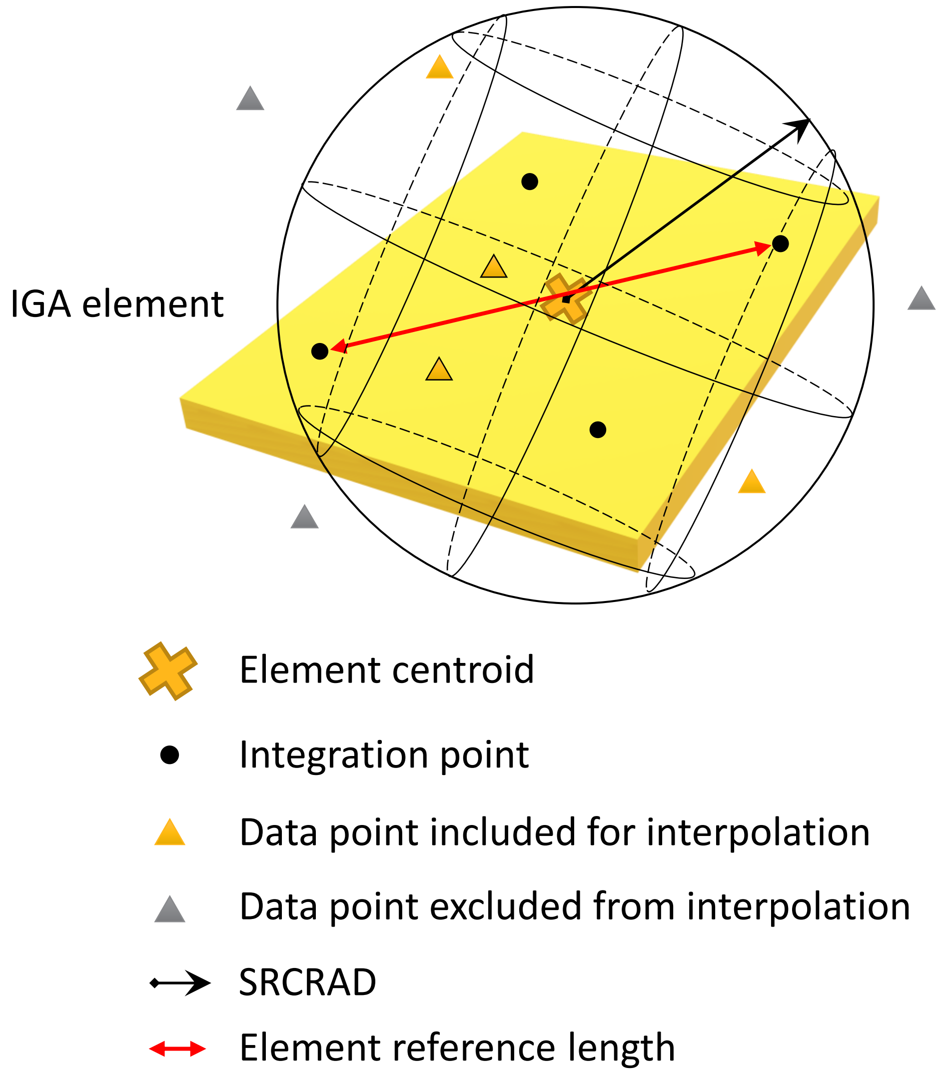
`*PART_FIELD`
Assign spatially varying baseline orientation field to part.

`*DEFINE_FIELD`
Assign field variables, to spatial locations collected in a point cloud. Define parameters for RBF-based interpolation.

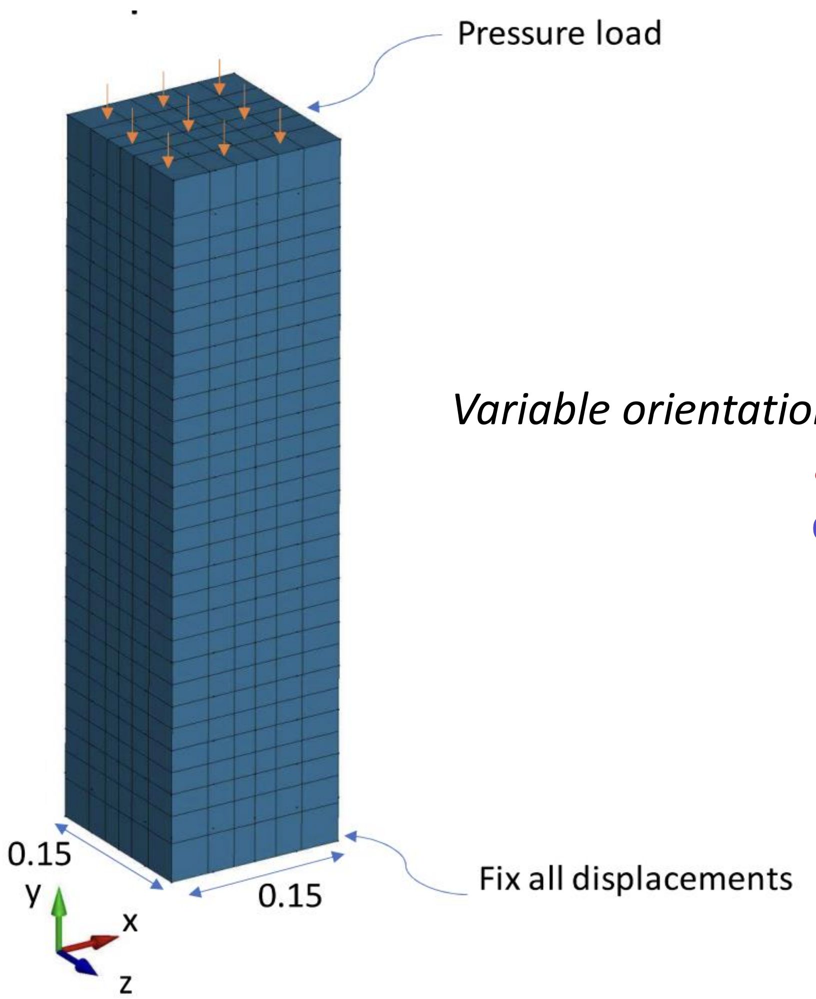
`*DEFINE_POINT_CLOUD`

Remarks:

- Invoking the `*PART_FIELD` keyword supersedes baseline vectors activated on the material cards (AOPT).
- The framework is designed to be used beyond defining baseline orientations in the future.
- Compatible with classical finite element implementation.



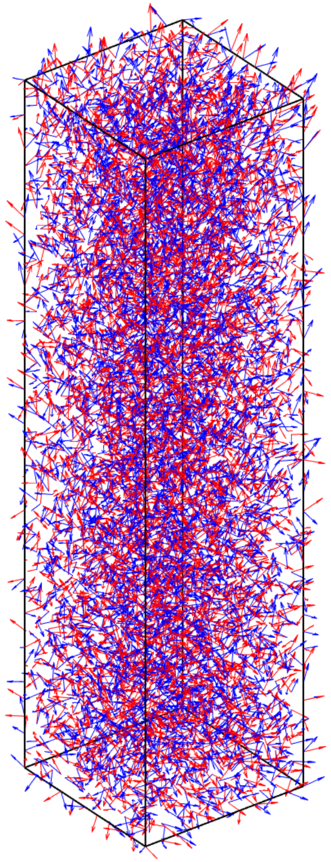
Slab Under Uniform Compression with Spatially Variable Baseline Orientation

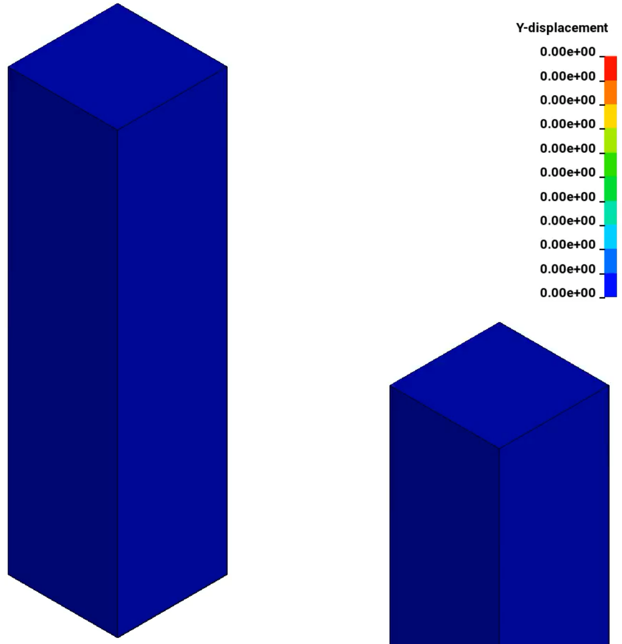


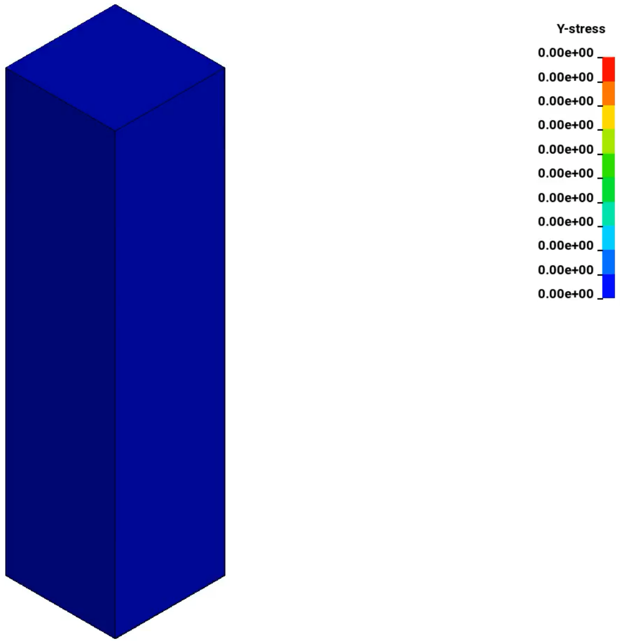
Variable orientation

a

d

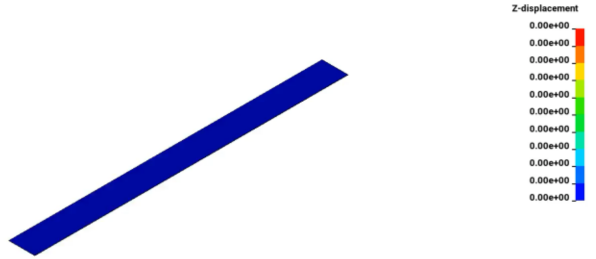




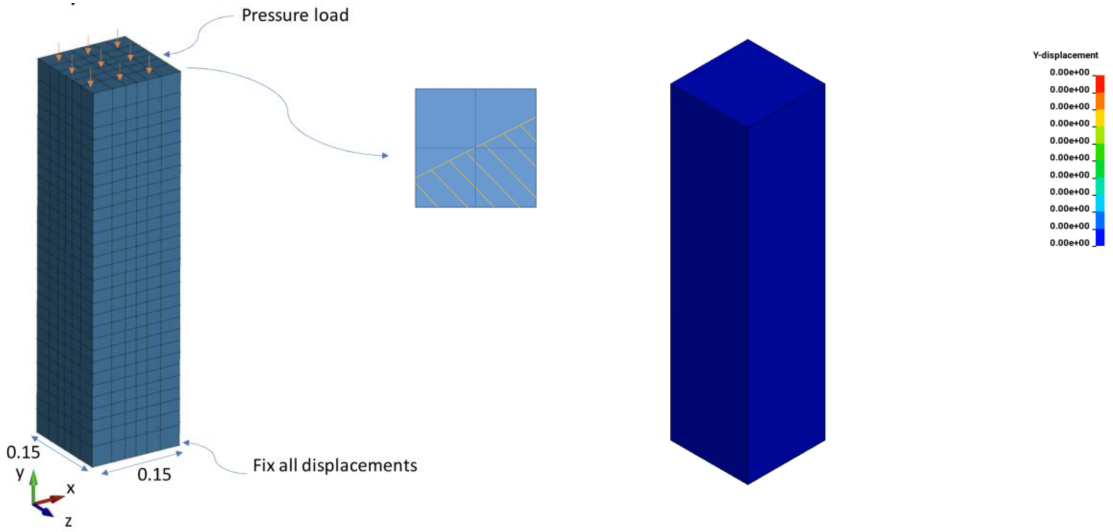


Feature-based Boundary Conditions and Constraints

- Control points and elements have no user IDs using the `*IGA` keywords.
- Boundary conditions are imposed on geometric entities defined in the parameter space of the geometry e.g., points, edges, faces, and volumes.
- Lower-level parametric entities assigned to patch at input.
- Constraints between parametric entities e.g., CNRB.

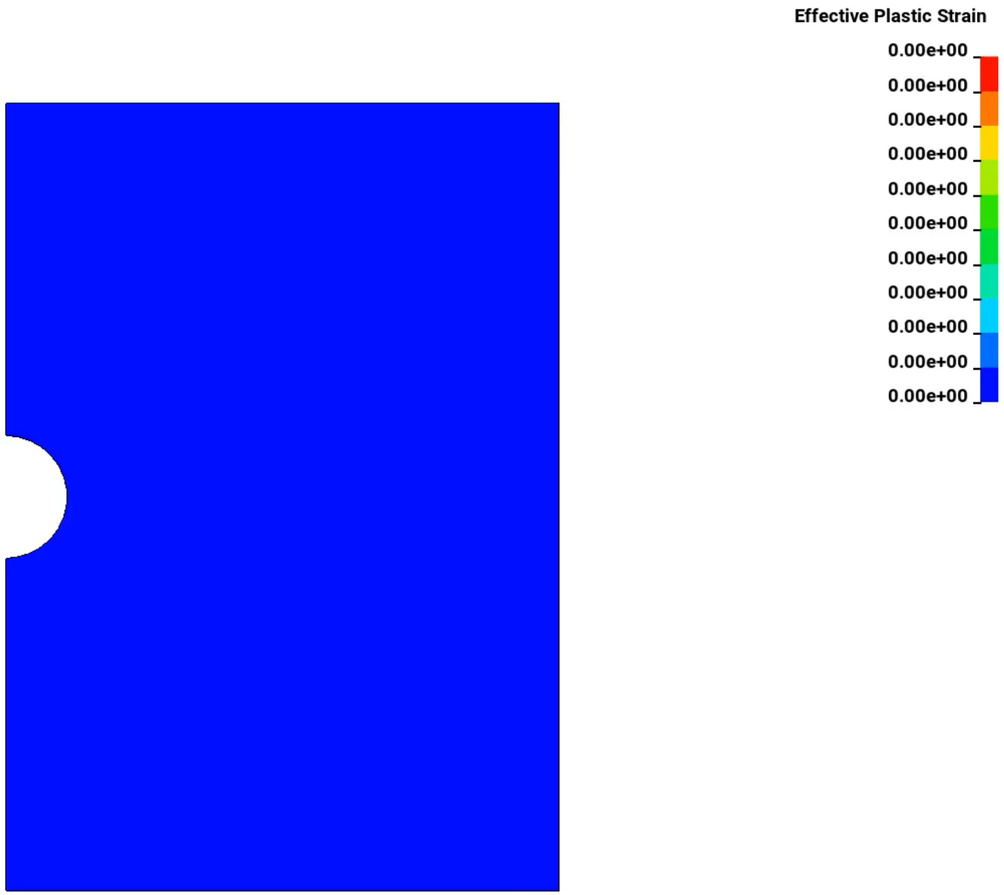


Clamped plate
Bending moments defined at three parametric points along the opposite edge.

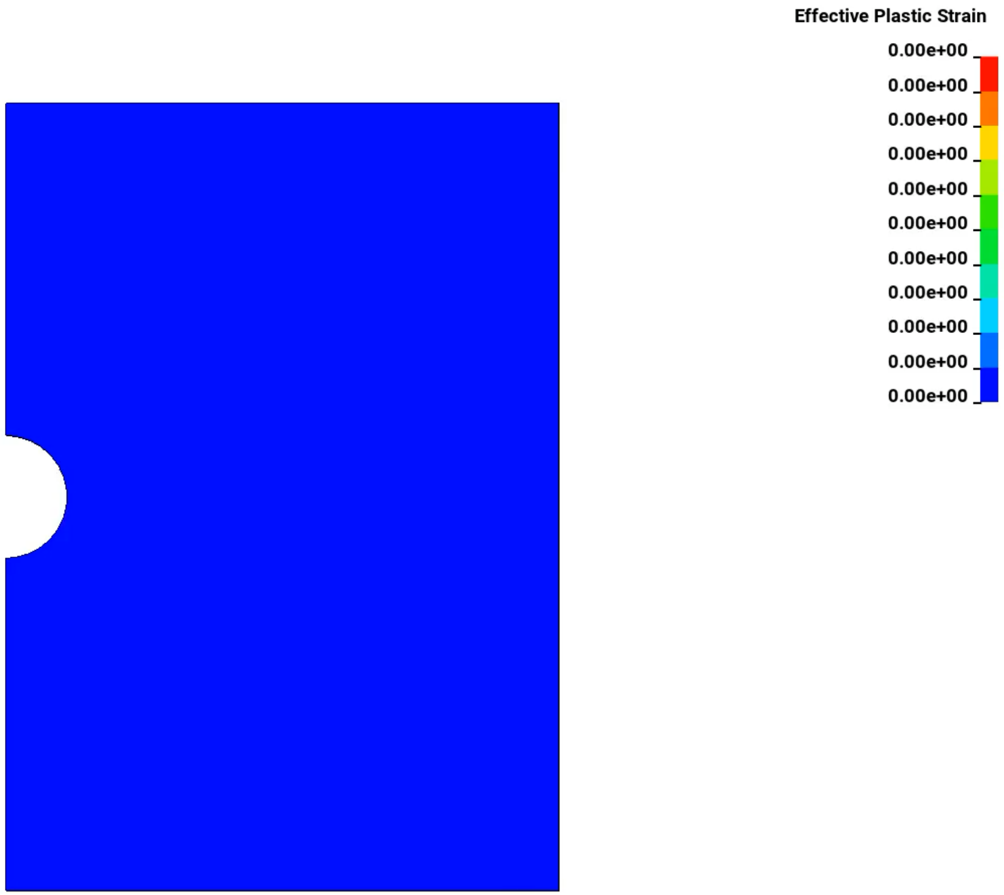


Slab Under Non-Uniform Compression

Revision of IGA Element Erosion (1)

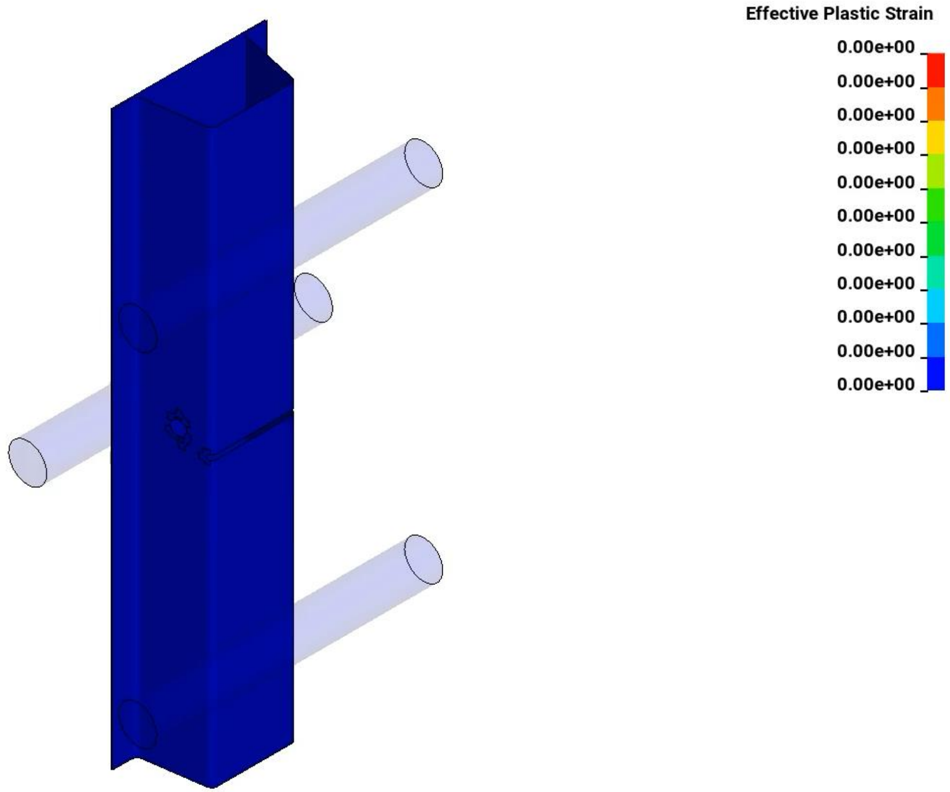


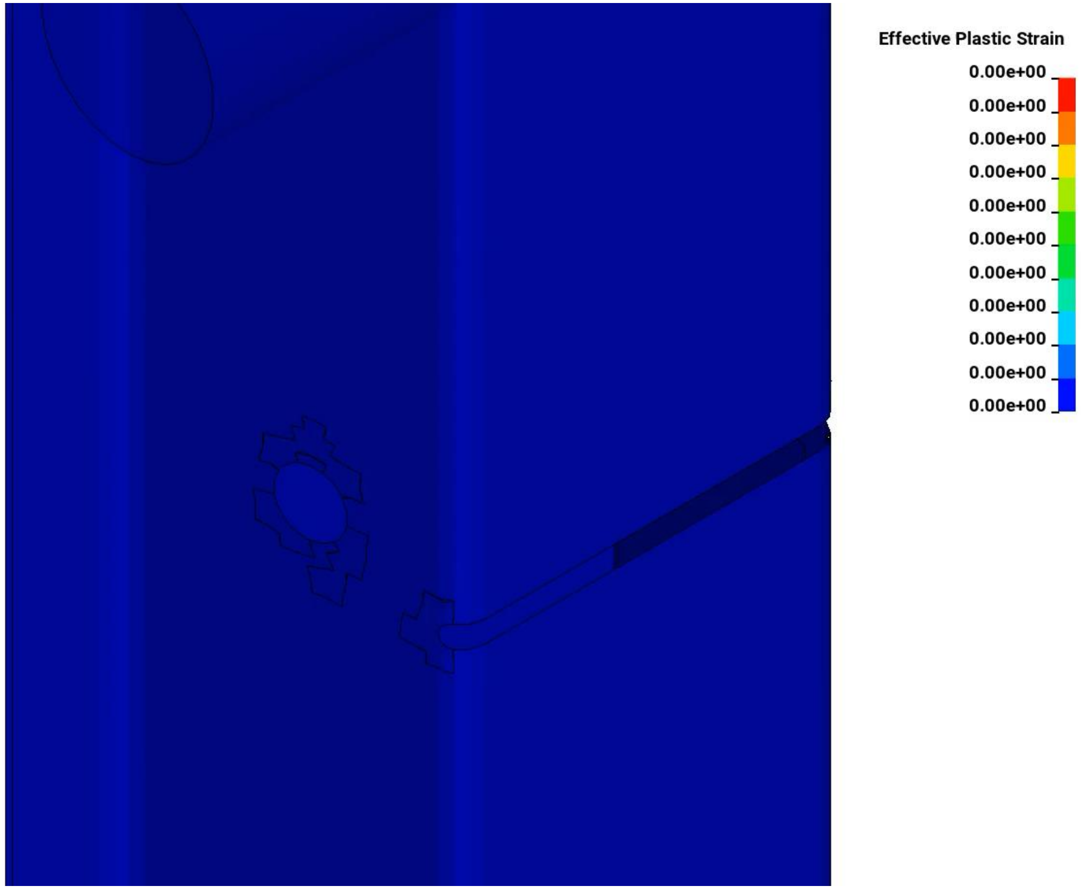
Scheme developed for Lagrange polynomial-based FEA



Revisited scheme for IGA

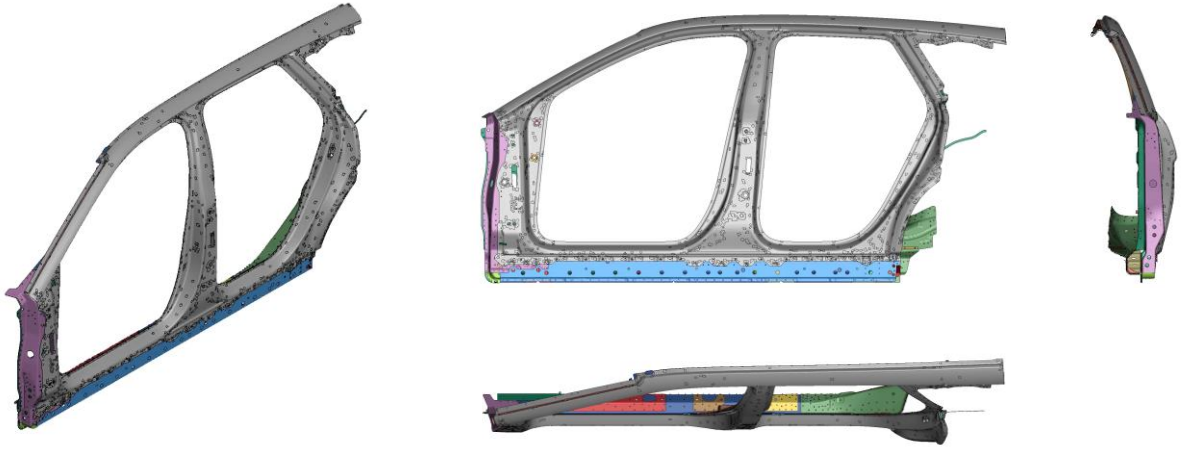
Revision of IGA Element Erosion (2)



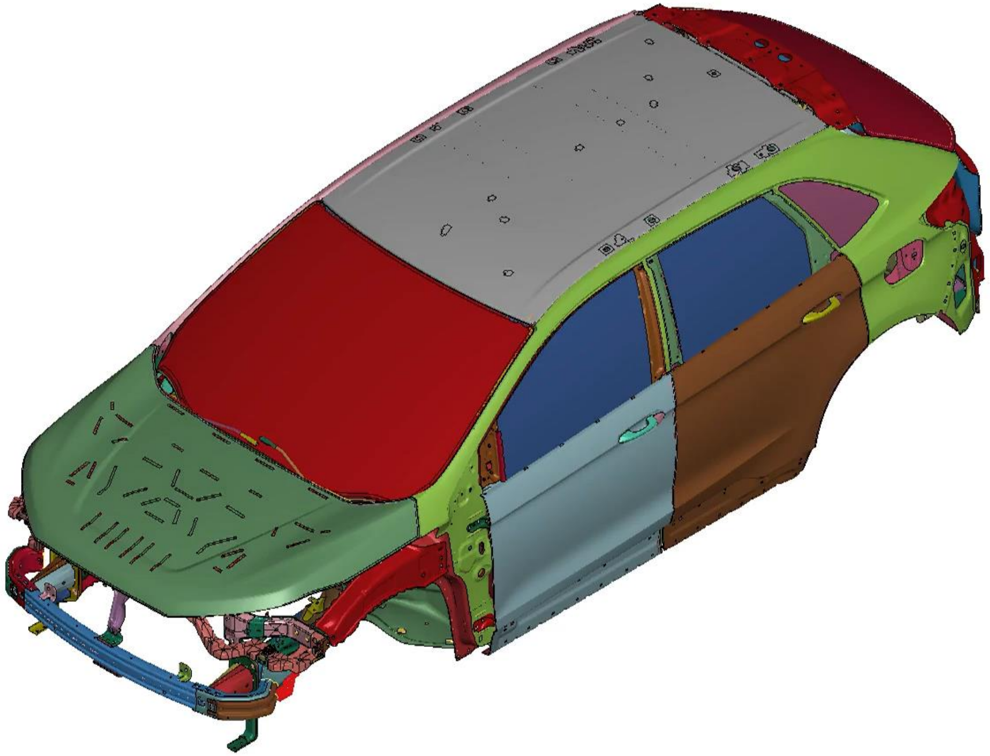


General scheme applicable to (un)structured shells and solids.

Hybrid assemblies



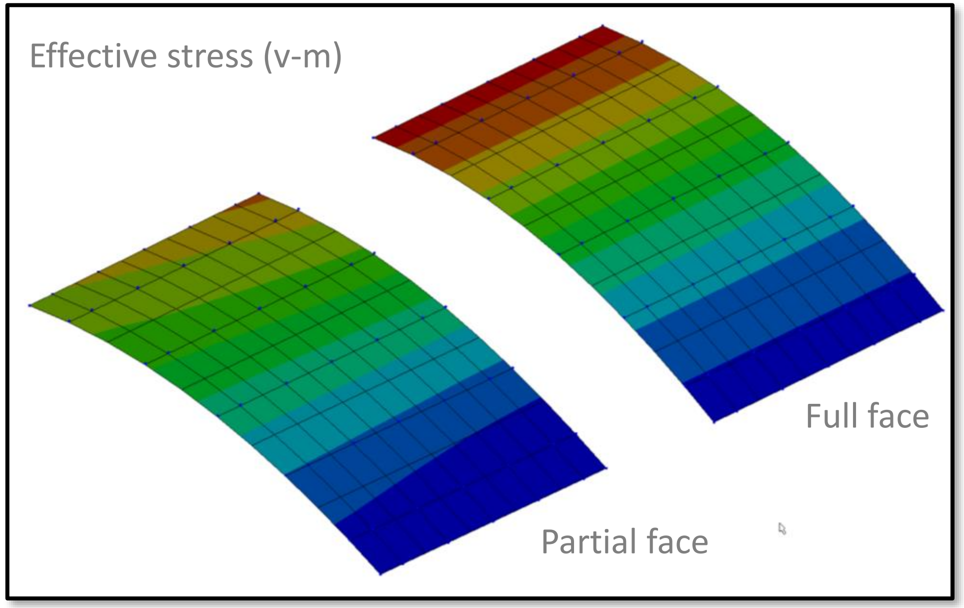
courtesy of General Motors

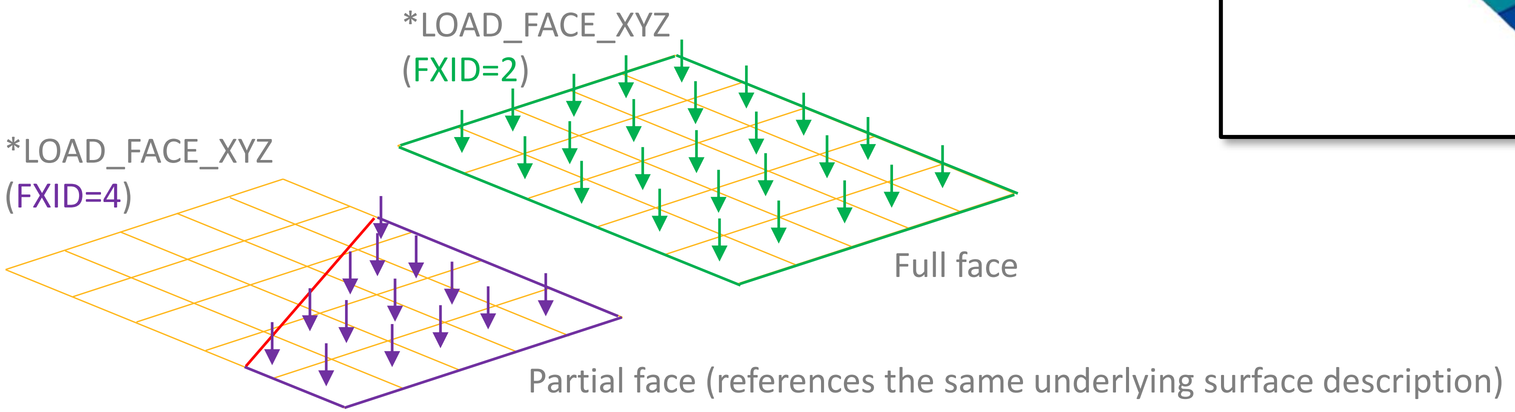


courtesy of Ford Motor Company

Feature-based boundary conditions and constraints

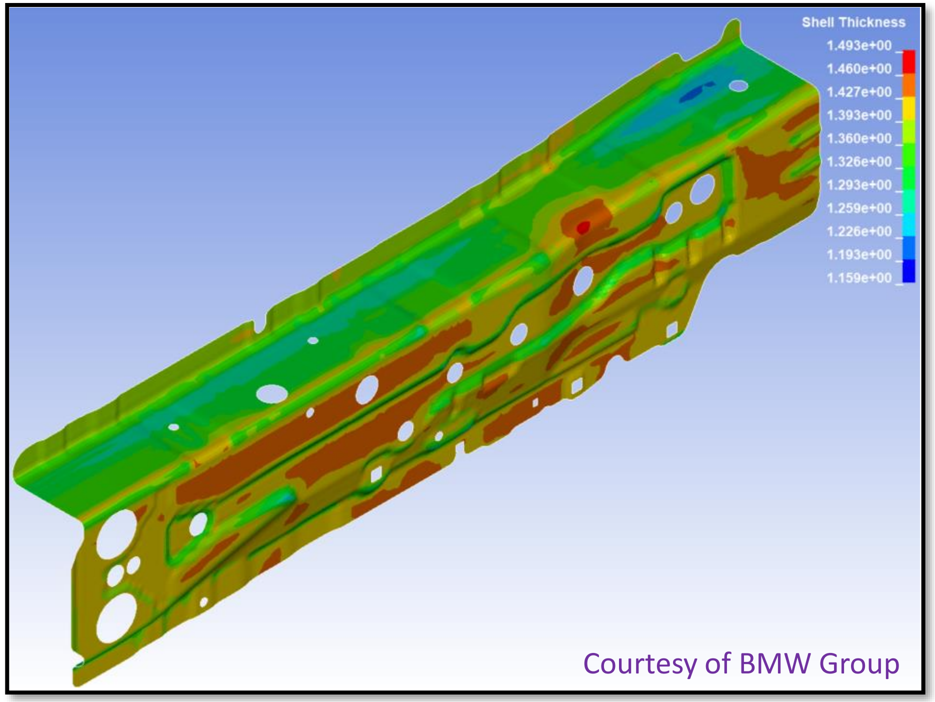
- BCs are imposed on geometric entities (points, edges, faces and volumes)
- Available keywords
  - *BOUNDARY_PRESCRIBED_MOTION_OPTION
    - Options: POINT_UVW, EDGE_UVW, FACE_XYZ
  - *LOAD_POINT_UVW(_SET)
    - Apply a point load at any location within a patch (not restricted to finite element nodes)
  - *LOAD_FACE_XYZ(_SET)
    - Apply a uniform pressure load on a selected subface of a surface





Mapping/Initializing

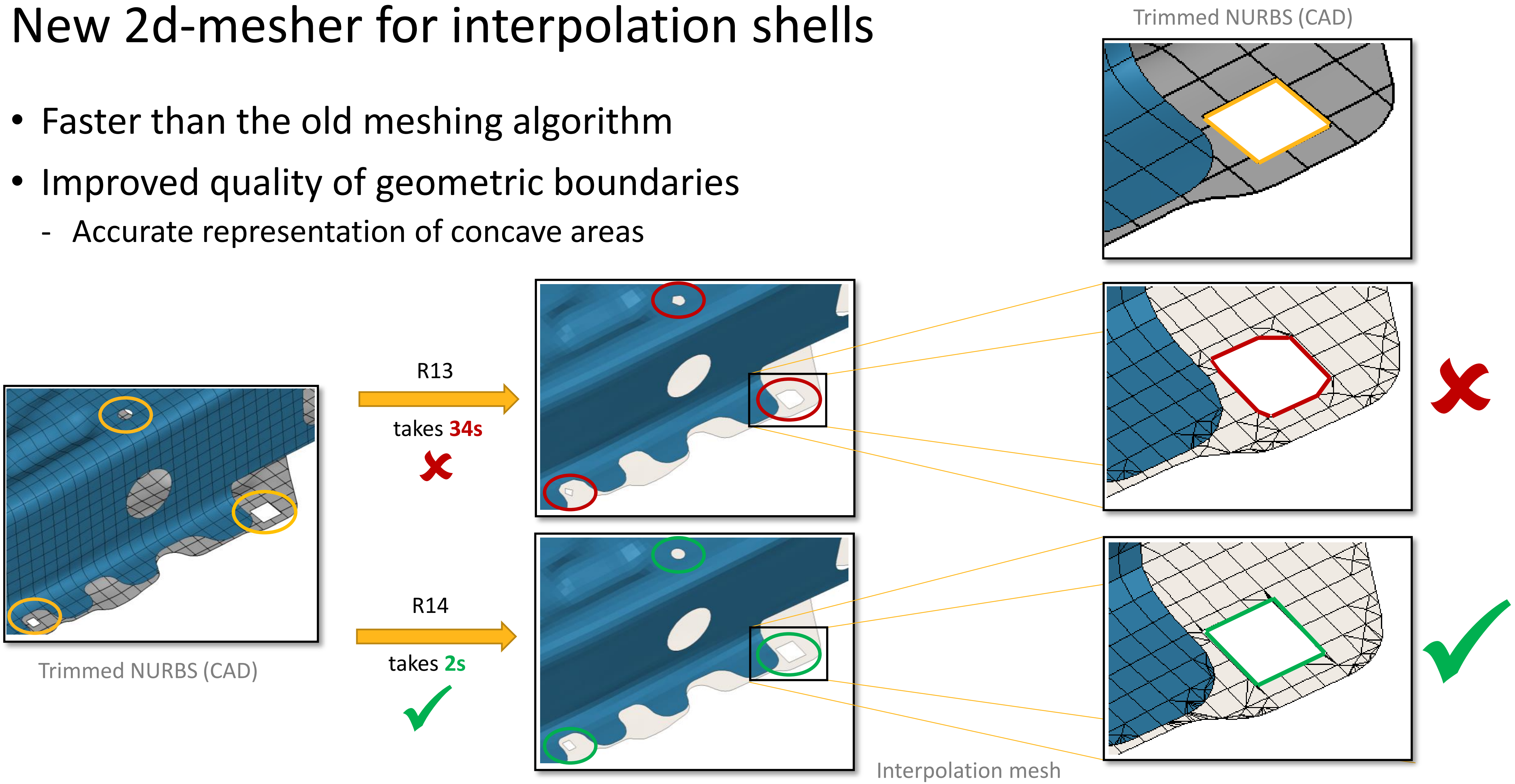
- Support multistage analysis (i.e. stamping) via dynain-file
- Keyword *INITIAL_STRESS/STRAIN_IGA_SHELL
- Allows the initialization of the following quantities at integration points:
  - Shell thickness
  - Initial stresses
  - Initial strains
  - Initial plastic strains
  - History variables



Shell thickness mapped via dynain-file using ENVYO

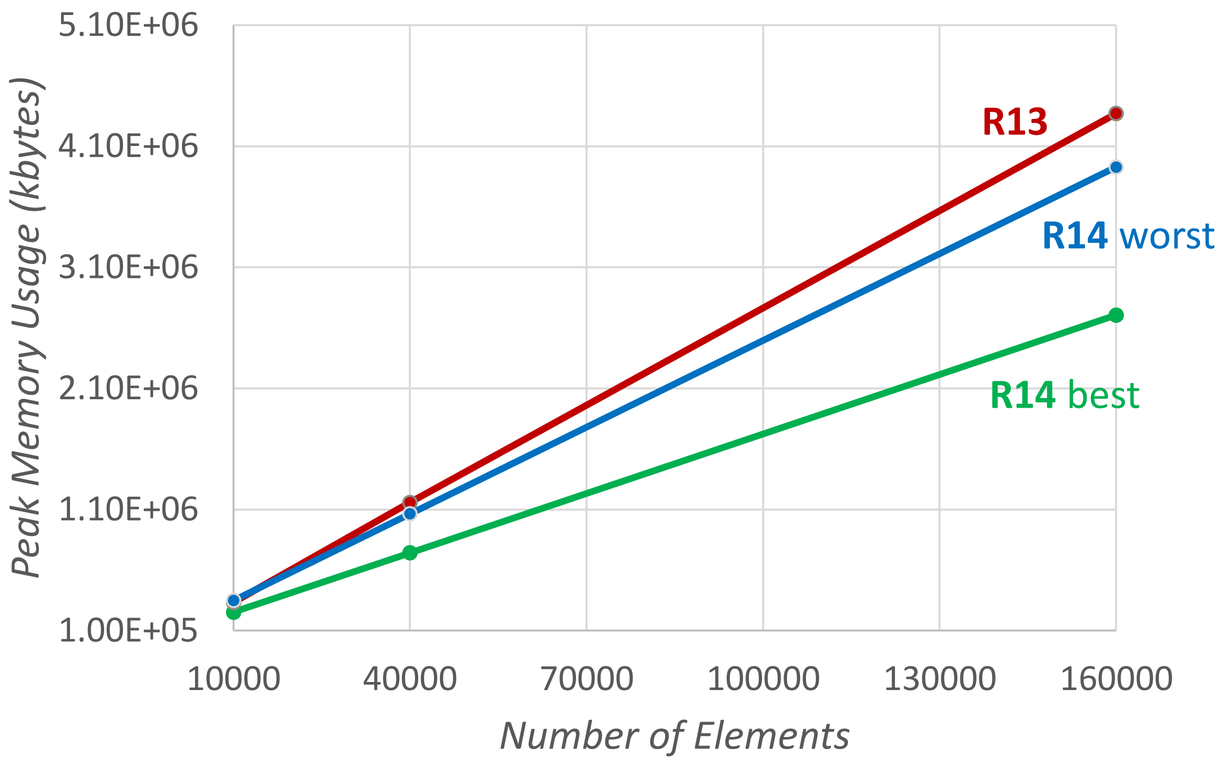
New 2d-mesher for interpolation shells

- Faster than the old meshing algorithm
- Improved quality of geometric boundaries
  - Accurate representation of concave areas



Restructured memory

- Complete restructuring of data structure and memory scheme
  - Reduce utilized memory, speed up computation *without* compromising on accuracy

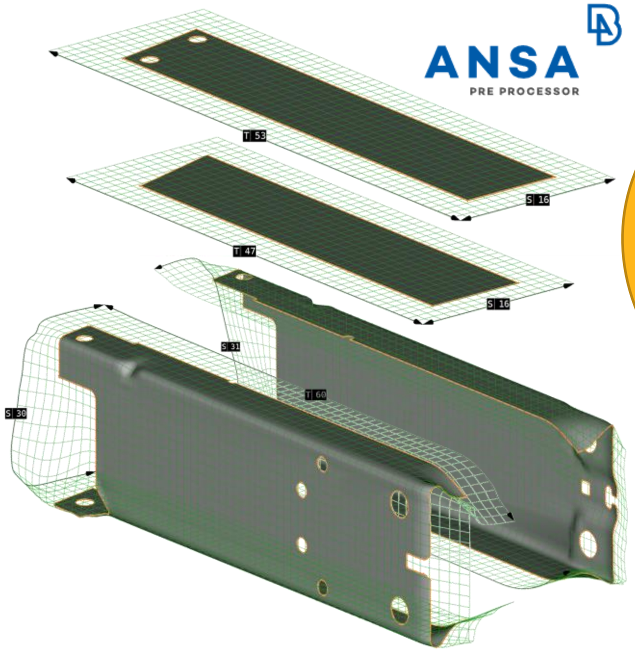


- Improvement is model dependent
- R14 saves up to 40% in memory and about 20% in computation time in comparison to R13

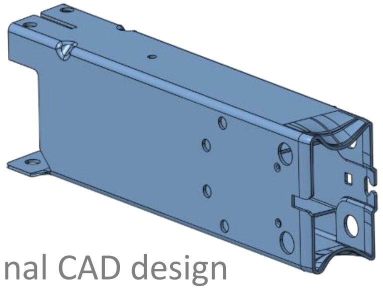
Mechanical Coupling

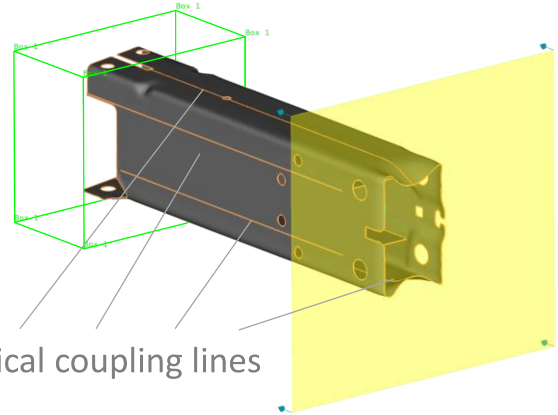
- Keyword *IGA_TIED_EDGE_TO_EDGE
- Improve domain decomposition for MPP speedup
- Improve estimate for penalty stiffness

Version	#of cores	Elapsed time	improvement
R13 (mpp, sp)	16	5min, 21s (321s)	-
R14 (mpp, sp)	16	3min, 11s (191s)	**40%**

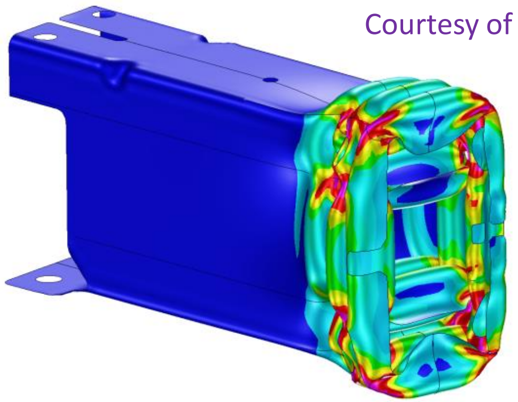


4 patches (using ANSA)


Original CAD design

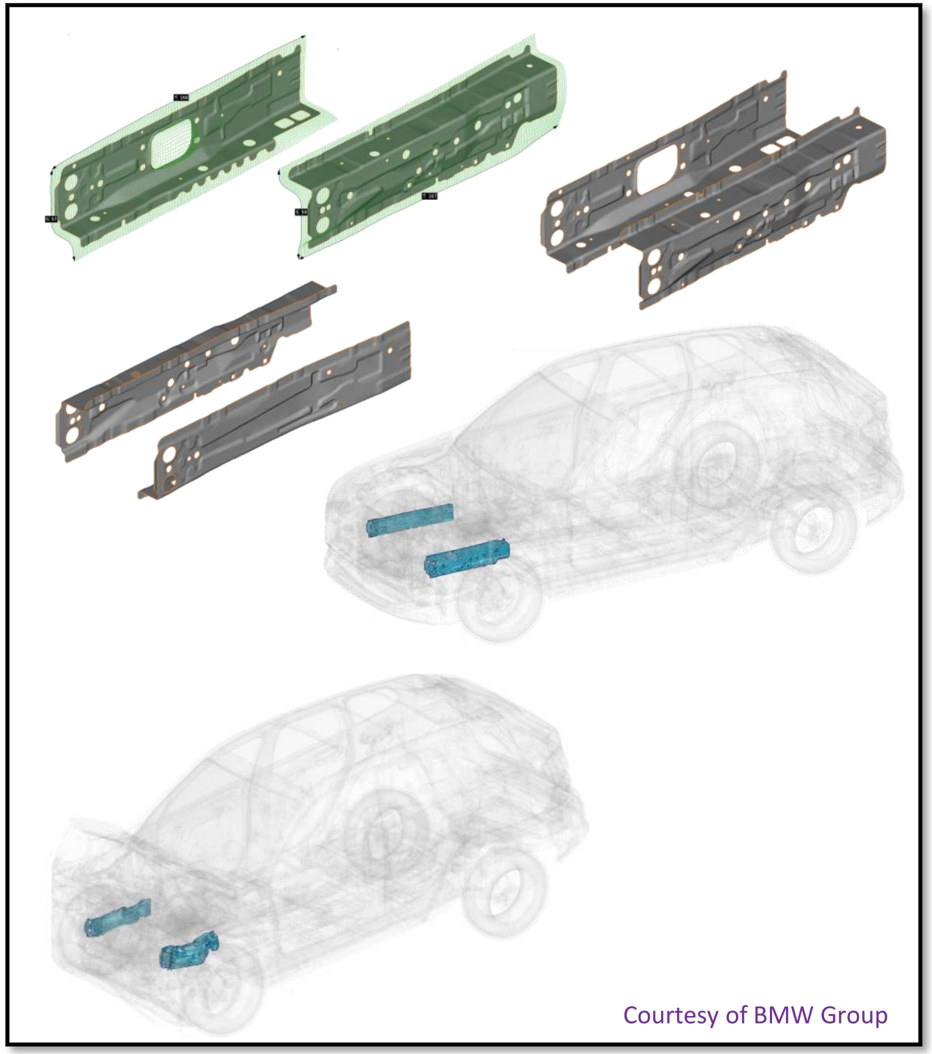

mechanical coupling lines

Courtesy of BMW Group


Analysis with LS-DYNA

Miscellaneous

- New timestep estimate (IGADO=1, *CONTROL_TIMESTEP)
  - May result in significantly (>50%) larger stable timestep!
- Allow the use of *IGA_SHELL elements as rigid bodies
- Support for *INITIAL_VELOCITY_GENERATION
- Enable *MAT_ADD_DAMAGE_DIEM/GISSMO
- Enable definition of HAZ (heat affected zone)
- Enable various connection modeling techniques to allow for hybrid models (standard FE and IGA)
  - Spotwelds (SPR3), Bolts, CNRBs
- Enable various implicit penalty contacts (via interpolation elements) for MPP
  - *CONTACT_xxx_MORTAR
  - *CONTACT_TIED_SHELL_EDGE_TO_SURFACE(_BEAM)_OFFSET
  - *CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET
- Various improvements for nonlinear implicit analysis

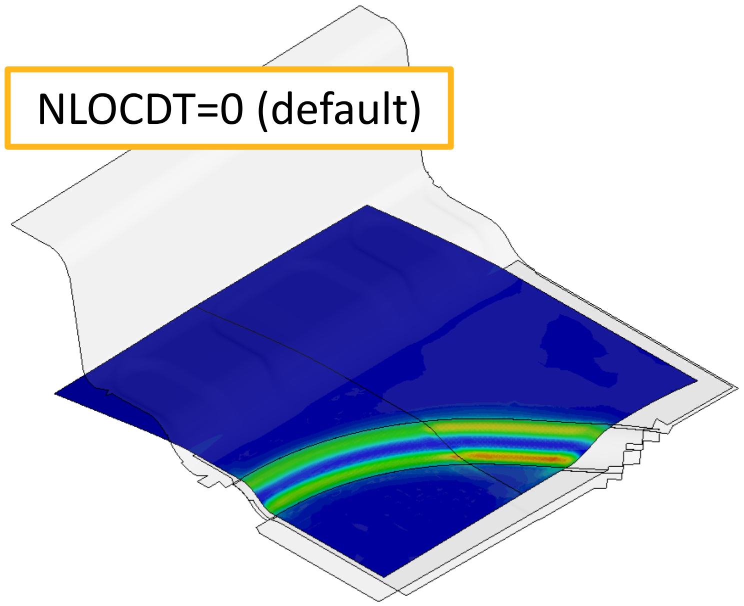


Hybrid assembly – Full vehicle front crash

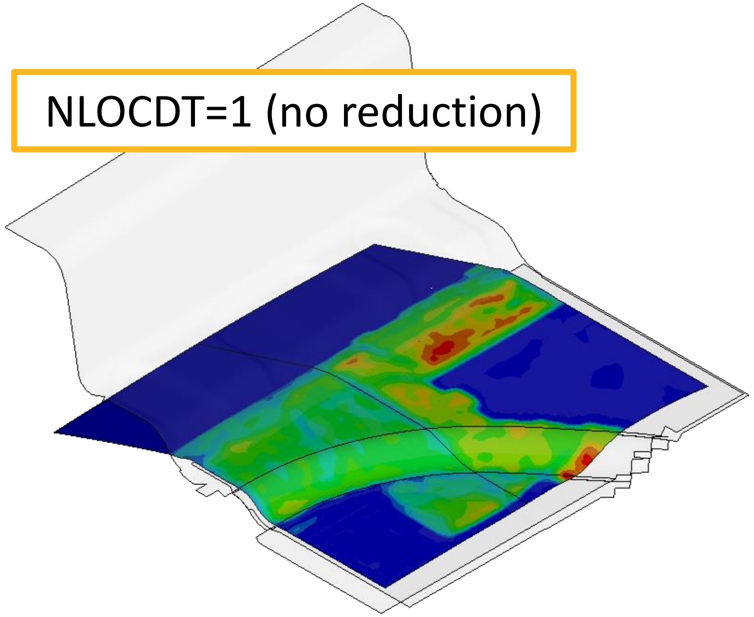
Elements

Shells with mid-plane offset

- New option NLOCDT for shells reduces time step size for stability
  - Active by default if NLOC (*SECTION_SHELL) or OFFSET (*ELEMENT_SHELL) is applied
  - Maximum reduction of 10% for large ratios of thickness to element length
  - NLOCDT=1 allows reproduction of old results



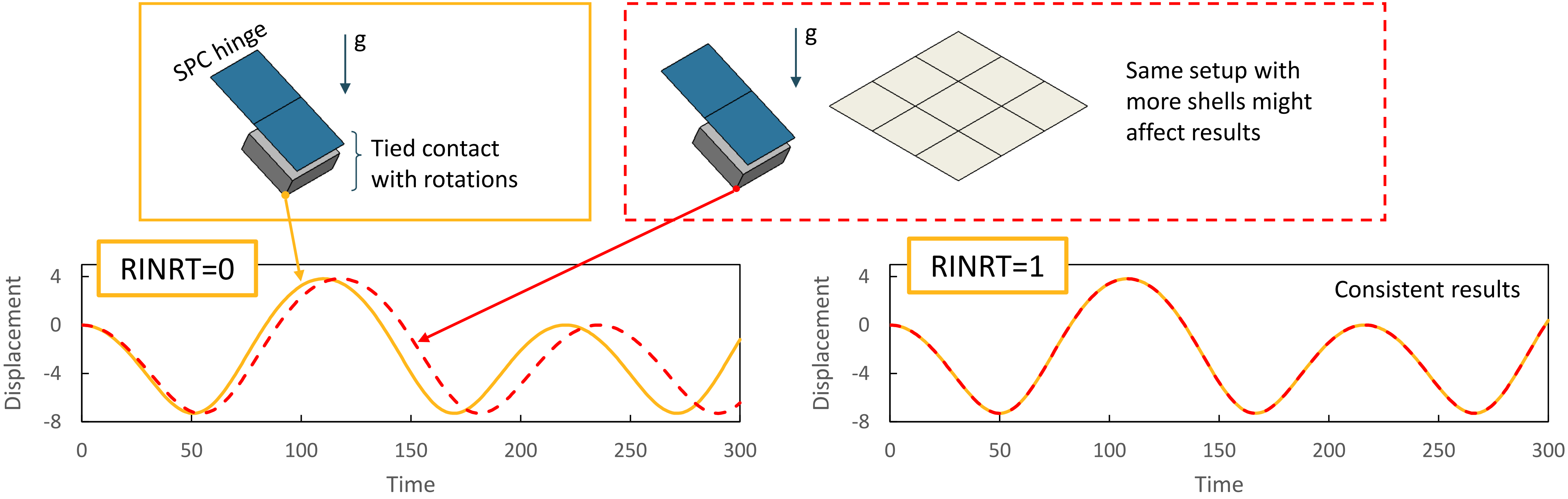
Realistic plastic strains



Spurious plastic strains due to instability

Solid elements in constraints with rotations

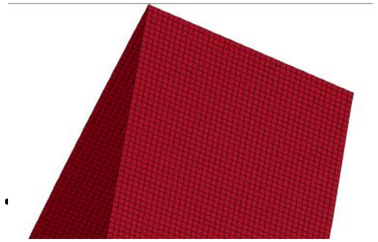
- New option RINRT for solids (*CONTROL_SOLID)
  - Compute rotary inertia for each solid to ensure consistent results
  - Otherwise, an average of rotary inertia from existing beams and shells is taken



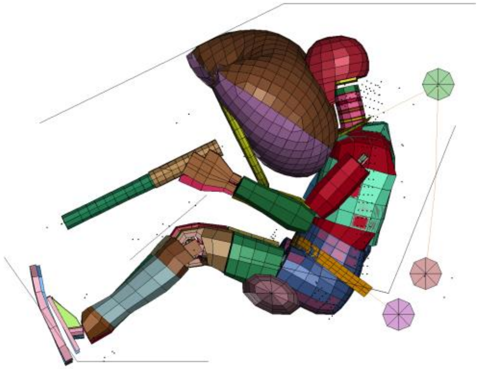
Seatbelt

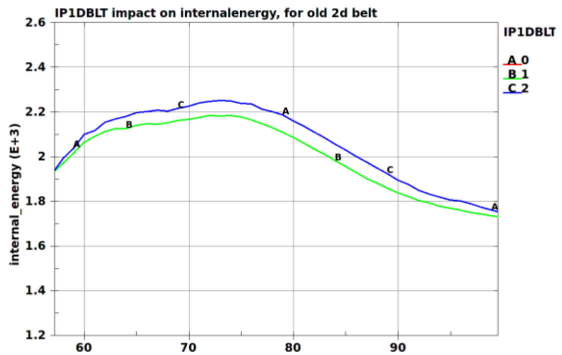
Ansys

SEATBELT



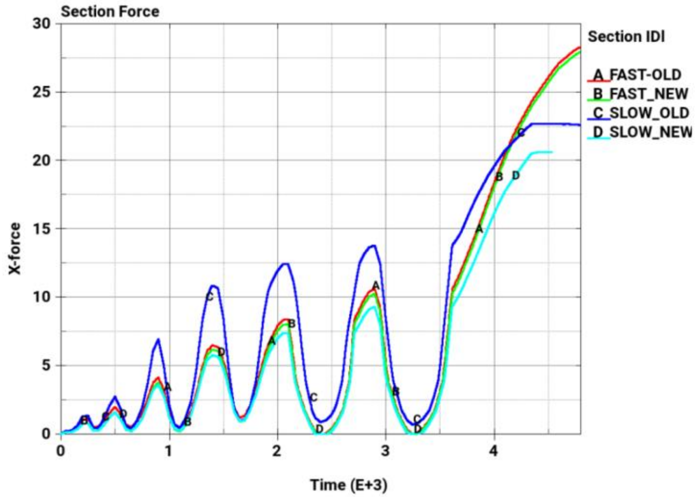
- allow edge node set of 2D belt, EDGSET of *SECTION_SHELL, to have up to 60 nodes.
- add option of IP1DBLT=2 for 2D belt, see *CONTROL_OUTPUT, for more consistent MPP results.





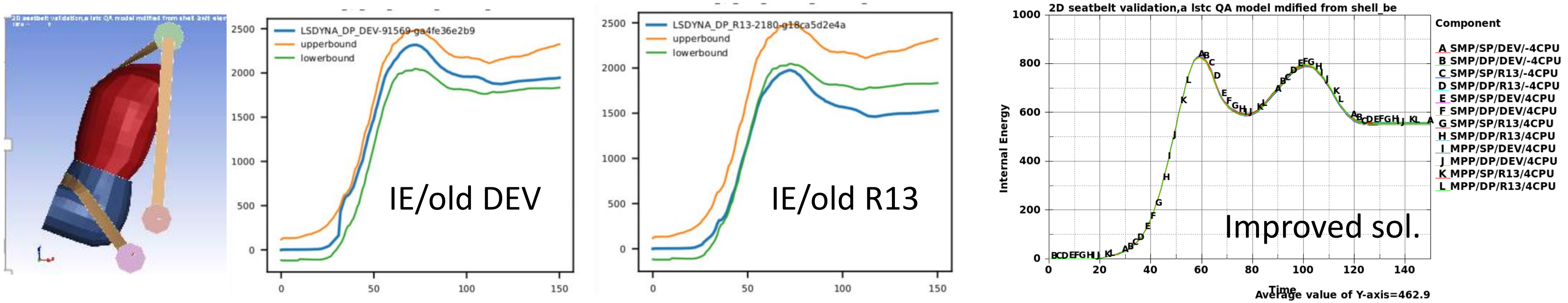
- fix a bug for belt that happened when *MAT_SEATBELT has loading or unloading curves defined in terms of table and the related strain rates are exponentially increasing.

*DEFINE_TABLE_2D
100
1.e-3, 101
1.e-2, 102
1.e-1, 103



SEATBELT

- zero-out inter energy of 2d belt inside the retractor. Such energy, even very minimal, could be annoying and confuses users.



- fix a bug that happened when the edge node set is defined using *SET_NODE_GENERAL and subject to *NODE_TRANSFORM
- add I.E. of 1D belt derived from 2D belt to the I.E. of the 2D belt it is derived from.
- fix a bug that, when ip1dblt=0, mistakenly turned off the output of regular 1D belt in SBTOUT

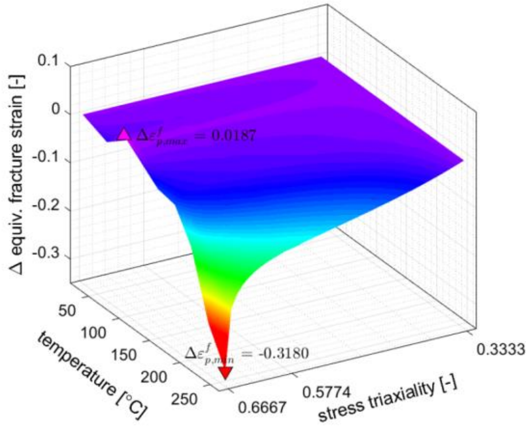
Ansys

Materials

Updates for *MAT_ADD_DAMAGE_GISSMO

- Properties depending on more and more variables
  - Failure/critical strain as function of plastic strain rate, temperature, Lode parameter, and triaxiality
  - Regularization factor as function of Lode parameter, triaxiality, and element size
  - Fading exponent as function of element size, triaxiality, and Lode parameter
  - Analytical failure strain, i.e. LCSDG<0 refers to *DEFINE_FUNCTION,
    got new arguments: plastic strain rate, temperature, history, element size.

  ⟶ **Improved failure prediction for large variety of applications**



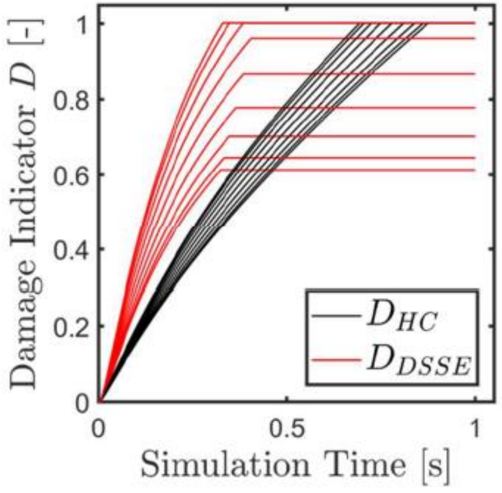
- Added new flag INSTF for instability treatment
  - This flag governs the behavior of instability measure, F , and fading exponent, FADEXP

  ⟶ **Better agreement with experimental data in post-necking behavior under various stress states**

Enhancements for generalized damage model

- Keyword *MAT_ADD_GENERALIZED_DAMAGE aka "eGISSMO"

- Domain of Shell-to-Solid Equivalence (DSSE) for shell elements
  - IFLG3=2: special model by Pack & Mohr (2017) for necking under bending



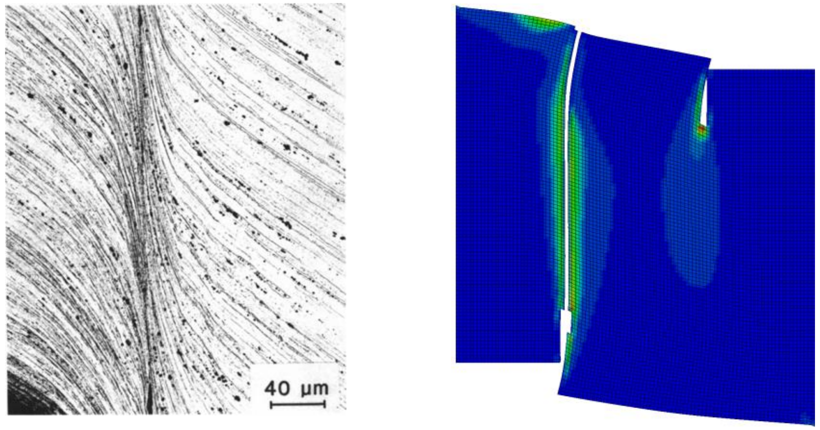
- Total strains as damage drivers (IFLG1=3)
  - This could be interesting for materials without plasticity

- Improvement for cyclic loading if damage driver drops now and then
  - New option IFLG4=1 prevents undesired damage evolution

- More solid material models supported
  - *MAT_058 (composites), *MAT_133 , *MAT_199, *MAT_233 (rolled/extruded metals)

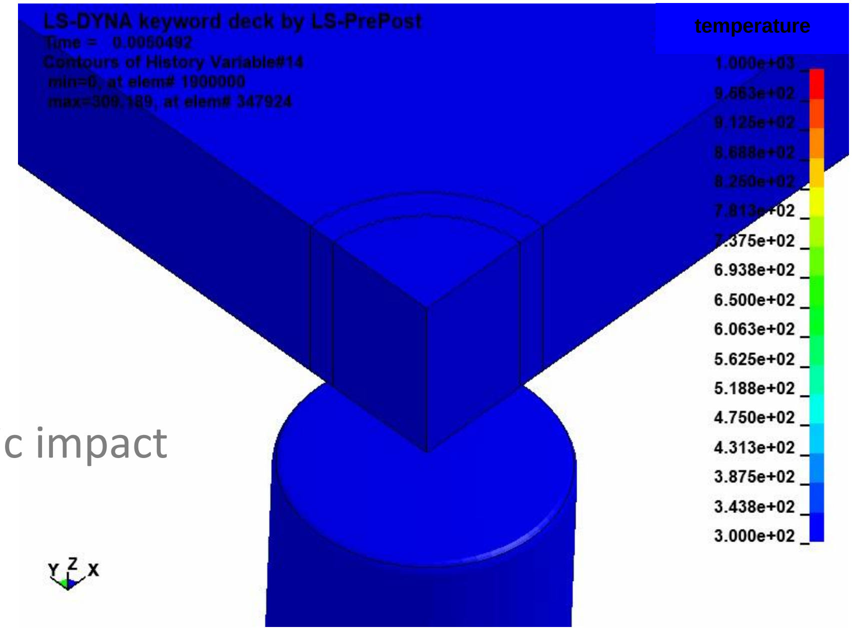
Adiabatic shear bands (ASB) in thick ductile metals

- New option for *MAT_TABULATED_JOHNSON_COOK (224)
  - BFLG=1: dissipation factor β (aka “Taylor-Quinney coefficient”) can now be a function of maximum shear strain, strain rate, and element size using a TABLE_3D
  - This allows simulating ASB initiation (thermal softening) using meshes with element sizes relevant to practical aerospace applications
  - Based on PhD research by S. Dolci (GMU) for AWG



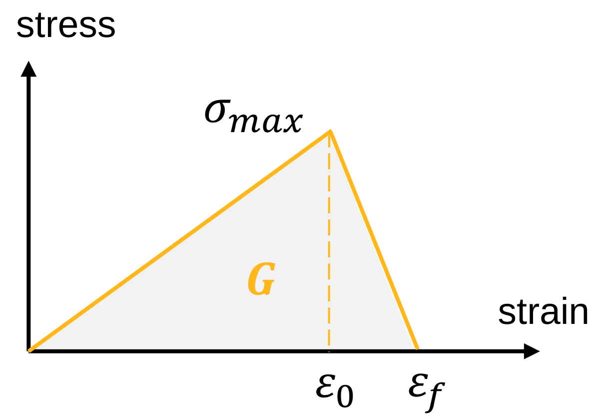
ASB: concentrated shear deformation

ballistic impact

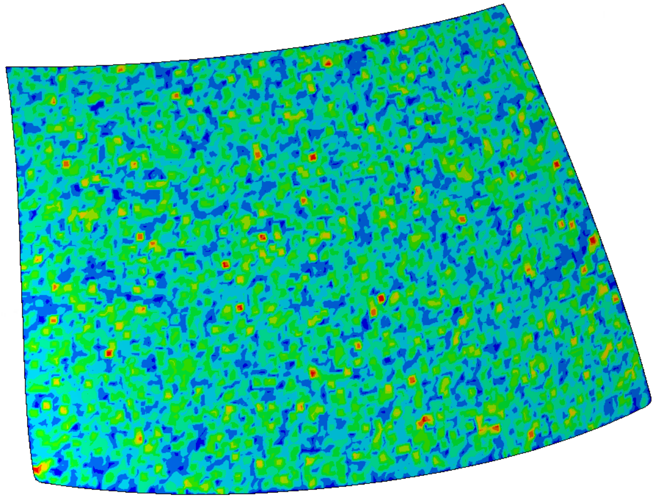


Glass model enhancements (*MAT_280)

- Optional damage model invoked by input of fracture energy
  - Orthotropic damage model with linear softening governed by crack opening strain
  - This can replace the existing approach of stress reduction over a few cycles



- Spatially varying distribution of properties
  - Scale factor for FT (tensile strength) on history variable #13 can be defined per element with *INITIAL_STRESS_SHELL
  - … or as automatically generated distribution by the new keyword option _STOCHASTIC (needs *DEFINE_STOCHASTIC_VARIATION)

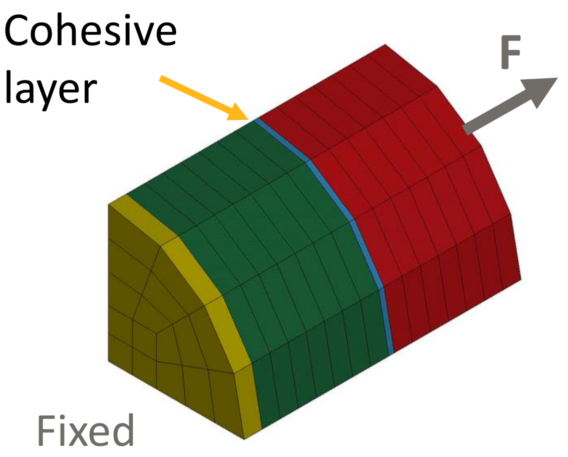


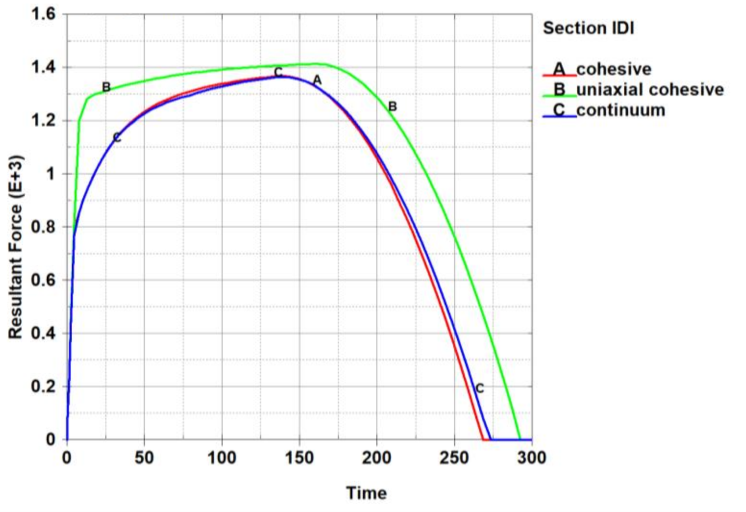
e.g. windshield with inhomogeneous defects

Enhancements for cohesive elements / materials

- New option UNIAX for *MAT_ADD_COHESIVE
  - Eliminates unused ("parasitic") stress components σ_{xx}, σ_{yy} and σ_{xy}, which result from most constitutive models due to Poisson's effect
  - Minimizes the effect of kinematic constraints in the cohesive zone

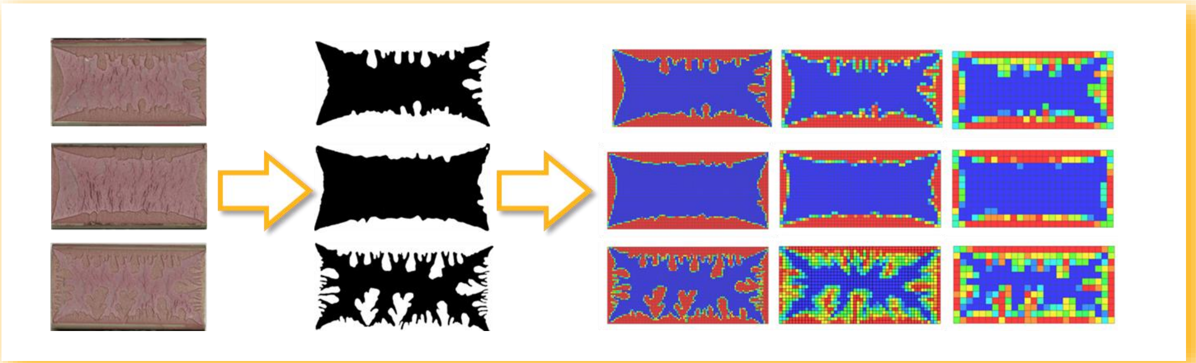
$$\begin{bmatrix} \dot{\varepsilon}_{xx} \\ \dot{\varepsilon}_{yy} \\ \dot{\varepsilon}_{zz} \\ \dot{\varepsilon}_{xy} \\ \dot{\varepsilon}_{yz} \\ \dot{\varepsilon}_{zx} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ \dot{\delta}_3/(t+\delta_3) \\ 0 \\ \dot{\delta}_2/(t+\delta_3) \\ \dot{\delta}_1/(t+\delta_3) \end{bmatrix} \xrightarrow{\text{Material model}} \begin{bmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{zx} \end{bmatrix} \xrightarrow{\text{*MAC}} \begin{bmatrix} t_1 \\ t_2 \\ t_3 \end{bmatrix} = \begin{bmatrix} \sigma_{zx} \\ \sigma_{yz} \\ \sigma_{zz} \end{bmatrix}$$

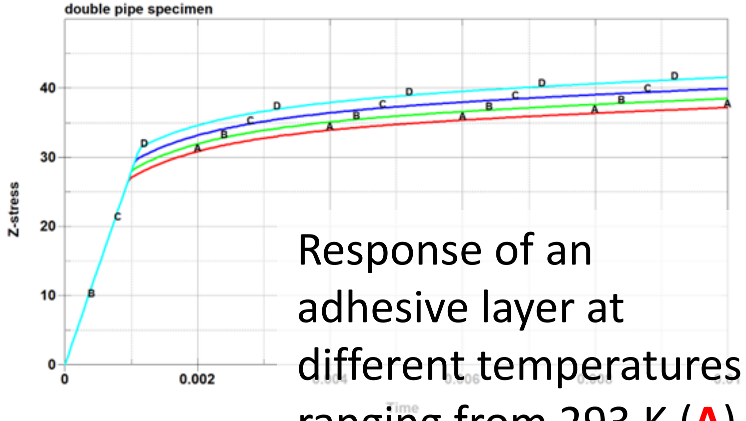


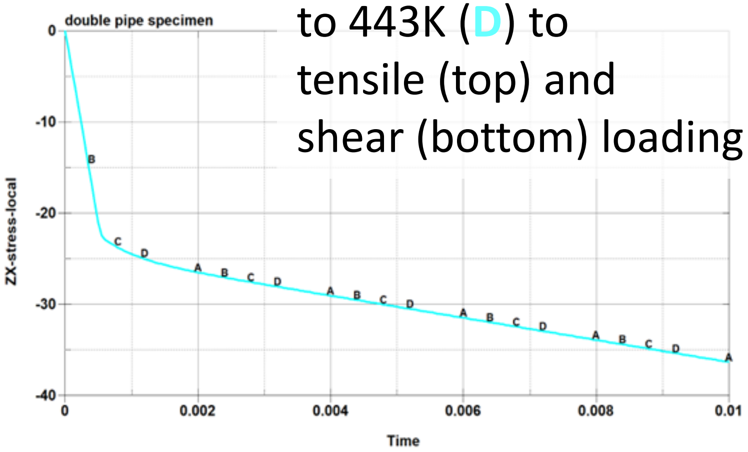


New options in *MAT_307 for thermoset adhesives

- Distortional hardening with respect to temperature variations
  - Temperature dependence for shape of yield surface (non associated $I_1 - J_2$ plasticity of TAPO model)
  - Yield strength depends on temperature and degree of cure
- Differentiation of damage mechanisms
  - Material damage mechanism based on TAPO model
  - Phenomenological approach for (pre-)damage due to viscous fingering in the liquid phase of the adhesive







Response of an adhesive layer at different temperatures ranging from 293 K (A) to 443K (D) to tensile (top) and shear (bottom) loading

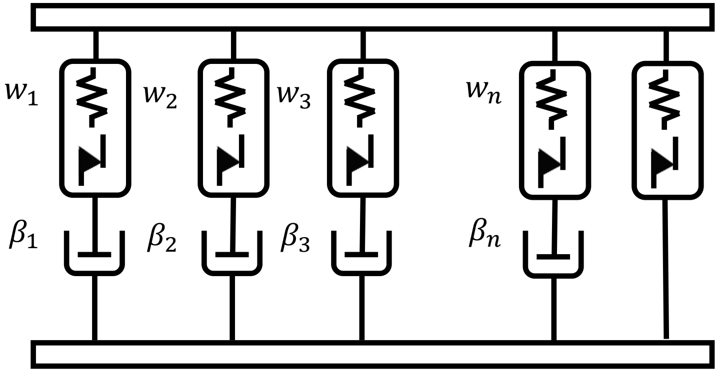
Miscellaneous material model enhancements

- *MAT_TABULATED_JOHNSON_COOK (MAT_224)
  - Thermal efficiency parameter BETA now supersedes the general data FWORK in the thermal solver in a coupled simulation
  - Allows to account for different thermal efficiencies in one model

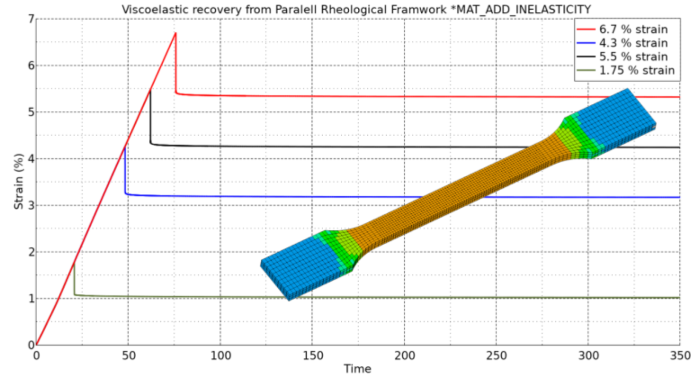
- *MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE (240)
  - New strain rate filtering method simplifies data calibration
  - User-defined exponent for the mixed mode failure criterion

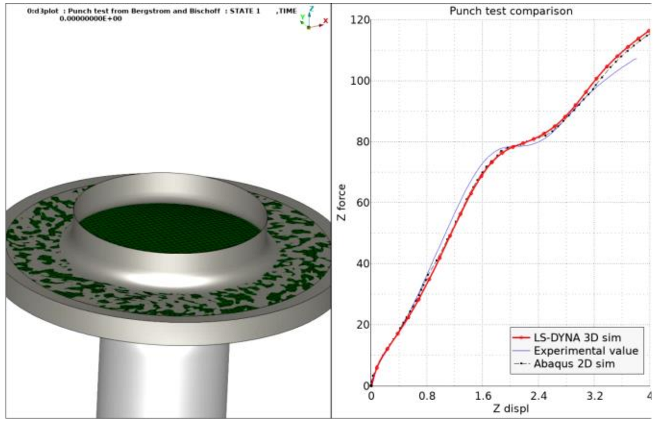
- *MAT_NON_QUADRATIC_FAILURE (258)
  - Set hardening variables R_i according to initial plastic strain
  - Speed-up of about 30% if exponents A and GAMMA are integer values

Nonlinear Viscoelasticity (Creep)



- *MAT_ADD_INELASTICITY supplemented with nonlinear viscoelastic laws, efficient variants of the creep laws
  - The relaxation coefficients in Prony series can depend on stress and strain to effectively support the Norton-Bailey and Bergström-Boyce creep laws
  - Bengzon F., Borrvall T., Jonsson A. and Lindvall M., *Using MAT_ADD_INELASTICITY for Modelling of Polymeric Networks*, 13th European LS-DYNA Conference 2021, Ulm, Germany, https://dynalook.com
- The Three Network Model (*MAT_TNM_POLYMER) available for explicit and implicit analysis
  - Model for thermoplastics https://polymerfem.com/three-network-model
  - Two viscous links with interdependence, and one elastic link
  - J.S Bergström and J.E Bischoff, *An Advanced Thermomechanical Constitutive Model for UHMWPE*, IJSCS, Volume 2, Number 1, April 2010, pp 31-39

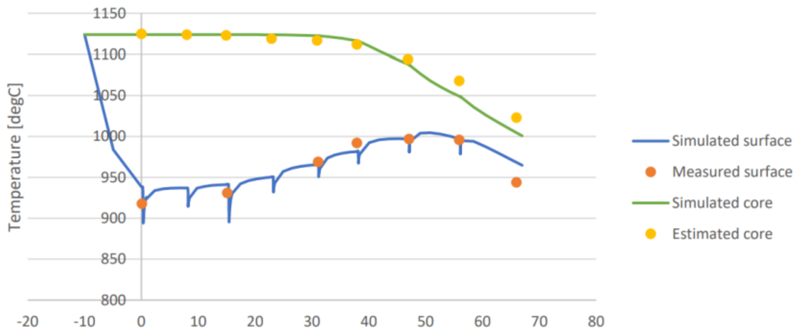


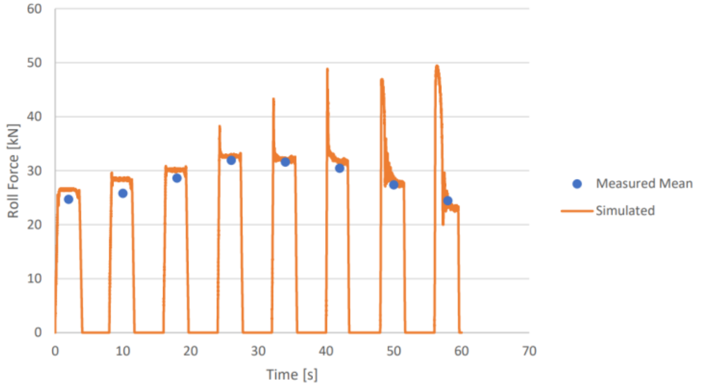


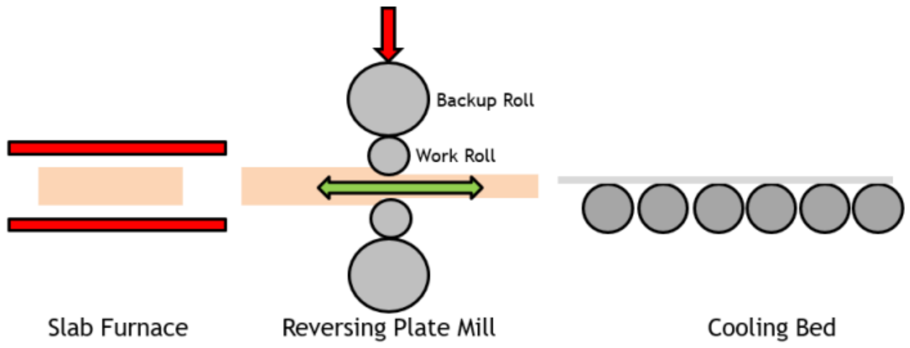
Ansys

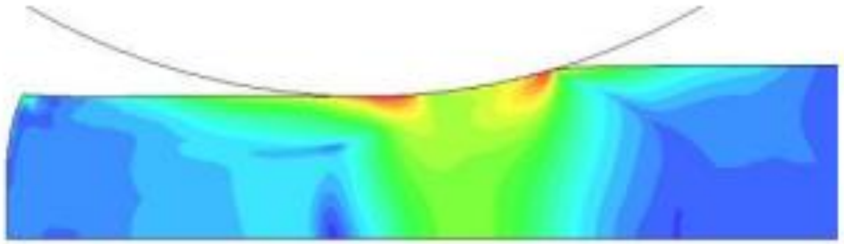
Hot Forming and Thermoplasticity

- MAT_HOT_PLATE_ROLLING (*MAT_305)
- Thermoelastoplastic material for hot rolling
  - Features: work hardening, dynamic softening, static recovery, and static recrystallization
  - Input parameters: calibrated from Gleeble tests at various deformation rates and temperatures
  - Developed in cooperation with Swedish steel industry to create virtual process lines for working and heat treatment processes
  - Schill M., Karlsson J., Magnusson H., et al., *Simulation of Hot Plate Rolling using LS-DYNA*, 13th European LS-DYNA Conference 2021, Ulm, Germany, https://dynalook.com





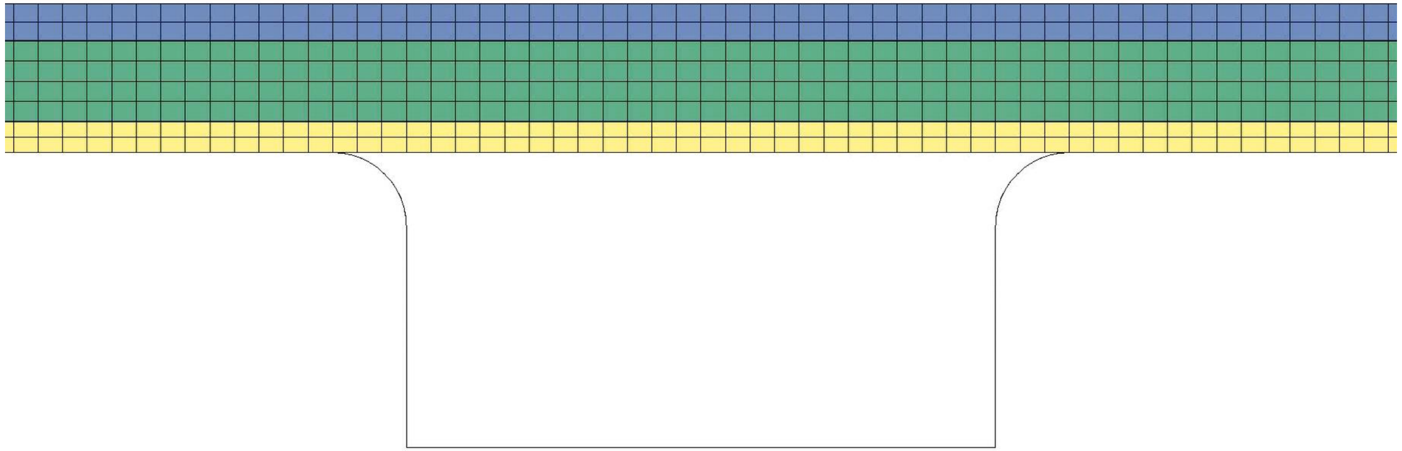


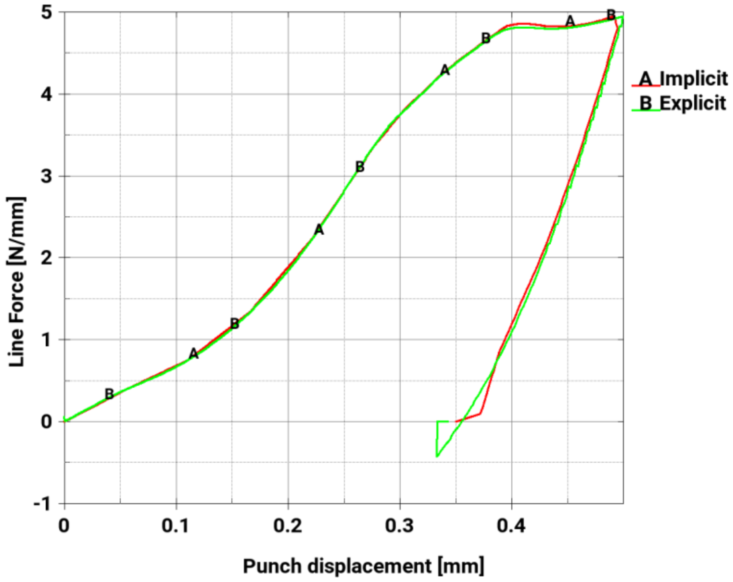


Paper and Delamination

- All cohesive materials (138/169/184/185/186/240/252/279/326) now supports element erosion for Newton-Cotes integration (INTFAIL < 0).
- *MAT_PAPER is now supported for implicit (shell and solid elements).

Creasing of paperboard





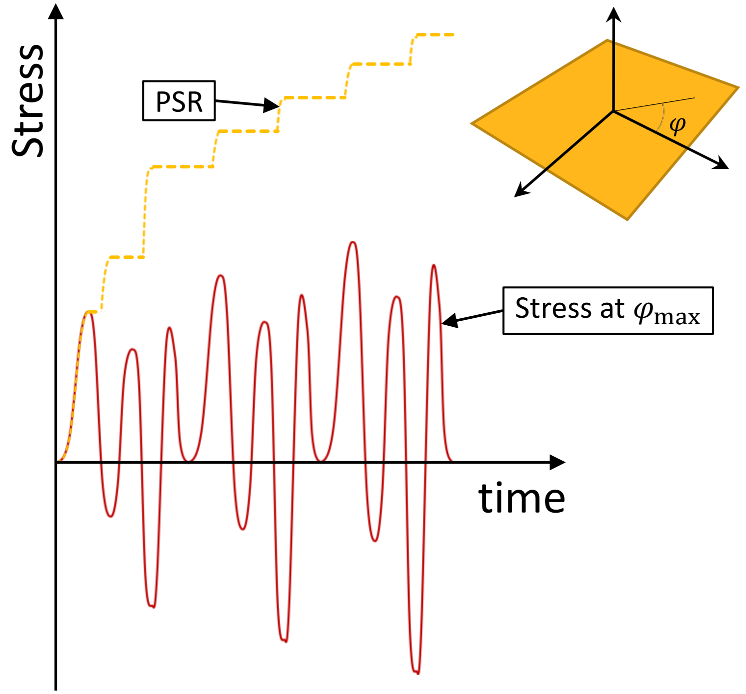
- Implicit runtime >25 times faster than explicit.

Material History Output

- Added features in *DEFINE_MATERIAL_HISTORIES:
  - *Effective Tresca Stress* (maximum shear stress).
  - *Principal Stress Range* - Compute the maximum (principal) stress amplitude. Intended as a scanning tool for fatigue analysis.
  - Standard operators (addition, multiplication, max, min, etc.) to enable user expressions on material histories.

- Output of history variable names (HISNOUT > 0):
  - Added support for more materials (including all EOS).
  - New name-mapping file (HISNOUT = 3), which supports more detailed info regarding, e.g., multi-material laminates and subsets of model.

Normal stress history and *Principal Stress Range* at angle of, over time maximum major principal stress, $\varphi_{\max}$



User Defined Features and Shared Object Files

- Info and compatibility checks
  - Prints information about loaded shared object files
  - Run-time compatibility checks between binary and shared object files
  - Prevents accidental use of incompatible shared object files
- Bug fixes
  - *MODULE_USE could not be used for user material numbers 1001 to 2000
  - Paths in *MODULE_PATH or LD_LOAD_LIBRARY_PATH did not work correctly
  - *MODULE_LOAD or the command line option "module=" could not be used with the default shared object filename
  - Users were barred from using subroutines umat42, umat48v, and umat49v

```
 Initial reading of file

                   ls-dyna mpp d DEV R14_491_c51a659f1

MODULE Loading :
 file: liblsdyna_s_R14-491-gc51a659f1d_sse2.so
 build: R14-491-gc51a659f1d
 version: smp
 precision: single
 NLQ:       112


 *** Error 21511 (STR+1511)
     *MODULE library file liblsdyna_s_R14-491-gc51a
     is incompatible with binary R14-491-gc51a659f1d
```

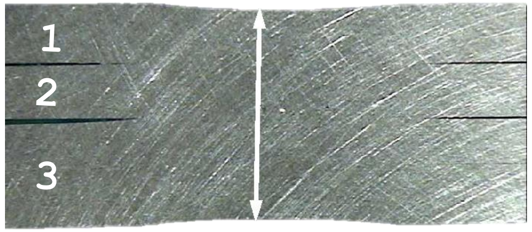
Connections & Contact

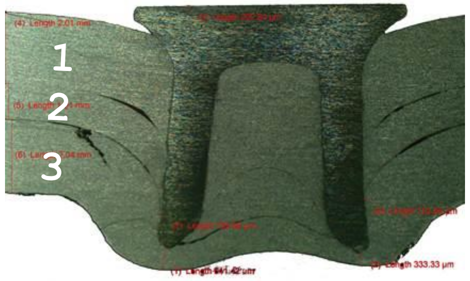
Ansys

Spot welds or rivets joining more than 2 flanges

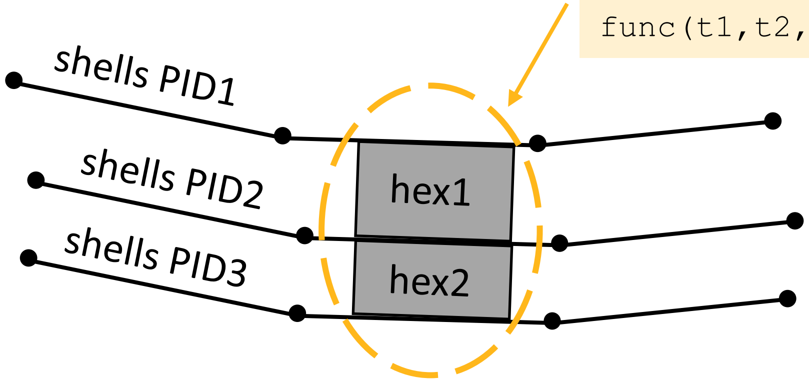
- New keyword *DEFINE_MULTI_SHEET_CONNECTORS
  - *n* sheets/panels connected by *n*-1 joining elements (current max. *n*=4)
  - Material and failure behavior of joining elements can be described based on geometric and material properties (thicknesses, yield stresses, etc.) of all *n* sheets involved
  - Better failure prediction through this information exchange
  - Currently available for single hex elements with *MAT_100_DA

3-sheet SW





3-sheet rivet

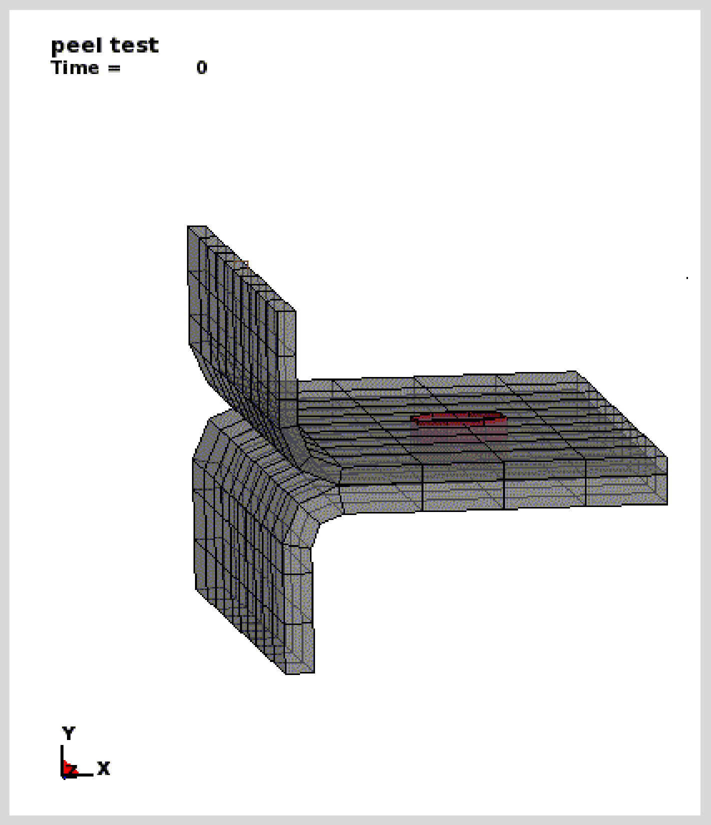



```
$                  D_SIGY
                      100
*DEFINE_FUNCTION
         100
func(t1,t2,t3,t4,sy1,sy2,sy3,sy4,...)=...
```

Updates for SPR3 connectors

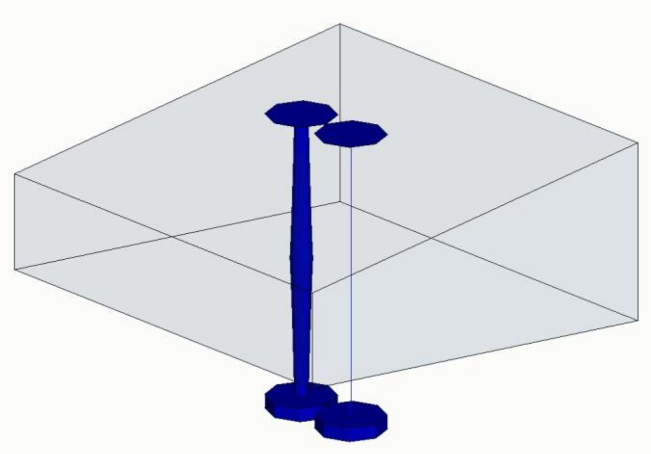
New options for *CONSTRAINED_INTERPOLATION_SPOTWELD aka "SPR3"

- Connection to thick sheets or volume components
  - Meets increased demand for using hex and tet elements
- Connection to in-plane composed parts, i.e., part sets
  - E.g. tailor welded blanks or other areas with different properties
- Introduction of "peel ratio"
  - Better load and failure prediction in bending-dominated cases
- Simplified scaling of properties
  - Modify strengths, but keep shape of load-displacement curve



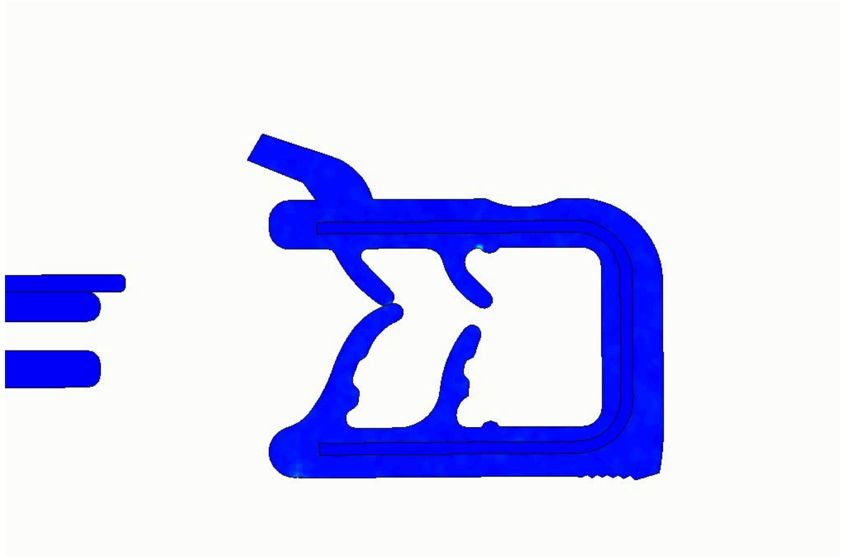
Preloading Bolts

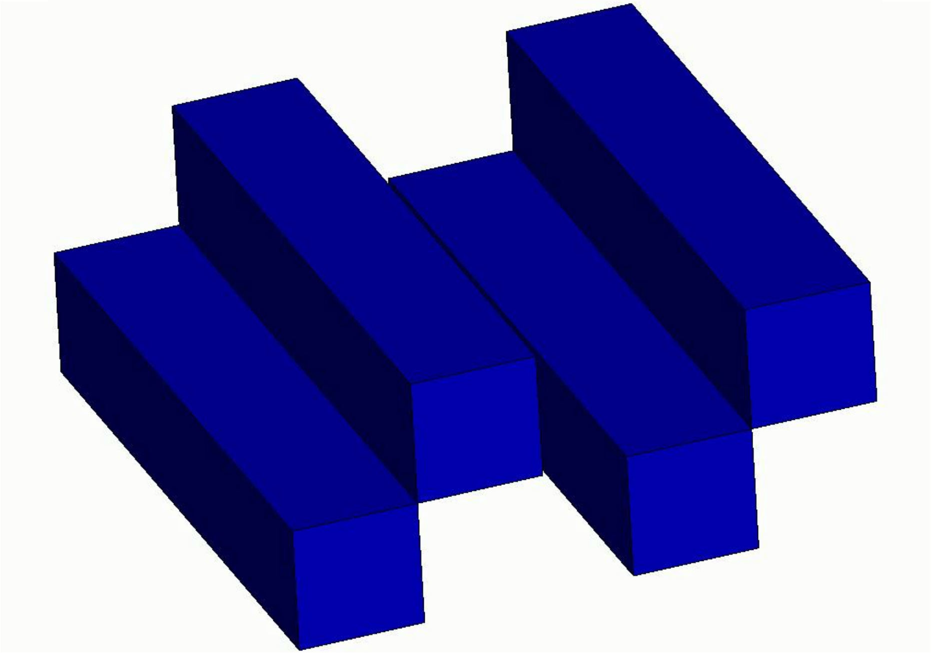
- IZSHEAR=2 for solid element bolts has been extended to KBEND=2 for beam element bolts
  - Bending resistance invoked to protect the structural integrity of the bolt
  - The prescribed force is distributed over all specified beams to avoid special purpose modelling techniques
    - Instead of prescribing one “long” beam element the force can be applied to all beams along the shank without introducing singularities
  - The contraction rate of beams has an upper limit to avoid dynamic effects as bolt heads may otherwise impact plates with arbitrary velocity. This applies to both solid (IZSHEAR=2) and beam (KBEND=2) element bolts.



Mortar Contact Developments

- Orthotropic friction in 3D Mortar Contact supports load curves for defining friction coefficients as functions of contact pressure and sliding velocity
- 2D Mortar Contact is supported in MPP as well as multistage analysis (dynain.lsda)
- Automatic 3D Mortar Contact supports “look-ahead” mesh adaptivity, meaning that elements on blank are refined as tools with sufficient curvature approaches
- 3D Mortar Tied Contact is now supported for full deck restarts and redecomposition
- Beam materials 66, 67 and 119 (discrete beam materials) are supported with 3D Mortar Contact
- TIME is introduced to Mortar Tied Weld Contact, meaning that welding can only occur if conditions are fulfilled for TIME consecutive time units, this to prevent “premature” welding situations with bad deformations as a result, principle shown to the right

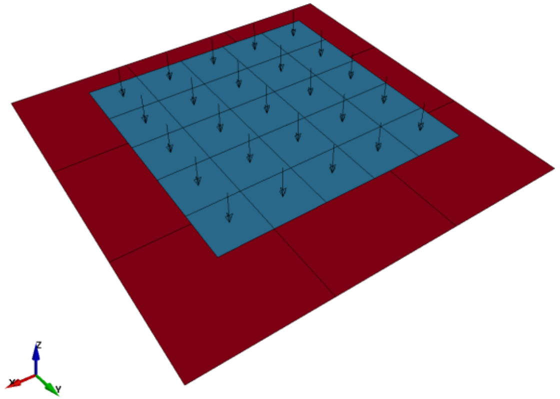




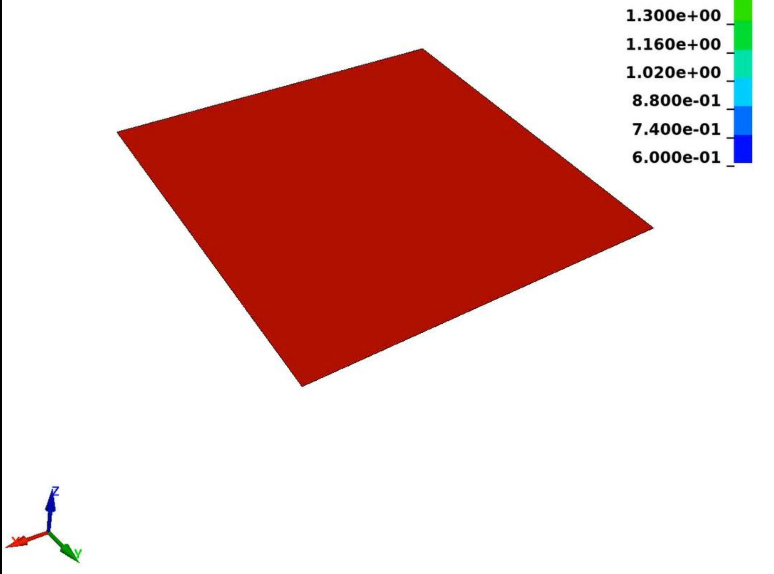
Hydroforming

- Pressure load from *LOAD_MASK/SEGMENT/SHELL can now act as surface boundary condition on shell elements with thickness stretch (ELFORM = 25-27 or IDOF = 3)

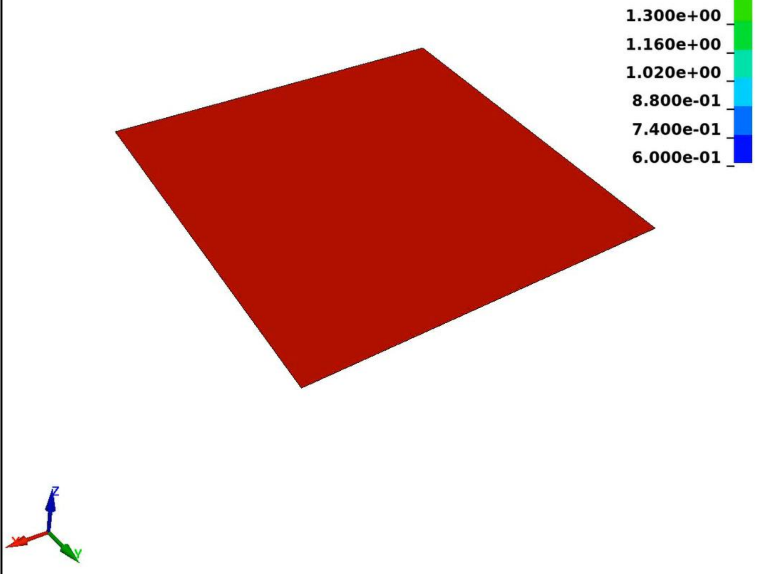
Pressure load on top and contact with support on bottom of blank.



Before:
Only contact pressure affected shell



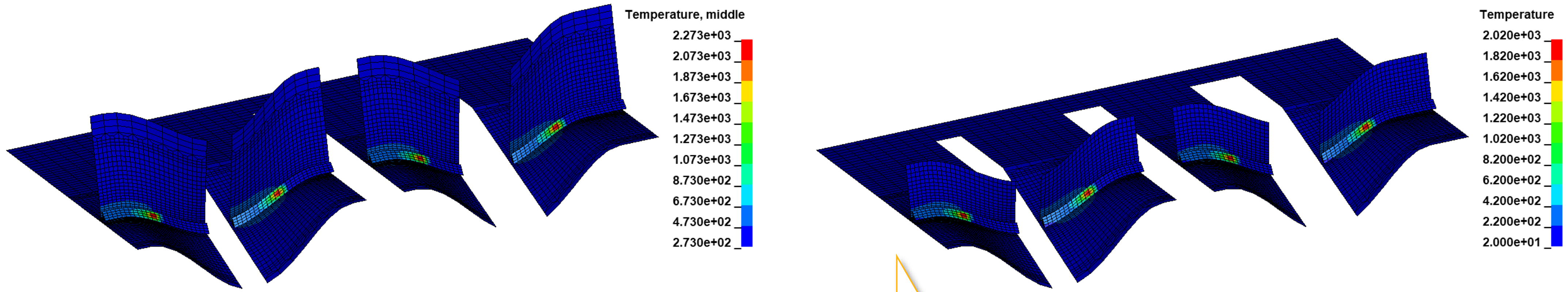
Update:
Both contact and load pressure affect shell



Thermal

New option for data transfer in one-way coupled simulations

- Offset of temperature results when used as thermal loading
  - New parameter TMPOFF in *LOAD_THERMAL_BINOUT
  - Enables the switch of temperature scales, such that results from a thermal-only simulation in the Kelvin scale can be applied in a structure-only simulation in the Celsius scale



Thermal-only welding simulation with temperatures in Kelvin scale

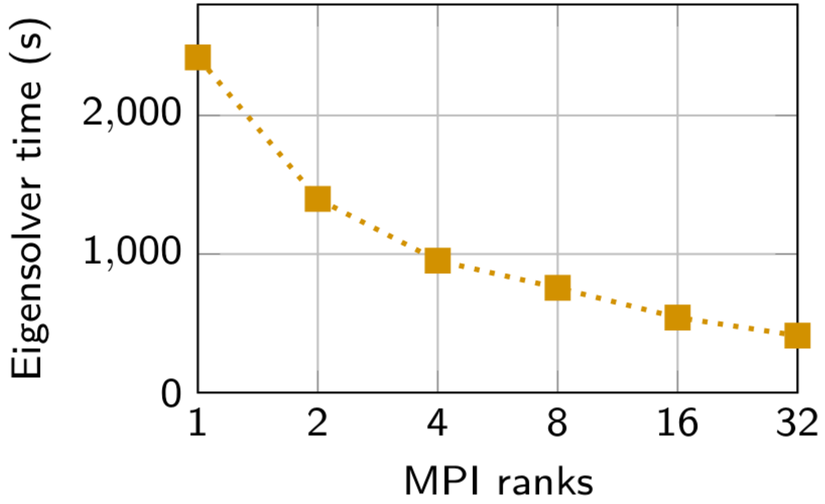
*LOAD_THERMAL_BINOUT (TMPOFF=-273)

Structure-only welding simulation with temperatures in Celsius scale and an adapted mesh

Implicit

MPP LOBPCG eigensolver

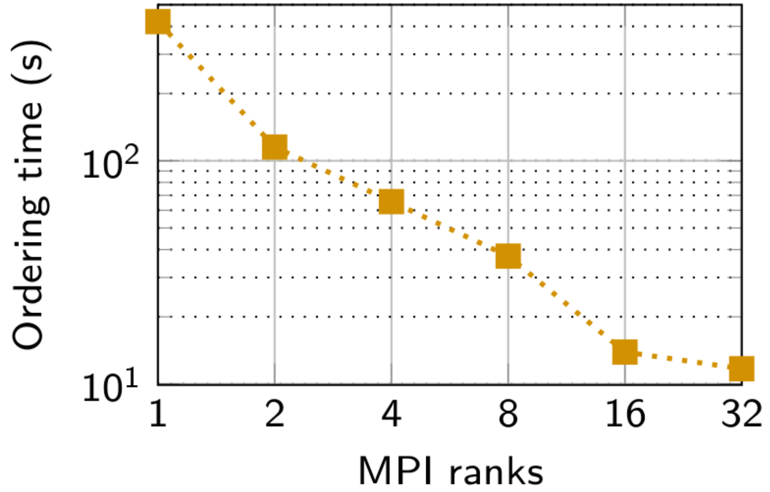
- LOBPCG preconditioned eigensolver:
  - Leverages a Block Low-Rank factorization preconditioner, less expensive than the exact factorization used by Lanczos;
  - Effective for small numbers of modes (<100).
- In LS-DYNA (METHOD=102 in *CONTROL_IMPLICIT_EIGENVALUE):
  - SMP implementation released in R12;
  - MPP implementation released in R14.
    Example, 10 modes of a 25M electric pickup truck model:



ParMETIS for fill-reducing ordering

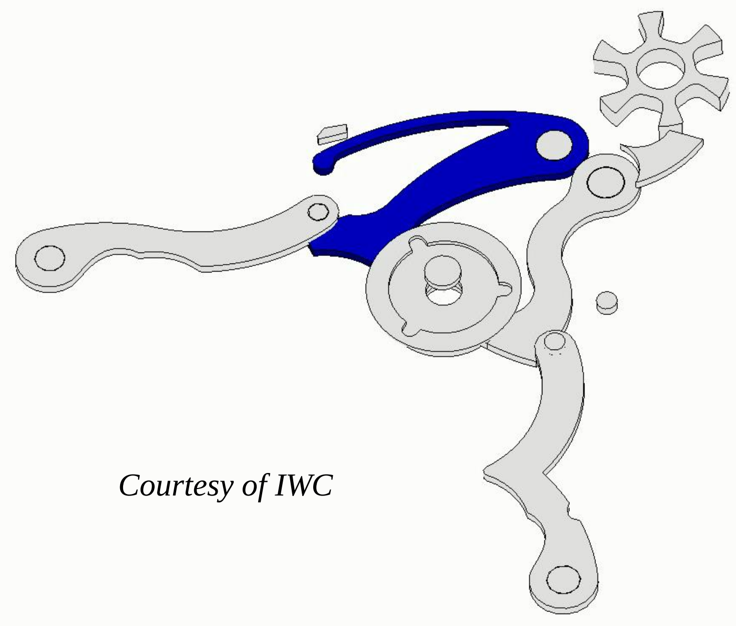
- Fill-reducing ordering: critical component of the Implicit sparse direct solvers.
- Options in LS-DYNA (ORDER in *CONTROL_IMPLICIT_SOLVER):
  - MMD: for small problems;
  - METIS: default option for most problems, serial algorithm;
  - LS-GPart: in-house MPP algorithm for very large problems and very large number of MPI ranks (500+).
  - ParMETIS: new in R14, MPP algorithm, recommended for most users.

  Example, ordering time for a 33M dof jet engine model from Rolls-Royce:

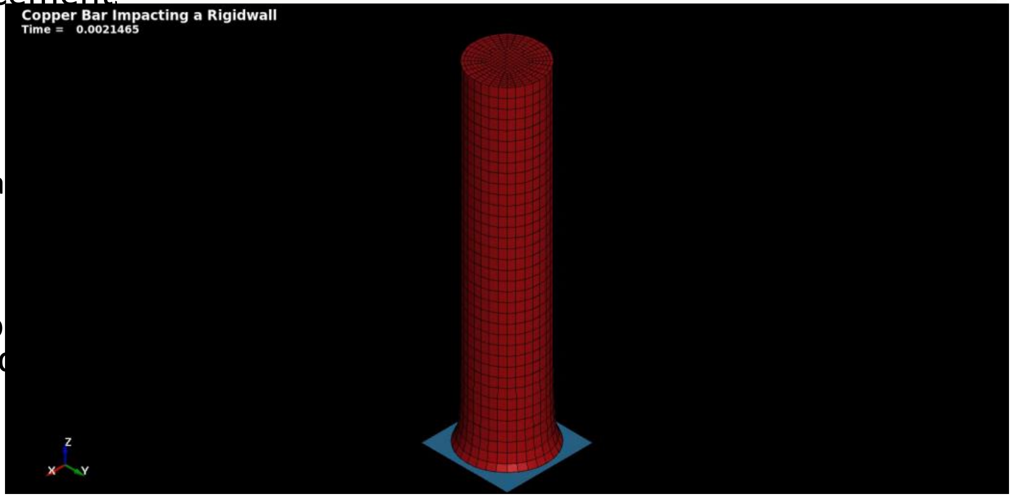


Implicit Developments

- New line search approach, excluding dependent degrees of freedom
  - Activate by LSTOL<0
  - Avoids choking due to "unfulfilled bc" and potentially reduces simulation time
  - Simulation to right finishes in less than 15% of the time reqiured by default approach
- Drilling Energy and Numerically dissipated energy reported to glstat
  - Used to be in hourglass and eroded energy slots
- Various inequality constraints are now supported by way of Lagrangian Multipliers
  - Rigid Body Stoppers
  - Rigid Walls (right)
  - Contact Entity
- Element formulations properly supported for linear implicit analysis (small displacement
  - High order shells
  - Discrete elements
- Minor enhancements for debugging implicit models
  - Warning if support of the eigenvector is small, indicating possible spinning beams or simila
  - Output of 100 worst elements, solids and shells, wrt aspect ratio
  - Removing time dependent effects in eigenvalue analysis when computing modal stress
- The option IACC=2 on *CONTROL_ACCURACY is introduced for explicit analysis, fo switching between the two. For instance will this invoke the strongly objective tie



Courtesy of IWC



NVH and Acoustics

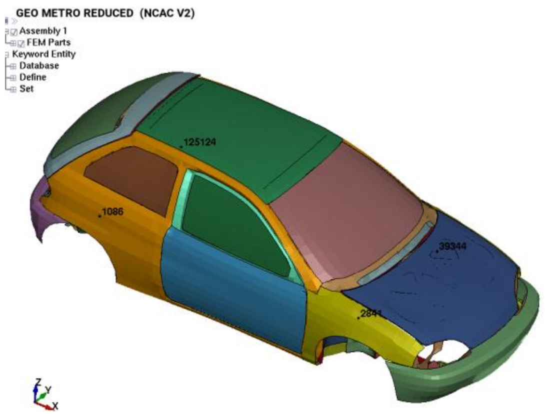
Summary of new features in R14

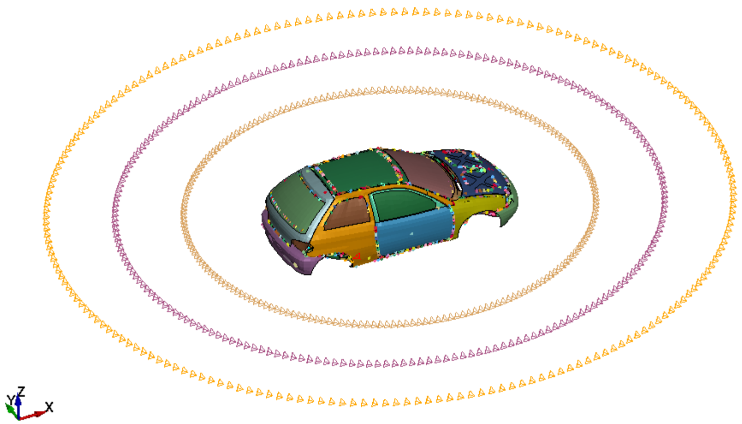
Acoustics	• Implementation of *FREQUENCY_DOMAIN_ACOUSTIC_DIRECTIVITY to create acoustic directivity plot from BEM acoustic computation
	• Improvement on performance
	  • Improved multithreading for variational indirect BEM
	  • Fast matrix assembly for variational indirect BEM
	• New options and boundary conditions
	  • Sound absorption coefficient boundary
	  • Symmetric boundary
	  • Option _POWER to calculate and output acoustic power
	  • Restart to get acoustic results at new locations

Random vibration	• OASPL computation for random pressure and plane wave load • GRMS computation for base acceleration PSD load
SSD	• NODOUT_SSD & ELOUT_SSD for direct SSD with frequency-dependent material properties. • Support new loading (torque and base rotational motion) in direct SSD.
ERP	• Support frequency-dependent ERP radiation loss factor • Performance improvement by allowing running ERP without SSD output
d3max	• Support ALE output in d3max

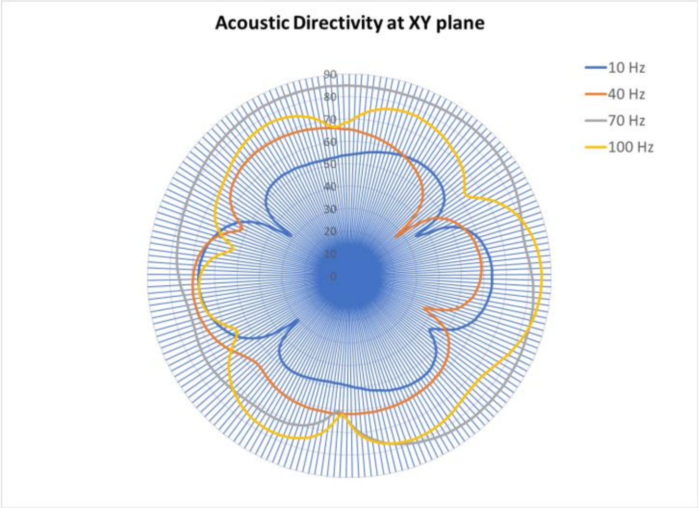
Acoustic Directivity Plot

- *FREQUENCY_DOMAIN_ACOUSTIC_DIRECTIVITY is implemented to provide acoustic directivity computation and output
- Multiple directivity plot can be provided in one run





Multiple directivity plot circles can be defined

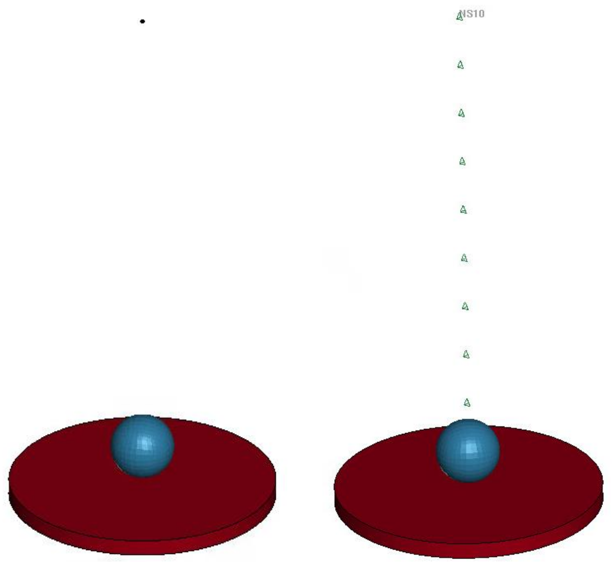


Directivity for multiple frequencies can be included

BEM: restart to get acoustic pressure at new location

- User requested to retrieve acoustic pressure at new location, after solving original acoustic problem
- RESTRT = 2 in *FREQUENCY_DOMAIN_ACOUSTIC_BEM is improved to allow reading BEM solution from last run and computing acoustic pressure for new location

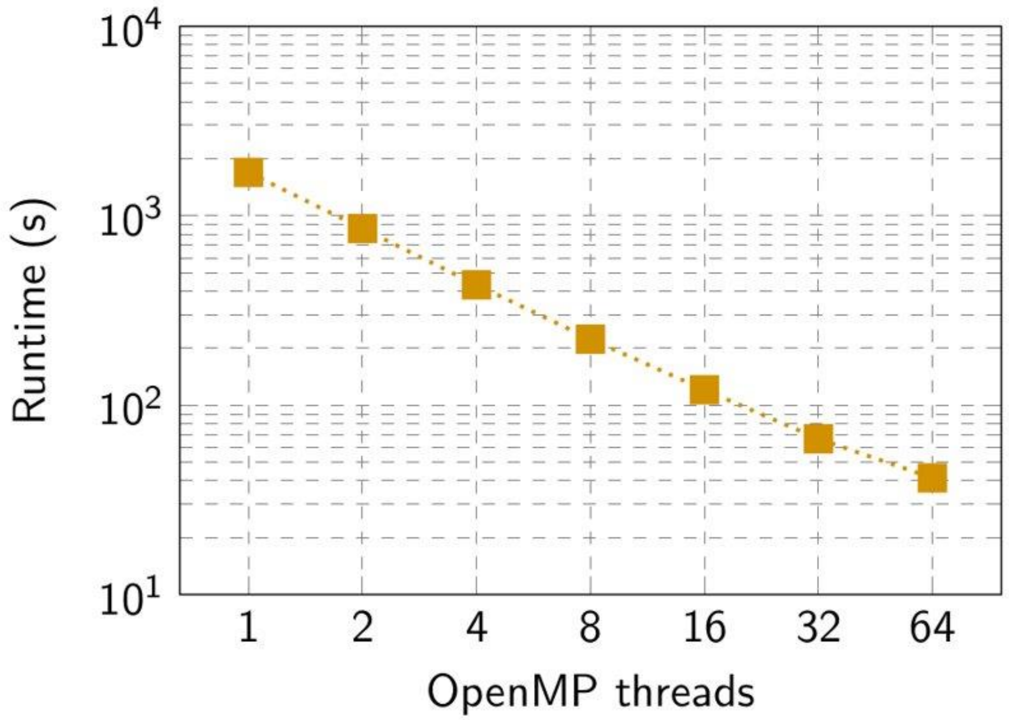
run jobs	No. of field points	No. of CPUs	Total CPU cost
original run	1	24	46 min 34 sec
restart run	9	1	0.3 sec



Variational BEM – Improved multithreading

METHOD=2 in *FREQUENCY_DOMAIN_ACOUSTIC_BEM (Variational BEM):

- Multithreading was introduced in R13;
- Improved in R14, up to 2x faster.
- Example, strong scaling experiment for a 78k dof engine block model:
  - Speed-up of 41 using 64 cores.



Variational BEM – Fast matrix assembly

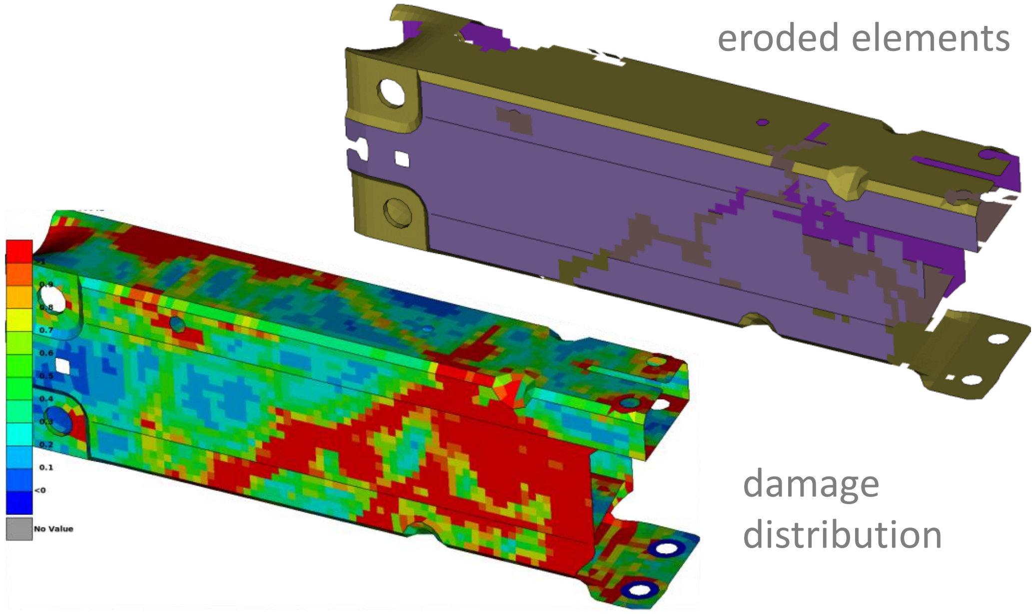
METHOD=211 in *FREQUENCY_DOMAIN_ACOUSTIC_BEM:

- New version of Variational BEM with fast low-rank matrix assembly.
- Example, 240k dof automotive cabin model using 64 cores:
  - R12, METHOD=2: 12 days (estimated – sequential);
  - R13, METHOD=2: 10 hours (using multithreading);
  - R14, METHOD=2: 4.5 hours (improved multithreading);
  - R14, METHOD=211: 1 hour (fast matrix assembly).

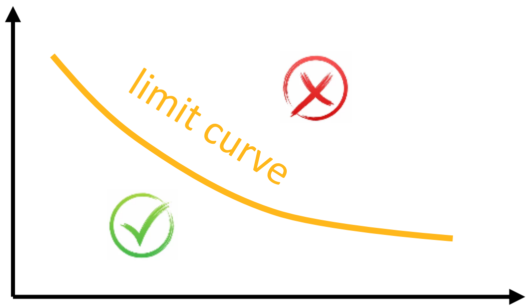
Miscellaneous

Miscellaneous enhancements

- Show results of deleted elements
  - New flag DELERES on *DATABASE_EXTENT_BINARY
  - E.g. to see damage value instead of gone elements
  - Associated new option in LS-Prepost 4.9: checkbox "Ignore Deleted Elements Flag“ in main menu "View"

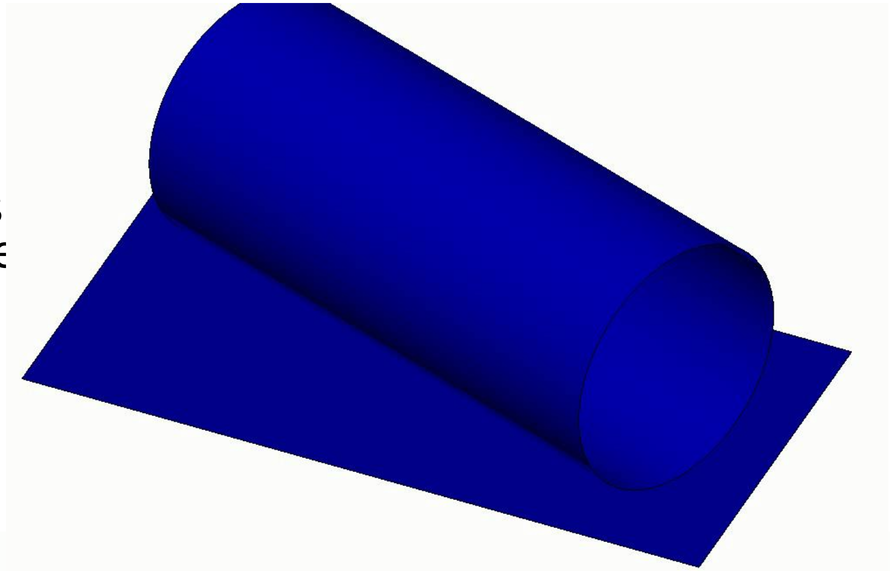


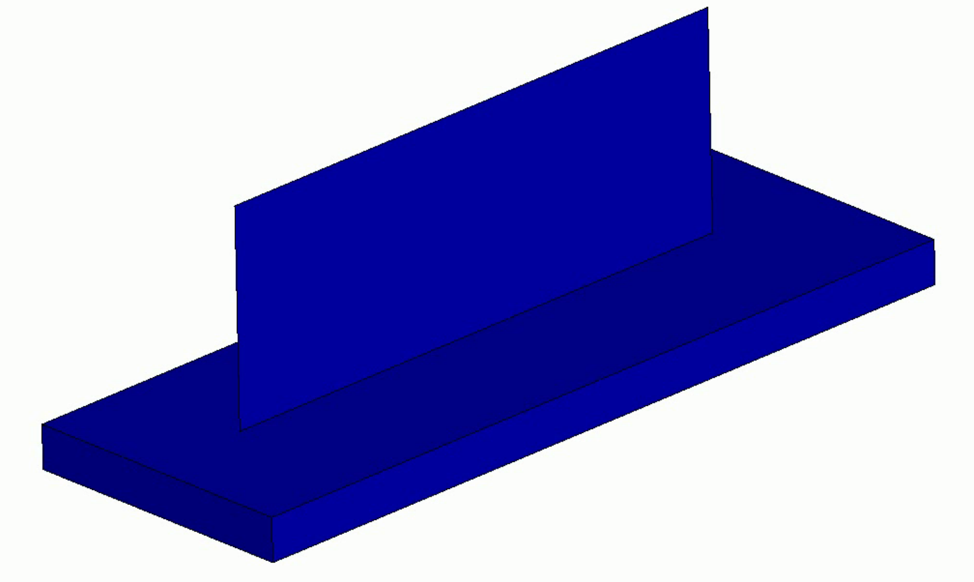
- Total mass change as stop criterion
  - ENDMAS on *CONTROL_TERMINATION: percent change in the total mass
  - This can now be defined as function (curve) of total mass
  - Full vehicle models as well as small sub-models can use the same setting



Mass Scaling Enhancements

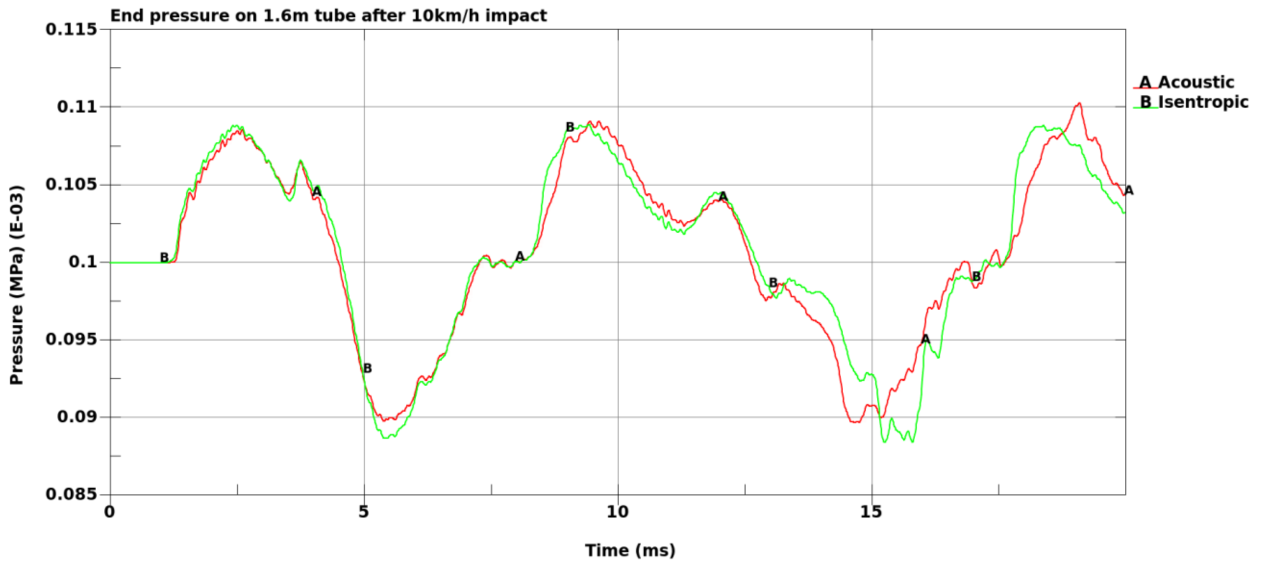
- EMSCL introduced on *CONTROL_TIMESTEP to scale gravity loads with scaled mass, by default the gravity load is proportional to the nominal mass. See also *LOAD_BODY
- Moving rigid walls are now properly supported in selective mass scaling, IMSCL on *CONTROL_TIMESTEP, by incorporating the motion of the rigid wall into the set of unknown variables in the mass acceleration system, example showed to the above right
- Tied shell edge to solid contact is now supported in selective ma scaling, IMSCL on *CONTROL_TIMESTEP, by incorporating the rotation degrees of freedom of SURFA into the set of unknown variables in the mass acceleration system, example showed to t below right
- Inertia element on rigid bodies are now supported in selective mass scaling, IMSCL on *CONTROL_TIMESTEP, these were inadvertently omitted in the past
- Mass by part is output to matsum files, both for conventional ar selective mass scaling

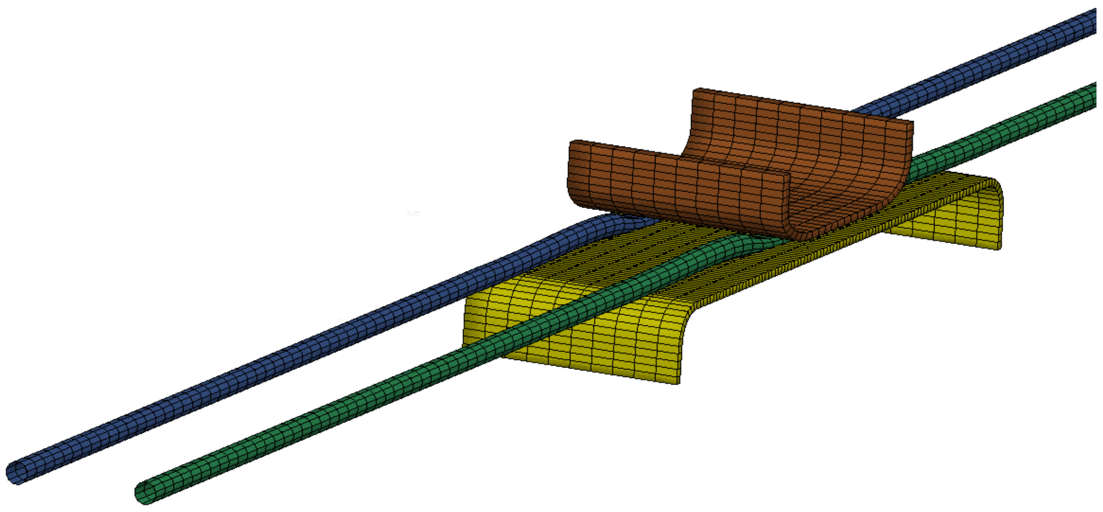


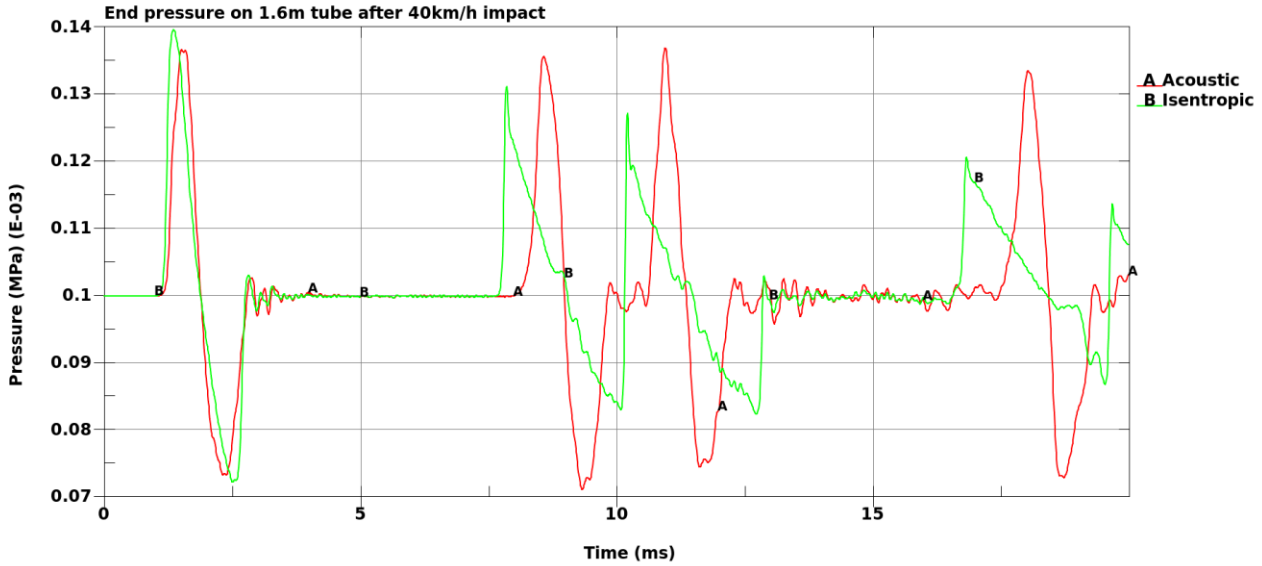


*DEFINE_PRESSURE_TUBE

- Supports isentropic Euler (MTD=2) with adiabatic index ≥1 (GAMMA)
- Captures non-linear effects, primarily in high velocity impacts







*SENSOR

- SENSOR_CONTROL
  - allow using TIMEOFF=1 to redefine the reference length of a discrete element as its length when it is turned on.
  - fix a bug for MPP contact that happen if a contact is initially turned off by*SENSOR_CONTROL
  - fix a MPP bug for CNRB, that happened when the rigid body to be controlled is not shared by all cores.
  - fix a bug for *SENSOR_CONTROL, when ELESET of type DISC.
- SENSOR_DEFINE_FORCE
  - add the option of tracing local force/moment of TYPE=JOINTSTIF
  - fix a bug for * SENSOR_DEFINE_FORCE when TYPE=X-SECFORC, that happened when *DATABASE_SECFORC is not defined
  - fix a MPP bug for SPC of *SENSOR_DEFINE_FORCE.
- SENSOR_DEFINE_MISC
  - fix an uninitialized variable issue for MTYPE=BNDOUT
  - add CVBAG to monitor control-volume airbag volume and temperature
  - output pressure and volume of incompressible control volume, *DEFINE_CONTROL_VOLUME.

New Features in LS-PrePost 4.9/4.10

Outlines

- LS-PrePost Version Overview
- Recent Developments
  - Pre-processing
  - Post-processing
  - Miscellaneous
- Conclusions

LS-PrePost Version Overview

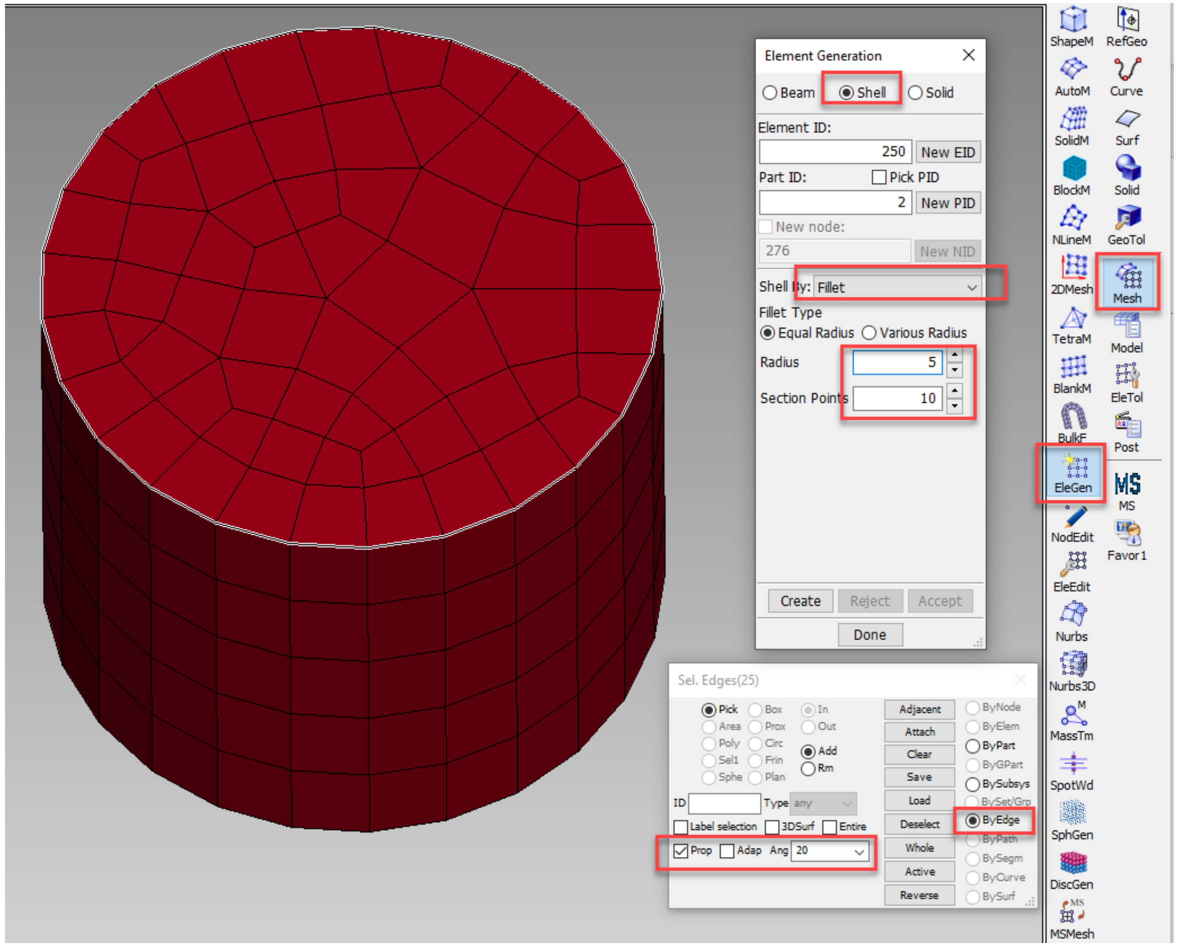
- LS-PrePost is delivered *free* with LS-DYNA. As of today, still *no* license key needed to run LS-PrePost
- LSPP 4.9 has been officially released in March 2022
- Development version is LSPP4.10
- LSPP 4.8/4.9 will only be updated for bug fixes
- One can download LS-PrePost from:
  https://ftp.lstc.com/anonymous/outgoing/lsprepost/dev/
  https://ftp.lstc.com/anonymous/outgoing/lsprepost/4.9/
- LS-PrePost is developed on Windows and ported to Linux…
  - Windows - LS-PrePost-4.9.7-x64-10Mar2022_setup.exe
  - Linux - lsprepost-4.9.7-common-10Mar2022.tgz
  - Apple Mac - We will not continue to support Apple Mac with the new version

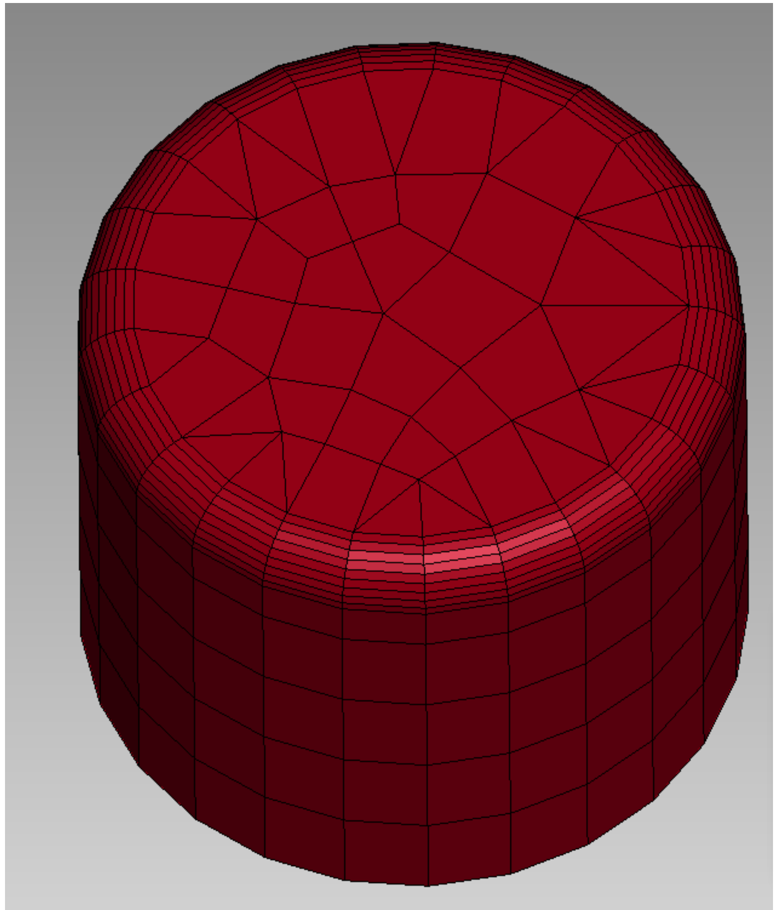
LS-PrePost Special DP (Double Precision) Version

- LS-DYNA allows Part IDs, Material IDs, and Property IDs to be longer than 10 Digits when using the double precision version of LS-DYNA
- A special DP version of LS-PrePost 4.9 (and later) is available for users to handle such input data or d3plot files
- User can download this version from:
  - https://ftp.lstc.com/anonymous/outgoing/lsprepost/dev/lsprepost_dp.exe
  - Put this file in the installation directory of the regular version
  - One will need to build a separate link to this version
  - Check the README file
    https://ftp.lstc.com/anonymous/outgoing/lsprepost/dev/README_lsprepost4.10_dp.txt

Pre-Processing – Mesh modification

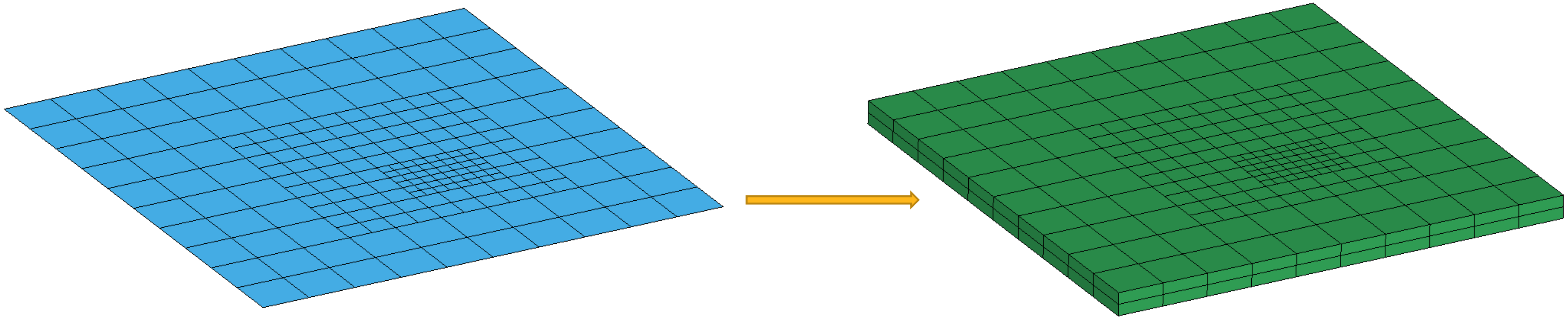
- Create Shell Fillet





Pre-Processing – Mesh modification

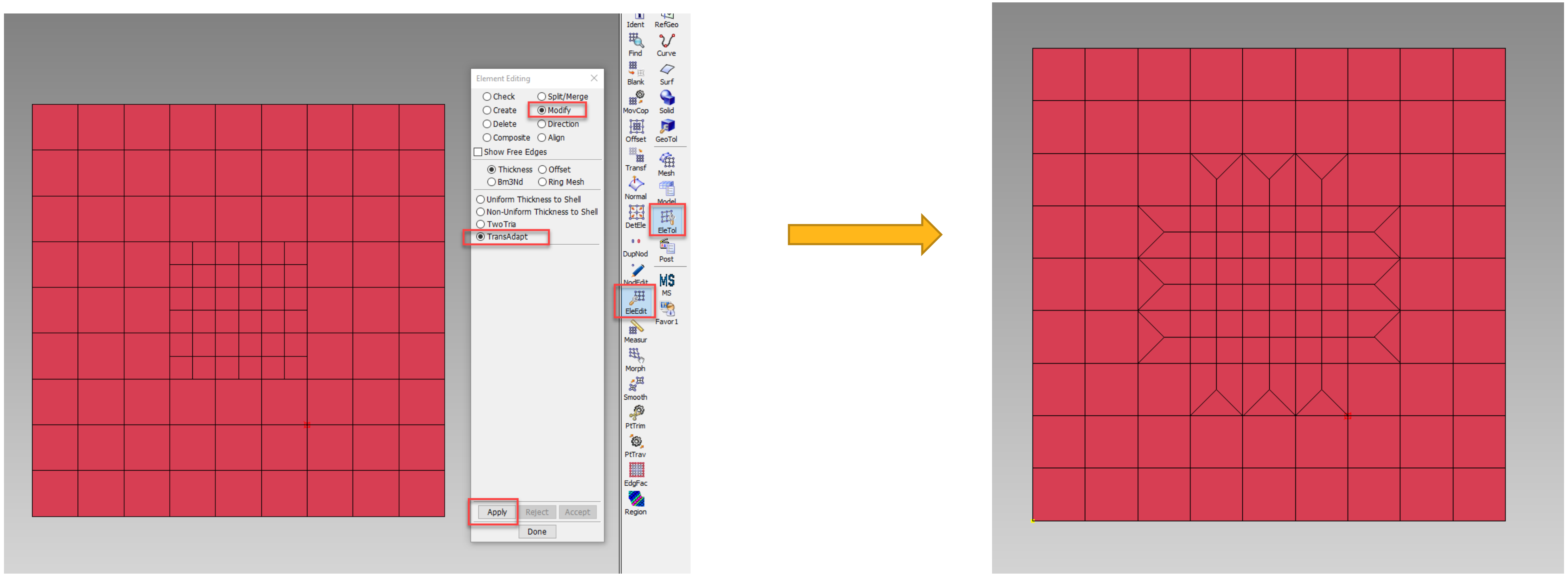
- Extent Shell to Solid with Adaptive mesh
  - Mesh→*ElGen* →*Solid* →*Solid By: Shell_Thickness*



- Transfer thickness, stress and strain (*INITIAL_xxx) from shell to solid when doing.
- Create *CONSTRAINED_ADAPTIVITY in solid part when creating solid by *Shell_Thickness* and the selected shell elements uses *CONSTRAINED_ADAPTIVITY

Pre-Processing – Mesh modification

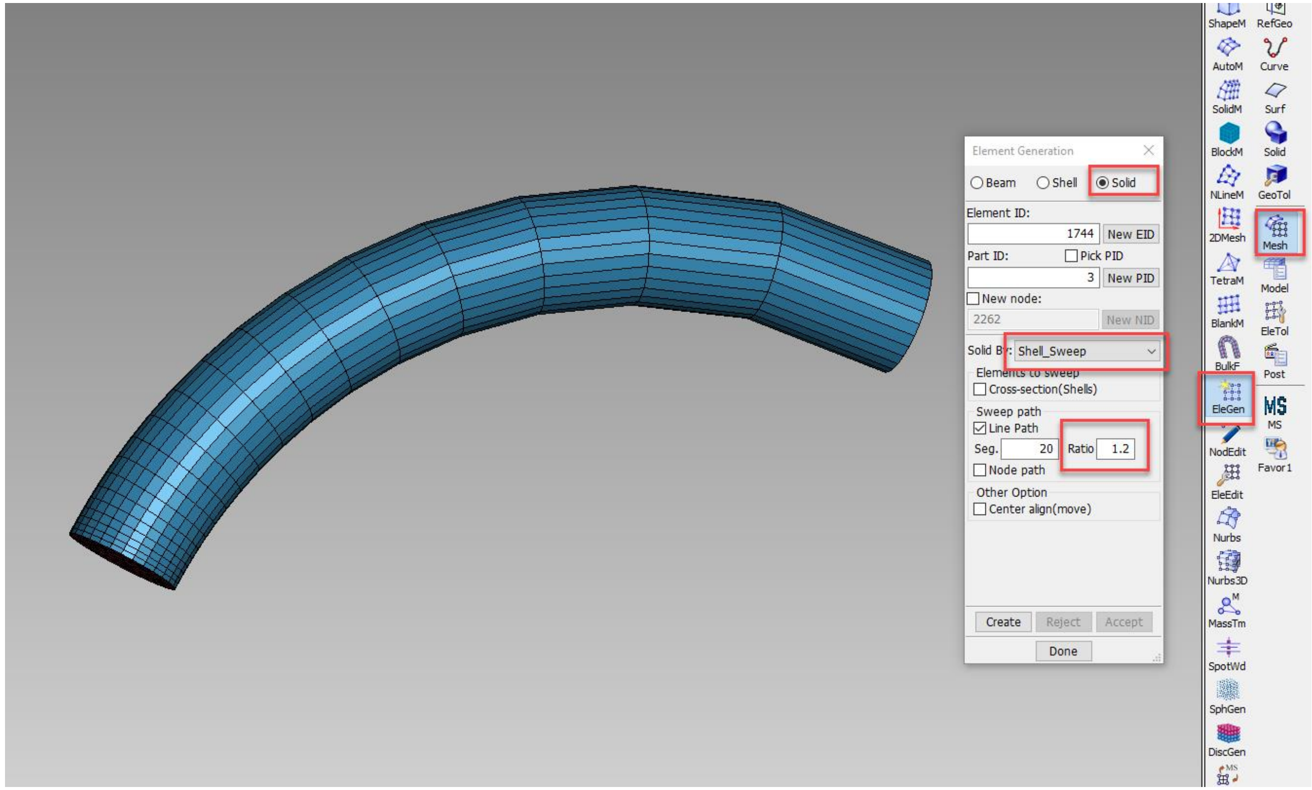
- Change adaptive mesh to transition mesh (with triangular elements)



- The stress/strain data will be created for the transition elements

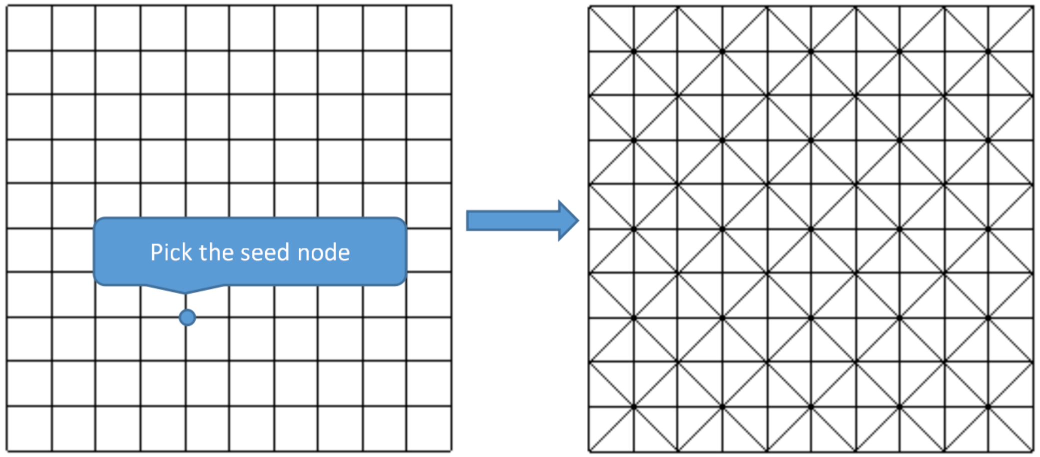
Pre-Processing – Create Solid Mesh

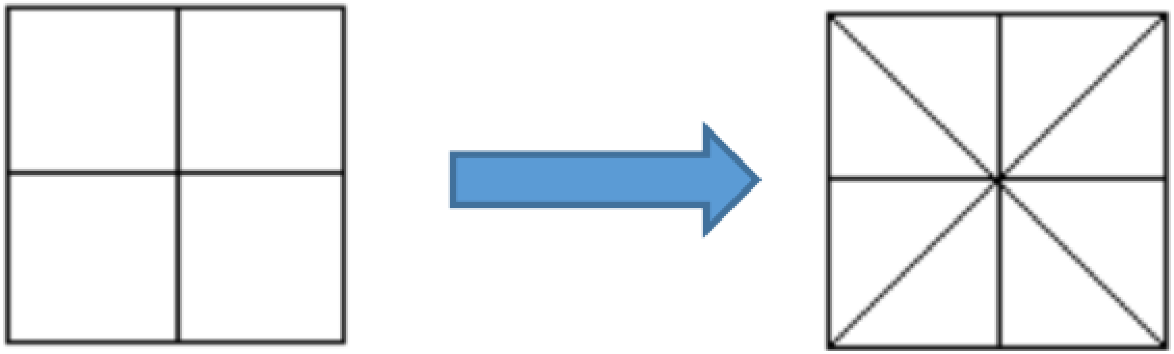
- Create Solid by sweeping shell along a curve
- Add Ratio to the spacing of the mesh along the curve



Pre-Processing – Shell Splitting

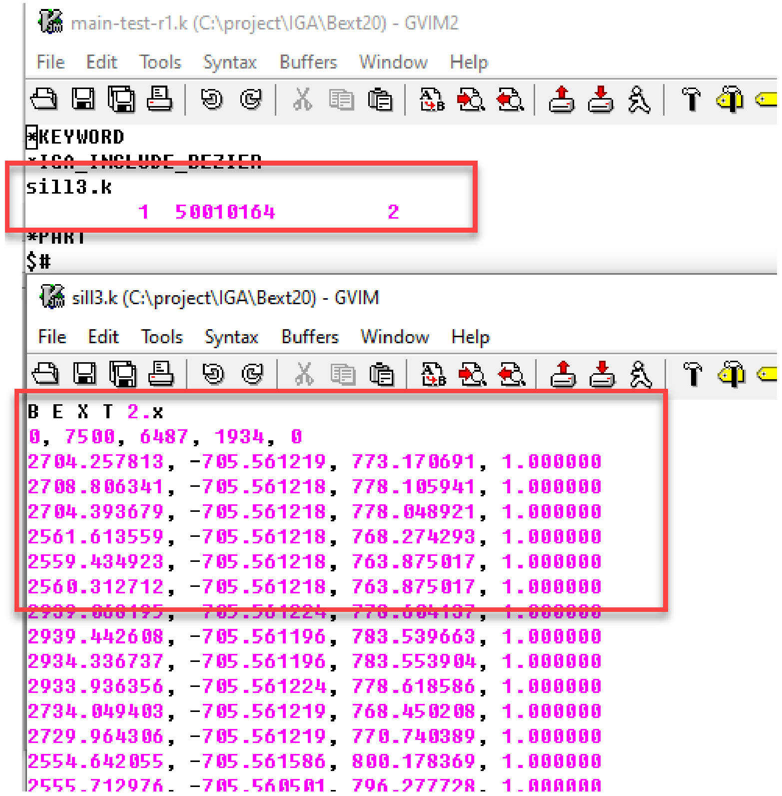
- Split Shell quad element into Triangular element to form Union Jack pattern
- It takes 4 quads to form one Union Jack pattern

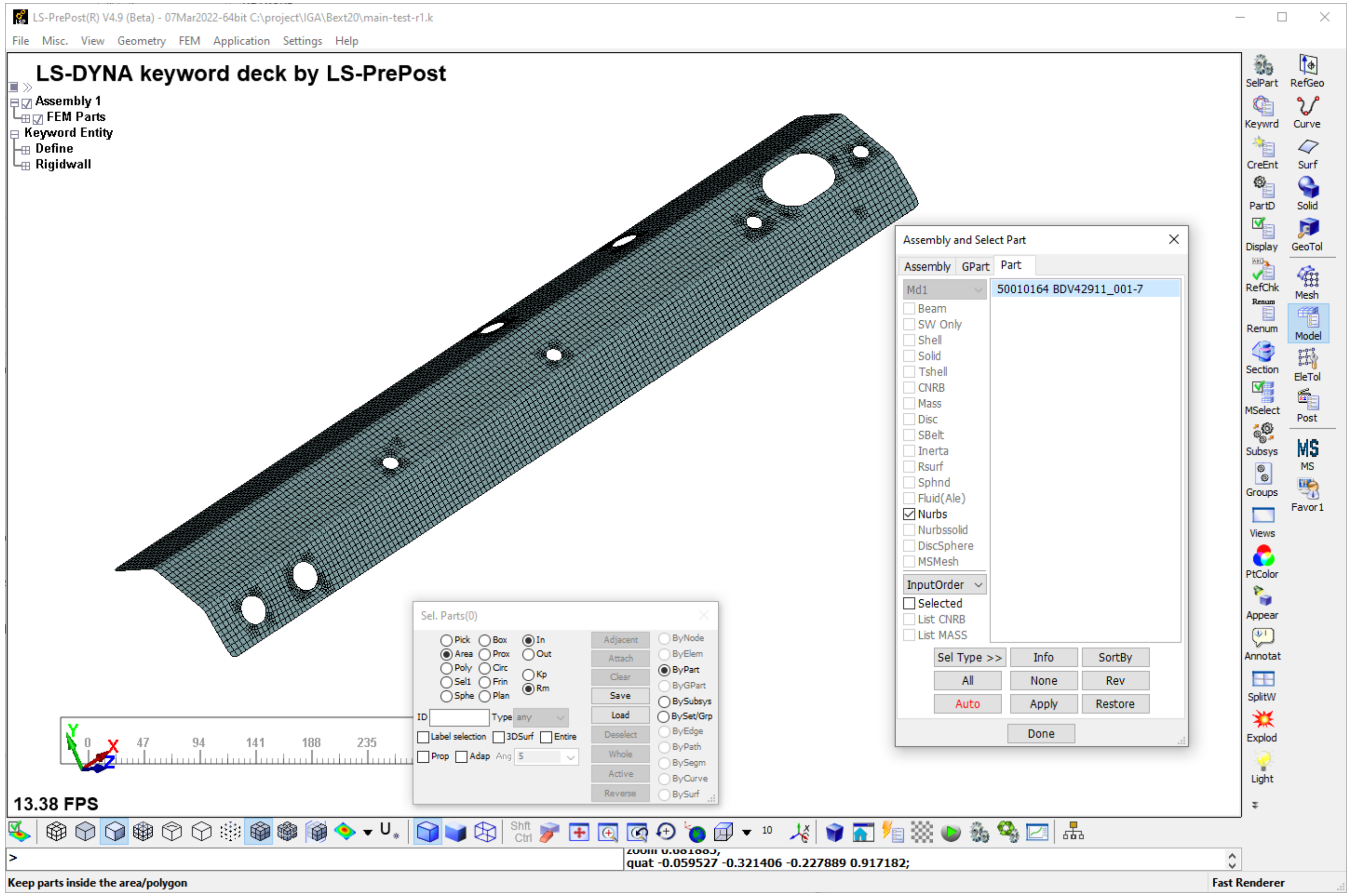




Pre-Processing – Read and Process Latest IGA data

- IGA element data have been changed, the new BEXT formats have been introduced



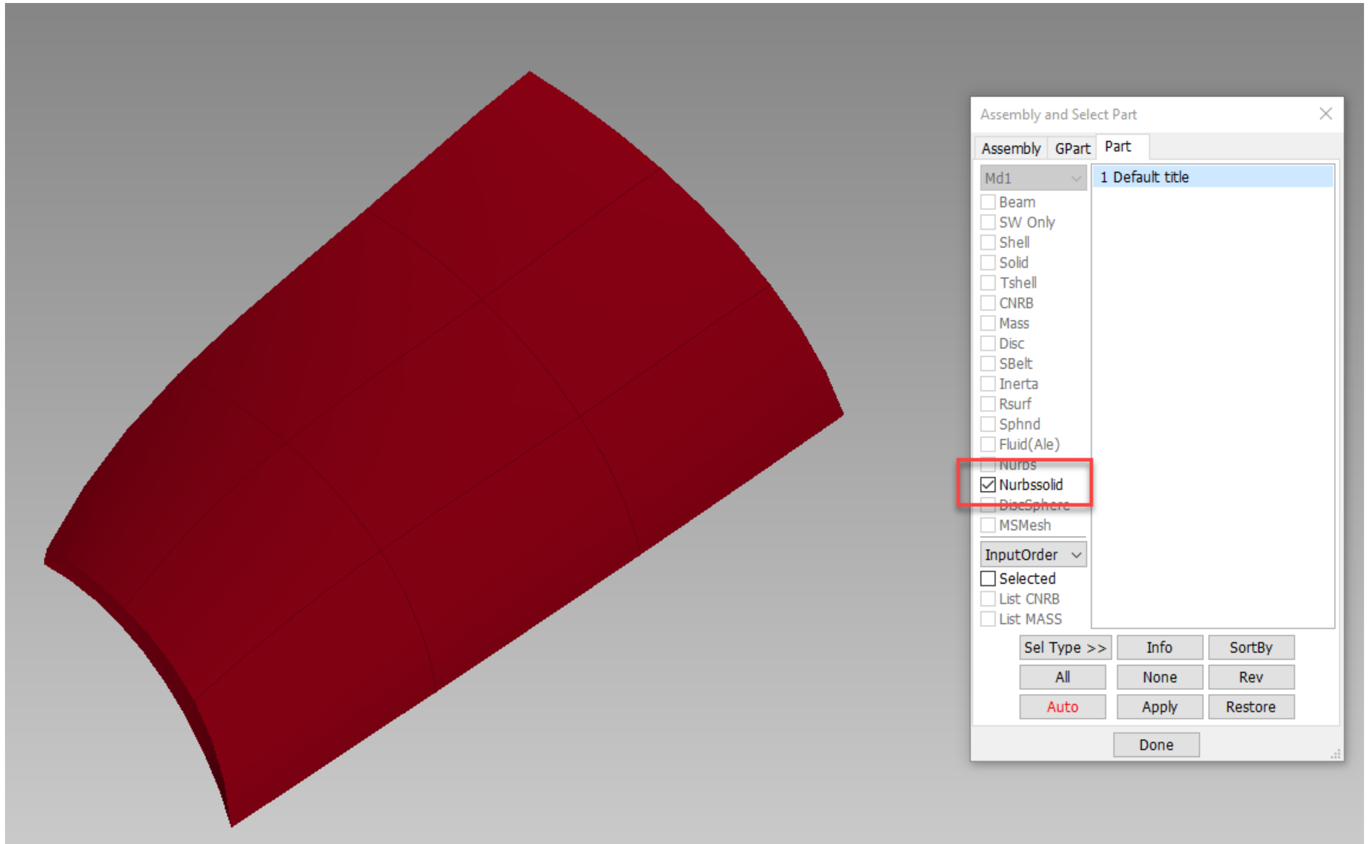


Pre-Processing – Read and Process Solid IGA (3D Nurbs) Data

- 3D Solid IGA data can be read and rendered

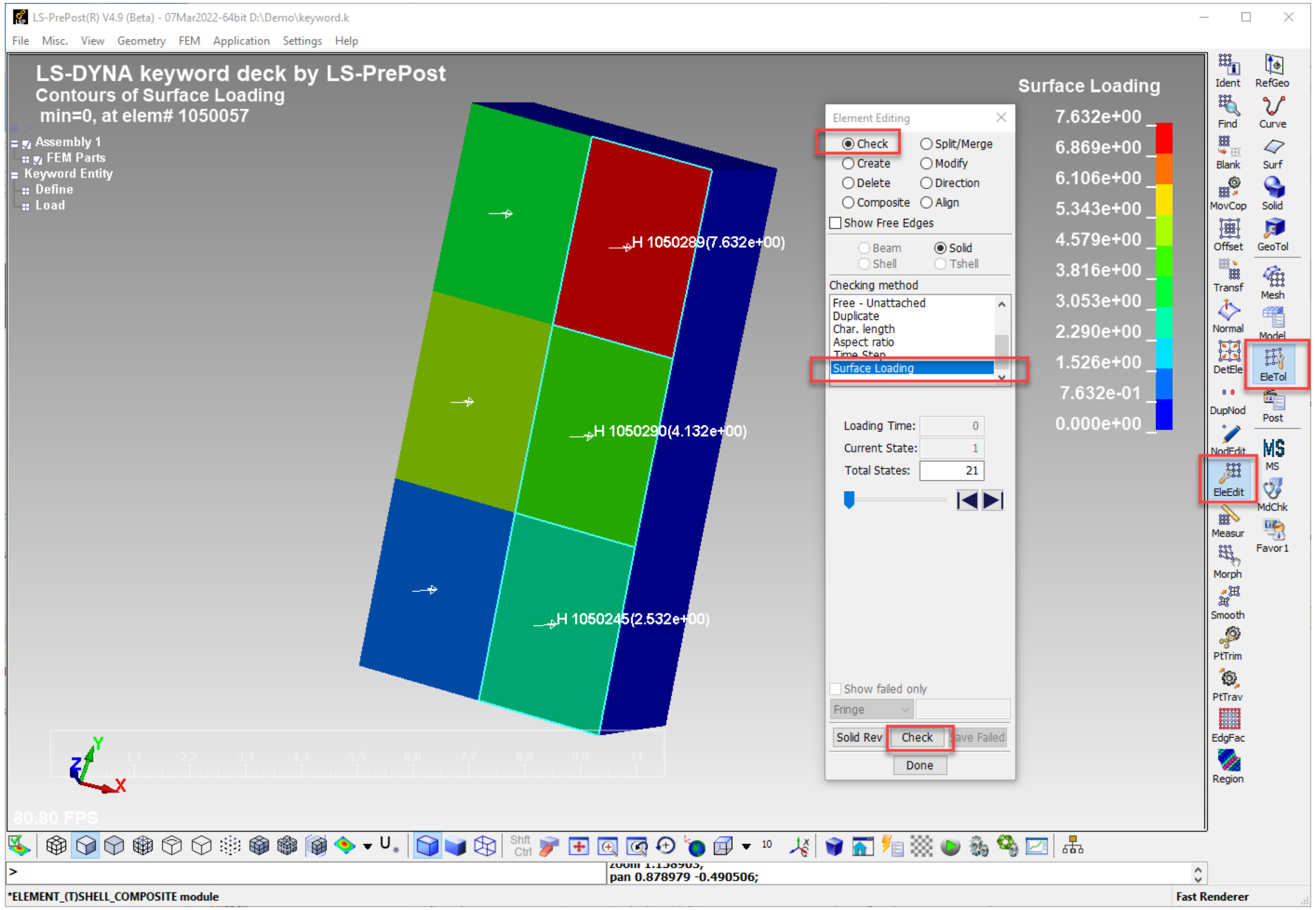
```
*PART
$TITLE
Default title
$      PID     SECID       MID
         1         1         1
*MAT_ELASTIC
$      MID        RO         E        PR        DA        DB
         1 0.0000078    200.00      0.30      0.00      0.00
*SECTION_IGA_SOLID
$    SECID    ELFORM        IR     IMASS
         1         0         1         0
*IGA_SOLID
$ SOLID
$      SID       PID      NISR      NISR      NIST       UIR
         1         1         4         4         4
*IGA_VOLUME_XYZ
$  IDV_XYZ   ID3_XYZ      IDPS      IDES      IDFS      IDVS
         1         1
$  ID2DBR1   ID2DBR2   ID2DBR3   ID2DBR4   ID2DBR5   ID2DBR6   ID2DBR7   ID2DBR8

*IGA_3D_NURBS_XYZ
$  ID3_XYZ        NR        NS        NT        PR        PS        PT
         1         6         4         6         3         3         3
$     UNIR      UNIS      UNIT
         1         1         1
$           RFIRST              RLAST
      0.0000000000       1.0000000000
$           SFIRST              SLAST
      0.0000000000       1.0000000000
$           TFIRST              TLAST
      0.0000000000       1.0000000000
$                X                  Y                  Z                WGT
     -1.7834530000       1.9310080000     -19.8415300000                1.0
     -1.5645735567       1.4277348056     -19.7942453887                1.0
     -0.8562420594       0.5053088153     -19.6733420628                1.0
```



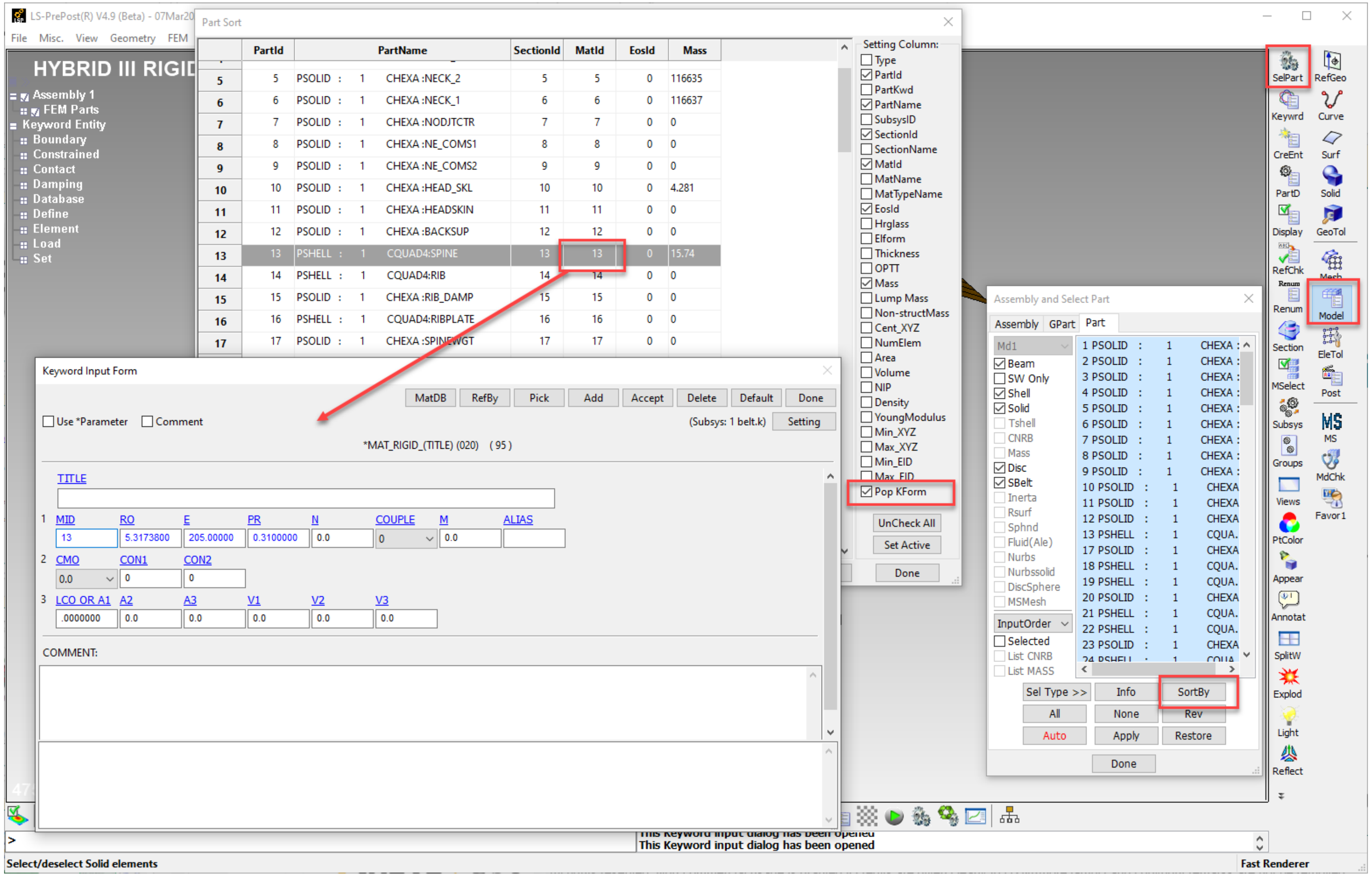
Pre-Processing – Fringe *LOAD_SEGMENT

- Segment load (Pressure) now can be visualized with Fringe color
- Can be activated in ElEdit->Check->Surface Loading
- If load curve is used, it can be step through time to see the value
- Left mouse click the segment will also show the pressure value



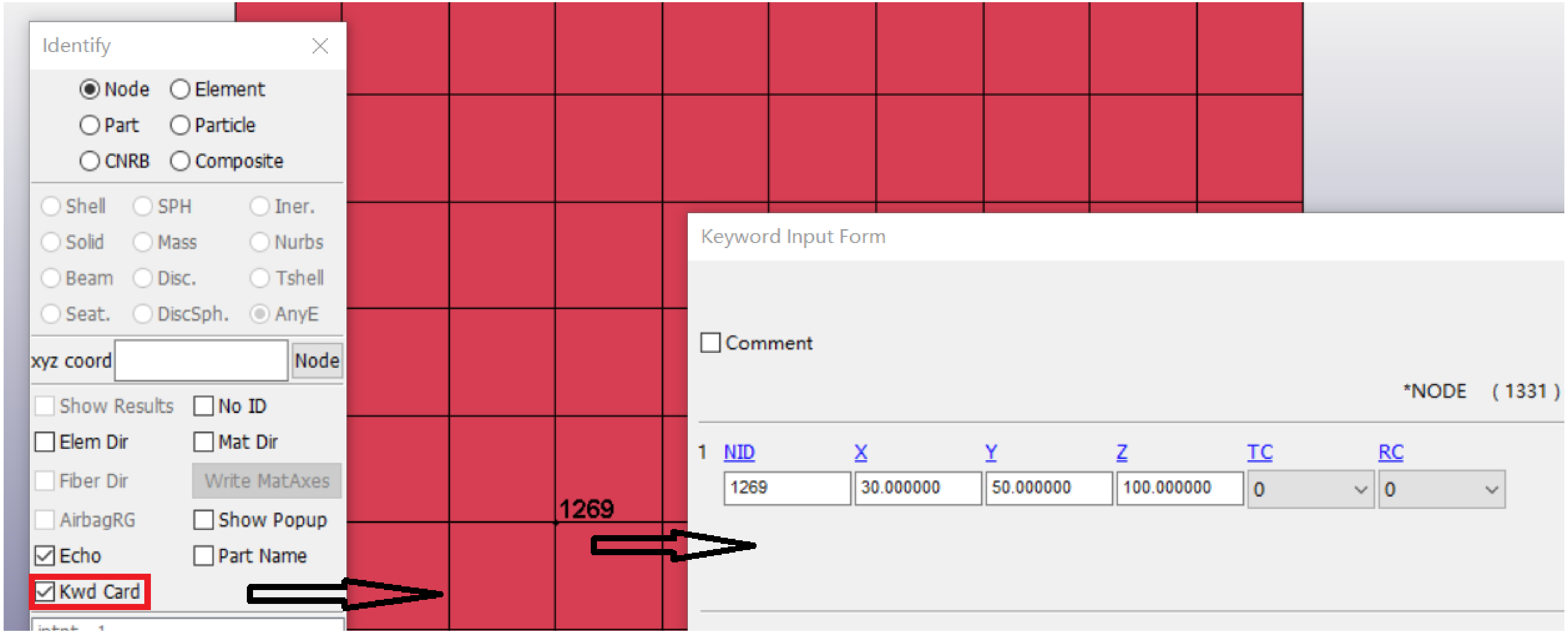
Pre-Processing – Popup Keyword Form in Part Selection

- Support Keyword Form popup when right click on one of the field ID in the SelPart ->SortBy table



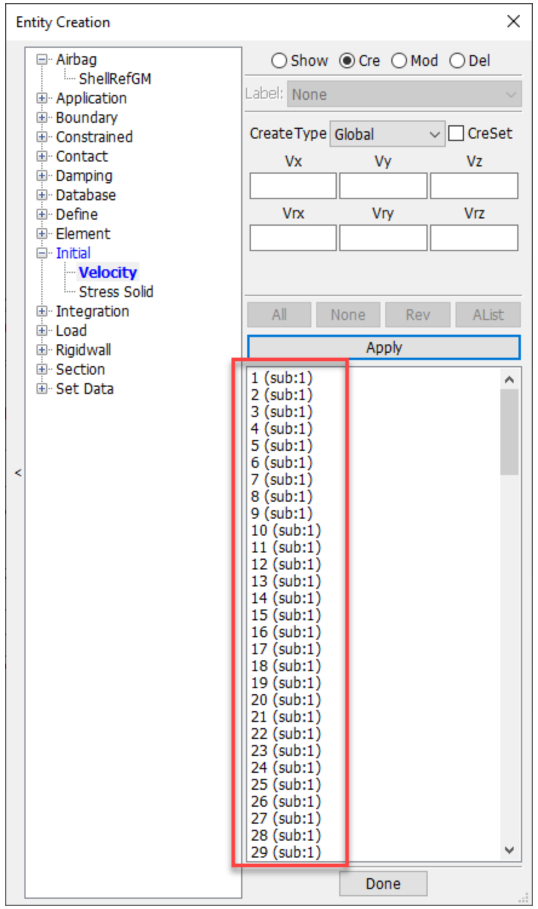
Pre-Processing – Popup Keyword Form in Ident Interface

- support pop-up keyword form of selected Entity (node/element/part) in the Identify interface

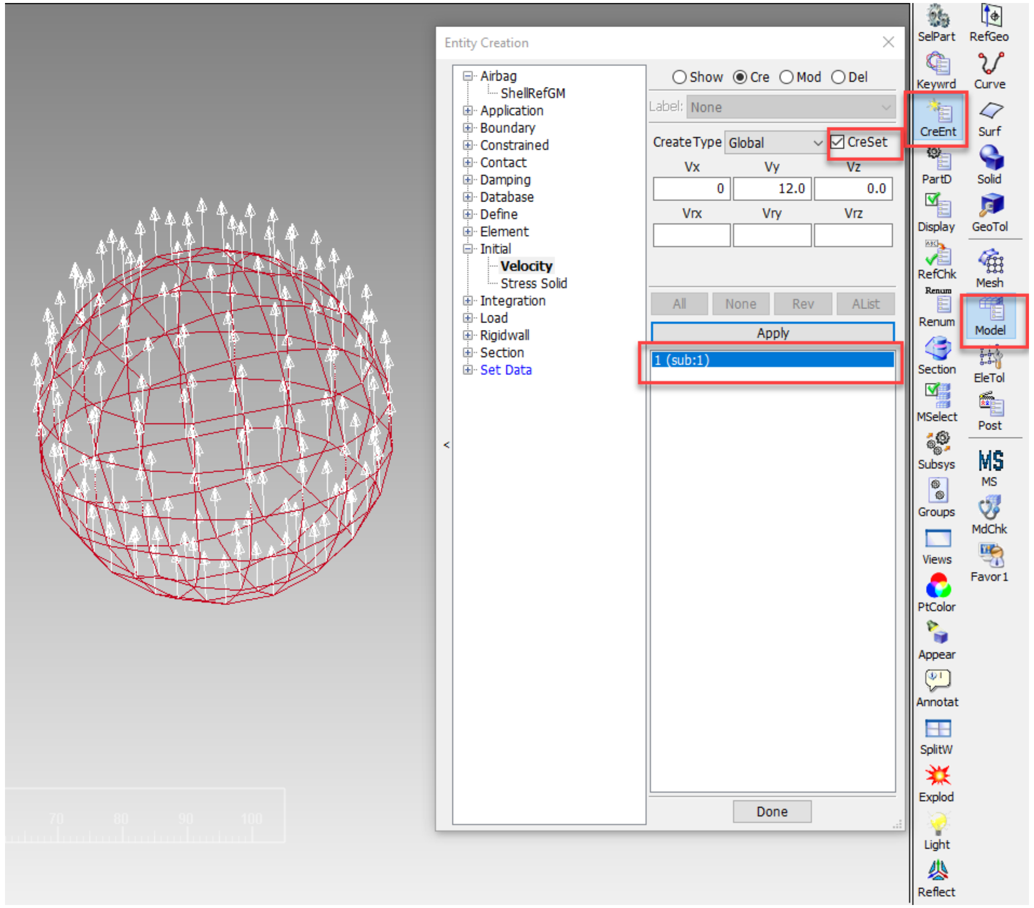


Pre-Processing – Entity Creation Improvement

- Support *INITIAL_VELOCITY data creation on a node set instead of individual nodes



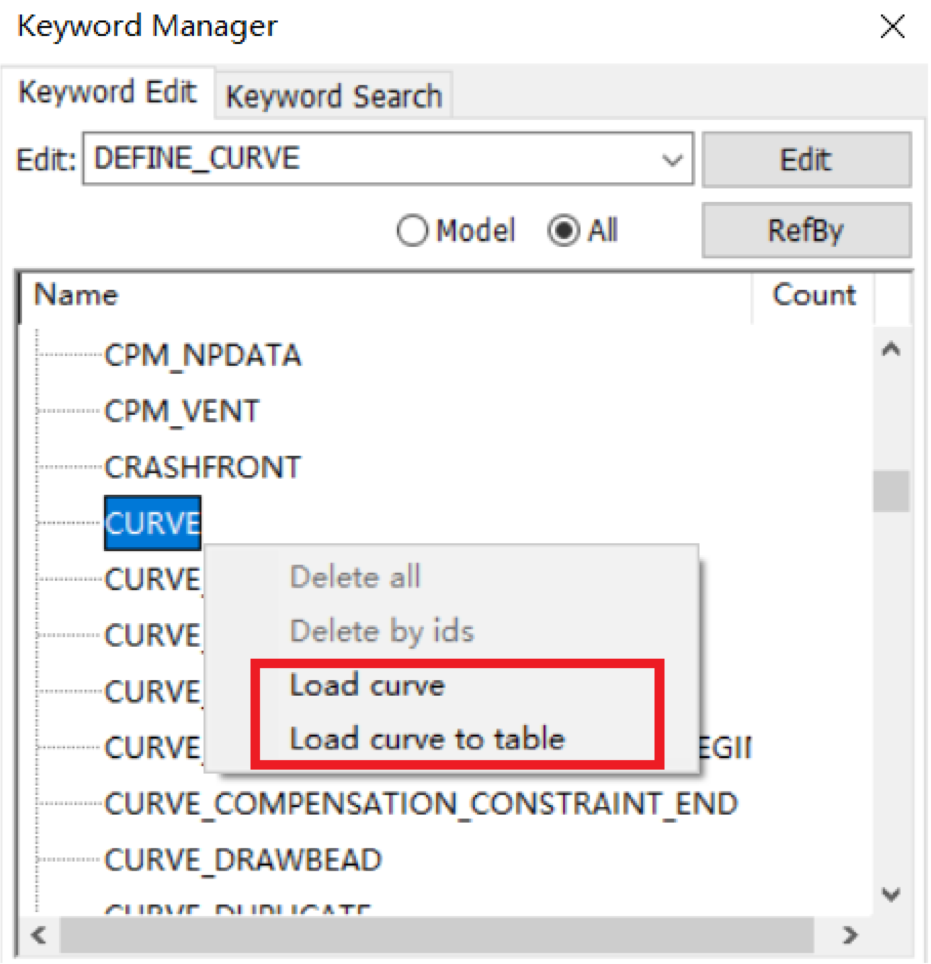
Old Way



New Way

Pre-Processing – New Feature in Keyword Manager

- Right menu of *DEFINE_CURVE to load curve from xydata file, "Load Curve" will load curve data from file and create *DEFINE_CURVE,
- "Load curve to table" will also use these curve ids to create *DEFINE_TABLE.



Pre-Procession - Solution Explorer

Multiphysics

- Mechanical
- ICFD
- Structured-ALE
- Thermal
- ISPH
- EM

Translation Tools

- Keyword data
- Tree structure & Properties
- Material databases

One Flow Line

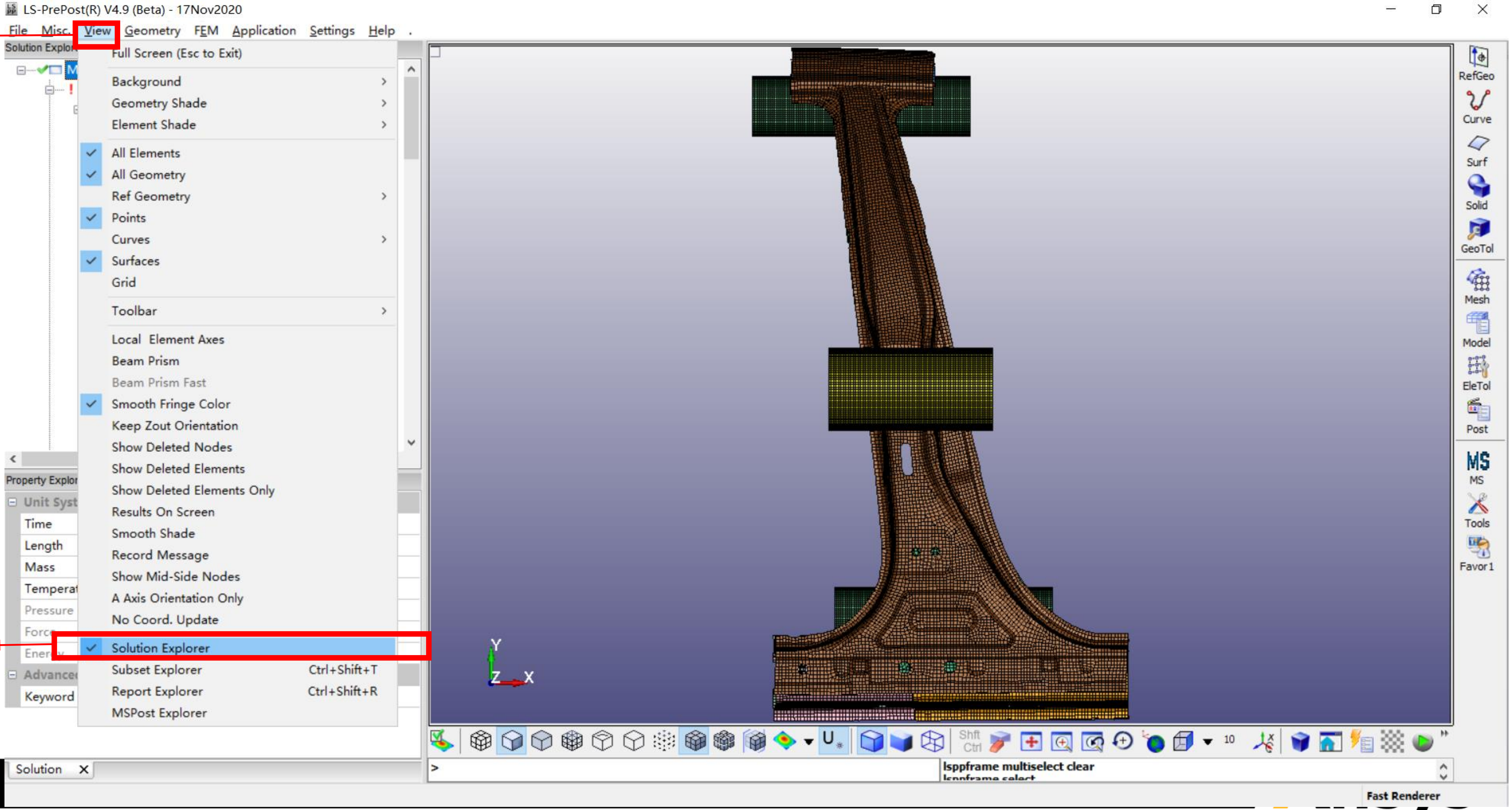
- No keyword editing and no result files loading by hand
- Whole flow line based on tree structure and friendly to users

Ansys

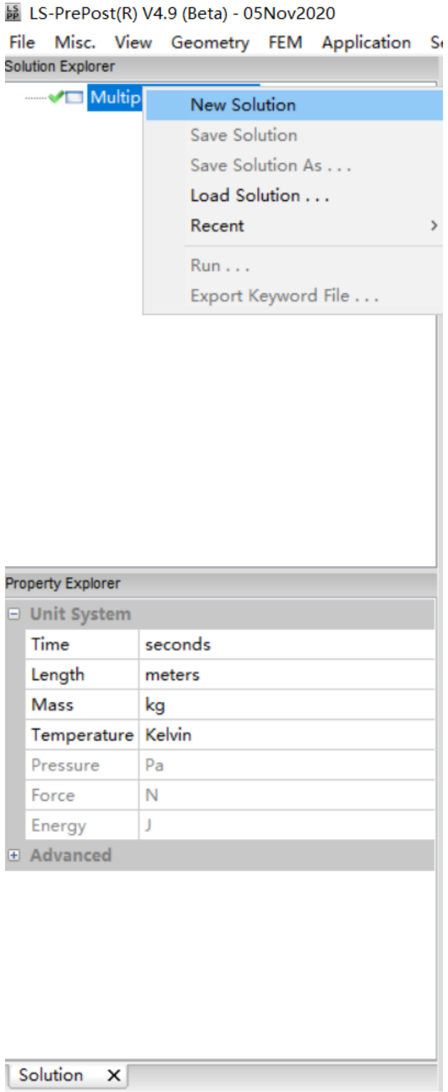
Solution Explorer - Activation

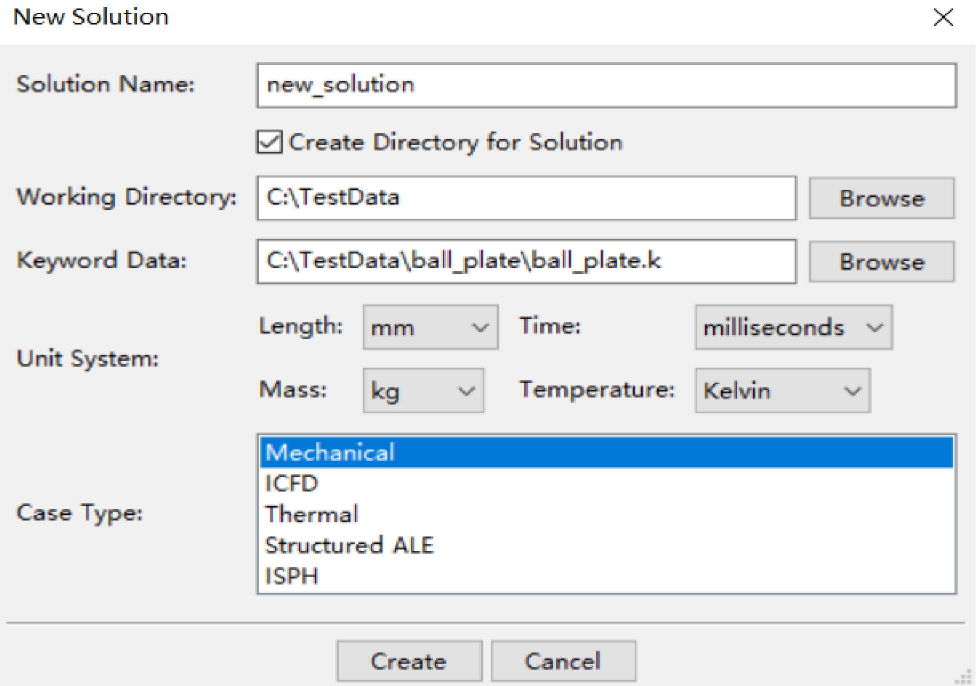
Step 1. View Menu

Step 2. Check on "Solution Explorer"



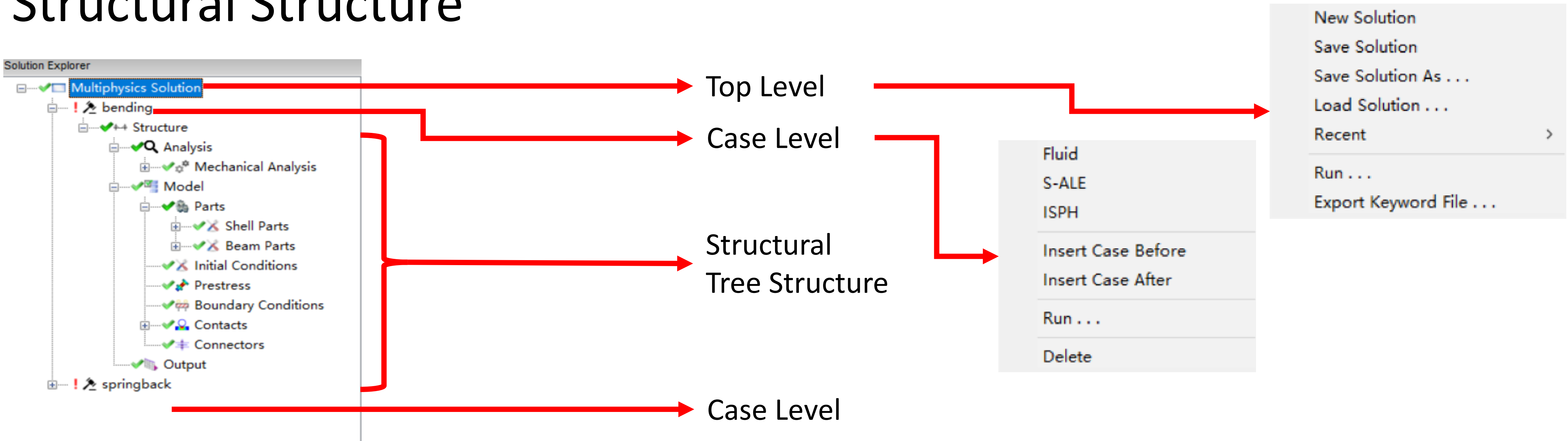
Solution Explorer - Create New Solution

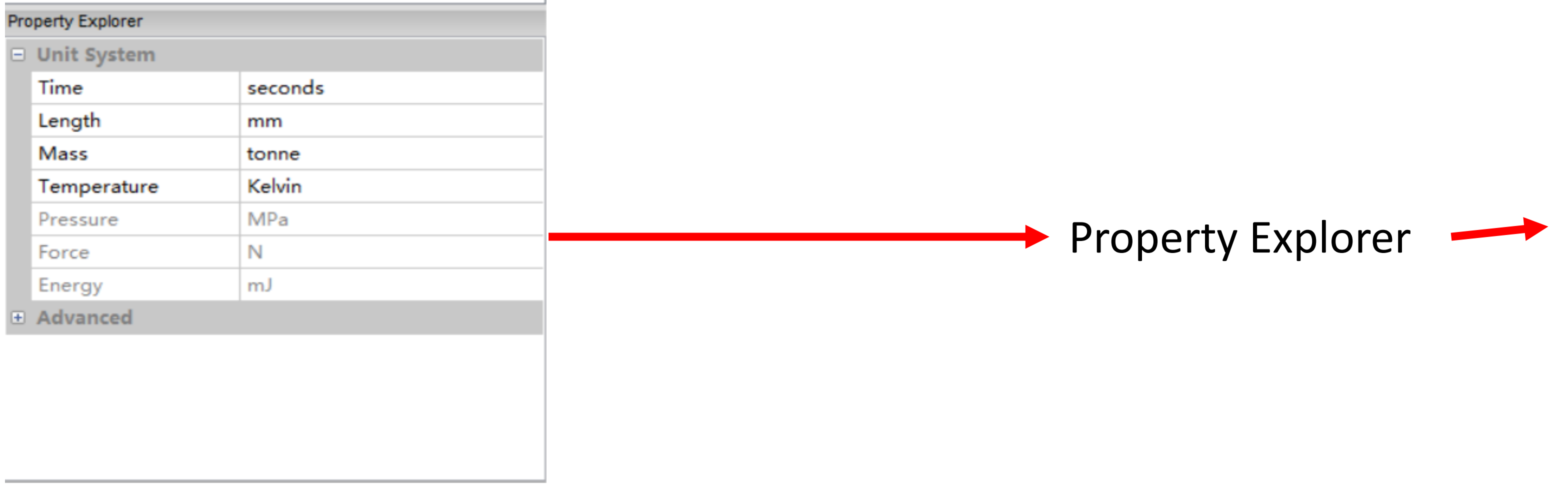




Folder	Keyword	Case
• Solution Name • Working Directory	• Mesh data (Part , Element data, Node data, Material…) • Unit System	• Mechanical • ICFD • … • ISPH

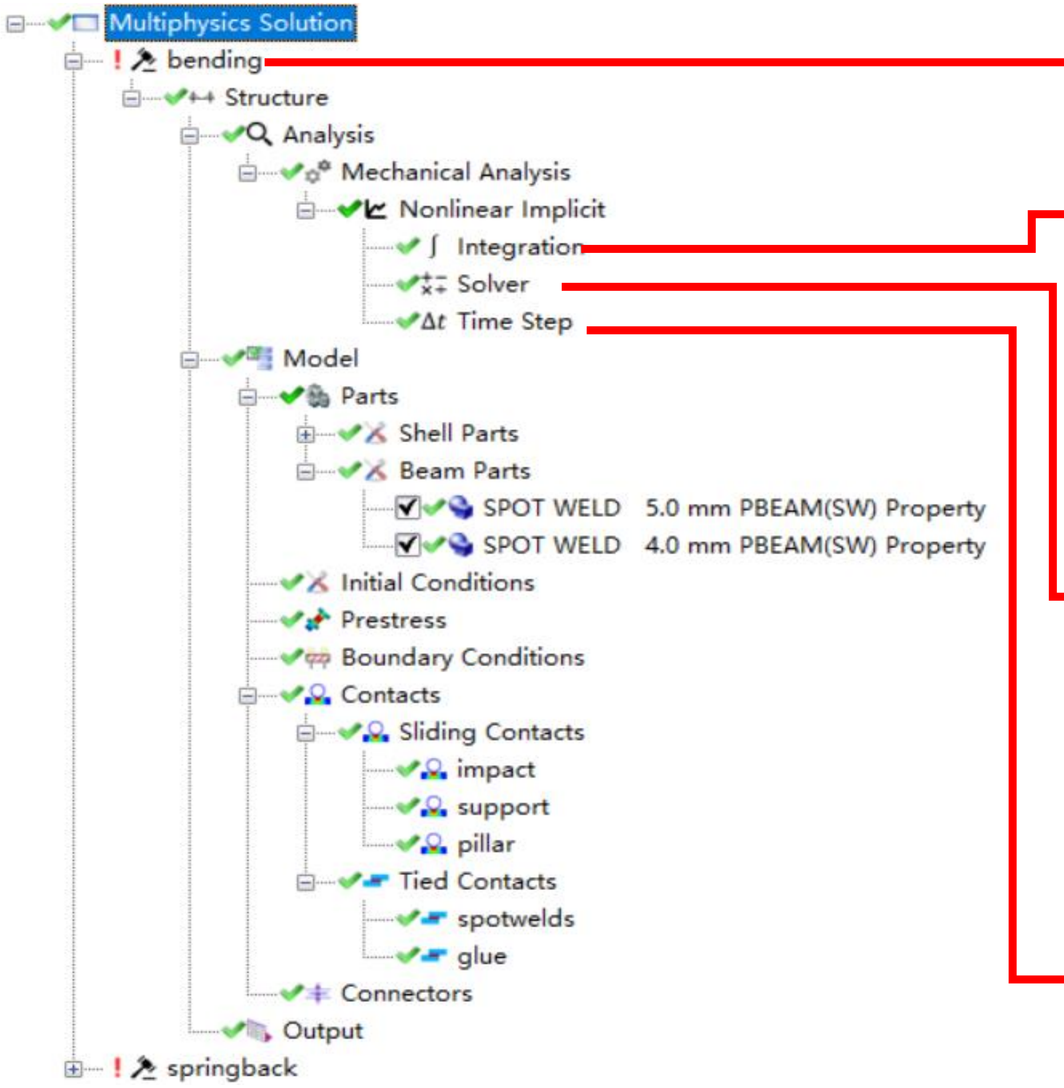
Structural Structure





1. Different tree item has different properties.
2. Edit the properties to change analysis data.
3. Many tools can be designed to the properties.
4. The descriptions of the properties are friendly to users.

Solution Explorer - Structural Structure



End Time	1 s
Database Plot Interval	0.01 s

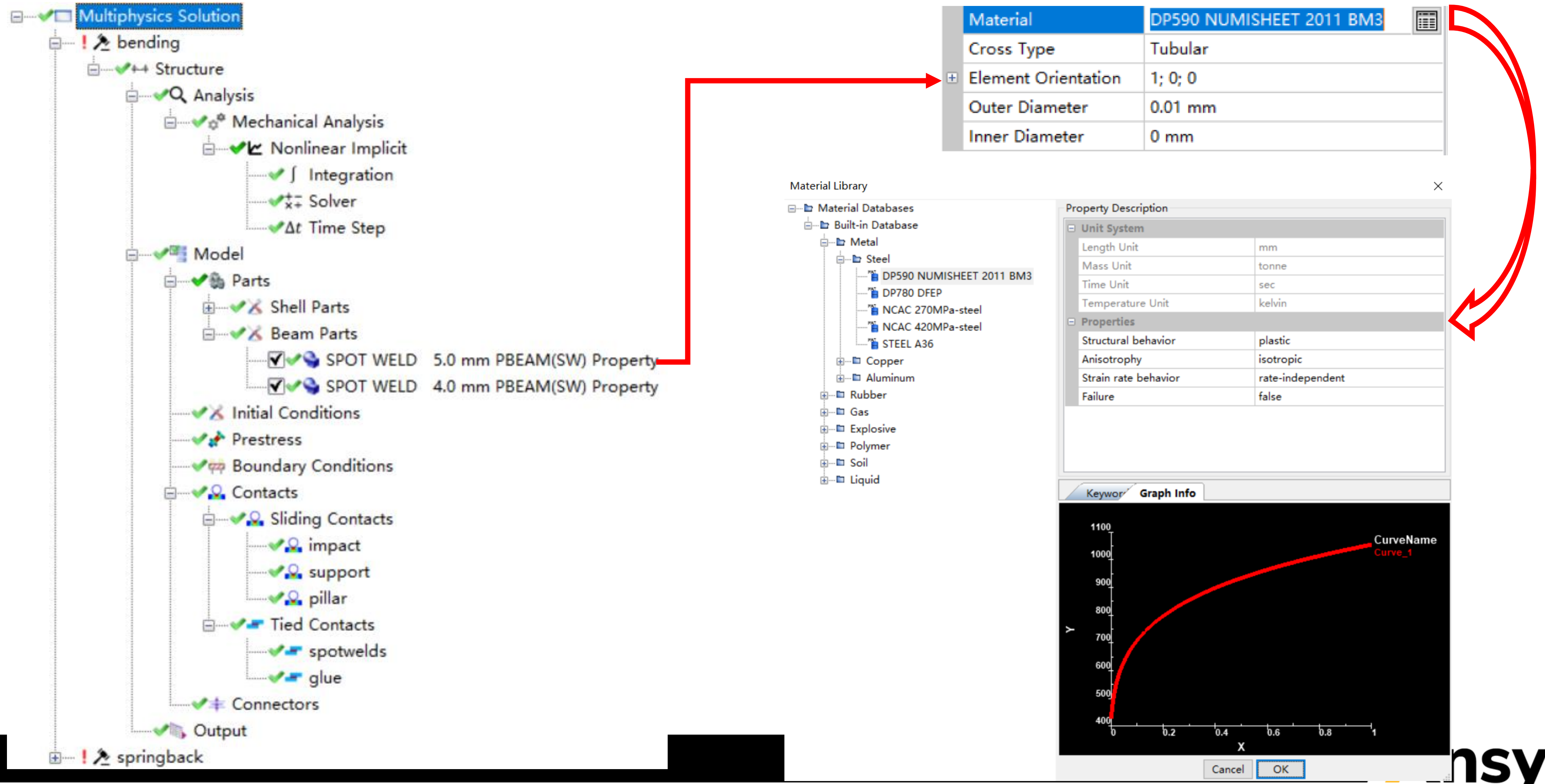
Types	Dynamics
Methods	Newmark Damped
Gamma	0.6
Beta	0.38

Solution Method	BFGS Slightly Nonlinear
Reformations Before Retry	15
BFGS Iterations Before Reform	11
Tolerance Type	Recommended
Relative Displacement	0.001
Advanced	
Geometric Stiffness Matrix	☐
Linear Solver	Symmetric
Use residual tolerances	☐

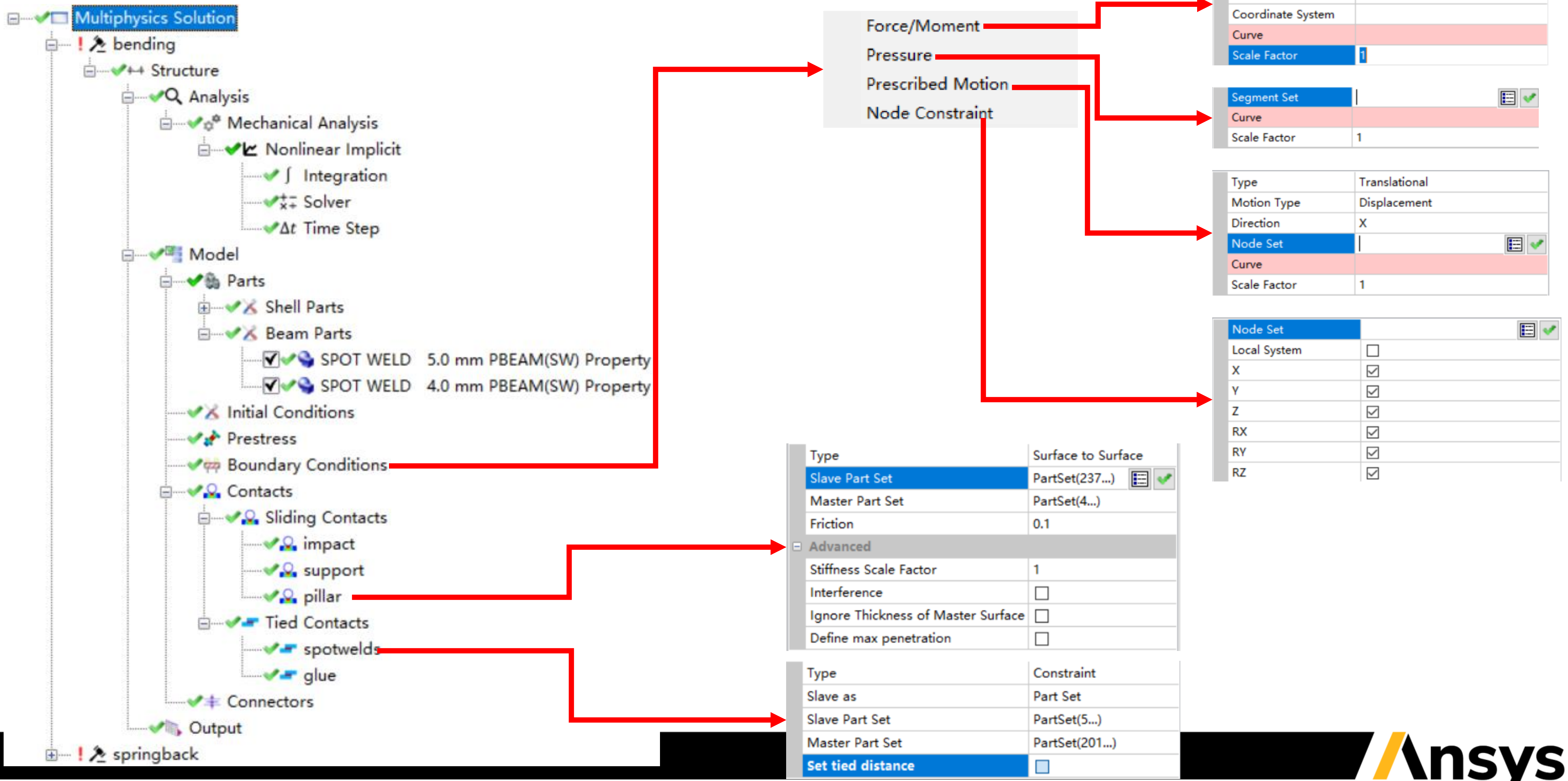
Initial Size	0.01 s
Step Size Option	Maximum Time Step Size
Maximum Time Step Size	0.05 s
Change Strategy	Aggressive
Increase T.S. if iter <	100
Decrease T.S. if iter >	110

Ansys

Solution Explorer - Structural Structure

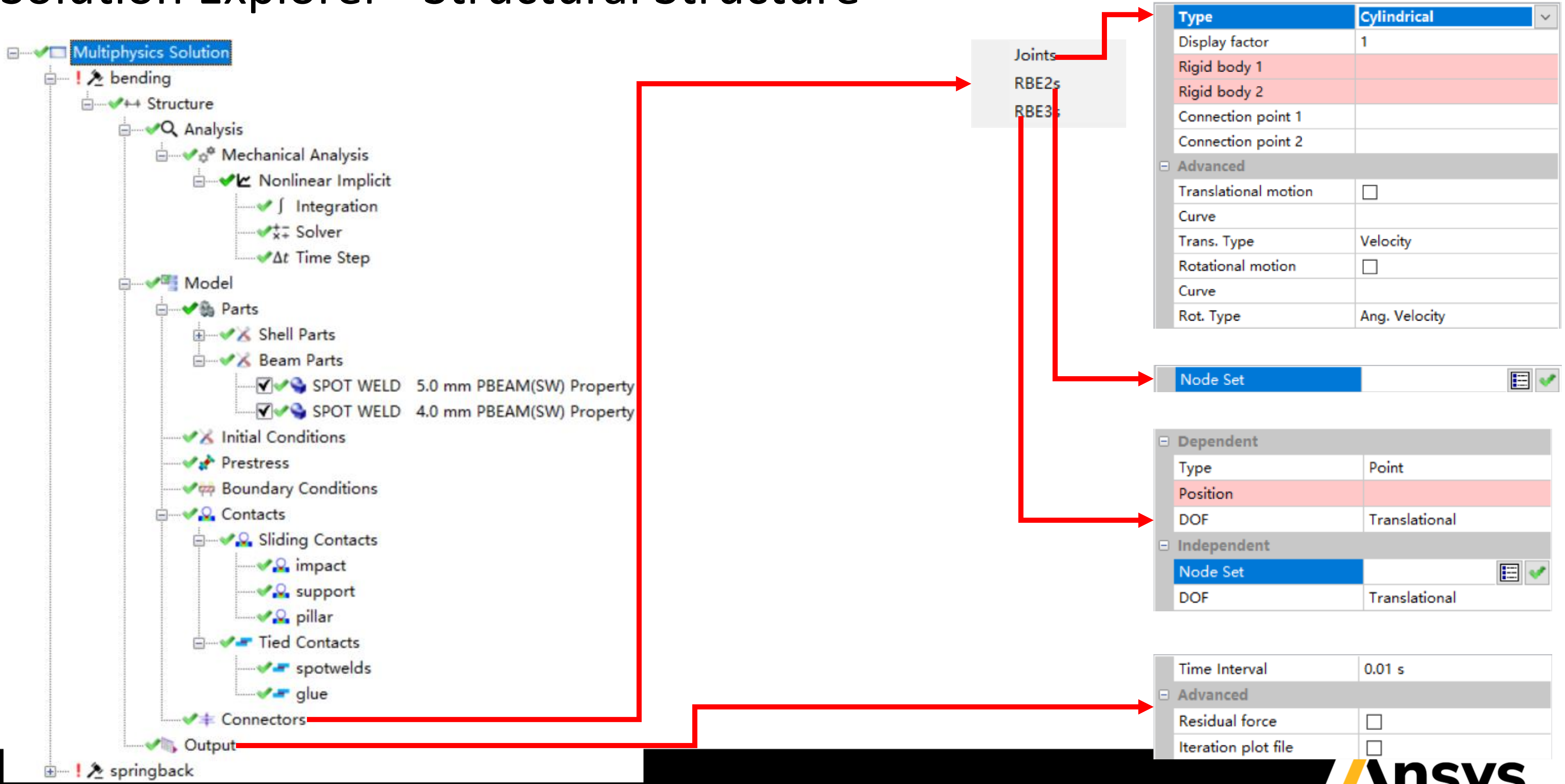


Solution Explorer - Structural Structure



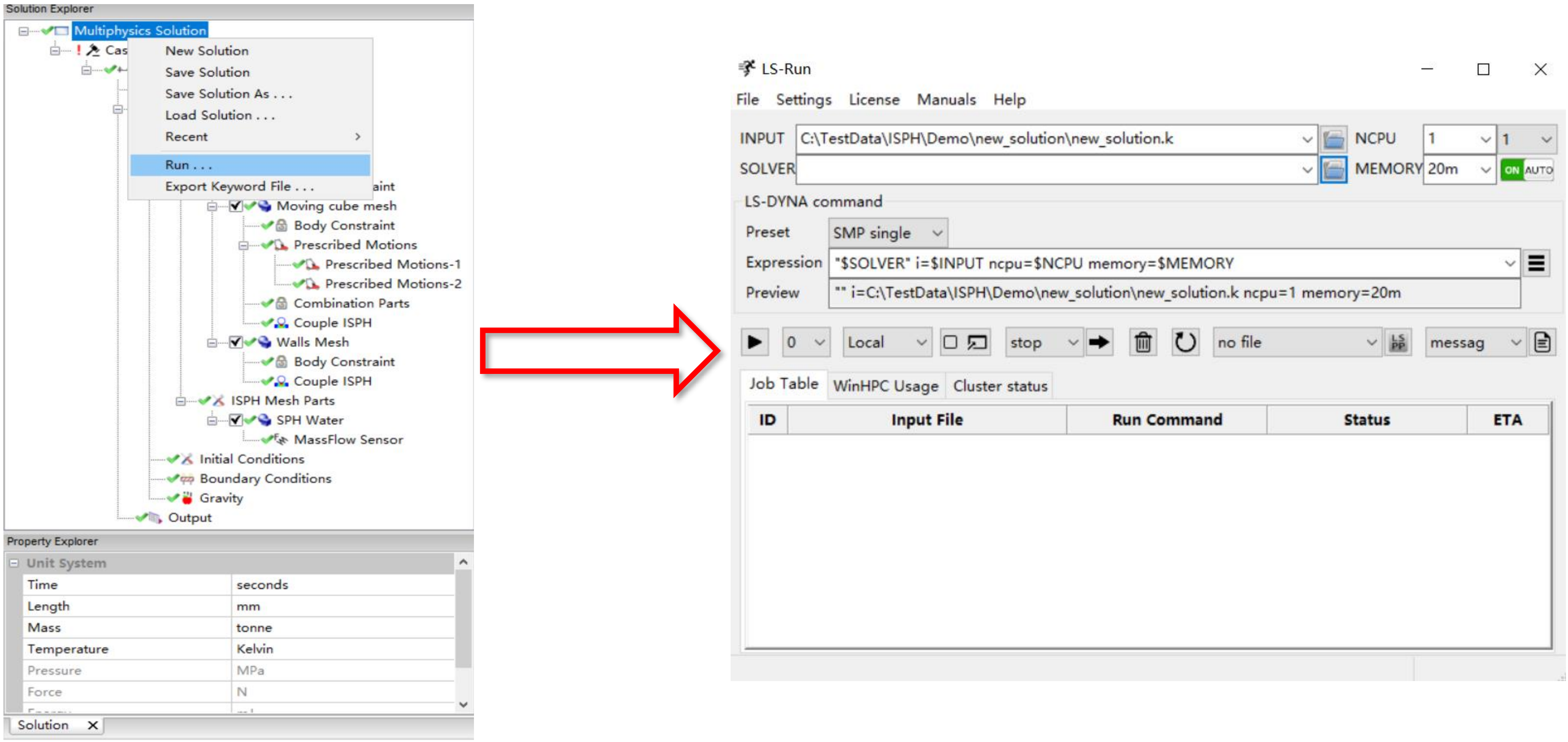
Ansys

Solution Explorer - Structural Structure



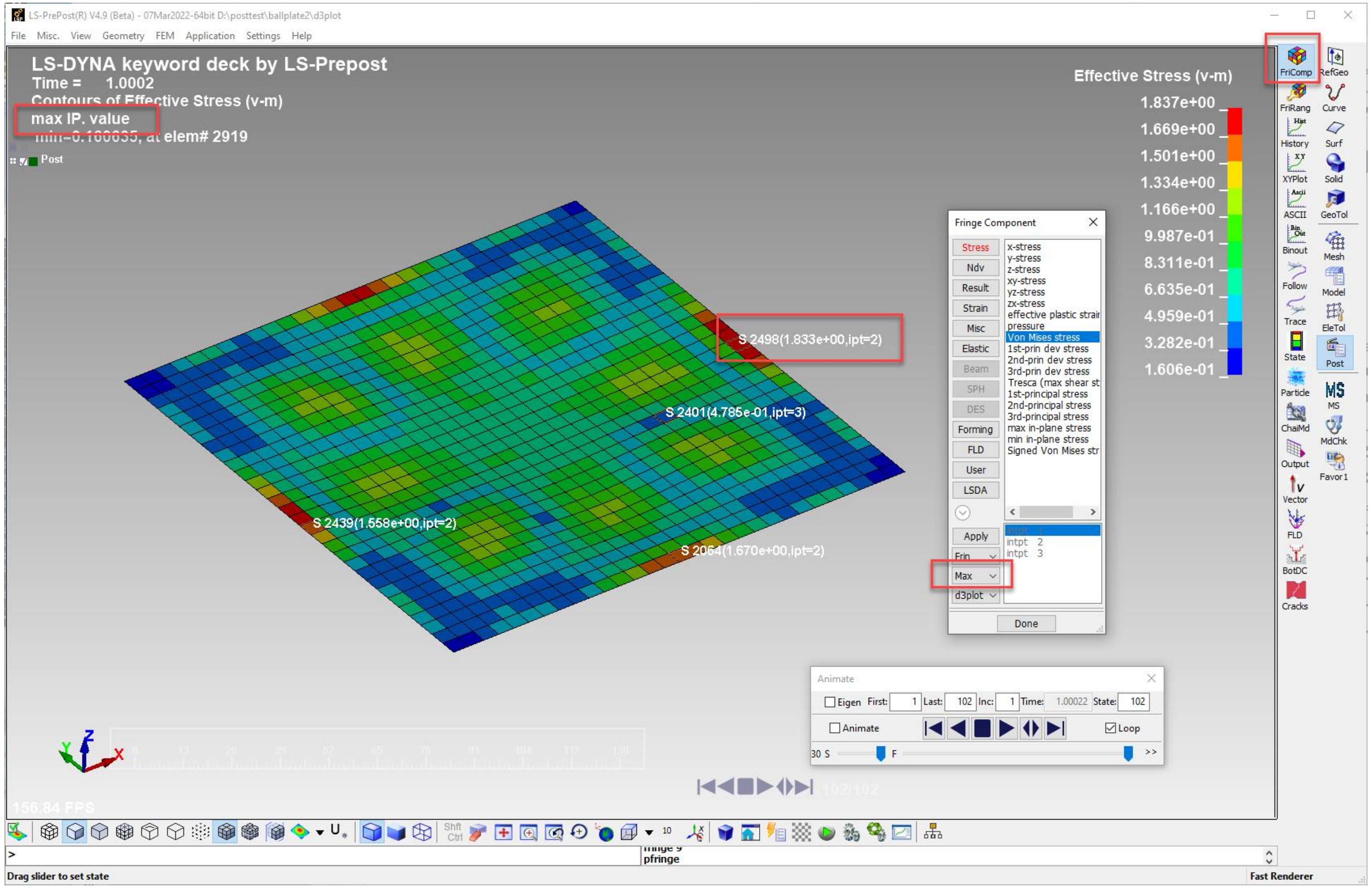
Ansys

Solution Explorer – Export to LS-DYNA



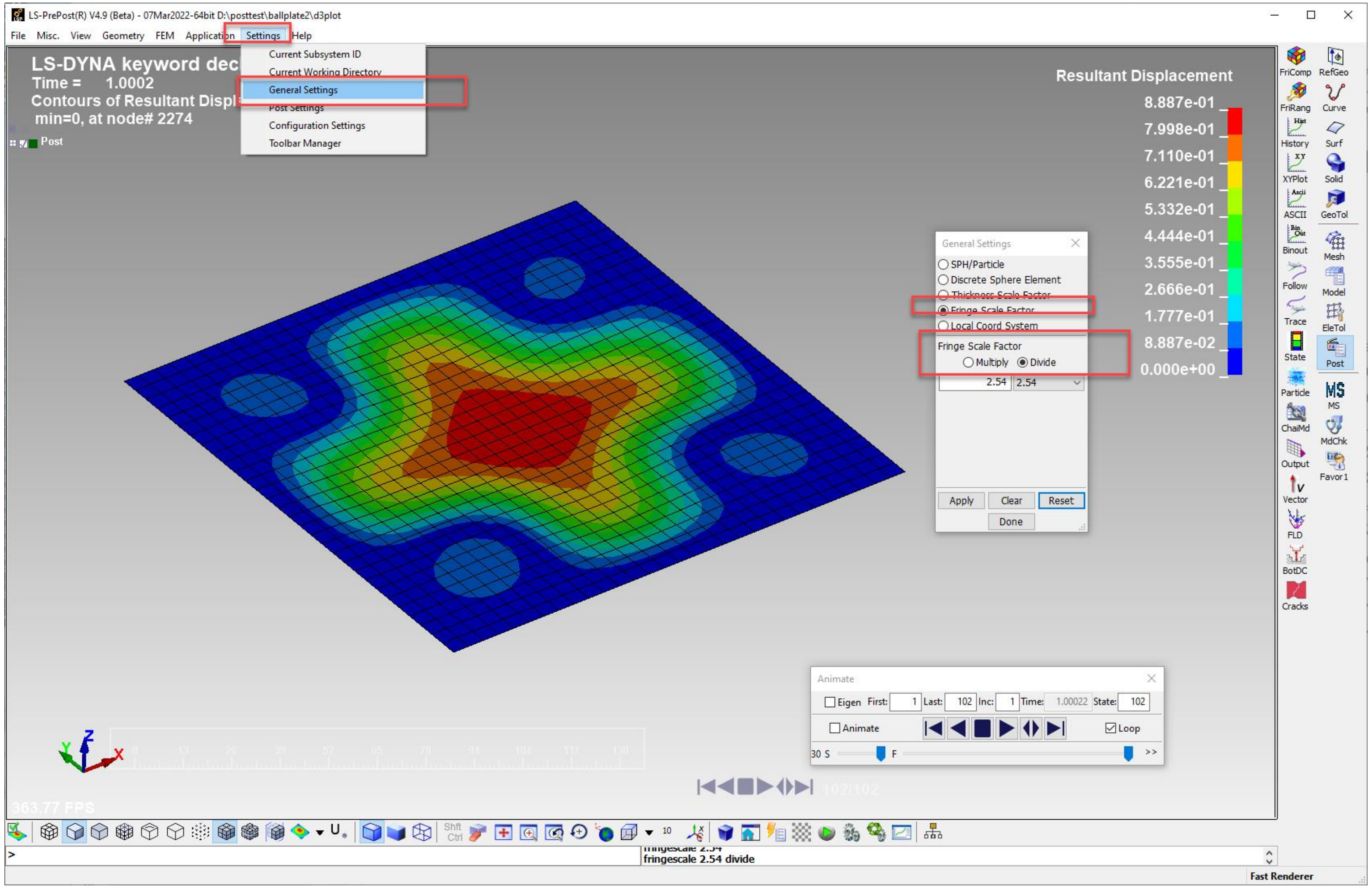
Post-Processing – Show fringe value location

- When Max. fringe value is being shown, identify the result will show max. value is on which layer (ipt)



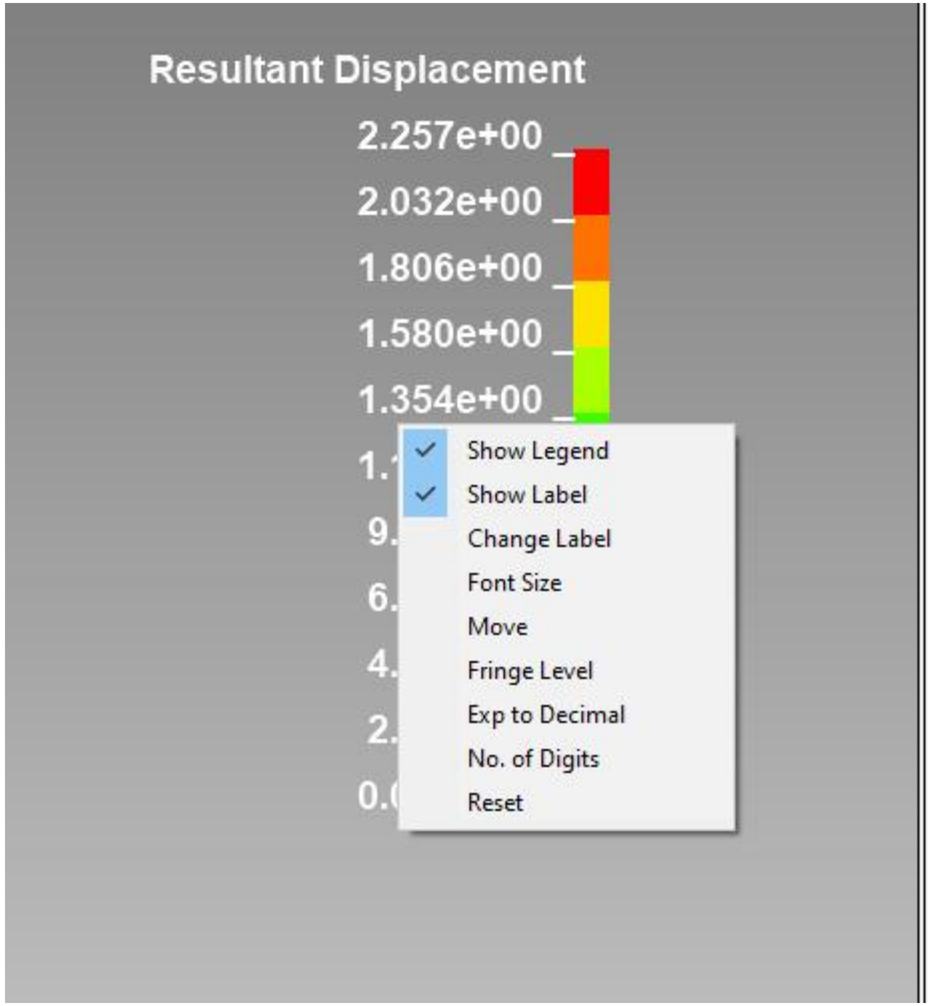
Post-Processing – Scale fringe values

- Applying fringe scale factor to the fringe plot can be Multiply or Divide



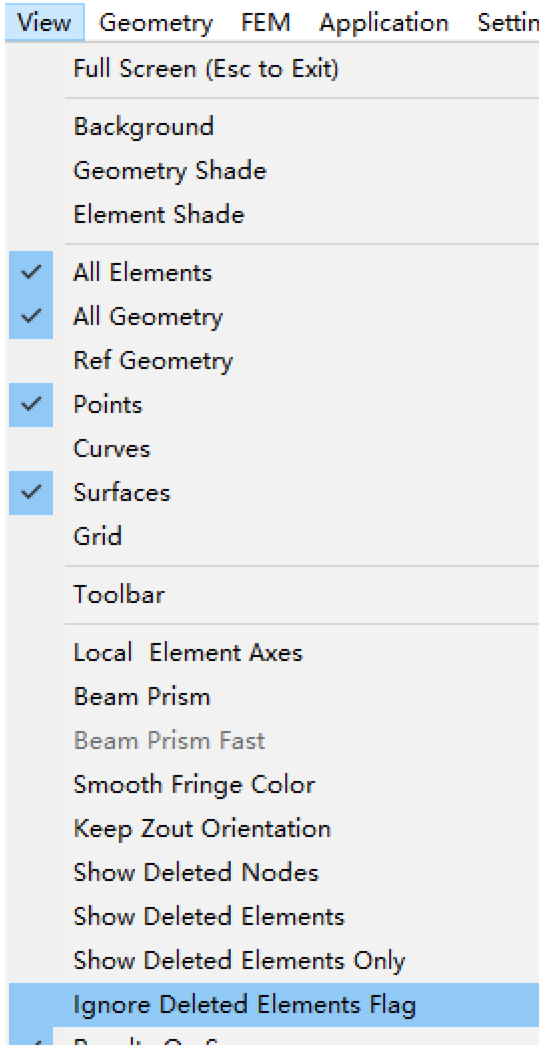
Post-Processing – Option to Change Fringe Legend

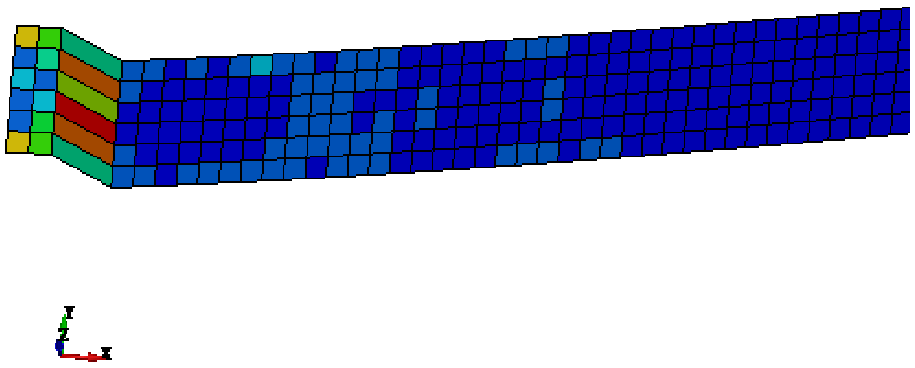
- Many options on the fringe legend can be changed
  - Label
  - Font size
  - Location
  - Fringe levels
  - Exp or decimal presentation
  - No. of digits
- Right click on the Fringe Legend area will popup the dialog

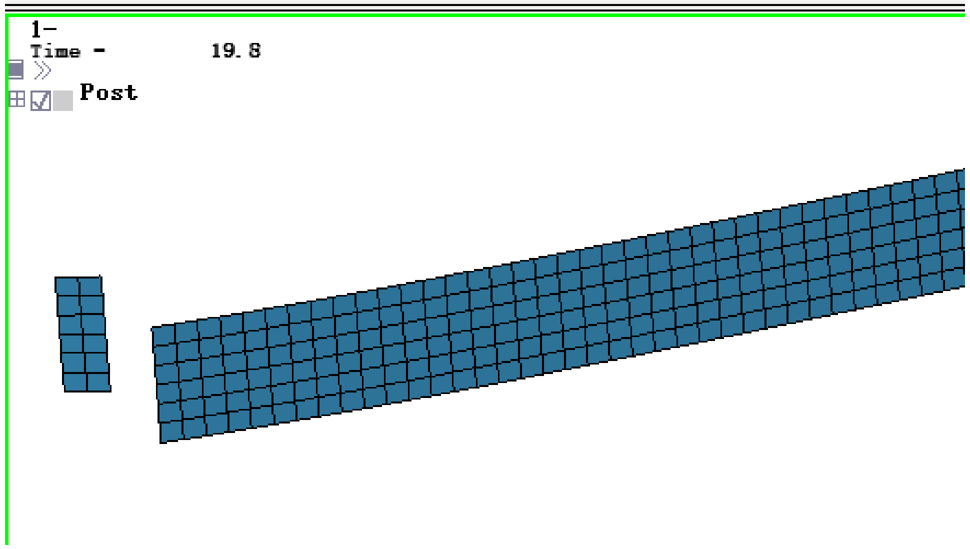


Post-Processing – Ignore "Deleted Elements"

Warning! Not all deleted elements can be visualized

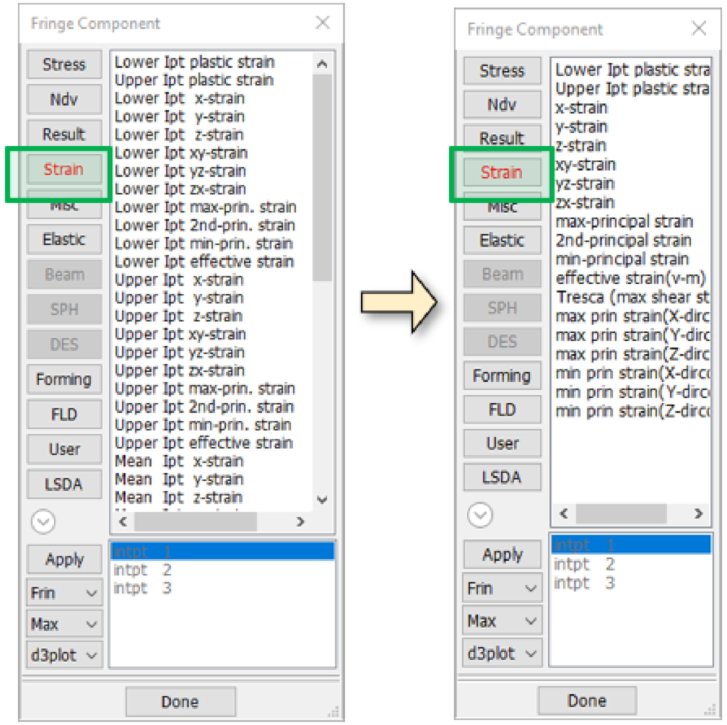


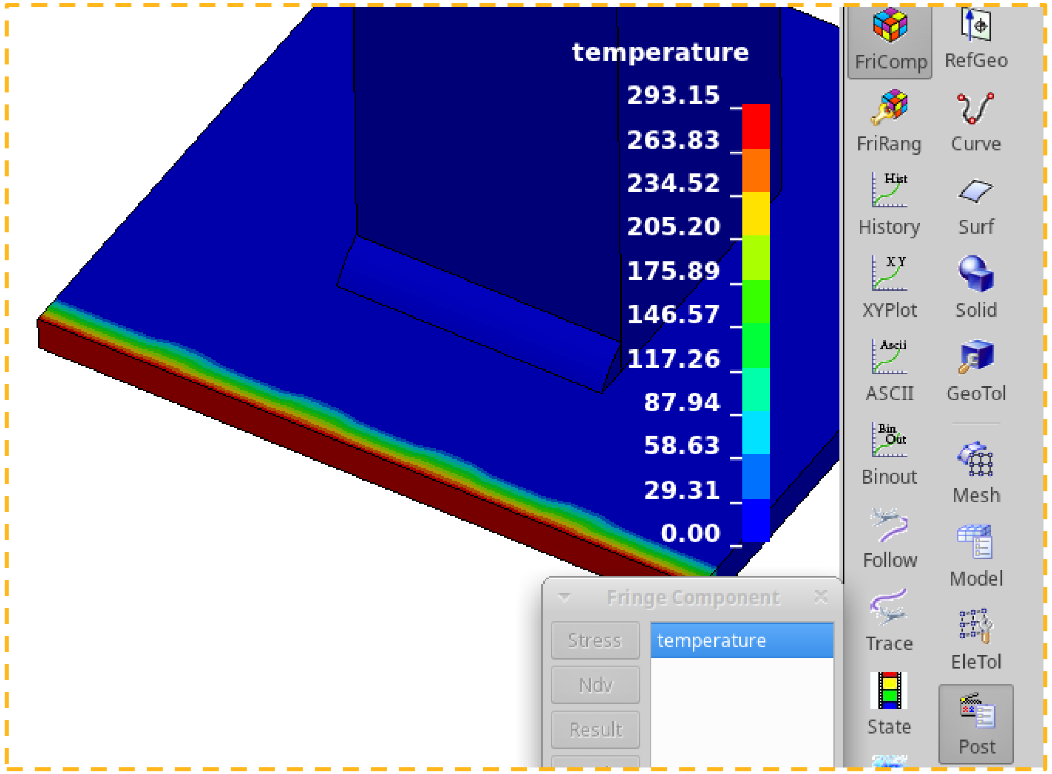




Post-Processing – Misc. Fringe features

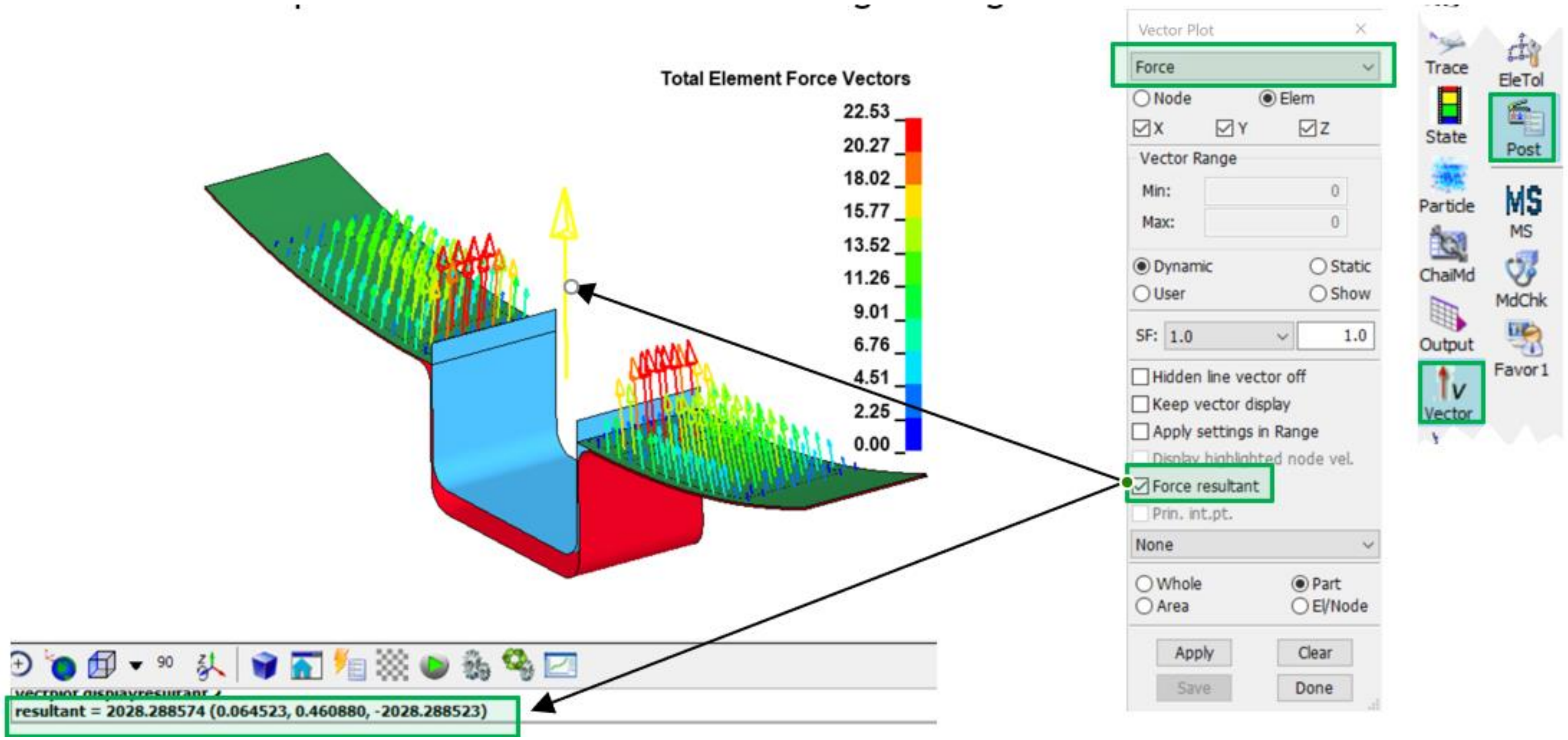
- Iso-surface: support TSHELL
- Fringe now recognizes *INITIAL_TEMPERATURE in LS-DYNA keyword input decks
- Fringe Strain: Reduced components





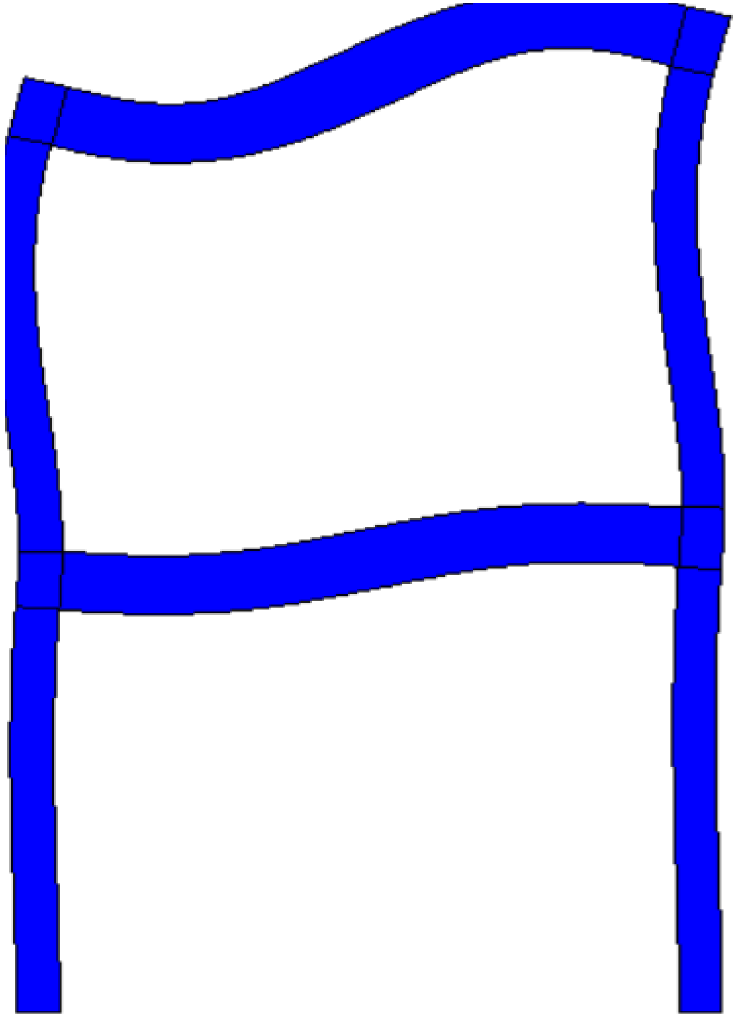
Post-Processing – Force Vector plot with Resultant

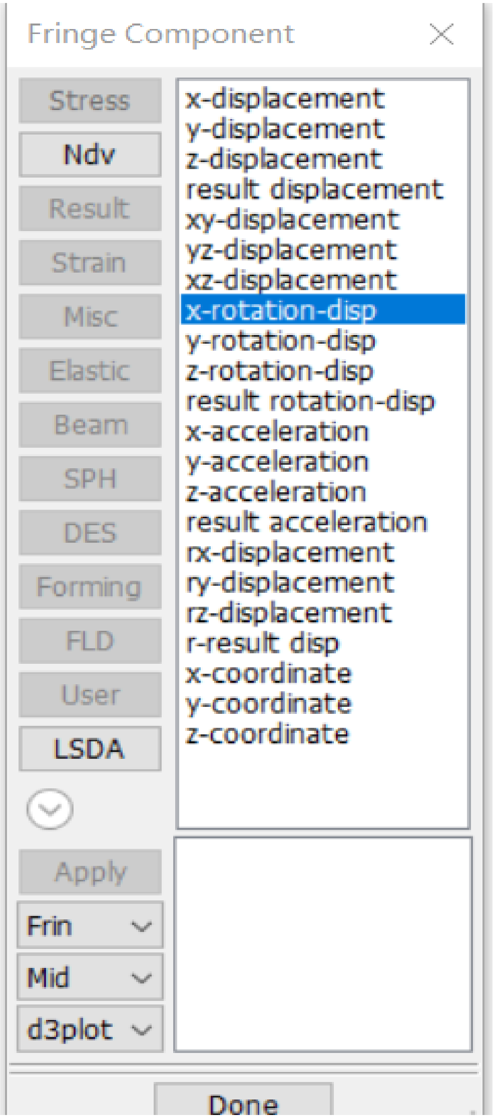
- Resultant force vector is now implemented
  - A yellow resultant force arrow is computed from the currently visible force vectors in an “intfor” file
  - The resultant value and its components are written to the message dialog when the checkbox is toggled ON



Post-Processing – D3eigv or D3mode output

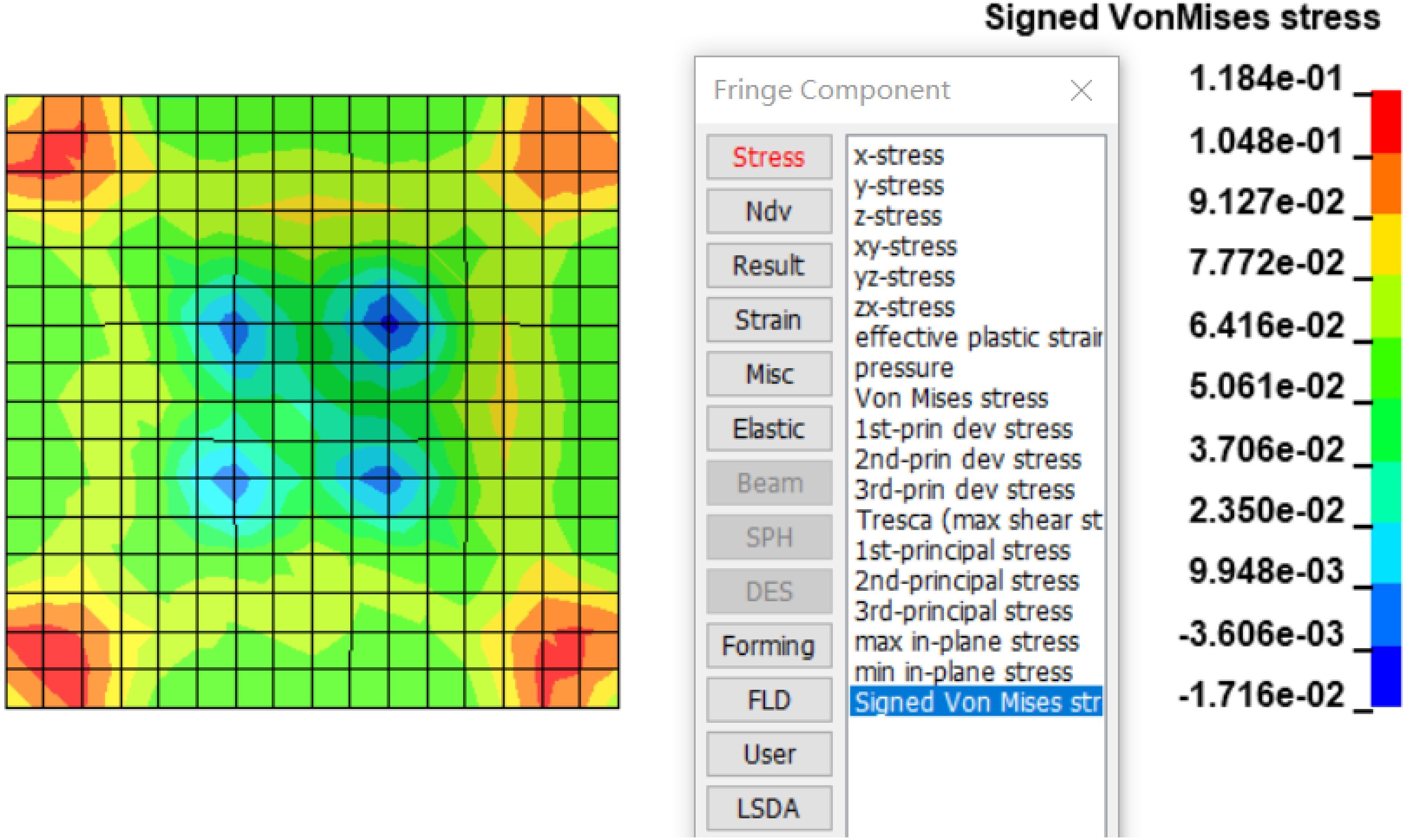
- For shell element rotational displacements have been output at the location used to store the velocity vector
- If the file name is d3eigv or d3mode, LSPP will replace the velocity label with rotation-disp





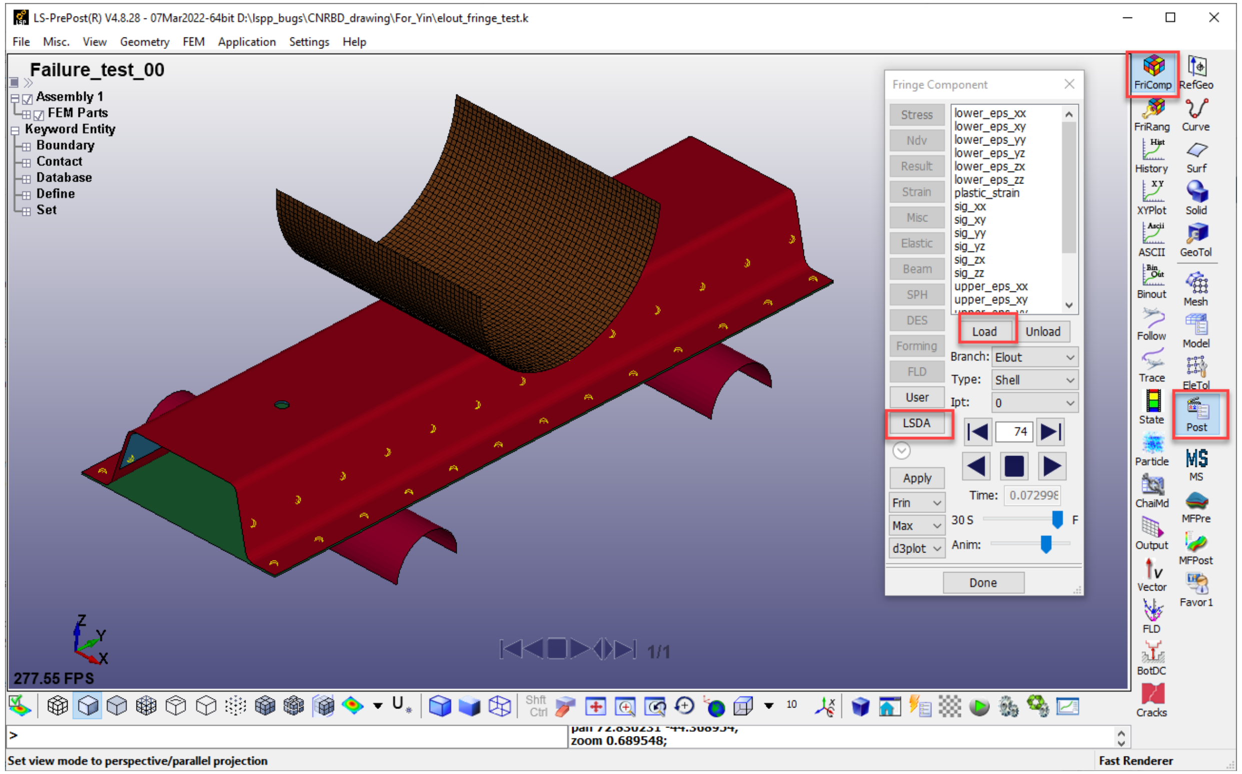
Post-Processing – Signed Von-Mises Stress Fringing

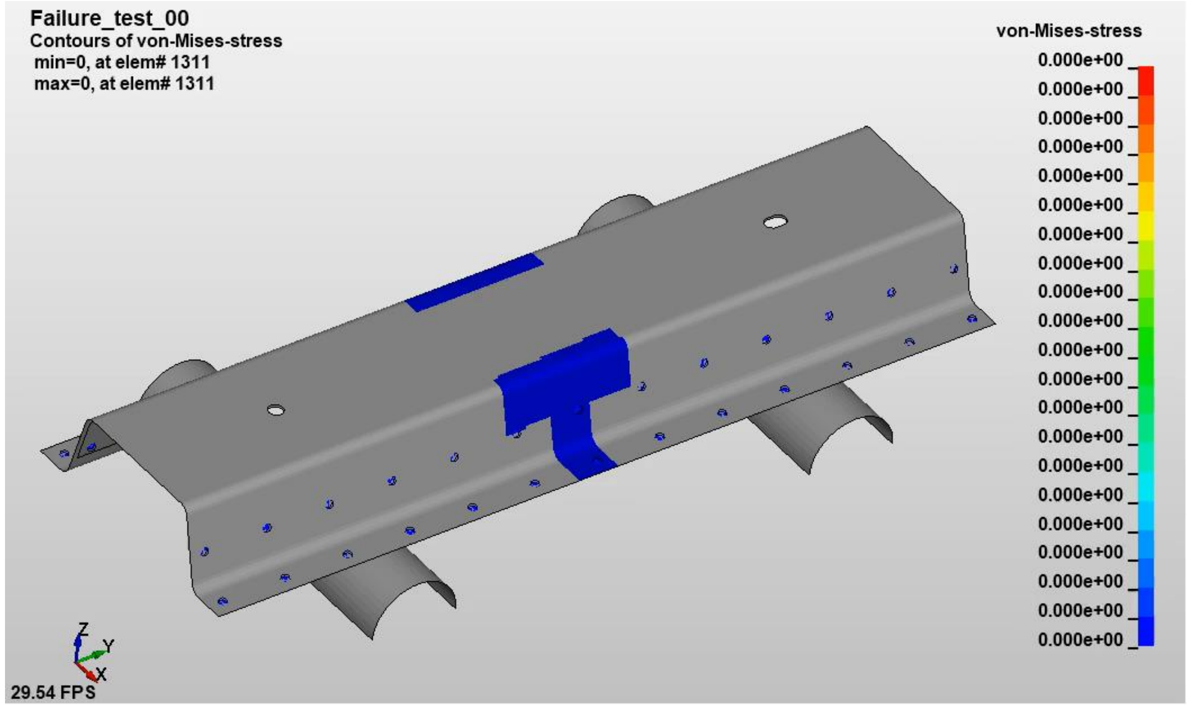
- Signed VM = VM * sign(SXX+SYY+SZZ)



Post-Processing – Fringe Binout (LSDA) data

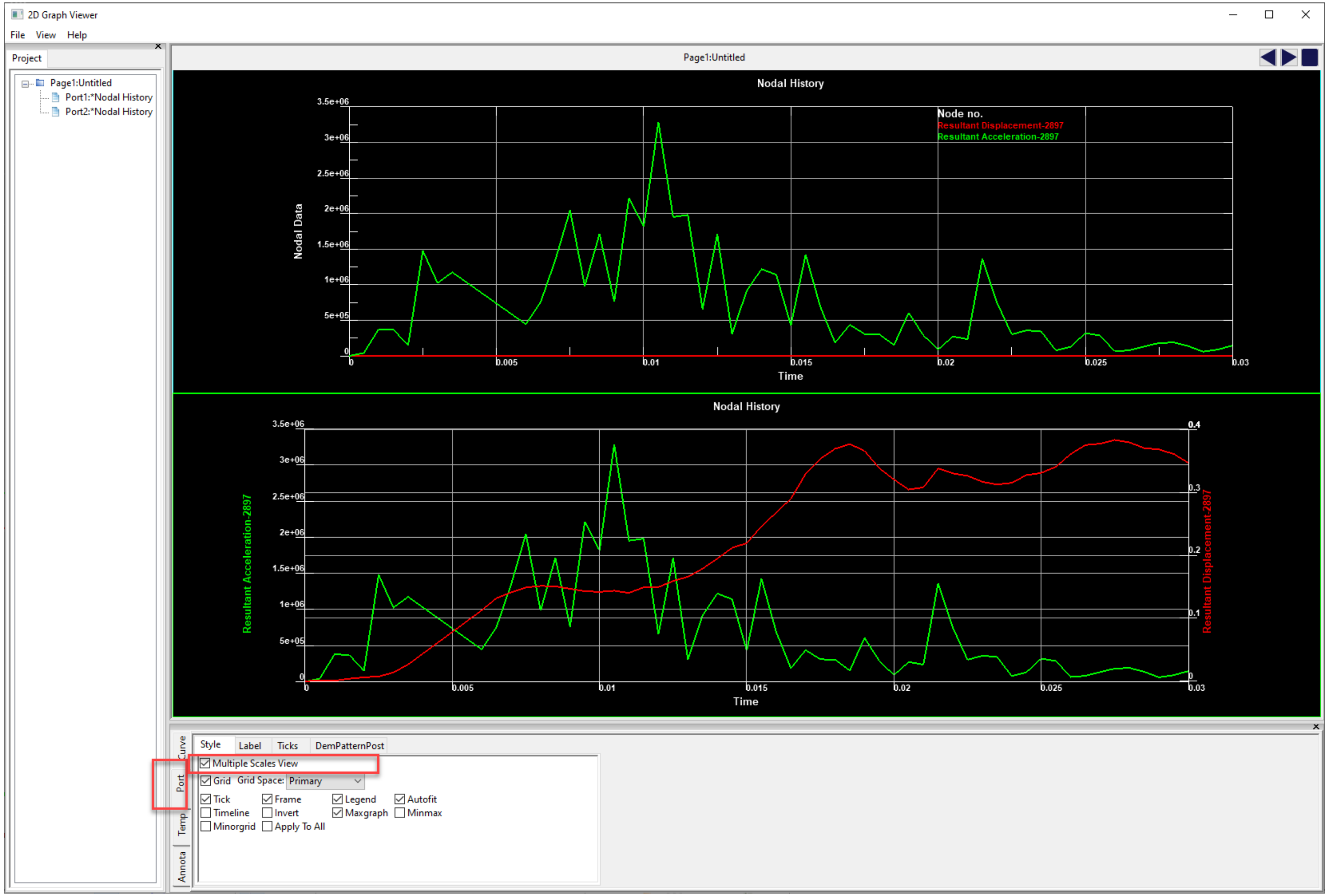
- The elout and tprint branches of the BINOUT data can also be Fringed
- Load the keyword file along with Binout data
- Can use this method to reduce the post data storage





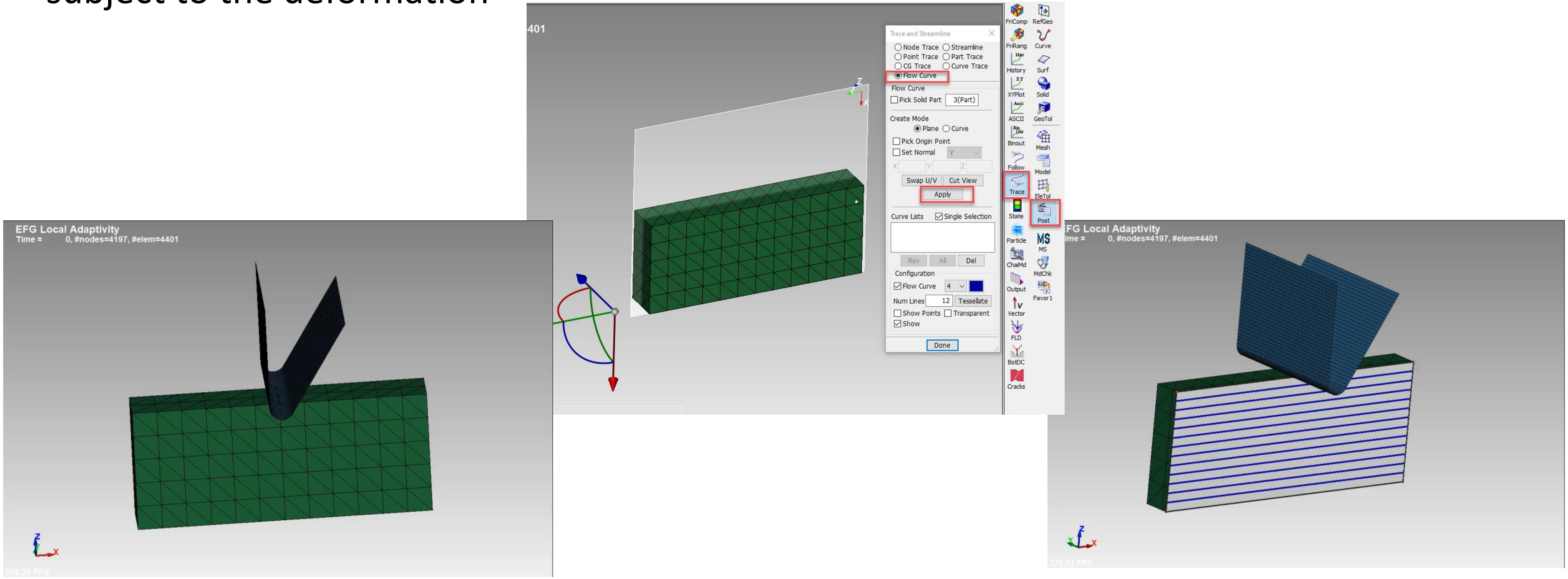
Post-Processing – Multi-Scale XY Plot

- When viewing 2 curves that have different data ranges, one can activate the Multi-Scale option
- Only available in the new XY plot interface



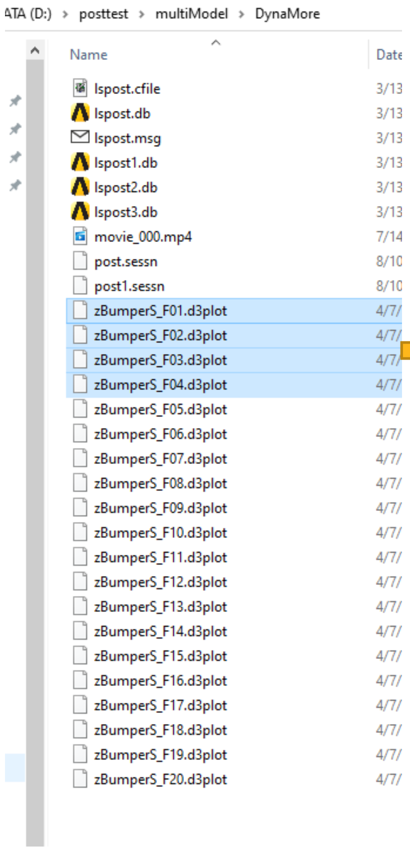
Post-Processing – Flow Line Trace

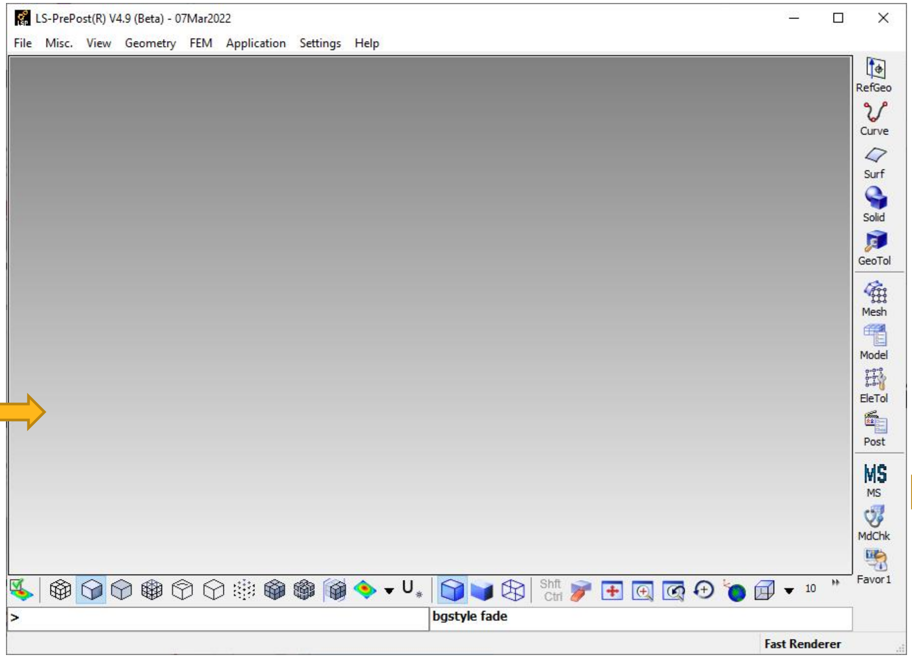
- On a very large deformation part, one would like to see how the material flow when subject to the deformation

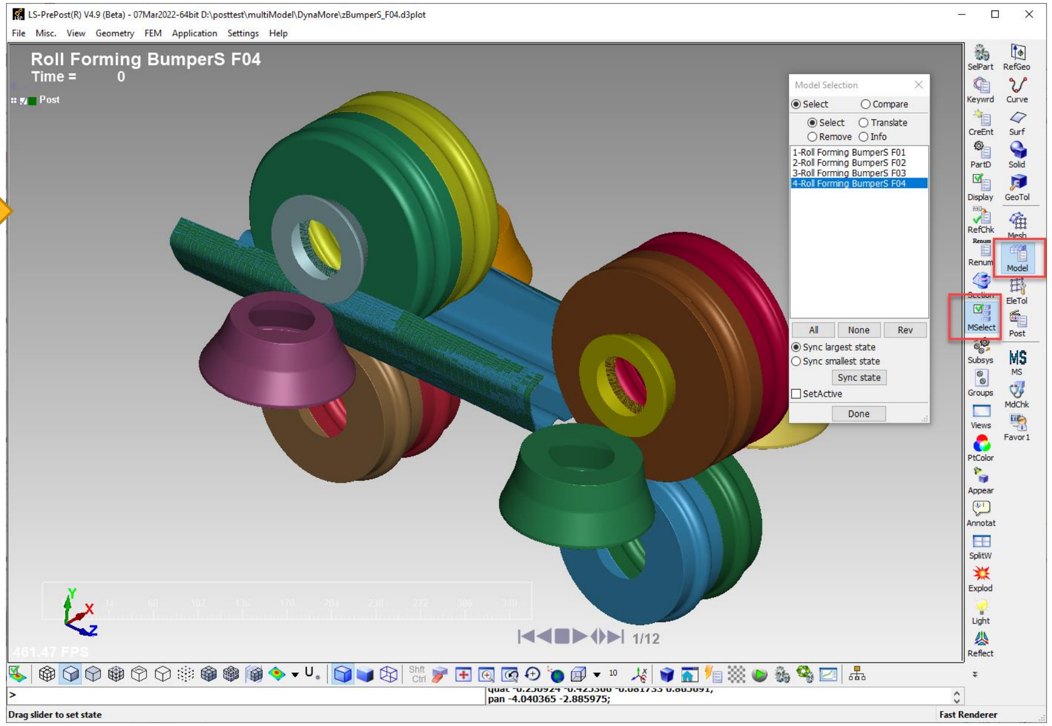


Post-Processing – Open Multiple sets of D3plot file

- One can drag and drop multiple d3plot files into LSPP Windows at once

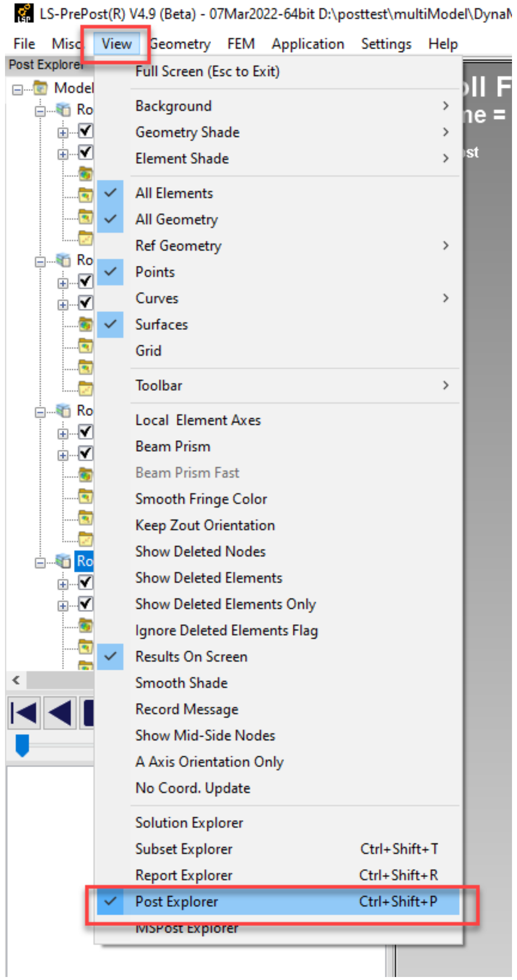


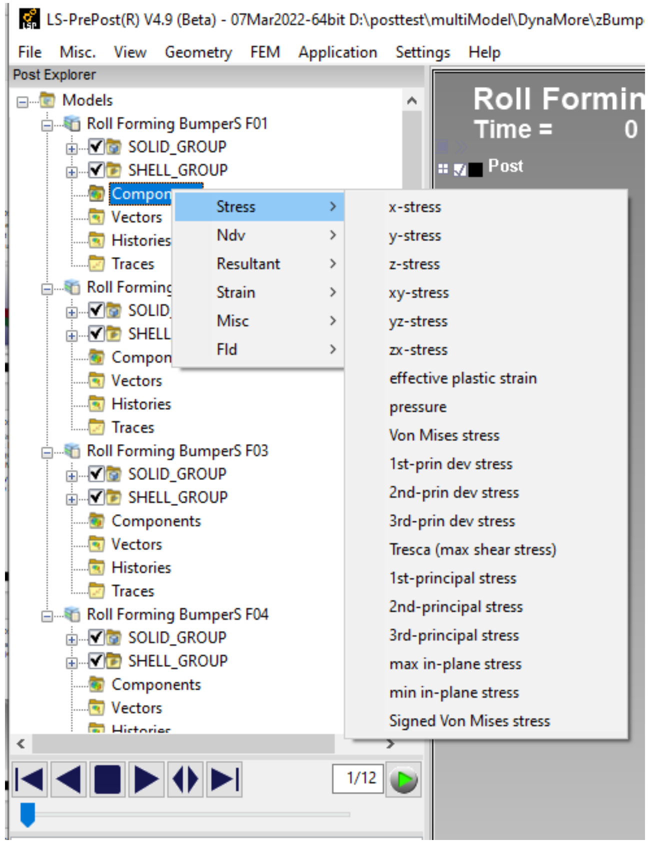




Post-Processing – Introduction of Post Explorer

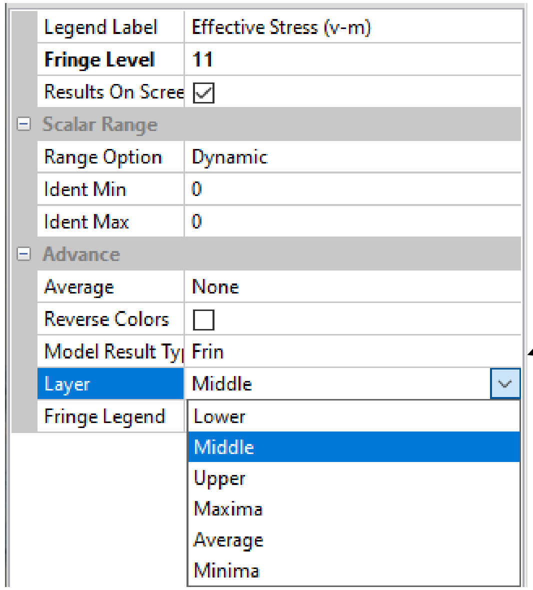
- A new paradigm of LS-DYNA post-processing in LS-PrePost
- It is feature tree based operations
- To activate: pull down menu “View”->”Post Explorer”
- Multiple models will be listed
- Each model has its own Components, Vectors, Histories, and Traces main objects
- Right click on these objects to select sub-object

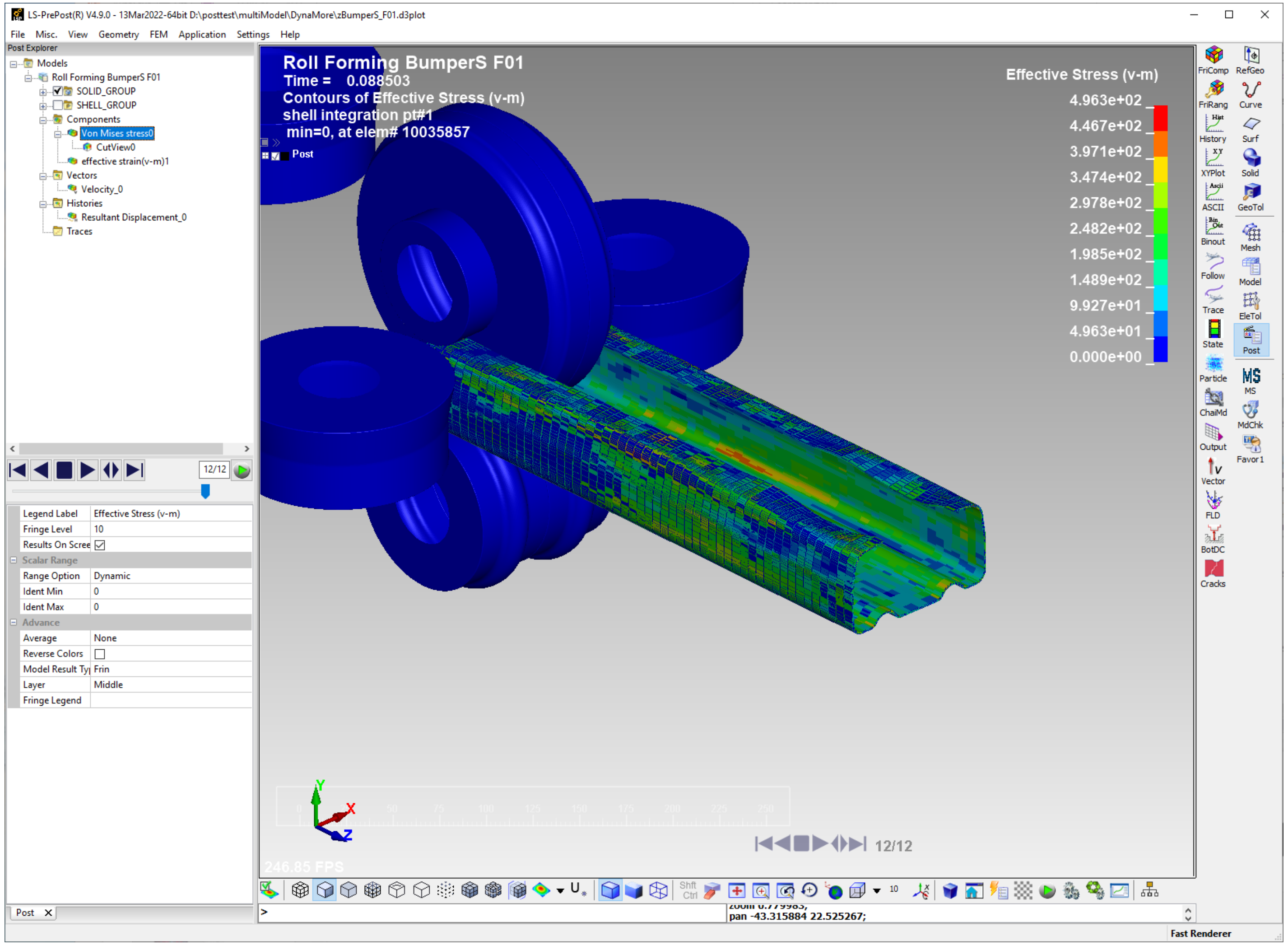




Post-Processing – Introduction of Post Explorer

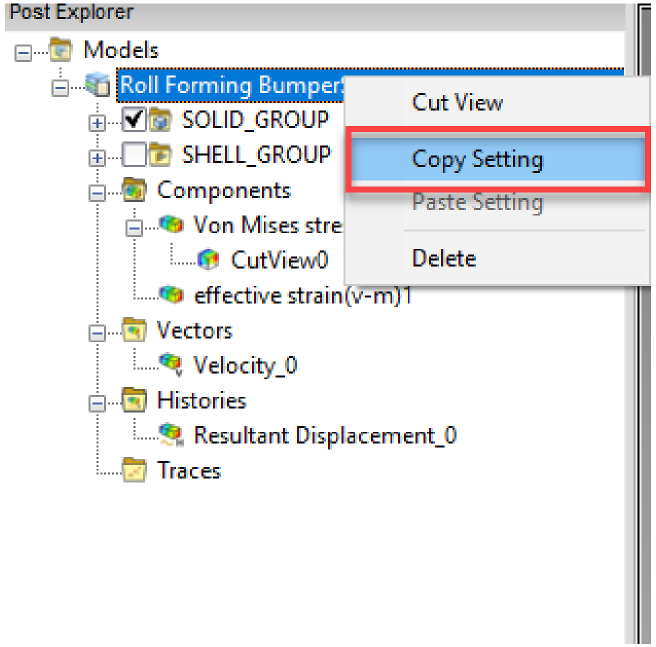
- Each selected object has its own property tree
- Many quantities associated with this object can be changed on the property tree

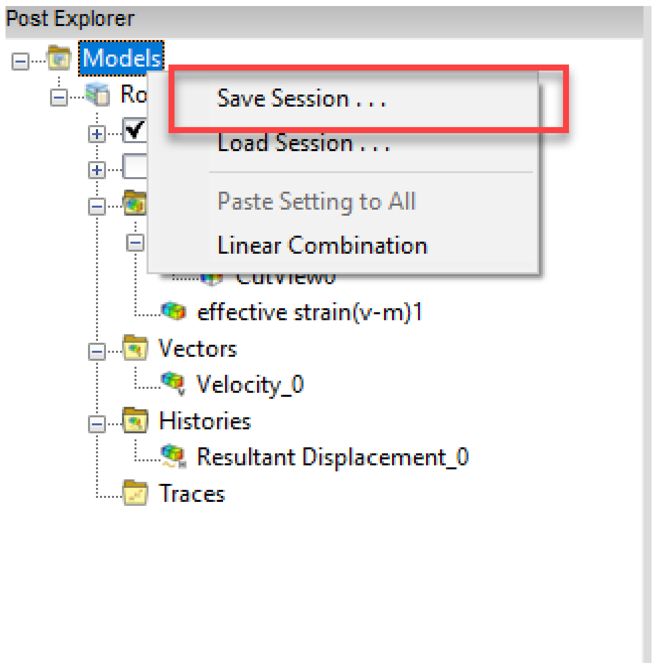




Post-Processing – Introduction of Post Explorer

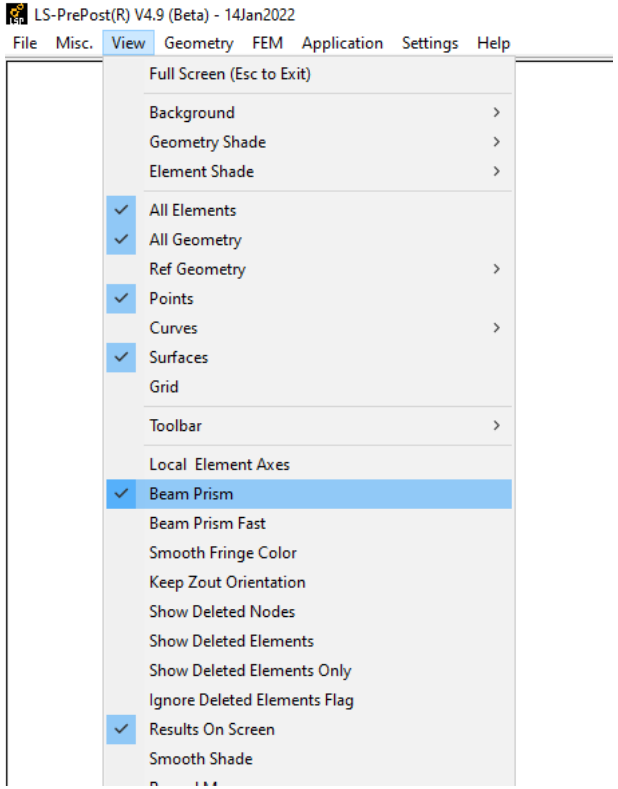
- Many objects with their properties can be setup in the feature tree
- The Settings for one model can be copied and pasted to other model (if multiple modes have been loaded)
- All settings for the session can be saved to file and be loaded back in the future

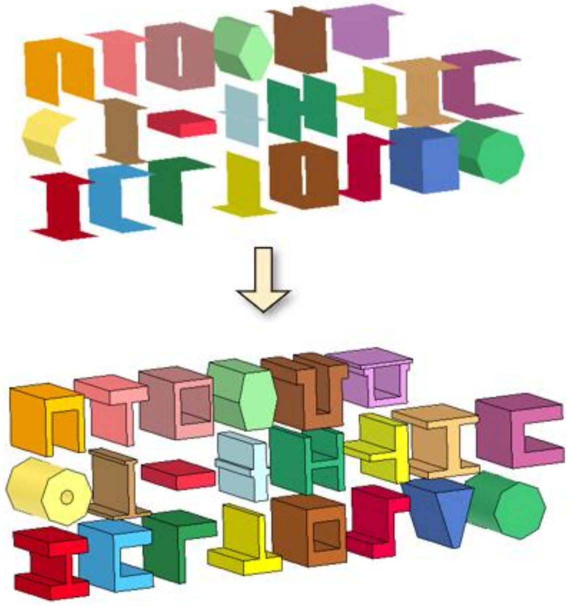




Misc – Beam Prism Visualization

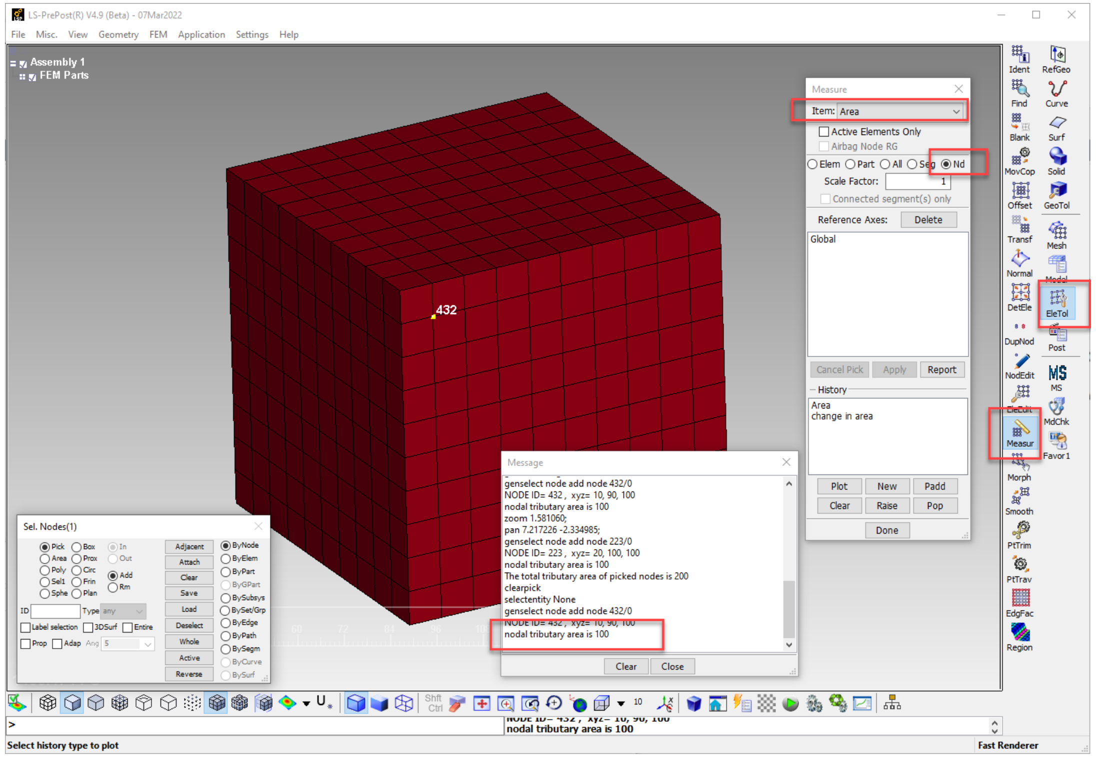
- Beam prism visualization for *INTEGRATION_BEAM has been revamped, showing the proper geometry
  - Various enhancements: *ELEMENT_BEAM_OFFSET/ORIENTATION well represented
- Turn on "Beam Prism Fast" if there are too many beam to be drawn as Prims. During rotation of the model, beam elements will be drawn as line until model stop rotating

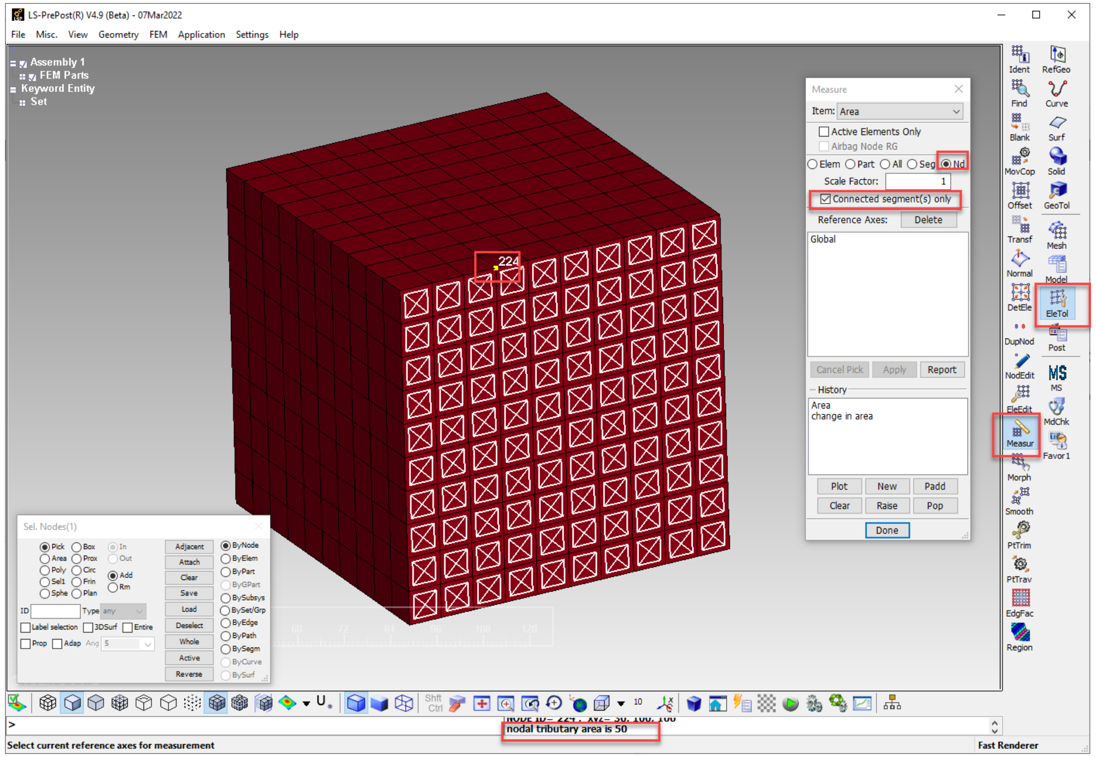




Misc. – Tributary Area Measurement

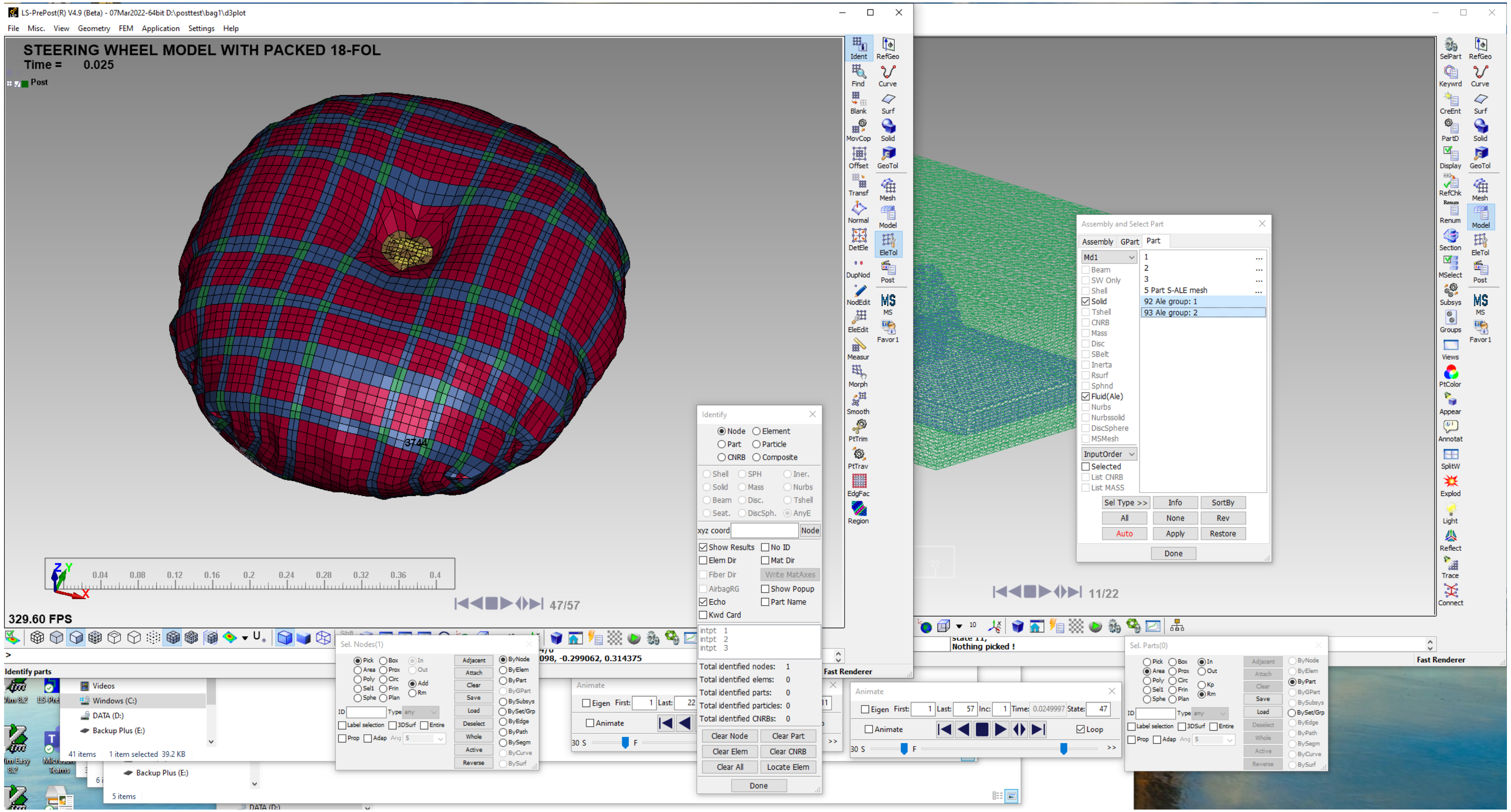
- Check “Nd” option to measure the nodal tributary area
- At the edge of the model, use “Connect Segment only” option and with segments to avoid including the unwanted neighboring element



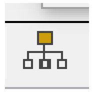


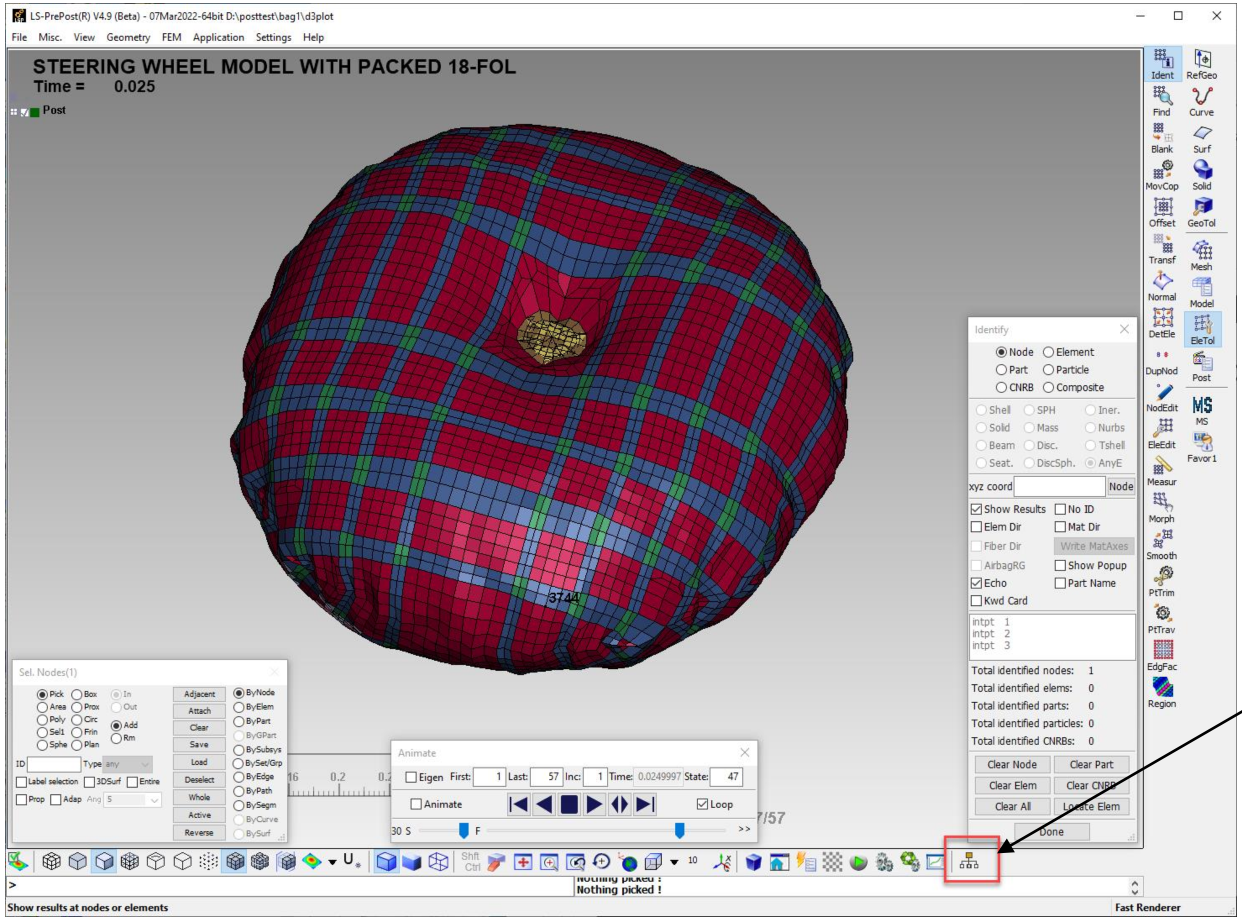
Misc – Multiple LSPP Sessions with Multiple Dialogs

- It may be confusing to identify which pop up belongs to which session



Misc – Multiple LSPP Sessions with Multiple Dialogs

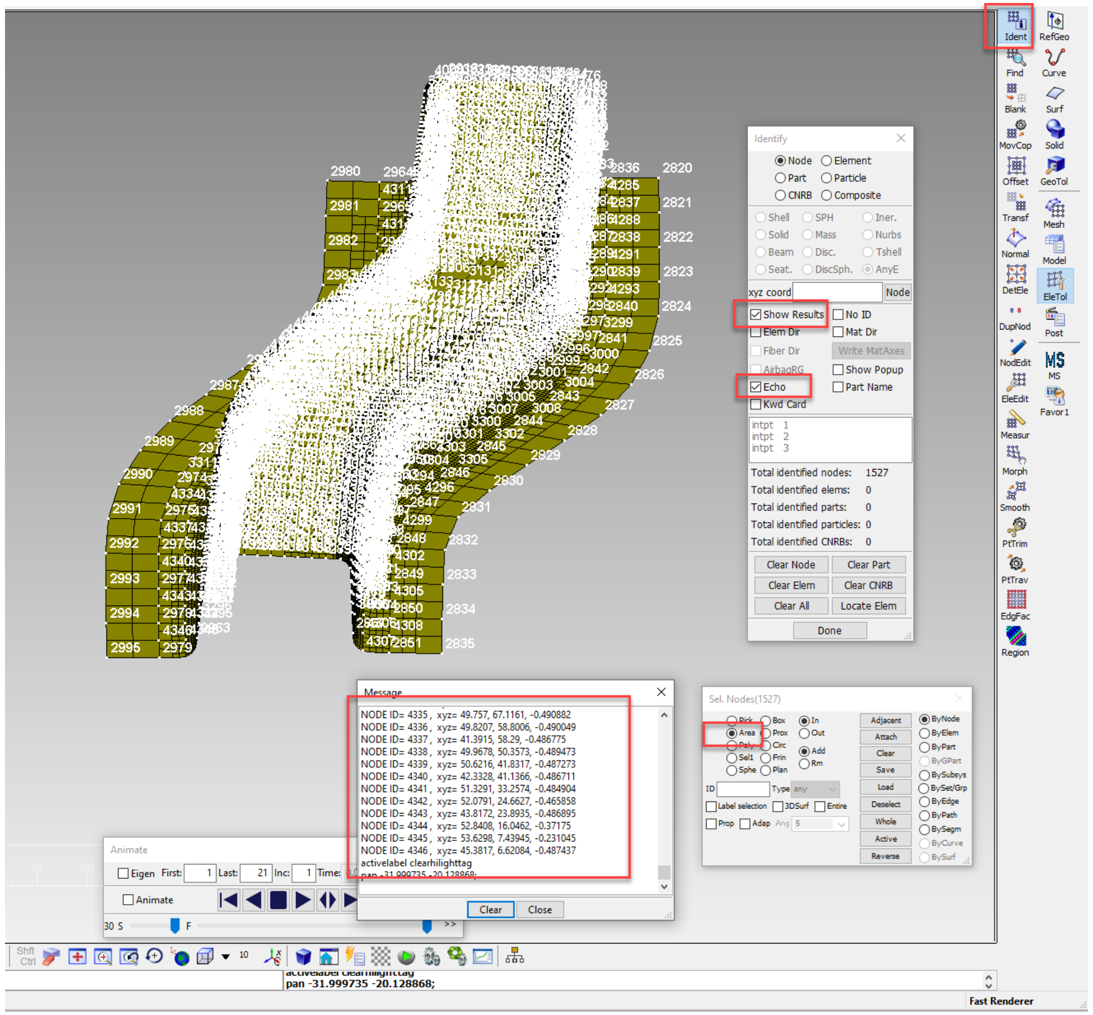
- The “Windows Relocate” icon  will bring all the sub-windows to the session’s man windows



The “Windows Relocate” Icon

Misc – Use ESC key to Stop Echoing large amount of data

- When identify node/element data in the IDENT interface, often using the area select will end up waiting for a long time for the output to finish
- Esc key will stop such operation

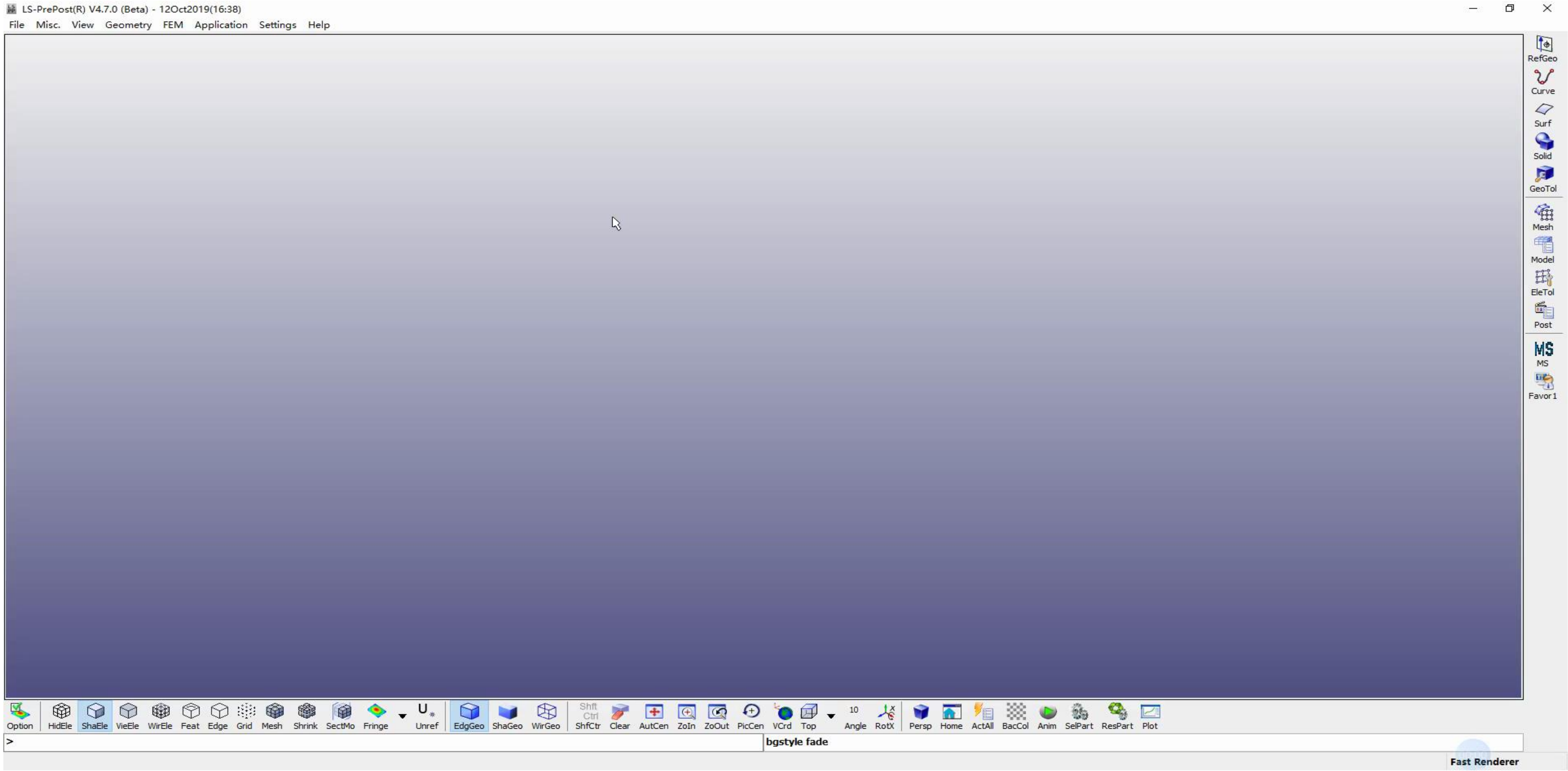


Ansys

Conclusions

- The development version LSPP4.10 will be formally released early 2023 (to match Ansys R2023R1)
- LSPP4.8 and 4.9 will only have updates for bug fixes
- Continue to focus in Solution Explorer and Post-Explorer developments
- Continue to keep up with the LS-DYNA's demand in pre- and post-processing requirements
- User's suggestions and requests make LSPP better and more robust

Solution Explorer Demo Video

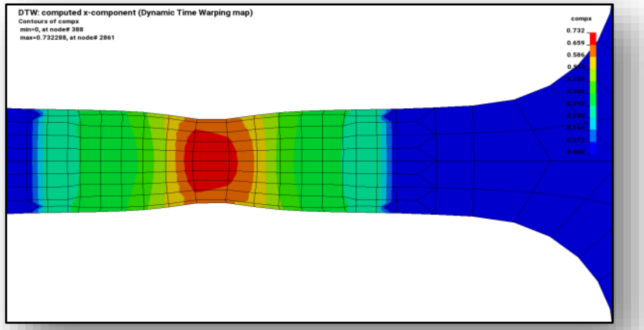


LS-OPT® 2023R1

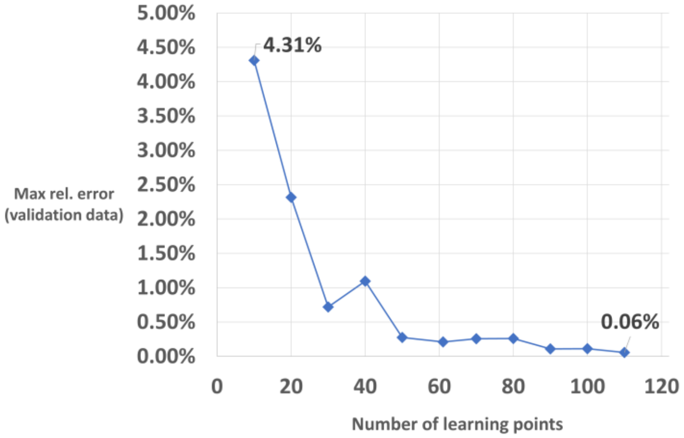
Ansys

Fast surrogate models with LS-OPT/LS-DYNA/Ansys Twin Builder

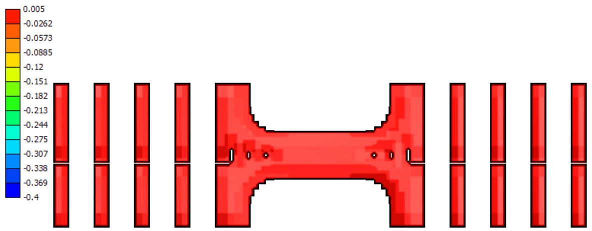
- ***Challenge: Interactive optimization using full-field LS-DYNA models***
- **Use *Digital Twin* as a surrogate**
- *Calibration example:* 2 static material parameters varying across scenarios:
  - *c [0.01 to 5], n [0.001 to 1]*
- Model: 122 spatial samples with LS-OPT
  - 376 strain values + 81 time frames
- ROM created with 61 learning time states evaluated at $n = 0.9$ and $c = 0.509$. Validation with the largest relative error: ***0.22%.***



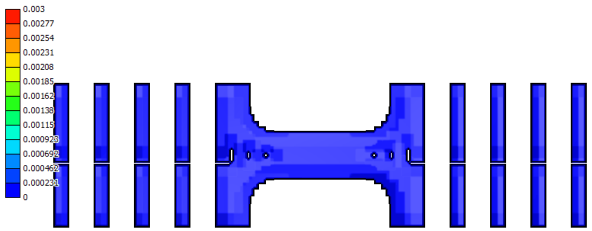
LS-DYNA FE model



Learning point relevance



yy-strain over time



Error in yy-strain over time

- Available as solver/metamodel for all *LS-OPT* tasks in **2023R2**

Twin Builder ROM model output with material [c =0.9, n =0.509]

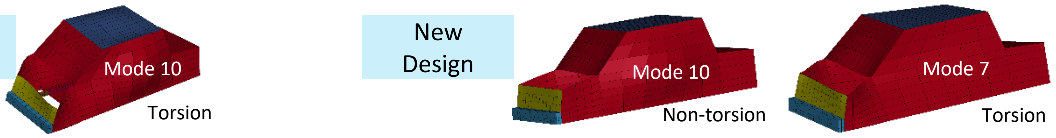
- *In collaboration with: V. Morgenthaler, C. Grivot, O. Crabbé (Ansys Digital Twin)*
- *Example by DYNAmore GmbH, Stuttgart-Vaihingen, Germany*

LS-OPT: Mode Tracking in the Presence of Shape and Meshing Changes

- Mode tracking is important because modal frequency can be part of a design problem.
  - Variable changes can induce changes in the structure
  - Mode ordering may change
  - If a particular mode shape is of interest then its ordering must be tracked.
  - The topic has seen interest from a leading electronics company.

Reference Mode 10

New Design



- Current Mode Tracking using Modal Assurance Criterion (MAC) in LS-OPT requires identical mesh for eigenvector comparison.

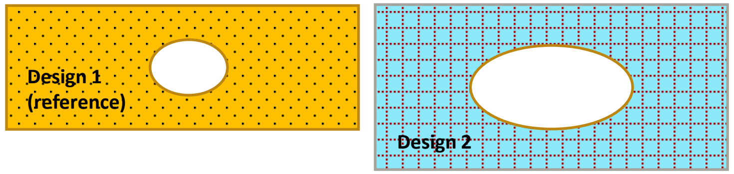
Reference mode

Compared mode

$$\max_j \frac{\{\varphi_0\}^H\{\varphi_j\}\{\varphi_j\}^H\{\varphi_0\}}{\{\varphi_0\}^H\{\varphi_0\}\{\varphi_j\}^H\{\varphi_j\}} = \max_j MAC_j$$

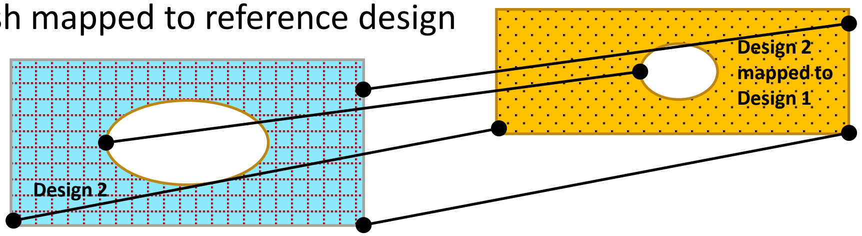
Eigenvectors φ_0 and φ_j must have same length and node order

- **Mode Tracking with Varying Geometries** – incompatible node sets → eigenvectors not comparable for MAC calculation

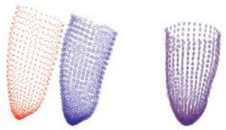


Capability requested by a leading electronic device company

- **New method enables MAC calculation for varying shape/mesh**
  - New mesh mapped to reference design



- Point clouds spatially mapped using Gaussian mixture model based *<u>non-rigid</u> point set registration (Coherent Point Drift)*
  - Mapping involves a regularized displacement function that can combine *rotation/translation/scaling/deformation*.



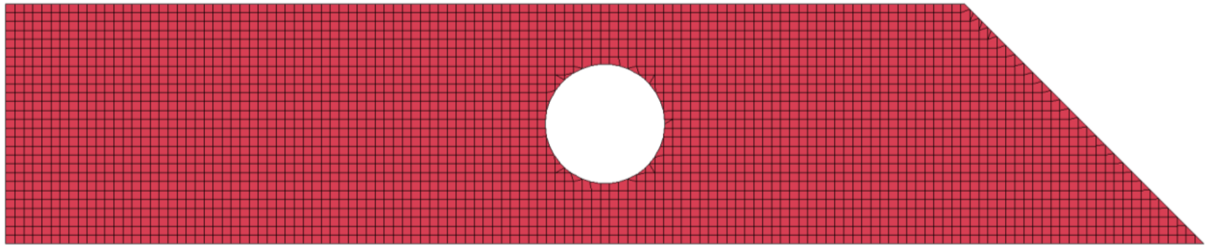
Myronenko A, Song X. Point set registration: Coherent point drift. IEEE transactions on pattern analysis and machine intelligence. 2010 Mar 18;32(12):2262-75.

Non-rigid point registration example: left ventricle

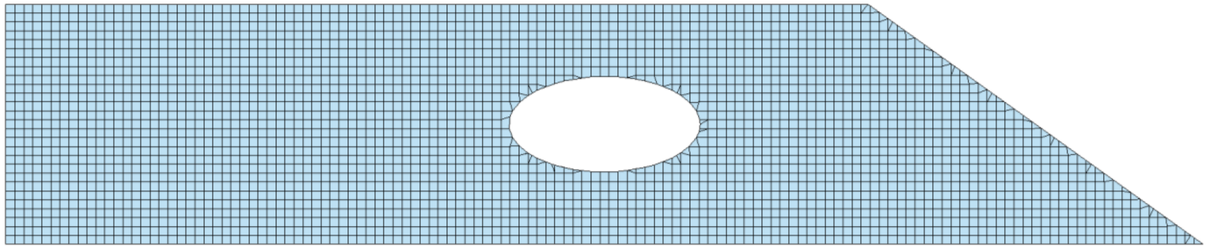
- Equivalent eigenvector interpolation using a Gaussian mixture model-based probability of association obtained as a byproduct of the CPD transformation.

- *Status:* Algorithm tested and Integrated to LS-OPT. *Release: **2023R1**. Working on performance improvement.*

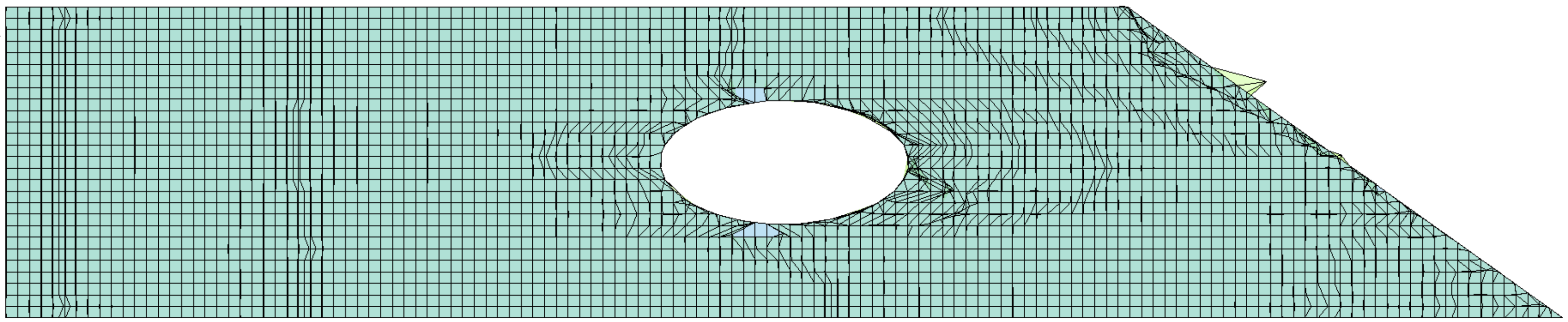
Shape Mapping Using Non-Rigid Coherent Point Drift



Reference Design



New Design

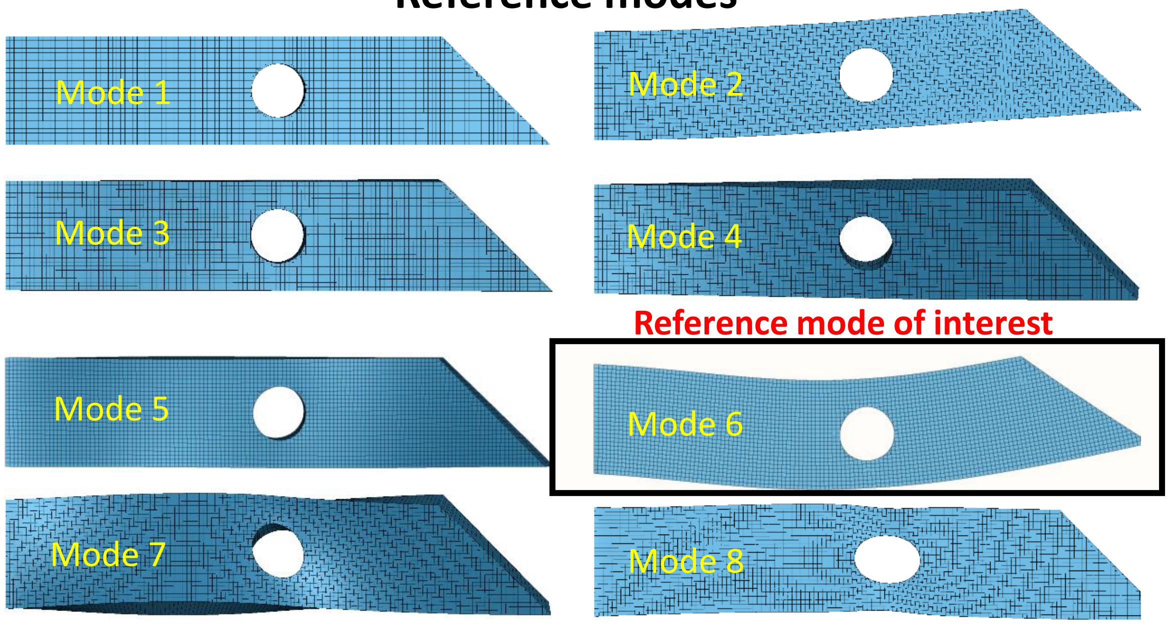


Reference Mesh Transformed to New Mesh

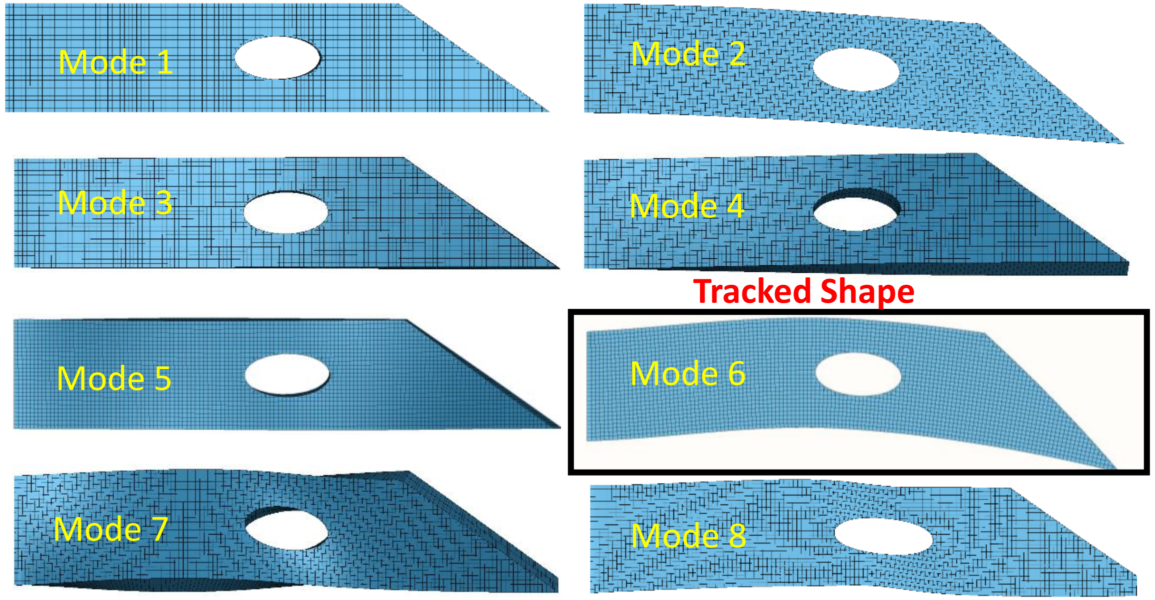
- Some mesh distortion is acceptable as transformed mesh is only used for interpolation, not for FE analysis
- Interpolation depends on pairwise probability of association, not on one to one map of the transformed location

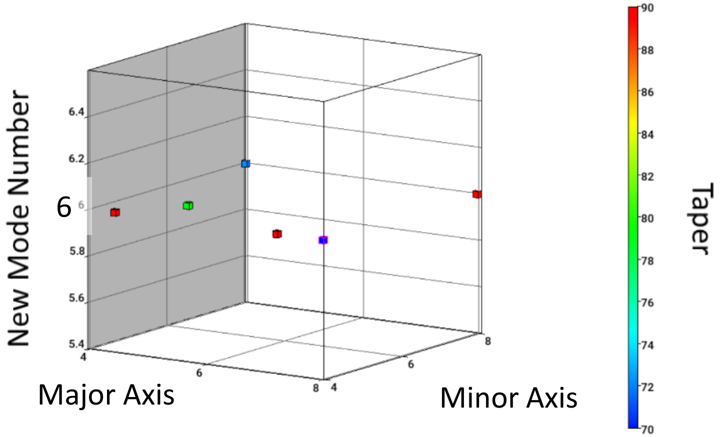
Mode Tracking Example with Varying Geometry

Reference modes



New design modes





- No mode switching in this particular example, but it is possible in general
- Correct mode number (6) corresponding to 6^{th} reference mode tracked after mapping the two shapes and subsequent interpolation

Ansys

Ansys