Release 2022 R1 Highlights

Ansys LS-DYNA

Agenda:

- **Ansys Mechanical Integration:**
  - **Imported Displacement**
  - **Localization (multiple language support)**
  - **Cyclic Symmetry Support**
  - **Fluent to LS-DYNA Thermal Transfer(1 way)**
  - **Restart Improvements** Displacements and Remote Displacement can now be modified in a small restart calculation. In addition, initial velocities can be modified for parts in small restarts and full restarts.
  - **Foam Material Support** New Material Models have been added in support of foam applications
  - **Support for Additional Contact Scoping Options**
  - **Additional Properties for Interference Contact**
  - **New Solver Version**
- **Meshing for Explicit**

- **LS-DYNA Solver R13:**
  - Iso-geometric Analysis (IGA)
  - Materials and Element Enhancements
  - Element Free Galerkin Enhancements
  - Smoothed Particle Galerkin (SPG) Features
  - Smooth Particle Hydrodynamics Enhancements
  - XFEM, Peridynamics
  - Multi-scale Modeling
  - Contact
  - Stamping
  - Acoustics
  - Fatigue
  - Thermal
  - ALE
  - Electro-magnetics (EM)
  - Incompressible Computational Fluid Dynamics (ICFD)
  - Miscellaneous enhancements

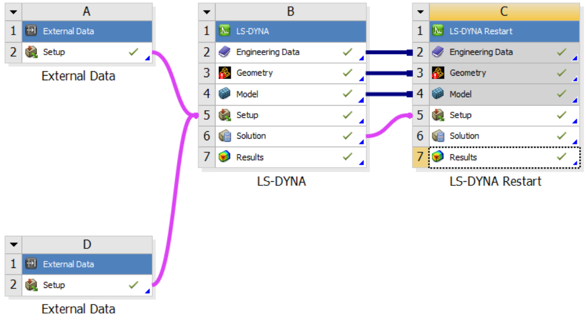
LS-DYNA Ansys Mechanical Integration

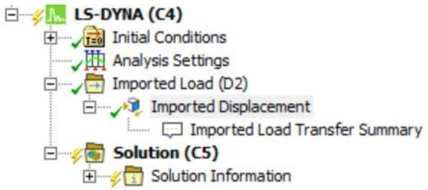
Ansys

Imported Displacement

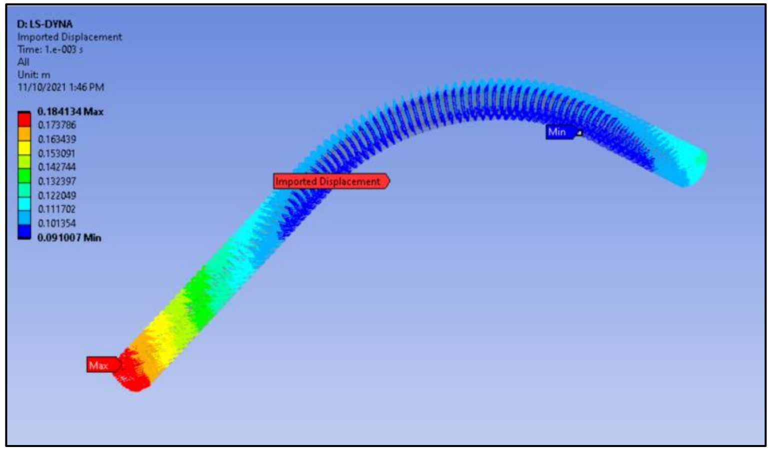
- LS-DYNA now supports Imported Displacement, allowing you to setup the model in non-trivial ways
- The mesh can either be initialized towards a final mesh, allowing the solver to prestress the model, or it can be used to specify the reference geometry in foams applications, if the user has the deformed mesh instead
- The solver will automatically calculate the stresses at the beginning of the calculation
- Create an external data system and import a file containing displacements defined by coordinates or mesh reference
- Link to an LS-DYNA system in the project schematic
- The Imported Load folder is automatically added and Imported Displacement is available for selection in the context menu along with the previously supported Imported Pressure
- Includes the standard options for mapping and display

	A	B	C	D	E
1	Column	Data Type	Data Unit	Data Identifier	Combined Identifier
2	A	X Coordinate	m		File1
3	B	Y Coordinate	m		File1
4	C	Z Coordinate	m		File1
5	D	Displacement	mm	Displacement1	File1:Displacement1



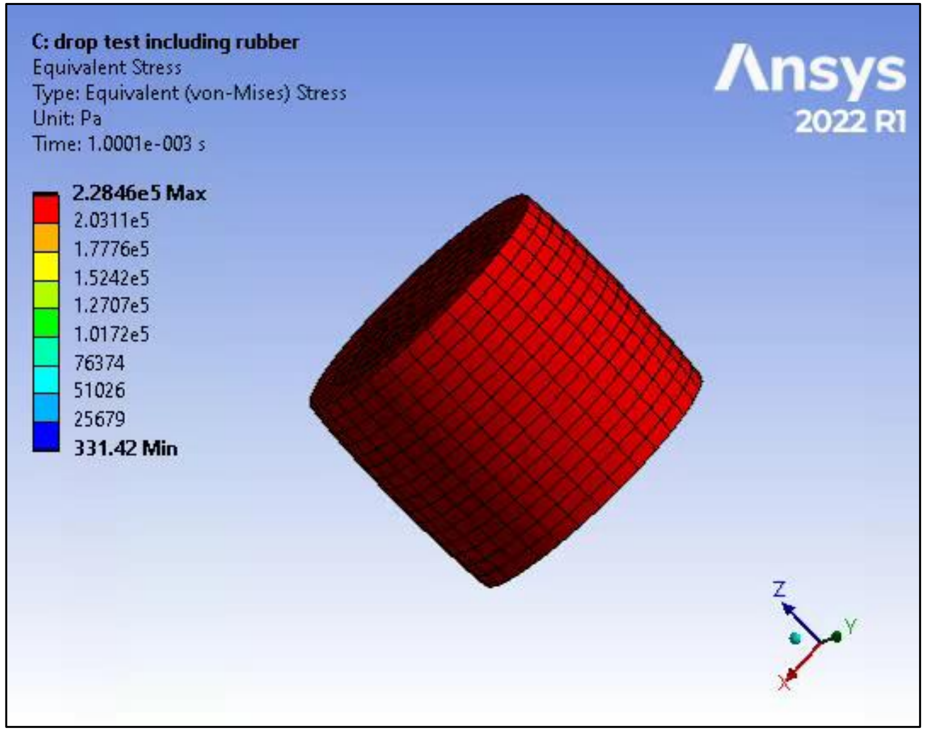


Graphics Controls	
By	Active Row
Active Row	1
Component	All
Display Source Points	Off
Display Source Point Ids	Off
Settings	
Mapping Control	Manual
Mapping	Profile Preserving
Weighting	Direct Assignment
Legend Controls	
Legend Range	Program Controlled
Named Selection Creation	
Unmapped Nodes	Off
Mapped Nodes	Off



Imported Displacement

- Two options are available:
  - **Boundary Prescribed Final Geometry** allows to move the scoped nodes from their initial positions to the final geometry along a straight-line trajectory. The geometry can be prestressed in a nontrivial way.
  - Loading can be applied in a series of Stepped or Ramped sections
  - A single displacement file is allowed for each component in the load table
  - **Initial Foam Reference Geometry**, which is only valid when used with foams materials. The mesh in Mechanical is the deformed geometry. The external file contains the coordinates of the reference (undeformed) geometry. The stresses are initialized by the solver
  - A single row is allowed in the load table
  - Tabular Loading is not applicable in this case. The solver determines the rate at which the displacement is applied
  - Reference field in material is automatically selected when the Imported Displacement is scoped to the relevant body. When using a material defined using a command snippet the reference field is not modified



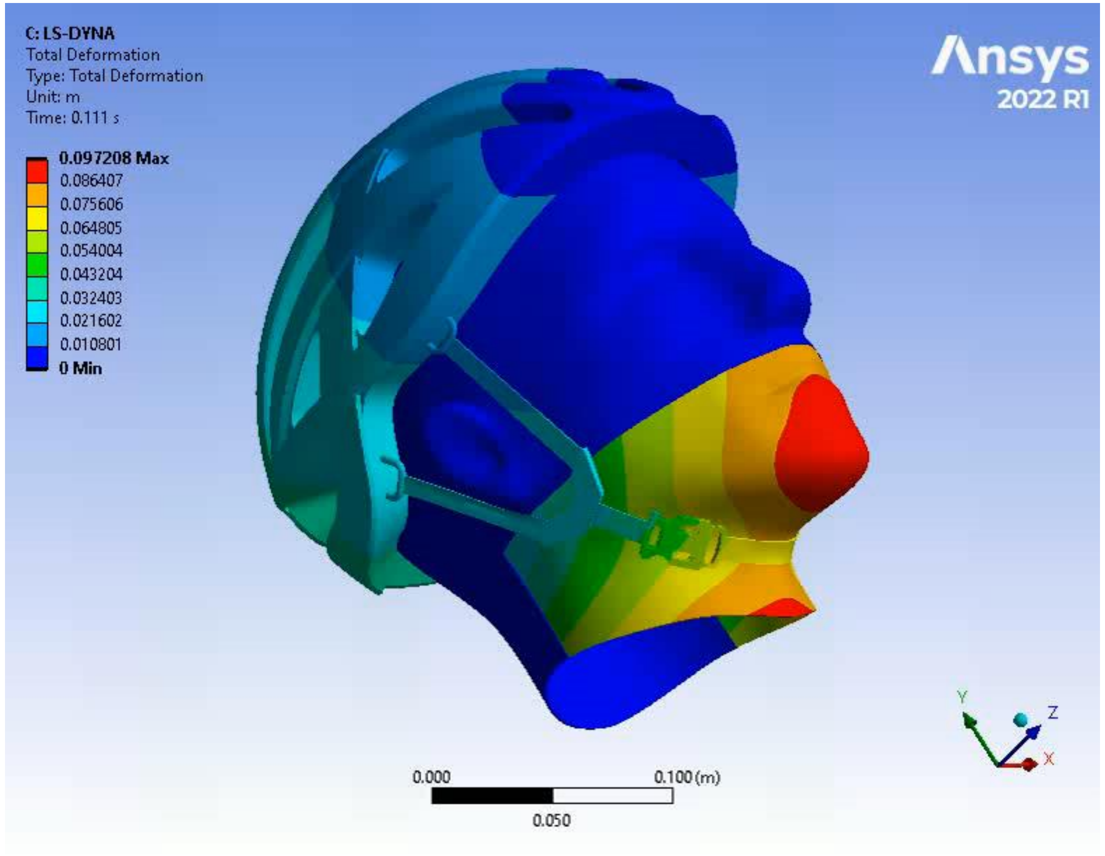
Details of "Imported Displacement"

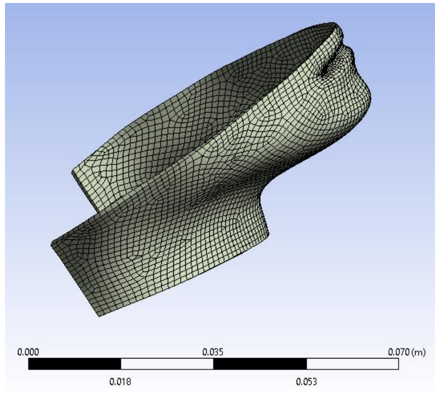
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Type	Imported Displacement
Displacement Type	Boundary Prescribed Final Geometry
Tabular Loading	Ramped
Suppressed	No
Override Constraints	No
Coordinate System	Source Coordinate System

	X Component (m)	Y Component (m)	Z Component (m)	Analysis Time (s)	Scale
1	File1:Displacement1	File1:Displacement2	File1:Displacement3	0.005	0.25
2	File1:Displacement1	File1:Displacement2	File1:Displacement3	0.001	1
*					

Imported Displacement – Boundary Prescribed Final Geometry

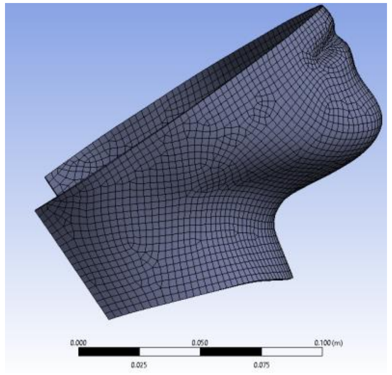
- The displacements between a scaled version of the chin and the full size geometry are calculated and imported via an External Data system
- An imported displacement load is used to apply the displacement from the initial to the final configuration pulling the helmet into place and pre-stressing the chin straps



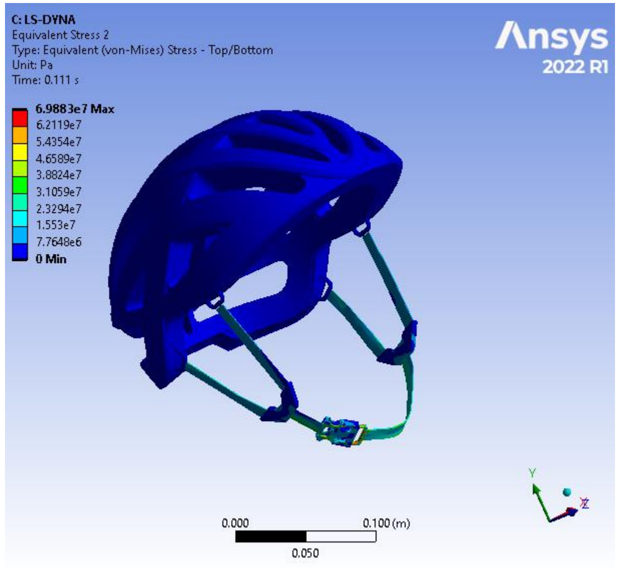


Initial Configuration

Apply Displacements

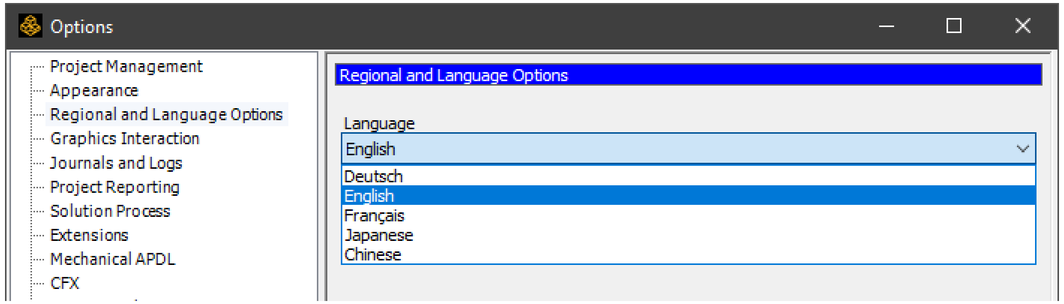


Final Configuration

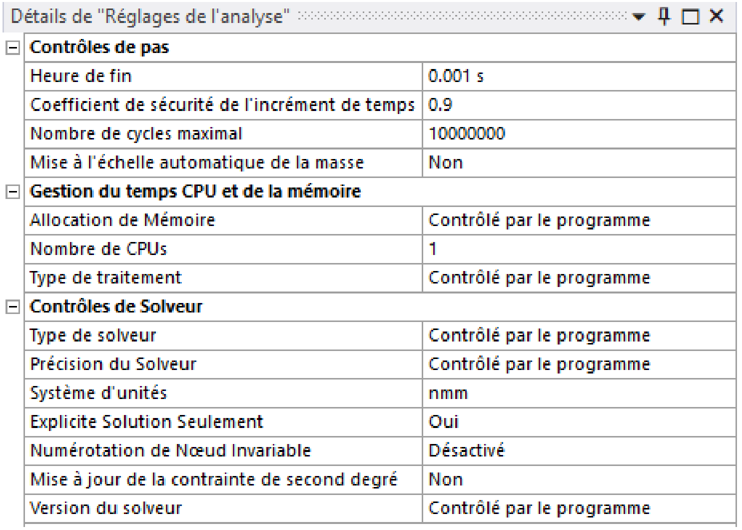


Localization

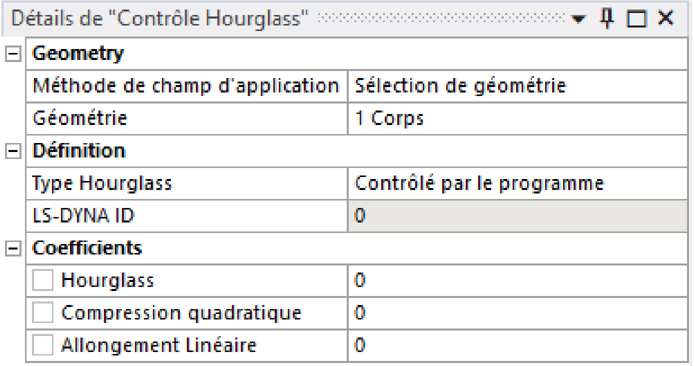
- The LS-DYNA Worbench system now supports additional languages
- Enhancement covers items specific to the LS-DYNA analysis system; Analysis Settings, Load Objects, Toolbars



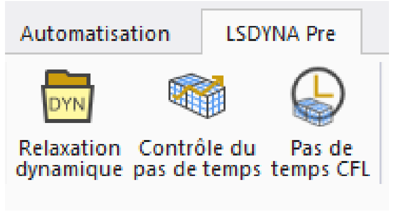
Language Selection



Analysis Settings



Analysis Specific Loads



Analysis Specific Toolbars

Localization

Includes all 4 additional languages that currently available in Workbench, French, German, Chinese, and Japanese

Détails de "Réglages de l'analyse"

Contrôles de pas	
Heure de fin	0.001 s
Coefficient de sécurité de l'incrément de temps	0.9
Nombre de cycles maximal	10000000
Mise à l'échelle automatique de la masse	Non
Gestion du temps CPU et de la mémoire	
Allocation de Mémoire	Contrôlé par le programme
Nombre de CPUs	1
Type de traitement	Contrôlé par le programme
Contrôles de Solveur	
Type de solveur	Contrôlé par le programme
Précision du Solveur	Contrôlé par le programme
Système d'unités	nmm
Explicite Solution Seulement	Oui
Numérotation de Nœud Invariable	Désactivé
Mise à jour de la contrainte de second degré	Non
Version du solveur	Contrôlé par le programme
Vitesses Initiales	
Contrôles de l'amortissement	
Amortissement Global	Non
Contrôles Hourglass	
Type Hourglass	Contrôlé par le programme
LS-DYNA ID	0
Coefficient de Hourglass défaut	0.1
Contrôles ALE	
Contrôles de Liaison	
Formulation	Contrôlé par le programme
Contrôles des composites	
Contrôles de sortie	
Format de sortie	Contrôlé par le programme
Facteur d'échelle de taille du fichier binaire	70
Contraintes	Oui
Déformations	Non
Déformation plastique	Oui
Historique des Variables	Non
Calculer les résultats à	Contrôlé par le programme
Fichier de contrainte pour pièces flexibles	Non
Contrôles de Sortie de l'Historique des Temps	
Calculer les résultats à	Non
Gestion de données d'analyse	

French

Details von "Analyseeinstellungen"

Schrittsteuerungen	
Endzeit	0.001 s
Zeitschritt-Sicherheitsfaktor	0.9
Maximale Zyklenanzahl	10000000
Automatische Massenskalierung	Nein
CPU und Speicherverwaltung	
Speicherallokierung	Programmgesteuert
Anzahl CPUs	1
Verarbeitungstyp	Programmgesteuert
Solver-Steuerungen	
Solvertyp	Programmgesteuert
Solver-Genauigkeit	Programmgesteuert
Einheitensystem	nmm
Nur explizite Lösung	Ja
Invariante Knotennummerierung	Aus
Spannungsupdate zweiter Ordnung	Nein
Solver Version	Programmgesteuert
Anfangsgeschwindigkeiten	
Dämpfungssteuerungen	
Globale Dämpfung	Nein
Hourglass-Steuerungen	
Hourglass-Typ	Programmgesteuert
LS-DYNA-ID	0
Standard-Hourglass-Koeffizient	0.1
ALE Steuerung	
Gelenkkontrollen	
Formulierung	Programmgesteuert
Composite-Steuerungen	
Ausgabesteuerungen	
Ausgabeformat	Programmgesteuert
Skalierungsfaktor für Binärdateigröße	70
Spannung	Ja
Dehnung	Nein
Plastische Dehnung	Ja
Zeitverlauf-Variablen	Nein
Ergebnisse berechnen für	Programmgesteuert
Spannungsdatei für flexible Teile	Nein
Ausgabesteuerungen für den Zeitlichen Verlauf	
Ergebnisse berechnen für	Nein
Analysedatenverwaltung	

German

"分析设置"的详细信息

步骤控制	
结束时间	0.001 s
时步安全系数	0.9
最大周期数量	10000000
自动质量缩放	没有
CPU和内存管理	
内存分配	程序控制的
CPU数	1
处理类型	程序控制的
求解器控制	
求解器类型	程序控制的
求解器精度	程序控制的
单位系统	nmm
仅显式解	是
不变节点编号	关闭
二阶应力更新	没有
求解器版本	程序控制的
初始速度	
阻尼控制	
全局阻尼	没有
沙漏控制	
沙漏类型	程序控制的
LS-DYNA ID	0
默认沙漏系数	0.1
ALE控制	
连结控制	
公式	程序控制的
复合控制	
输出控制	
输出格式	程序控制的
二进制文件大小比例因子	70
应力	是
应变	没有
塑性应变	是
历史变量	没有
计算结果	程序控制的
柔性部件的应力文件	没有
时间历史输出控制	
计算结果	没有
分析数据管理	

Chinese

"解析設定" の詳細

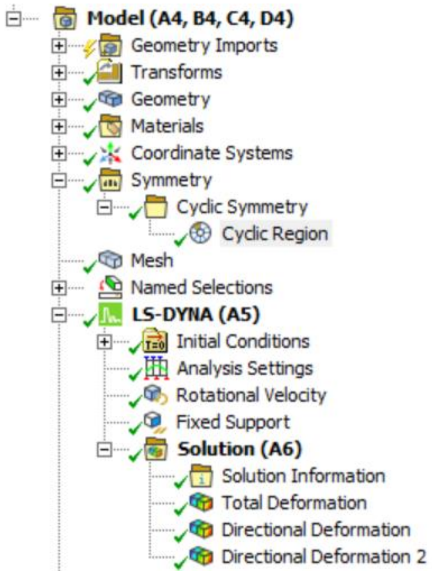
ステップコントロール	
終了時間	0.001 s
時間ステップ安全率	0.9
最大サイクル数	10000000
自動質量スケール	No
CPU およびメモリー管理	
メモリー割り当て	プログラムによるコントロール
CPU 数	1
処理タイプ	プログラムによるコントロール
ソルバーコントロール	
ソルバータイプ	プログラムによるコントロール
ソルバー精度	プログラムによるコントロール
単位系	nmm
陽解法解析のみ	Yes
不変節点番号	オフ
2 次応力の更新	No
ソルバーバージョン	プログラムによるコントロール
初期速度	
減衰コントロール	
グローバル減衰	No
アワーグラス制御	
アワーグラスタイプ	プログラムによるコントロール
LS-DYNA ID	0
デフォルトのアワーグラス係数	0.1
ALE コントロール	
ジョイントコントロール	
定式化	プログラムによるコントロール
複合材コントロール	
出力コントロール	
出力形式	プログラムによるコントロール
バイナリファイルサイズのスケールファクター	70
応力	Yes
ひずみ	No
塑性ひずみ	Yes
履歴変数	No
結果の計算点	プログラムによるコントロール
弾性体の応力ファイル	No
時刻歴出力コントロール	
結果の計算点	No
解析データ管理	

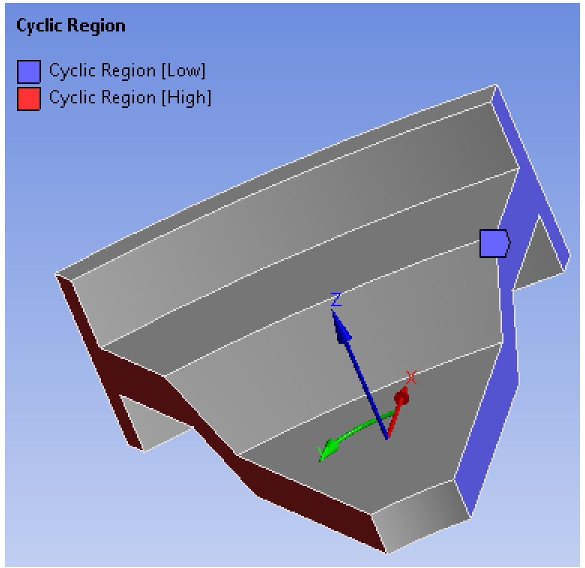
Japanese

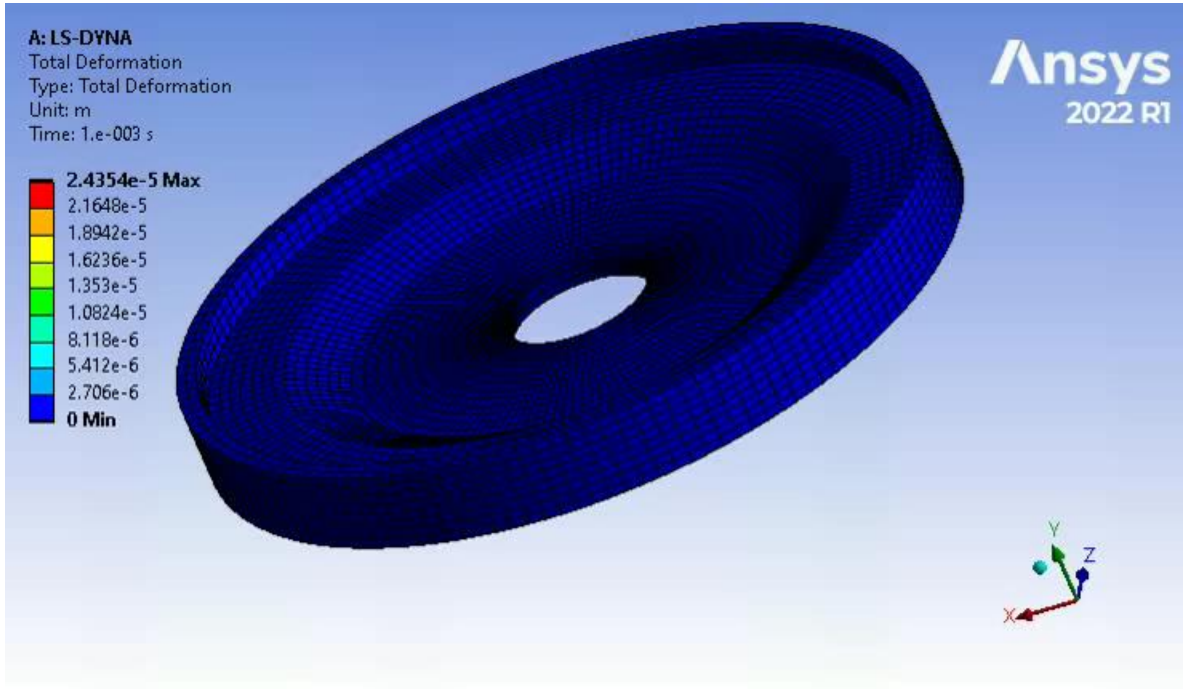
Ansys

Cyclic Symmetry

- Cyclic Symmetry is now supported in the LS-DYNA Workbench system and can be used in turbomachinery applications for faster running times
- Option to display the mesh and results with full symmetry (Beta)

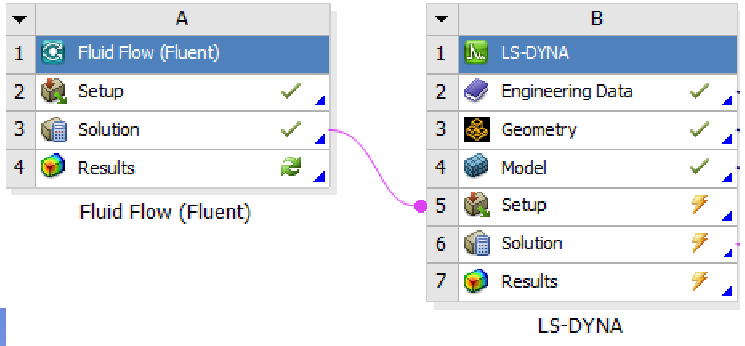


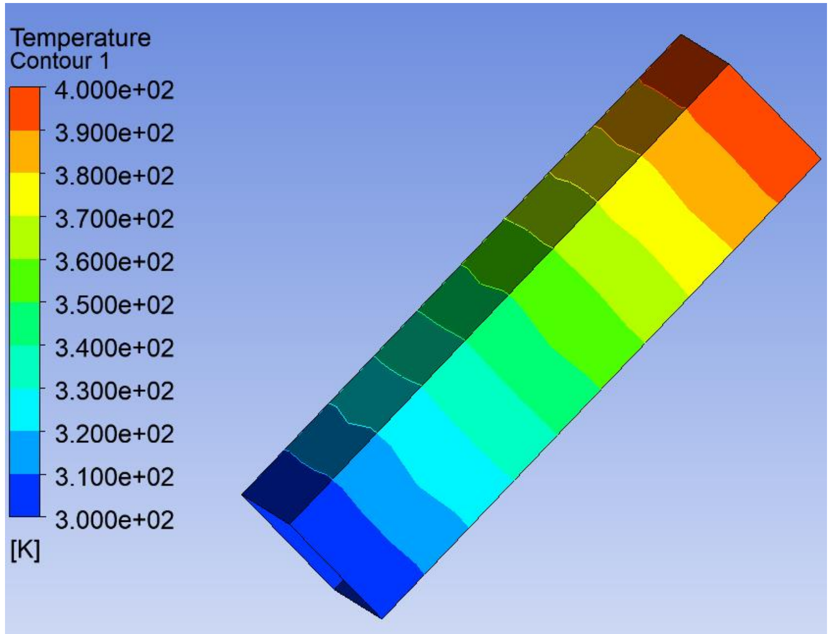




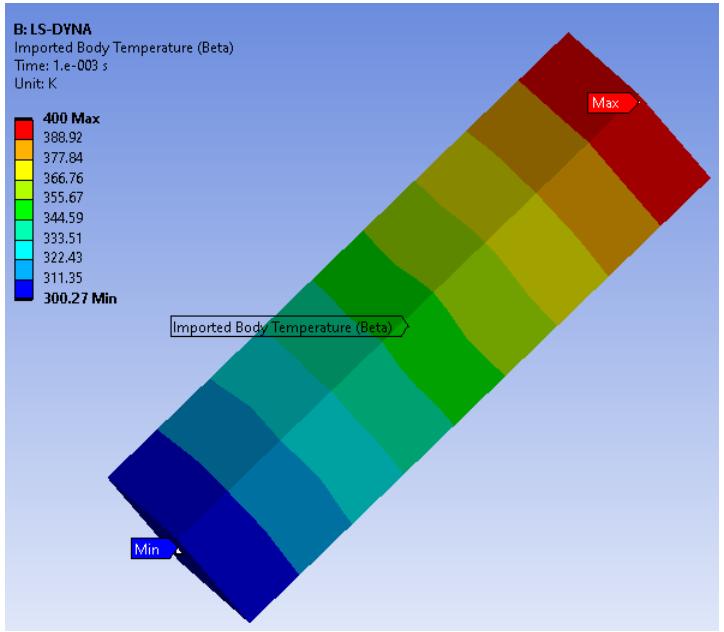
Fluent to LS-DYNA 1 Way Thermal Transfer

- Body Temperatures can be now imported from CFD calculations allowing to take in accurately temperatures effects in LS-DYNA simulations
- Link systems in the project schematic, imported load folder is added automatically, body temperature load can be inserted from the context menu
- Standard imported load features are available; Stepped\Ramped loading with the option to apply scale factor and offset

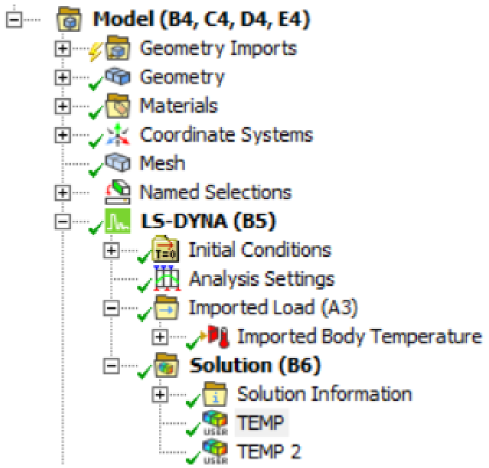




Fluent

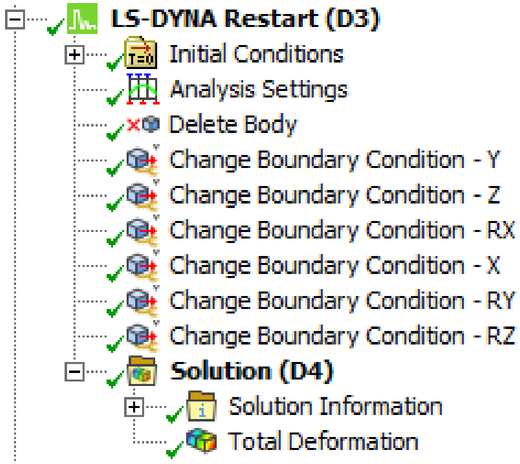


LS-Dyna



Restarts Improvements

- Displacements and Remote Displacements can be modified in restarts allowing to simulate complex movements
- Location method allows the selection of a boundary condition or a curve (added using the keyword manager)
- Each component of a boundary condition can be independently redefined by a curve

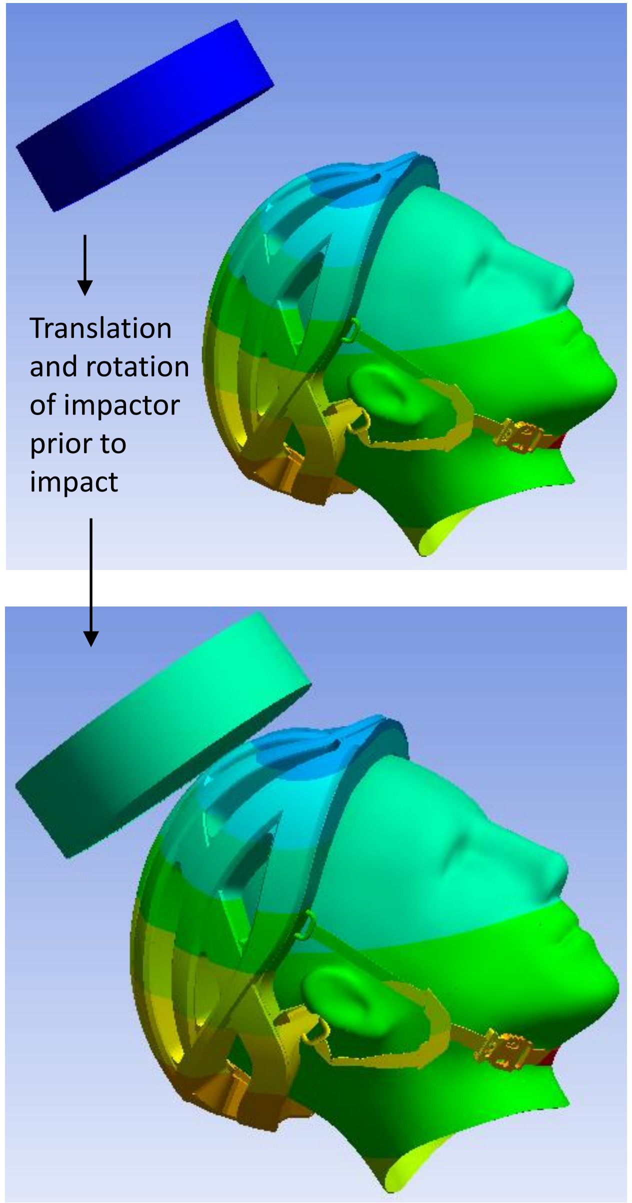


Details of "Change Boundary Condition 2"

Definition	
Location Method	Boundary Condition
Boundary Condition	Remote Displacement_Round_Impactor
Component	Y Component

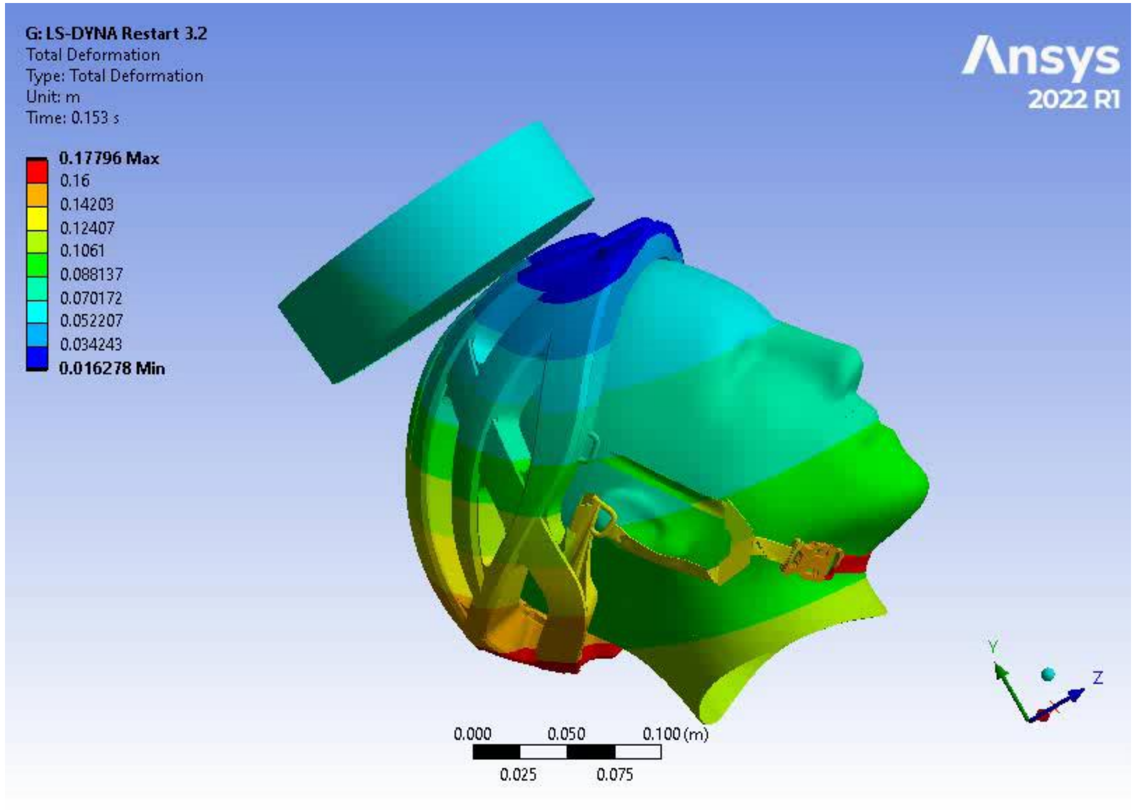
Tabular Data

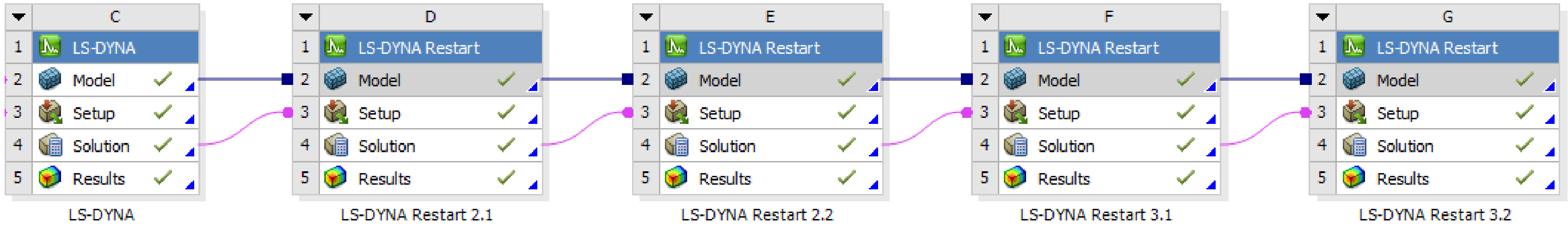
		Time [s]	Displacement [m]
1	1	0.	= 0.
2	1	0.132	0.
3	1	0.133	3.5e-002
4	N/A	3.	3.5e-002
5	N/A	4.	3.5e-002
*			



Restarts Improvements

- Multi-step impact analysis using a series of small restarts
  - Pre-Stress
  - Impactor 1 Positioning
  - Impact 1
  - Impactor 2 Positioning
  - Impact 2
- Uses the new Change Boundary Condition object along with the Delete Body and Change Velocity objects





Enhanced Material Support

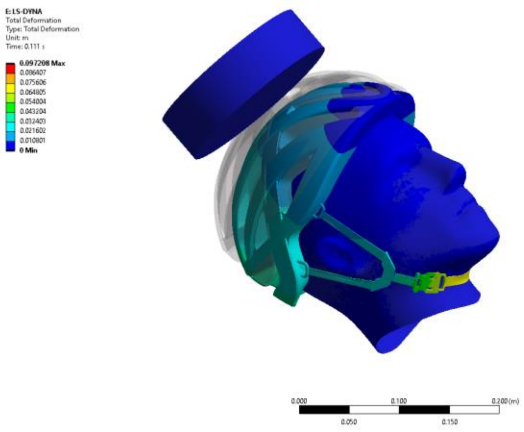
- 4 additional material models have been introduced for applications using Fabric\Foam
- Can be imported to Engineering Data from .k file using External Model
- These definitions generally follow the input card with the variable names added as a suffix

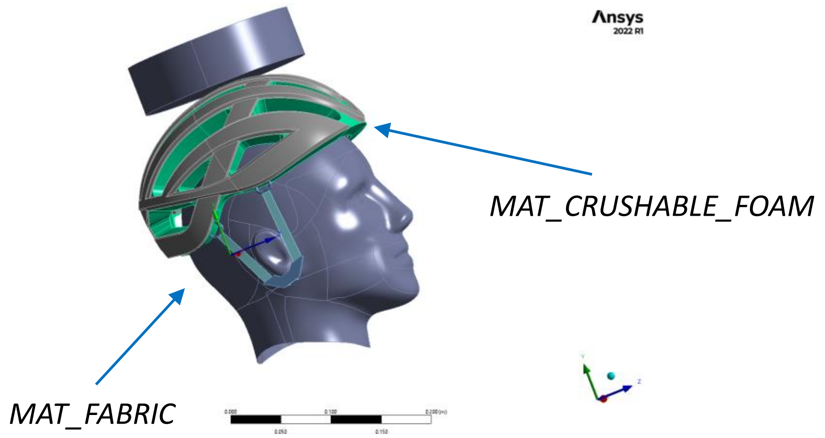
Properties of Outline Row 8: MAT_CRUSHABLE_FOAM

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	70	kg m^-3		
4	Isotropic Elasticity				
5	Derive from	Young's Mod...			
6	Young's Modulus	2.5E+07	Pa		
7	Poisson's Ratio	0.2			
8	Bulk Modulus	1.3889E+07	Pa		
9	Shear Modulus	1.0417E+07	Pa		
10	*MAT_CRUSHABLE_FOAM				
11	Definition				
12	Tensile Stress cutoff, tsc	3.9E+05	Pa		
13	Rate Sensitivity via damping coefficient, damp	0			
14	Yield Stress versus Volumetric Strain, lcid	Tabular			
15	Scale	1			
16	Offset	0	Pa		

LSDYNA External Model - MAT

- *MAT_ELASTIC
- *MAT_ORTHOTROPIC_ELASTIC
- *MAT_ANISOTROPIC_ELASTIC
- *MAT_PLASTIC_KINEMATIC
- *MAT_BLATZ-KO_RUBBER
- *MAT_HIGH_EXPLOSIVE_BURN
- *MAT_NULL
- *MAT_JOHNSON_COOK
- *MAT_POWER_LAW_PLASTICITY
- *MAT_PIECEWISE_LINEAR_PLASTICITY
- *MAT_CRUSHABLE_FOAM
- *MAT_SIMPLIFIED_JOHNSON_COOK
- *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY
- *MAT_SIMPLIFIED_RUBBER/FOAM
- *MAT_BILKHU/DUBOIS_FOAM
- *MAT_FABRIC



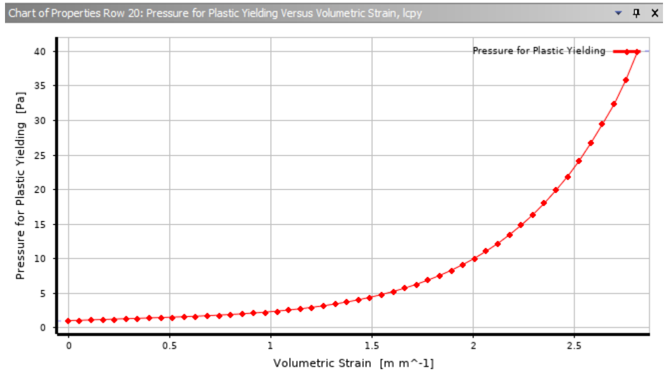


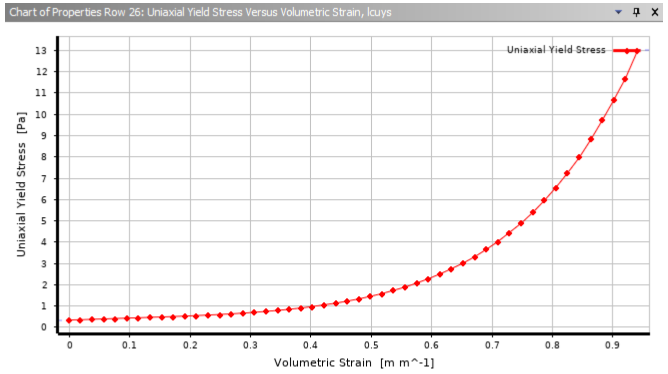
Enhanced Material Support

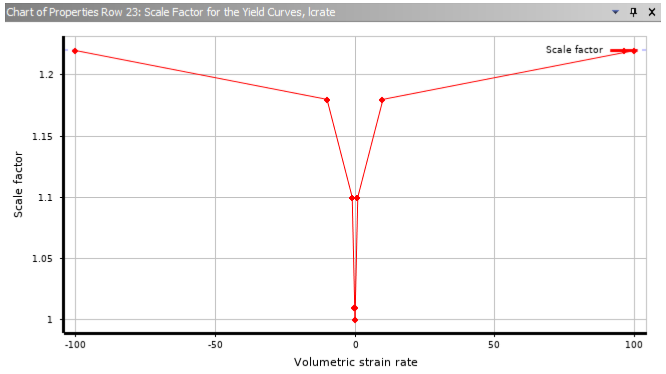
- Associated load curves are available
- Added by default or available under the additional tabular data when selecting the material
- Not all options are supported when there are optional cards

*MAT_BILKHU/DUBOIS_FOAM		
Definition		
Tensile Response	Volumetric Strain	
Viscous Damping Coefficient, vc	0	
Pressure Cutoff, pc	0	Pa
Variable Pressure Cutoff as a Fraction of Pressure Yield Value, vpc	0	Pa
Tension Cutoff for Uniaxial Tensile Stress, tsc	0	Pa
Variable Tension Cutoff as a Fraction of the Uniaxial Compressive Yield Strength, vtsc	0	Pa
Stiffness Coefficient for Contact Interface Stiffness, kcon	0	Pa
Number of Cycles to Determine the Average Volumetric Strain Rate, ncycle	0	
Pressure for Plastic Yielding Versus Volumetric Strain, lcpy	Tabular	
Uniaxial Yield Stress Versus Volumetric Strain, lcuys	Tabular	

Additional Tabular Data
- Scale Factor for the Yield Curves







Enhanced Material Support: Prony Series

- Prony Series (Shear Relaxation) are supported for the following material properties, allowing to simulate time-dependent stress relaxation effects:
  - Mooney-Rivlin
  - Ogden
  - Yeoh
  - Polynomial
  - Arruda-Boyce
  - MAT_SIMPLIFIED_RUBBER/FOAM (new addition at 2022 R1)

Viscoelastic
Prony Shear Relaxation

→

Properties of Outline Row 3: Admat_OD1

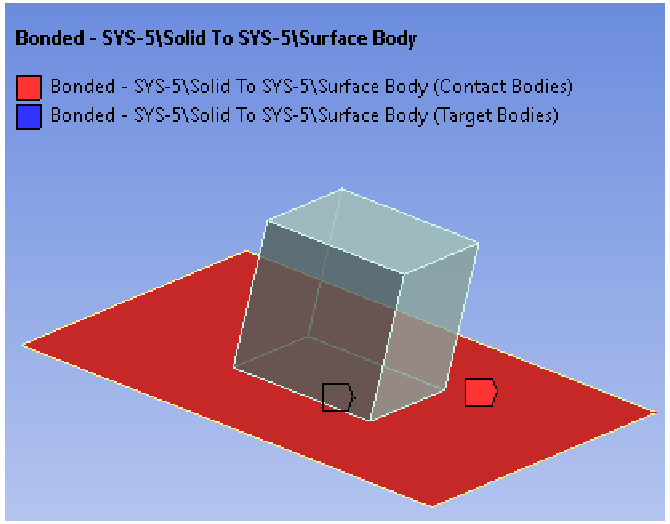
	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	1000	kg m^-3		
4	Ogden 1st Order				
5	Material Constant MU1	23845	Pa		
6	Material Constant A1	2.1151			
7	Incompressibility Parameter D1	1.4429E-07	Pa^-1		
8	Prony Shear Relaxation	Tabular			
9	Number of Terms	12			
10	Relative Moduli(i): Scale	1			
11	Relative Moduli(i): Offset	0			
12	Relaxation Time(i): Scale	1			
13	Relaxation Time(i): Offset	0	s		

→

	B	C	D
1	Index i	Relative Moduli(i)	Relaxation Time(i) (s)
2	1	1000	0.0001
3	2	10000	0.00011
4	3	9000	0.002
5	4	8000	0.003
6	5	7000	0.004
7	6	6000	0.005
8	7	5000	0.006
9	8	4000	0.007
10	9	3000	0.008
11	10	2000	0.009
12	11	1000	0.00111
13	12	2050	0.0123

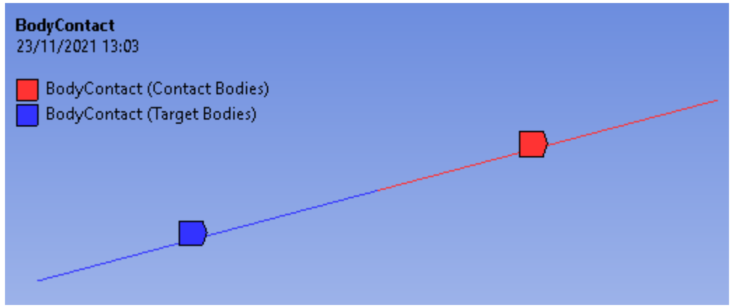
Contact Scoping

- Additional contact scoping options are available for LS-DYNA
- Contacts can now be scoped to Shell and Beam bodies in a 3D Analysis



Details of "BodyContact"

Scope	
Scoping Method	Geometry Selection
Contact	1 Edge
Target	1 Body
Contact Bodies	Beam2
Target Bodies	Beam1
Protected	No
Definition	
Type	Bonded
Scope Mode	Manual
Trim Contact	Program Controlled
Maximum Offset	1.e-002 m
Breakable	No
Suppressed	No
Advanced	
Formulation	Program Controlled



Details of "Bonded - SYS-5\Solid To SYS-5\Surface Body"

Scope	
Scoping Method	Geometry Selection
Contact	1 Body
Target	1 Face
Contact Bodies	SYS-5\Surface Body
Target Bodies	SYS-5\Solid
Contact Shell Face	Program Controlled
Protected	No
Definition	
Type	Bonded
Scope Mode	Manual
Behavior	Program Controlled
Trim Contact	Program Controlled
Maximum Offset	1.e-007 m
Breakable	No
Suppressed	No
Advanced	
Formulation	Program Controlled

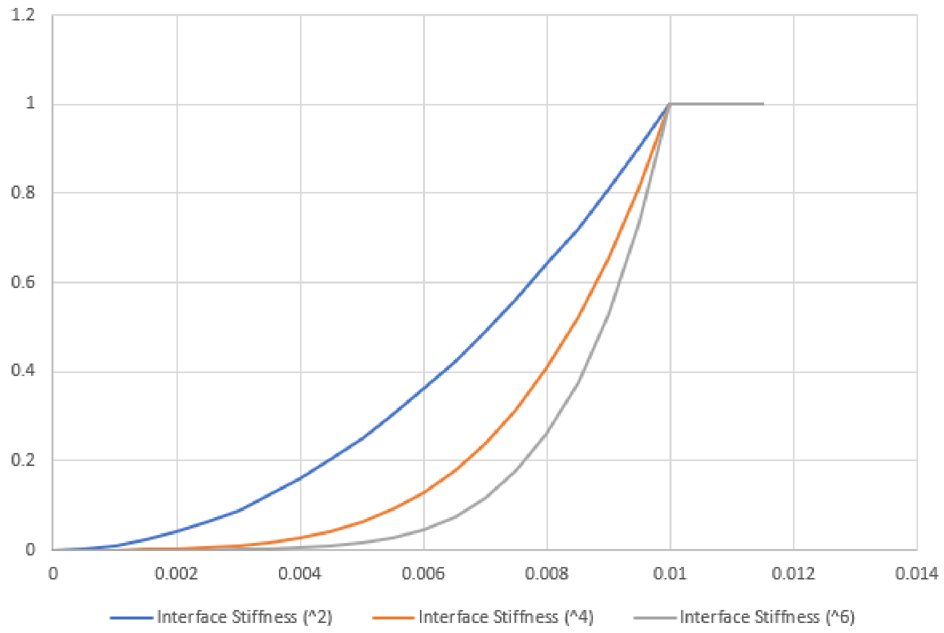
Interference Contact Properties

- New options added to give additional flexibility in contact stiffness curve definition
- End time for contact. Previously this just used analysis time
- Stiffness curve function exponent to vary the transition to peak stiffness

```
*DEFINE_CURVE_FUNCTION
$          ID        sidr
            3           0
min(1, (time / 0.01)**4)
```

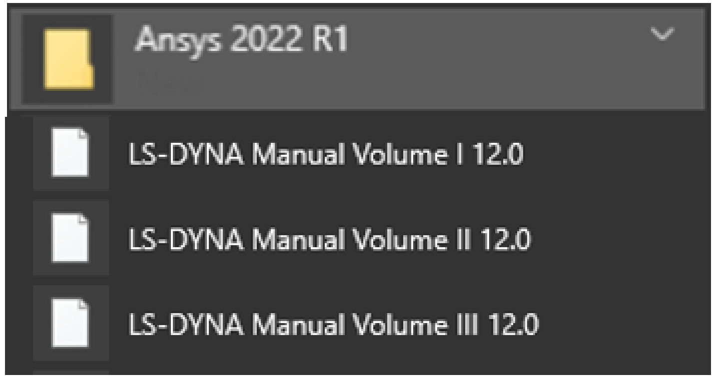
Details of "Contact Properties 2"

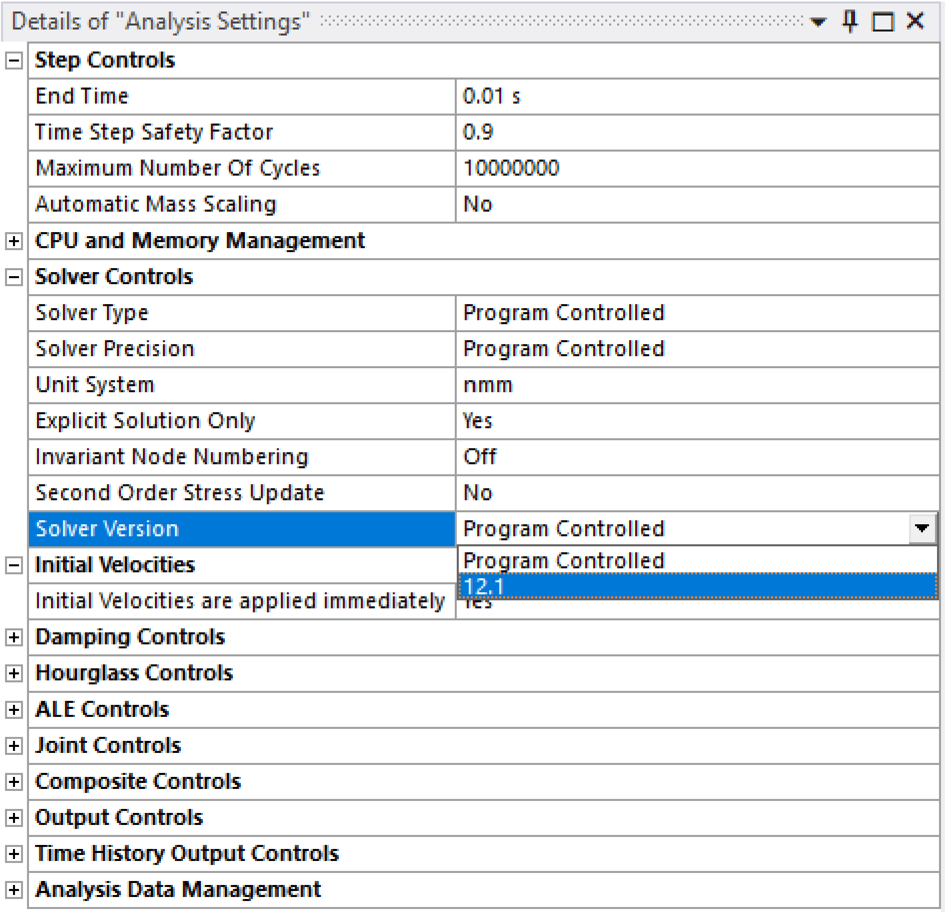
Definition	
Contact	Frictional - Multiple To Multiple (Interference)
Type	Interference
Formulation	SURFACE_TO_SURFACE_INTERFERENCE
Common Controls	
Birth Time	0 s
Death Time	0 s
Viscous Damping Coefficient	10
Contact Penalty Scale Factor	4
Target Penalty Scale Factor	4
Advanced Controls	
Optional Thickness for Contact Surface	0 m
Optional Thickness for Target Surface	0 m
Optional Solid Element Thickness	0 m
Soft Constraint Formulation	Program Controlled
Soft Constraint Scale Factor	0.1
Depth	5
Interference Controls	
Stiffness Scale factor At End of Dynamic Relaxation	1
Interference End Time	0.01 s
Stiffness Curve Exponent	4



New Solver Version

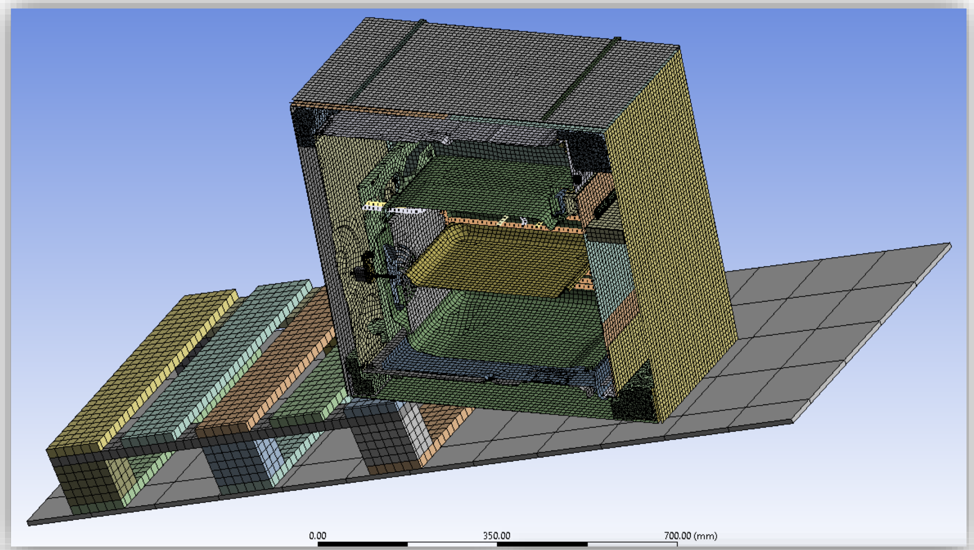
- Default solver version for 2022 R1 is 12.1
- Refer to the documentation for a summary of new developments \ improvements to stability and release notes for further information

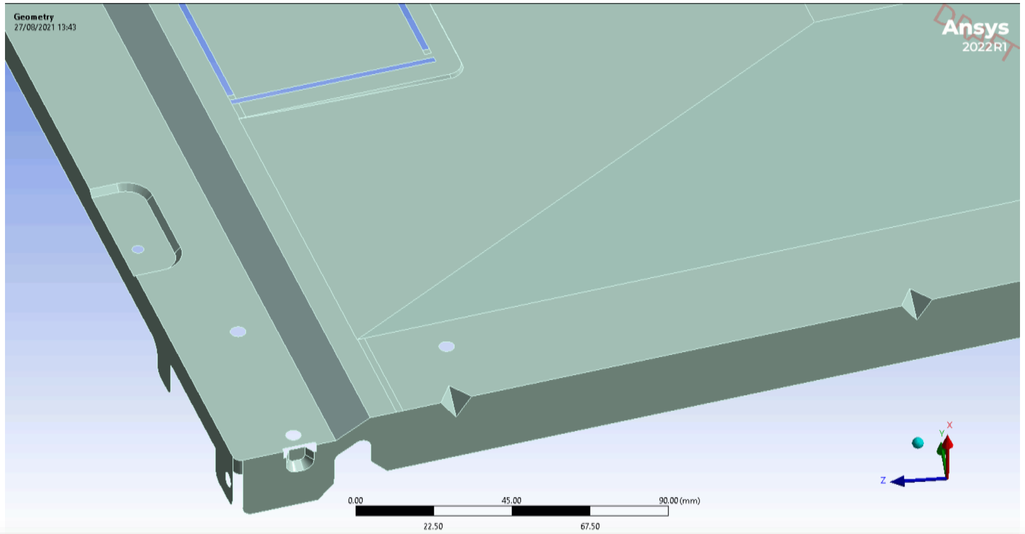


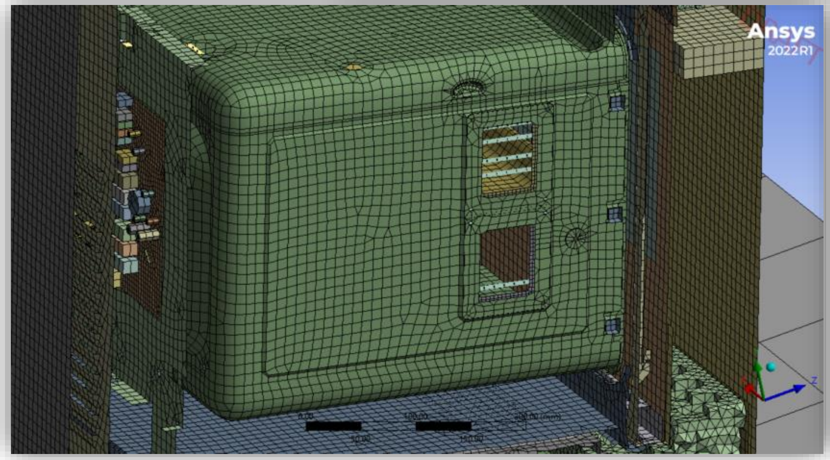


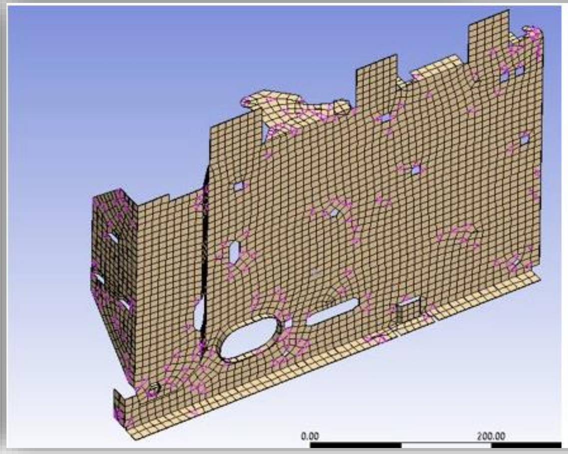
Meshing for Explicit

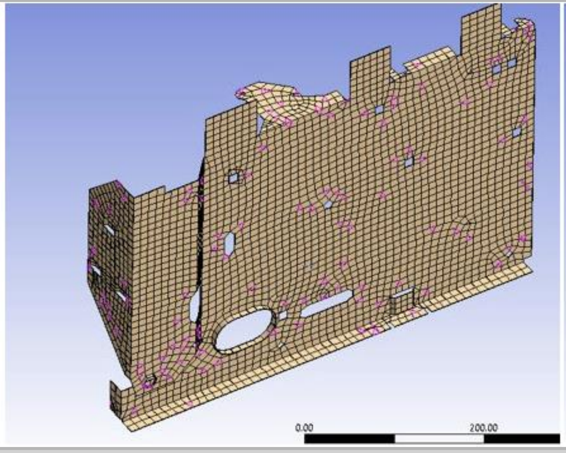
Steamer Shell Mesh – Tri Reduction

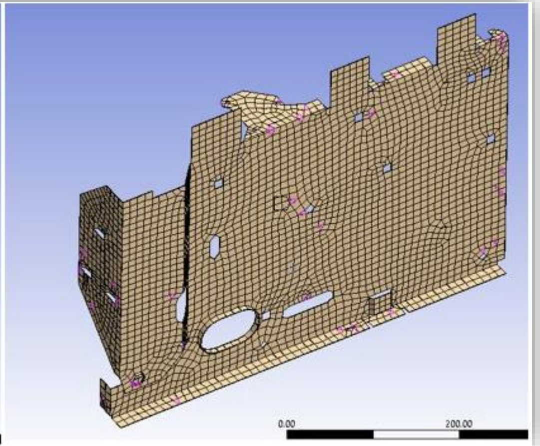












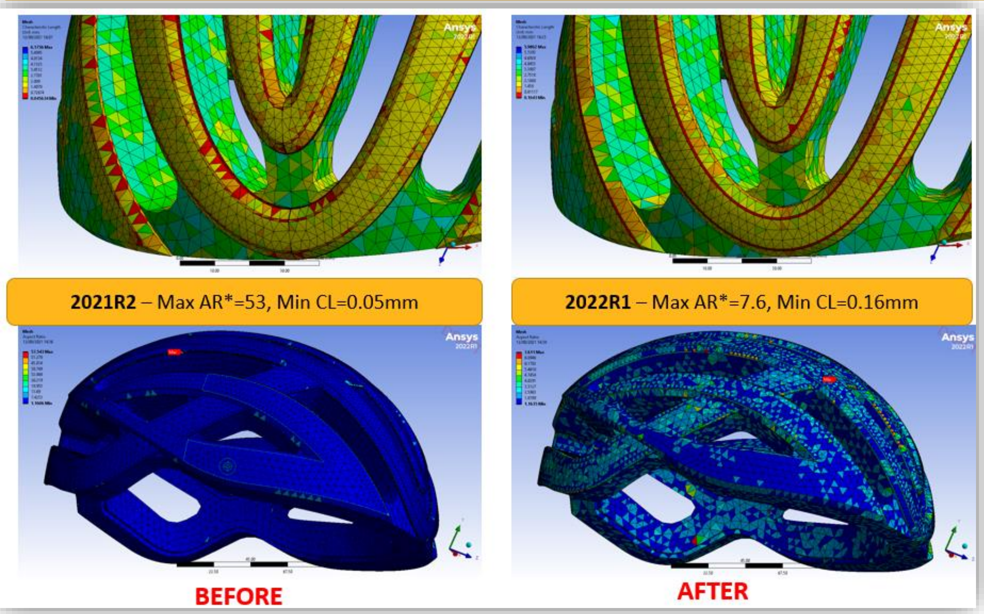
Tri Reduction = None
Number of tri = 184

Tri Reduction = Conservative
Number of tri = 126

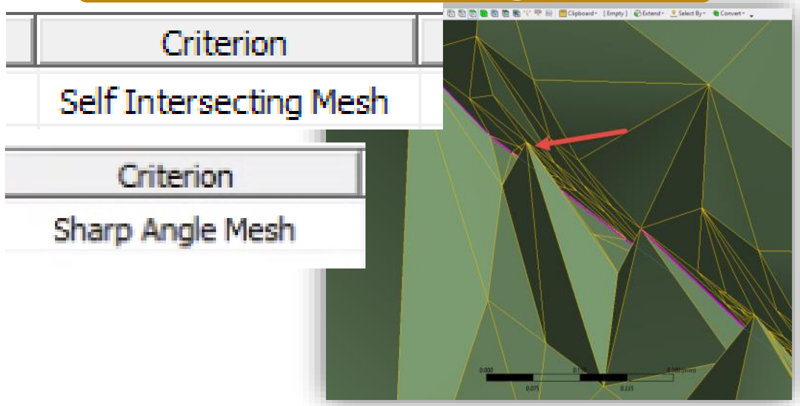
Tri Reduction = Aggressive
Number of tri = 48

Ansys Mechanical Meshing in 2022 R1

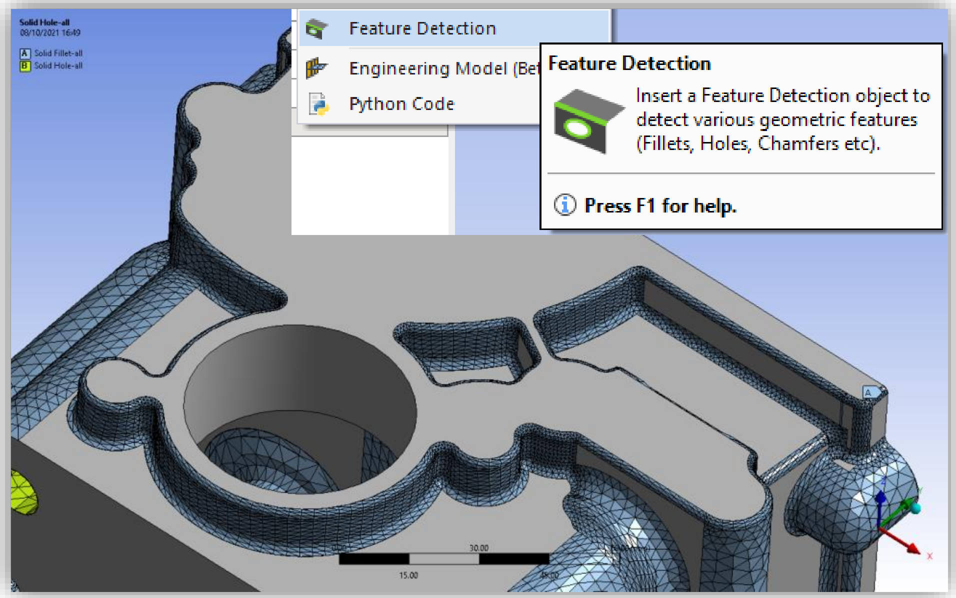
Tet Meshing for Explicit with AR & CL targeting



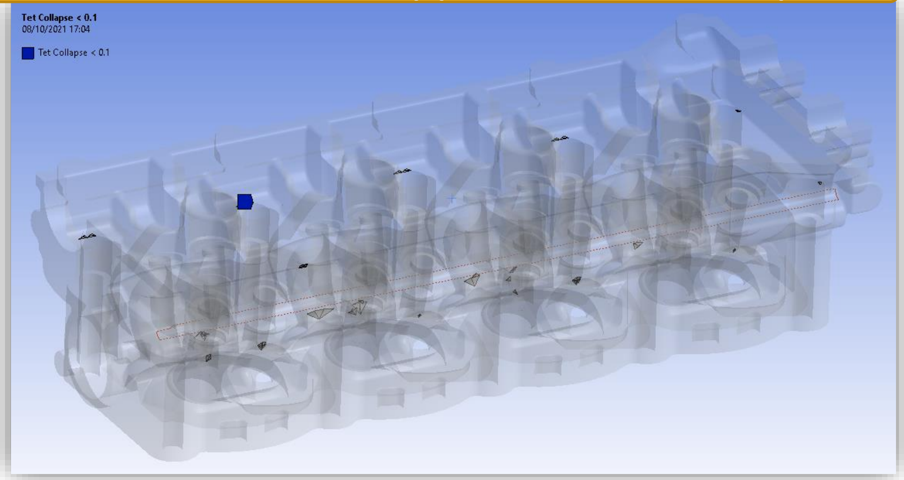
WS NS → Mesh Diagnostics



Feature Detection & Treatment for Solids

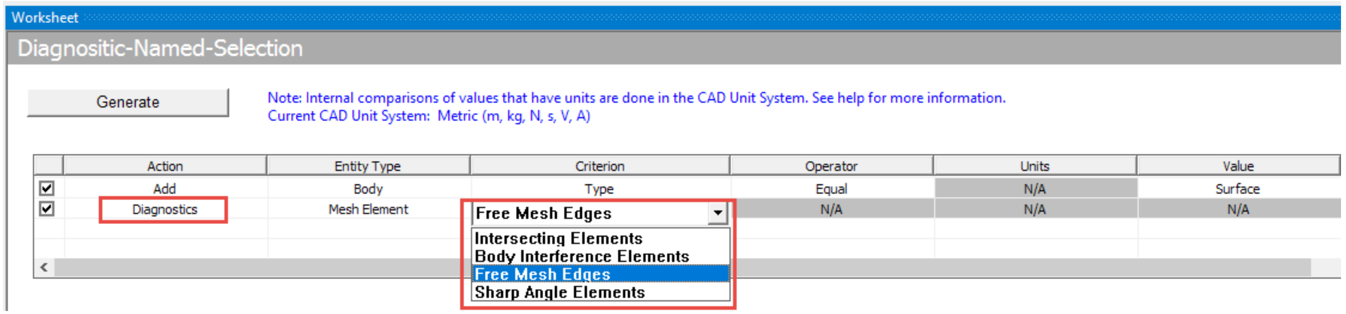


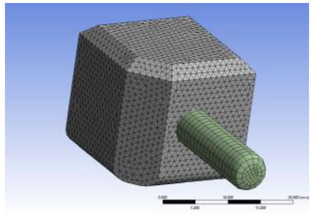
New Mesh Metric Support inc. Tet Collapse



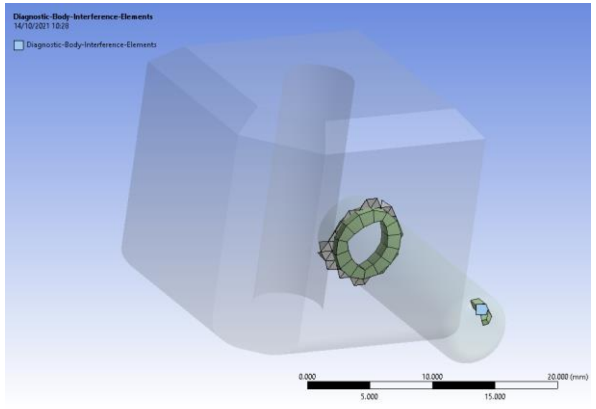
Diagnostics Tools

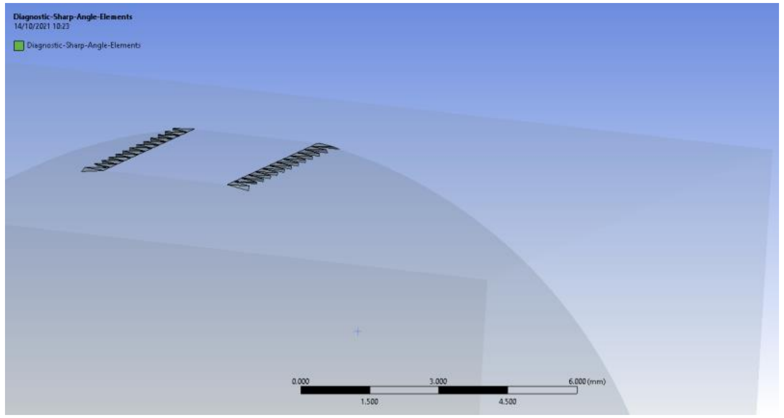
- Use worksheet Named Selection to select bodies and then run Diagnostics for visualisation of the problem
- New tools to find issues and fix via additional settings or return to geometry tool for modifications
- Options available at 2022 R1:
  - Mesh Element:
    - Intersecting surface mesh failures
    - Free edge mesh
    - Sharp angle
    - Body Interference
  - Topology
    - Defeatured Faces





Body Interference





Sharp Angle Elements

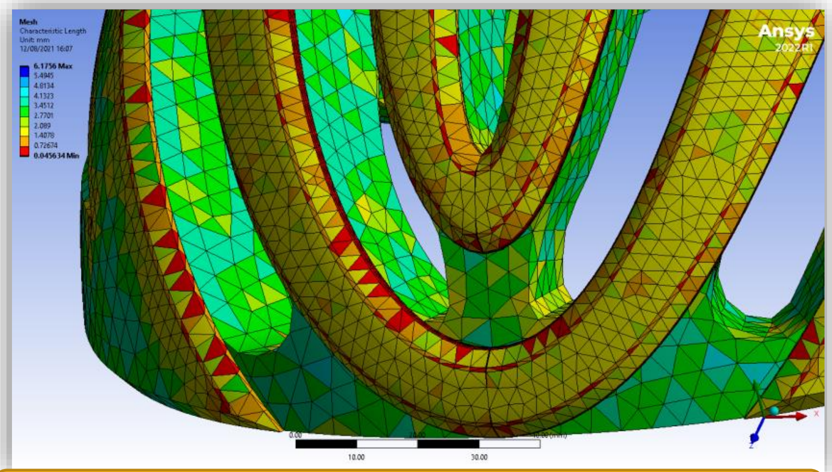
Tet Meshing Aspect Ratio Targeting (Explicit Physics Pref.)

- Aspect Ratio based meshing criteria drastically reduces the max. Aspect Ratio (AR)
- This help improve the Characteristic Length (CL)significantly which has a big impact of the Explicit CFL Time-Step ($\Delta t = \frac{Characteristic\ Length}{Speed\ of\ Sound}$)
- Help run the analysis without much mass-scaling

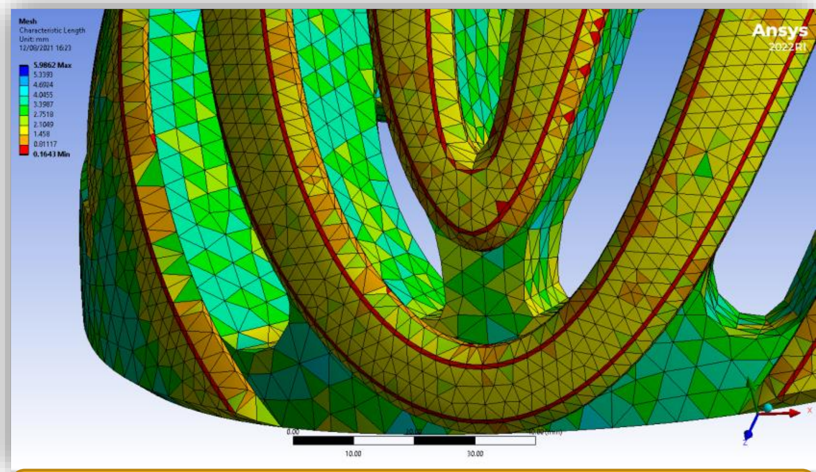
Quality	
Check Mesh Quality	Yes, Errors and Warnings
Target Element Quality	Default (0.200000)
Target Characteristic Length (LSDyna)	Default (0.5 mm)
Target Aspect Ratio (Explicit)	5.

Challenges:

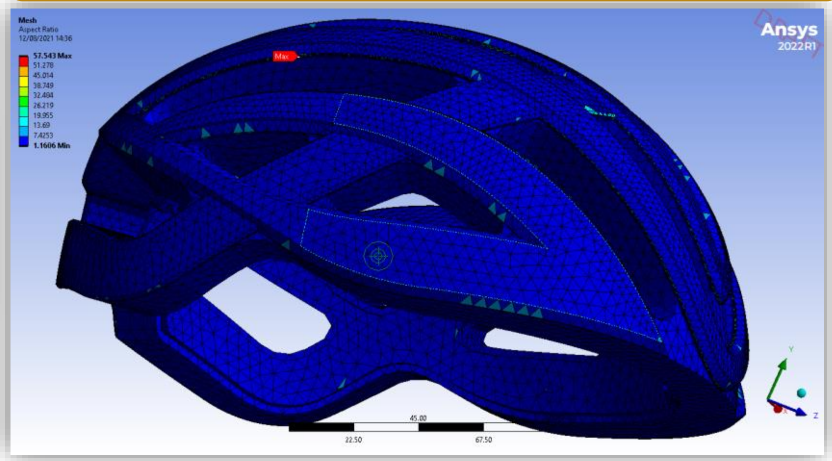
- Bike Helmet Example: A complex geometry with several intricate features



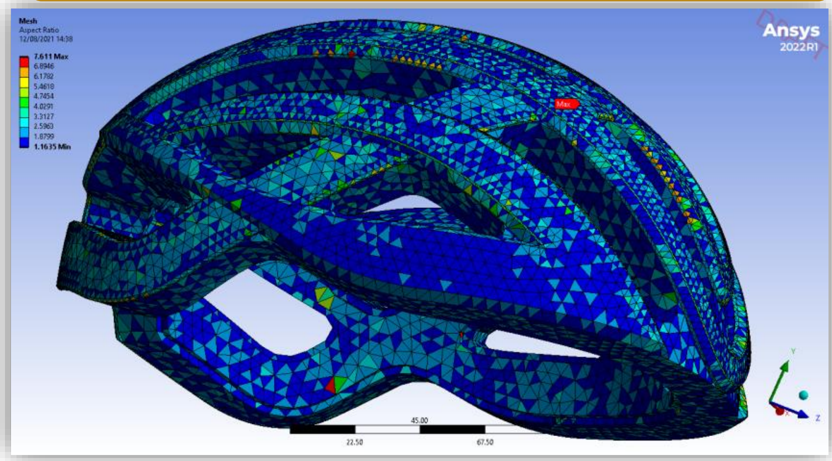
2021R2 – Max AR=53, Min CL=0.05mm



2022R1 – Max AR=7.6, Min CL=0.16mm



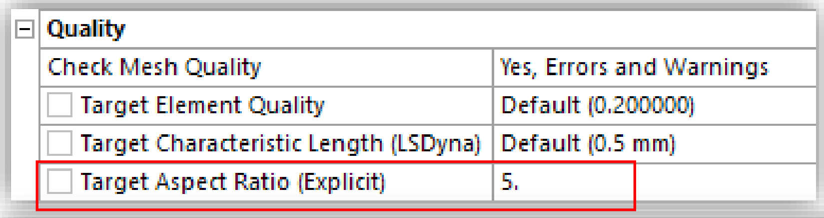
BEFORE

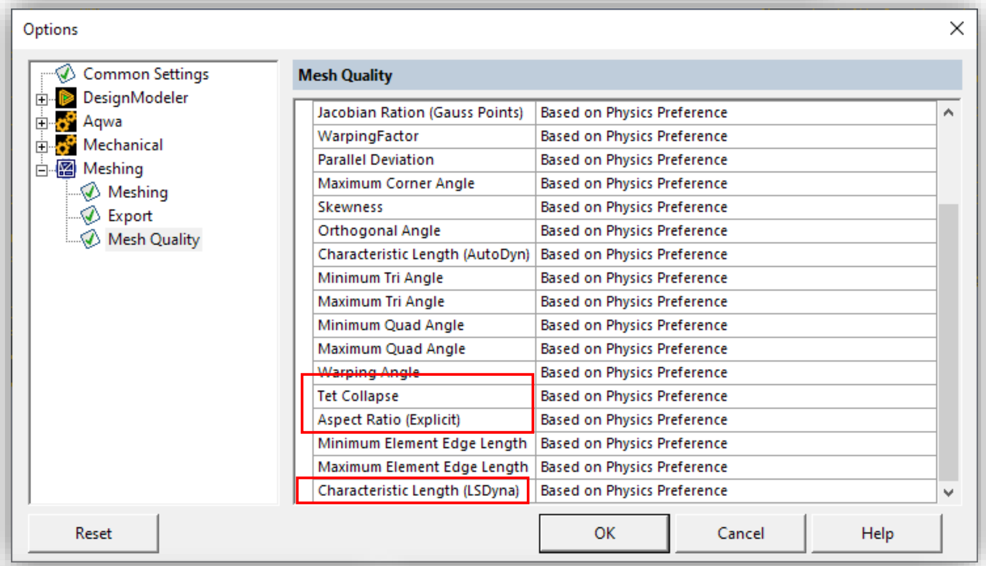


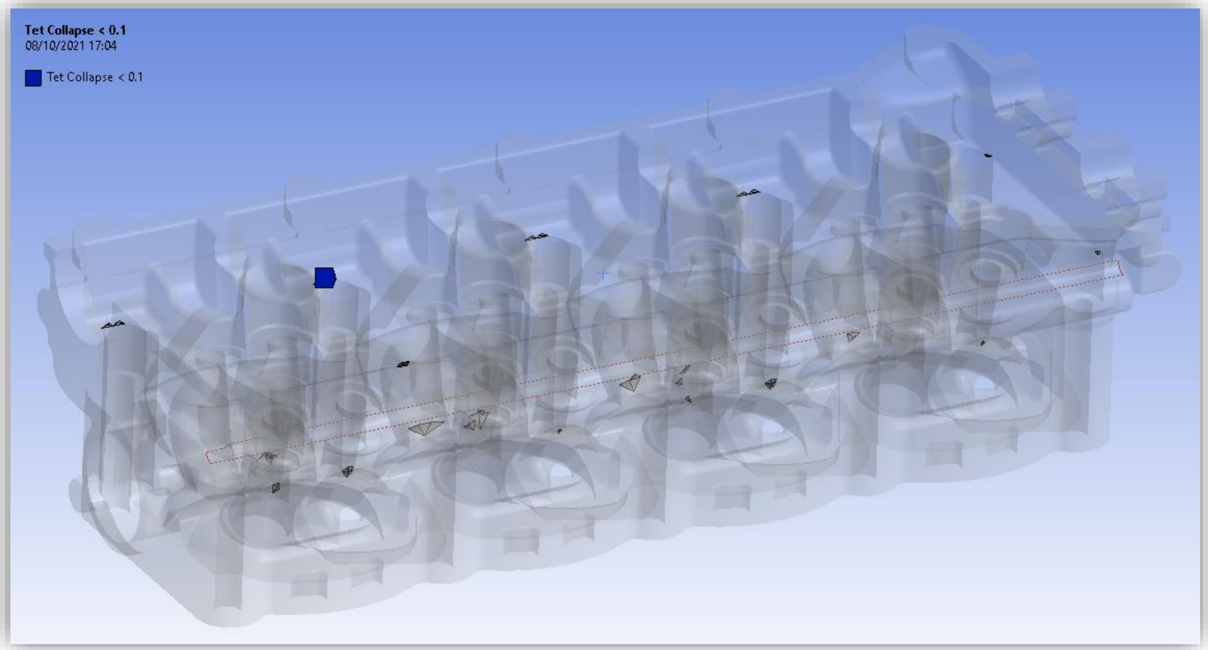
AFTER

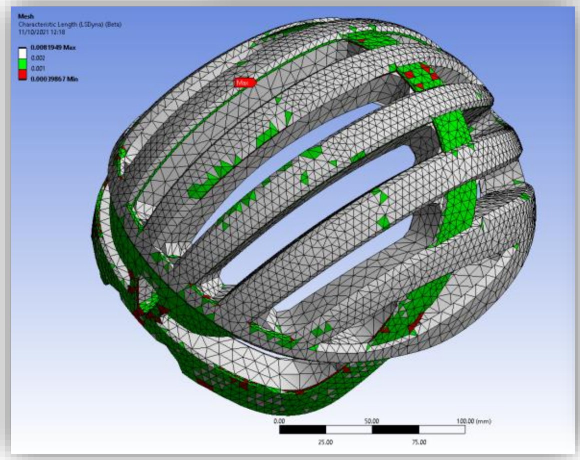
Quality

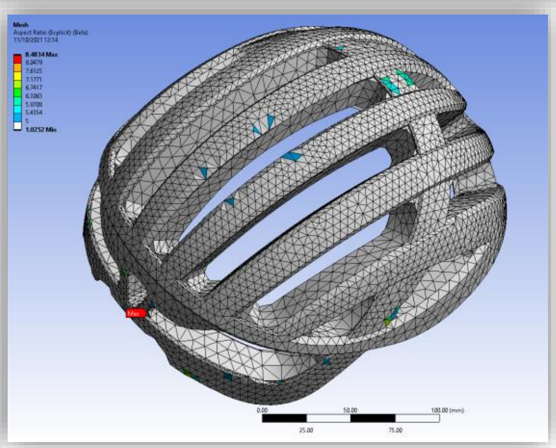
- Exposure of many more metrics
  - Visibility of metrics is based on physics preference
  - New include LS Dyna Characteristic Length and Explicit Aspect Ratio



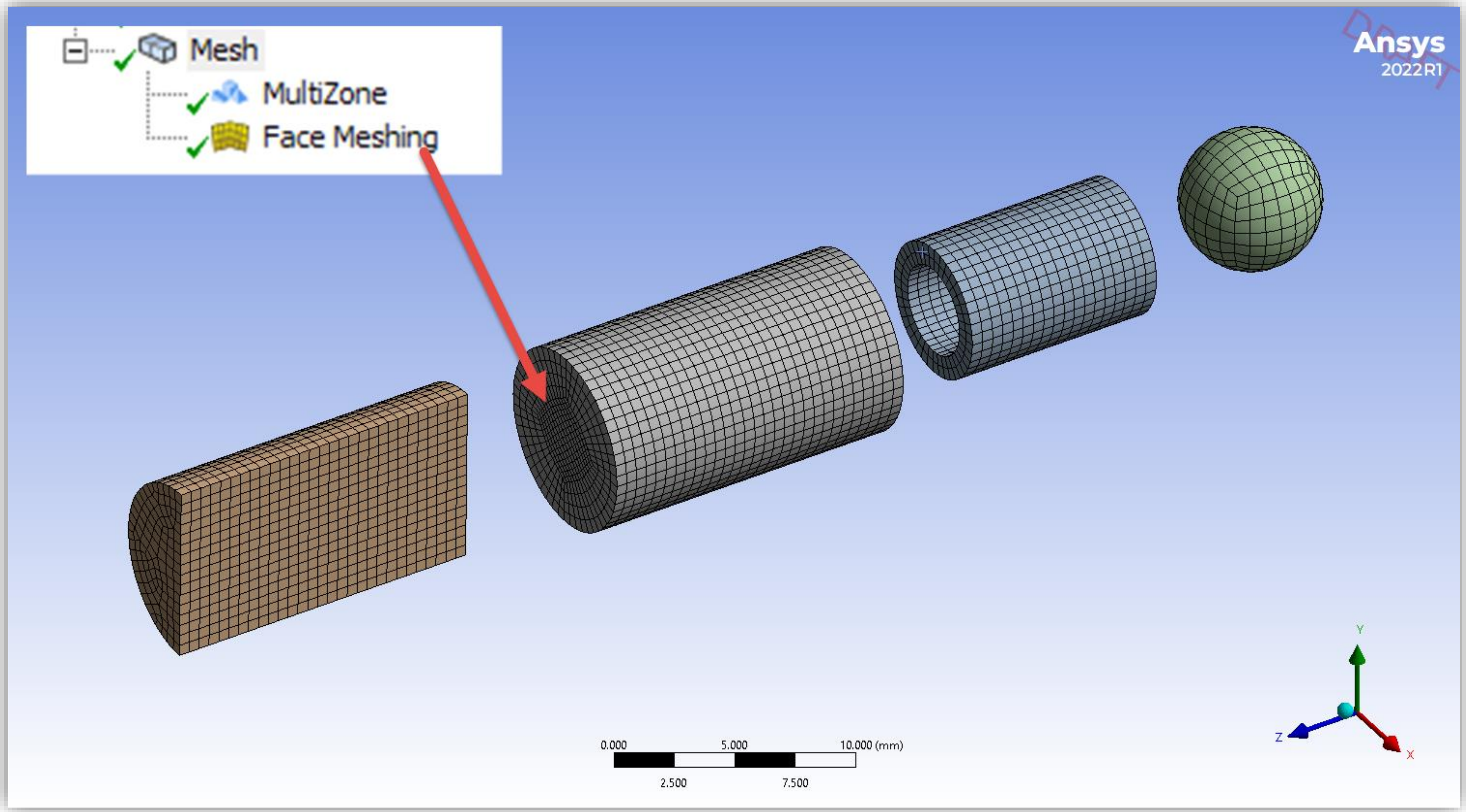




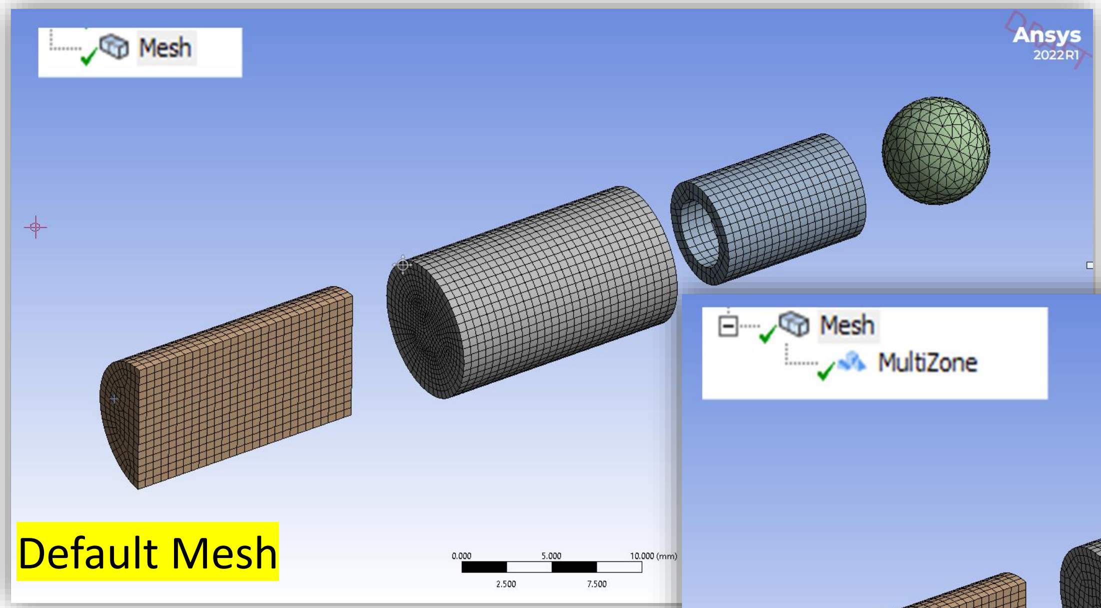


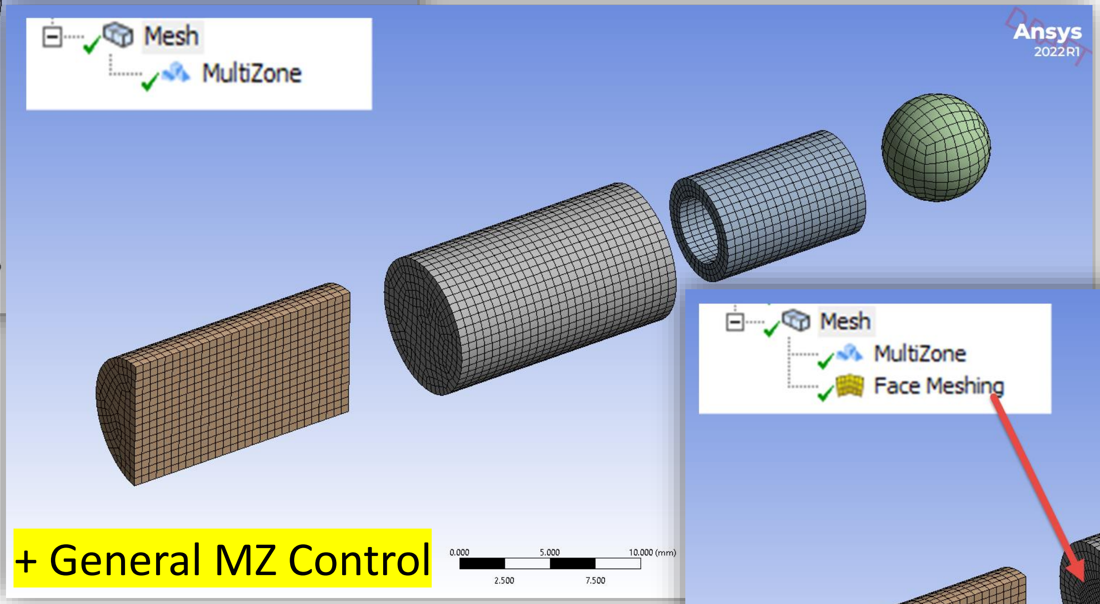


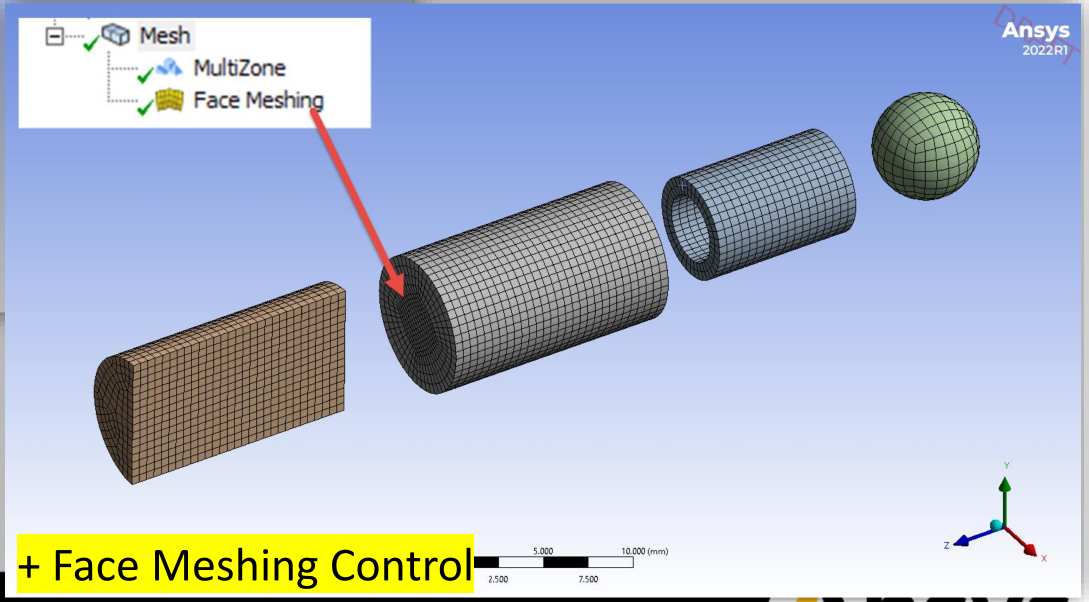
Less Decomposition in Mechanical MZ in 2022 R1



Better Orthogonality and Default Meshing with Explicit Prefs

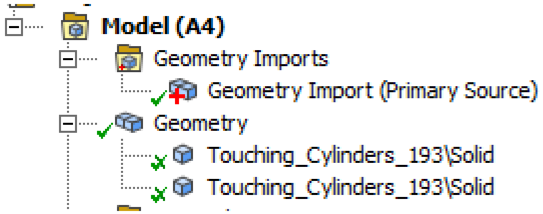


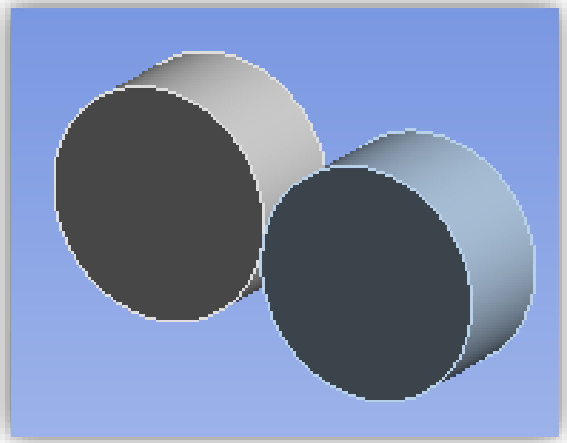




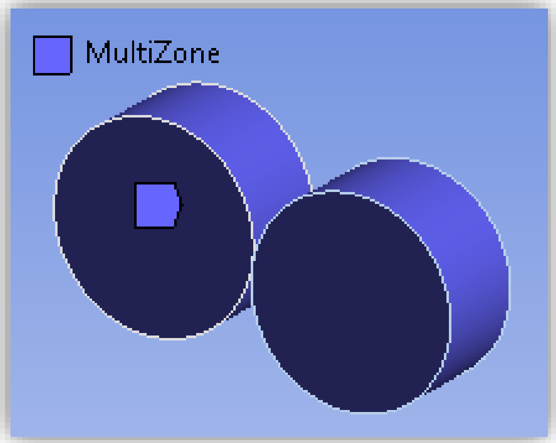
Better default mesh for cylinders, circles, and spheres

Touching Cyclinders

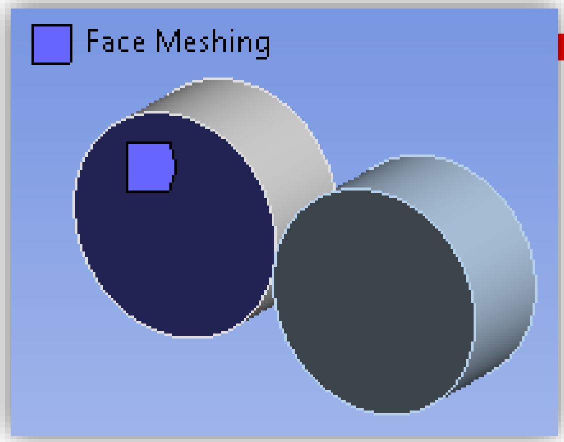


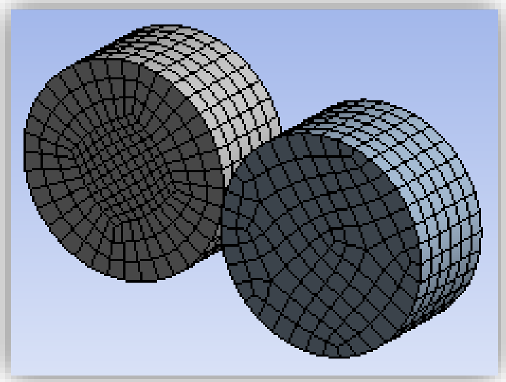


MultiZone Applied with no special inputs/selections – Automatic Hex



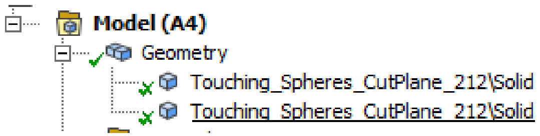
If Face Meshing is Applied O-Grid is automatic with no decomposition needed

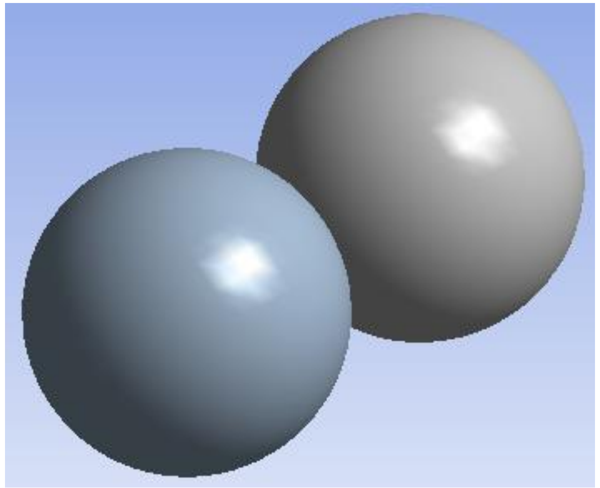




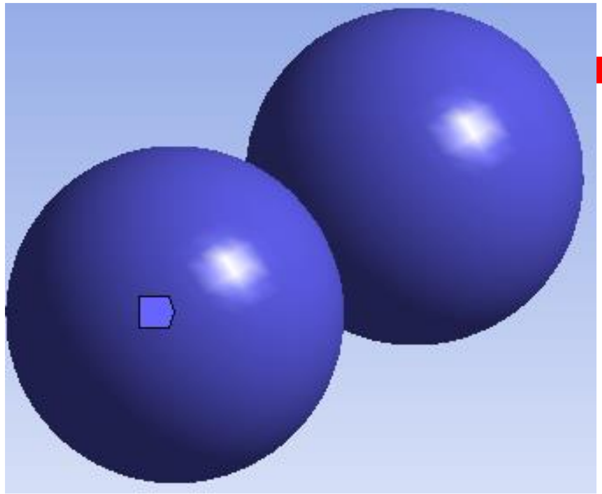
Better default mesh for cylinders, circles, and spheres

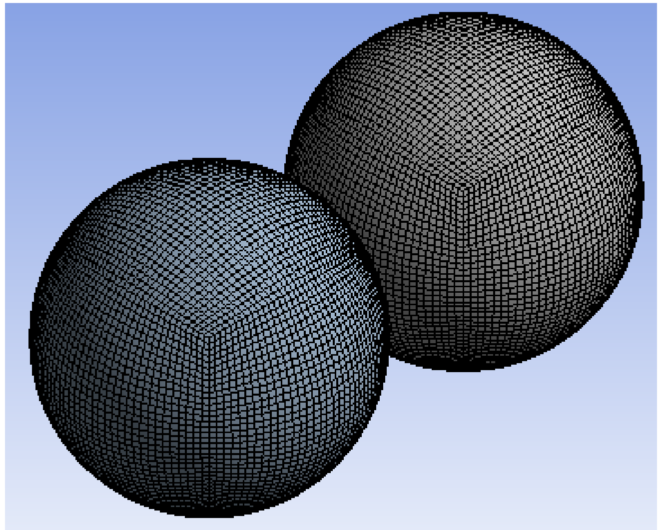
Touching Spheres

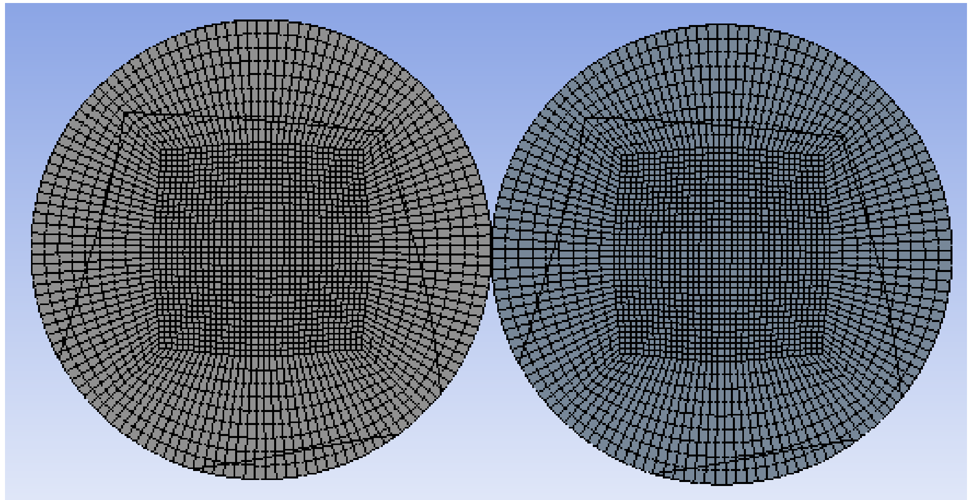




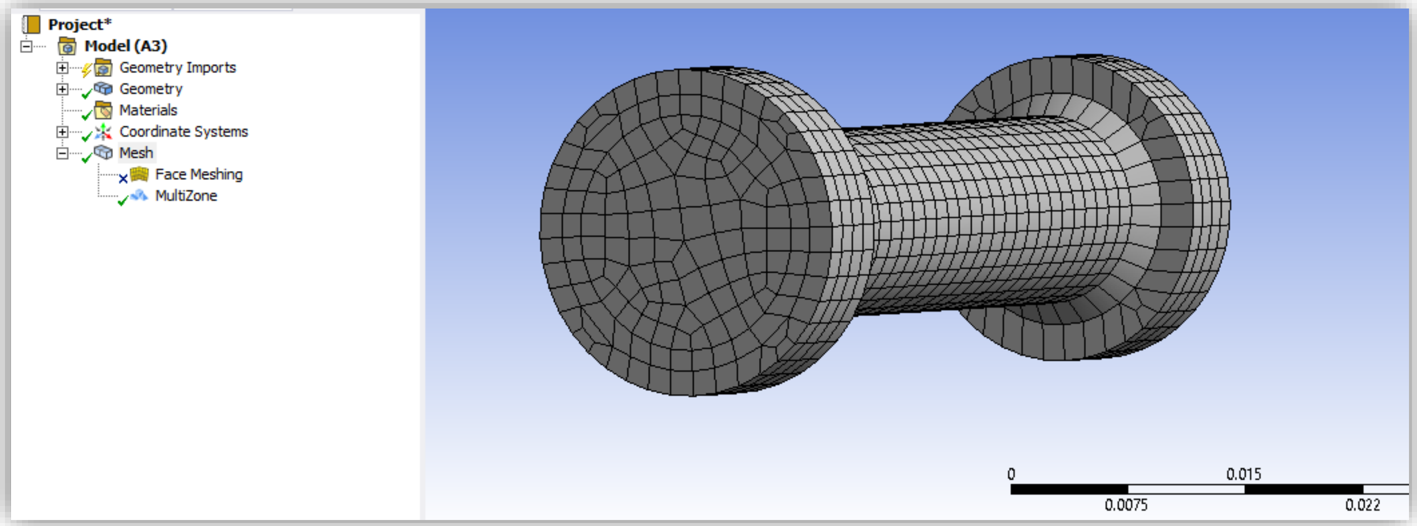
MultiZone Applied with no special inputs/selections – Automatic Hex



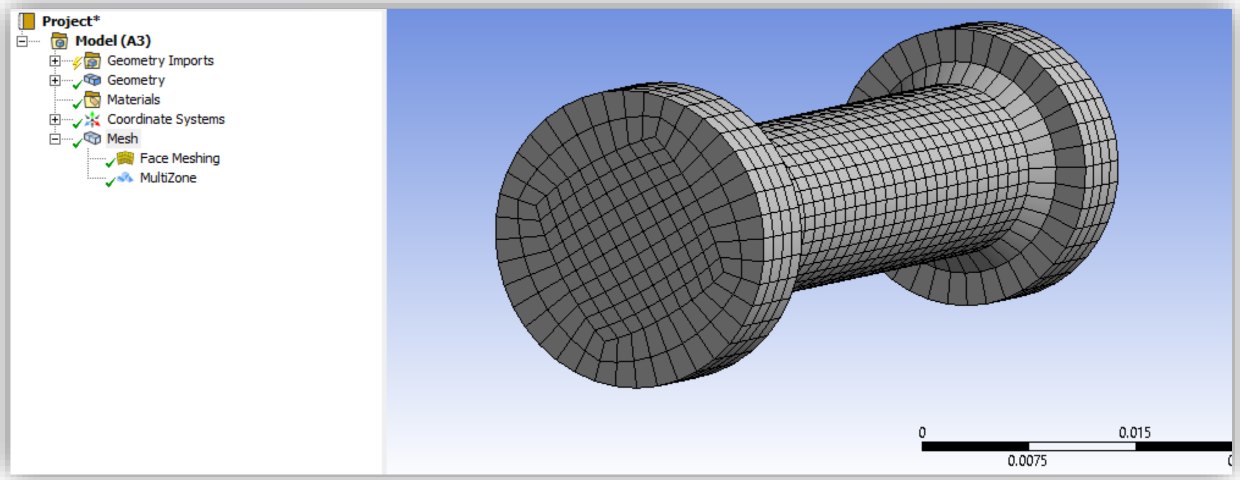




Improved Hex Mesh for Simple Shapes



O-Grid results are more smoothed for Explicit Phys. Pref.



LS-DYNA Solver Updates

Iso-geometric Analysis *IGA

NEW Keyword family dedicate for IGA

Full compatibility with standard CAD

Automatic connections through topology

Analysis suitable model generation

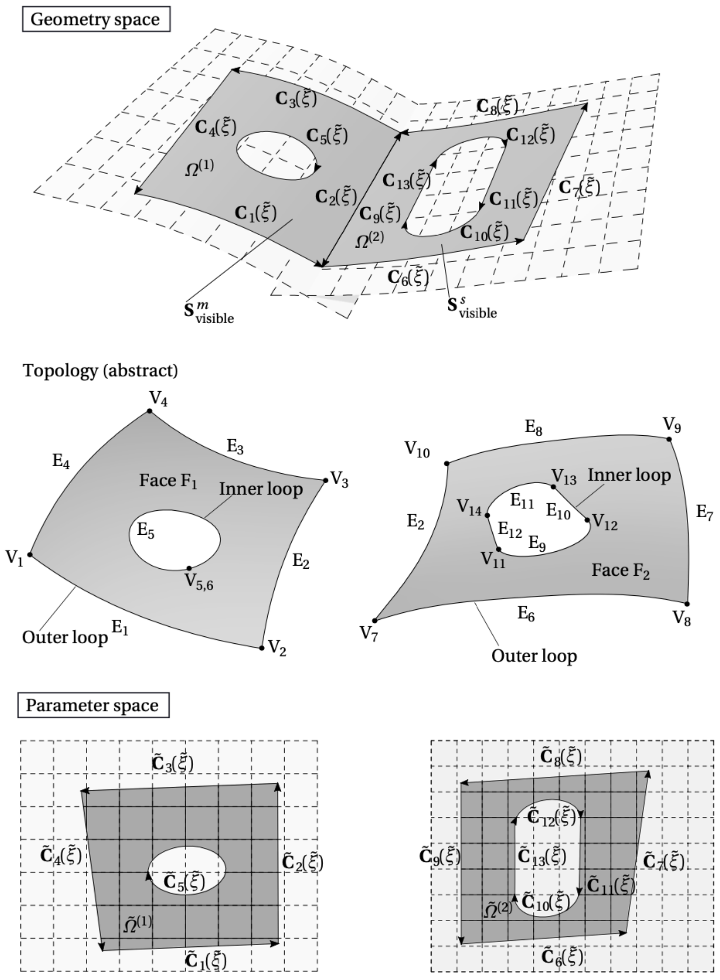
- Parameterization for improved analysis
- Associativity for analysis related features
- Extended topology, e.g., T-joints

Extensions to volumes (and curves)

Working on/with the exact (CAD) geometry

- Boundary conditions
- Contact
- Constraints and connections

- Available with LS PrePost and Ansa



LEIDINGER 2020

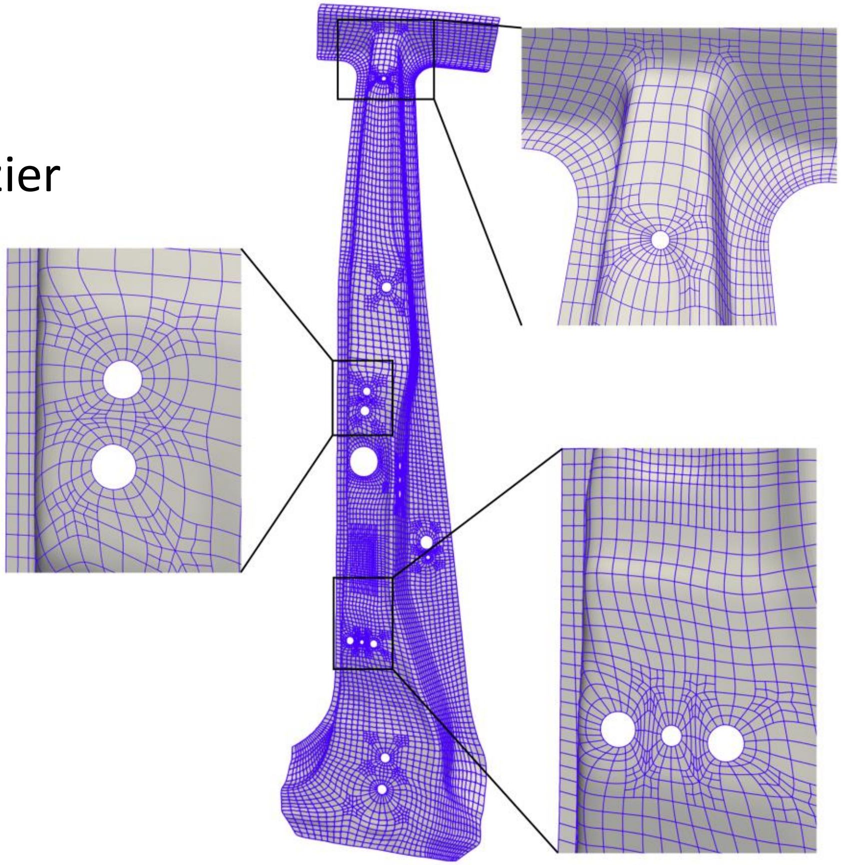
Unstructured splines

Represent arbitrary topologies with a single patch (at least *C*^{1})

Geometry input using elements in Bézier decomposed form, i.e., `*IGA_INCLUDE_BEZIER`

Model highlights:

- Advanced plasticity
- Contact with impactor (loading)
- Element erosion
- Spotwelds with HAZ



CASQUERO ET AL. 2020

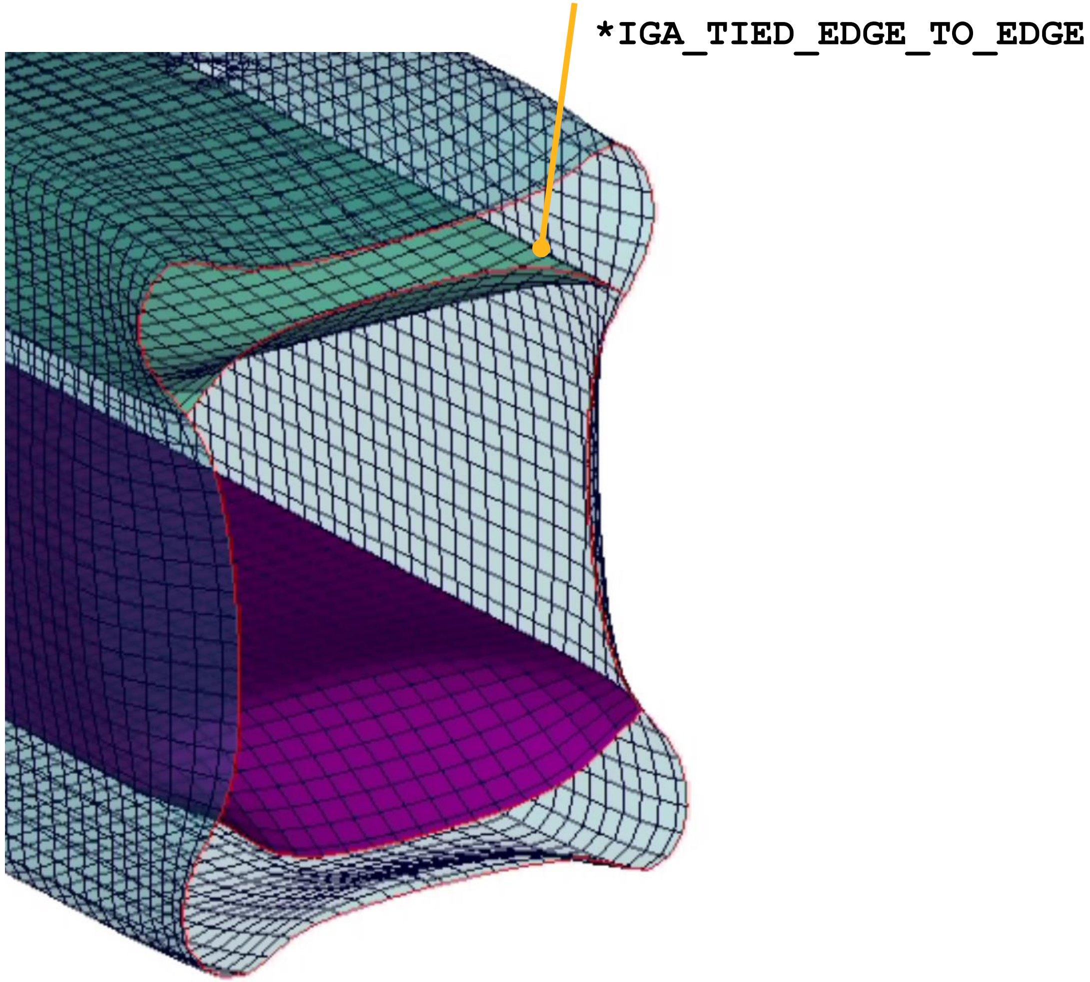
T-joints - analysis

Allows more flexibility in connecting different patches e.g. at T-junctions

Topology defines connections

Geometry defines mechanical coupling

- Merged control points along shared edges with conformal parameterization (untrimmed, C^0)
- Weak, penalty-based coupling along shared edges

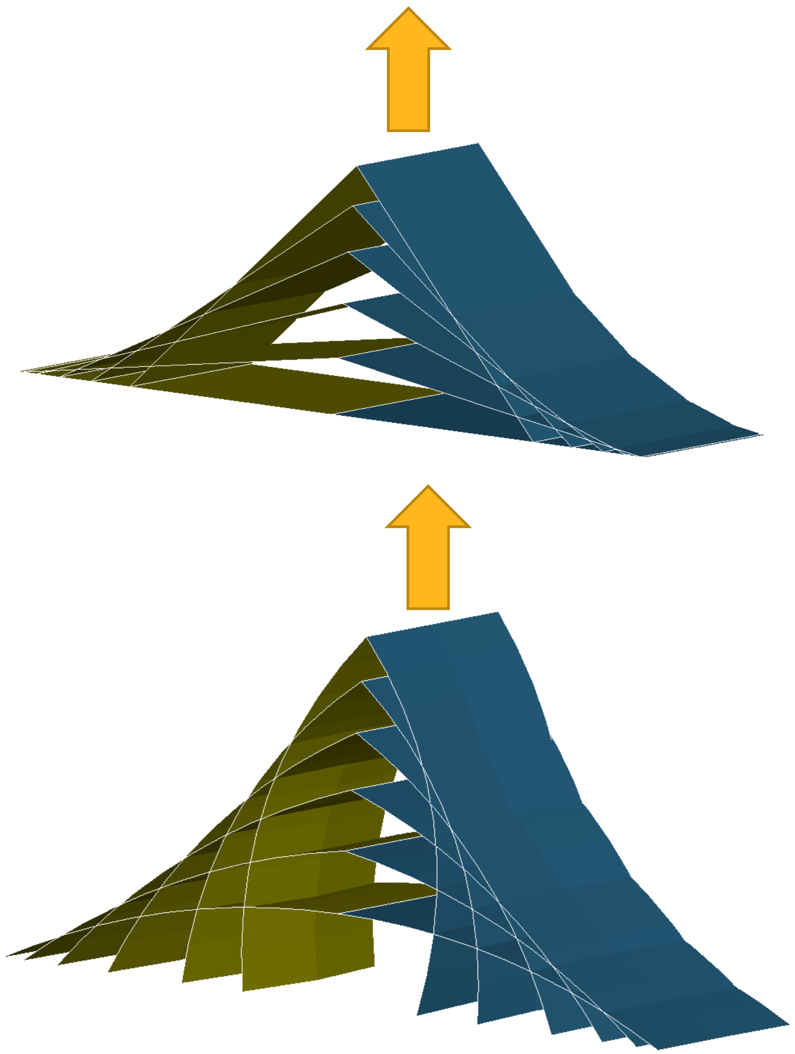


*IGA_TIED_EDGE_TO_EDGE with rotation free Shells

Allows to preserve original angles between patches

initial:

- 2 patches coupled via *IGA_TIED_E2E
- Different angles between patches at interface (0°, 10°, 20°, 30°, 40°, 50°)
- Rotation free shell formulation



deformed:

- Initial angles are preserved with improved *IGA_TIED_E2E

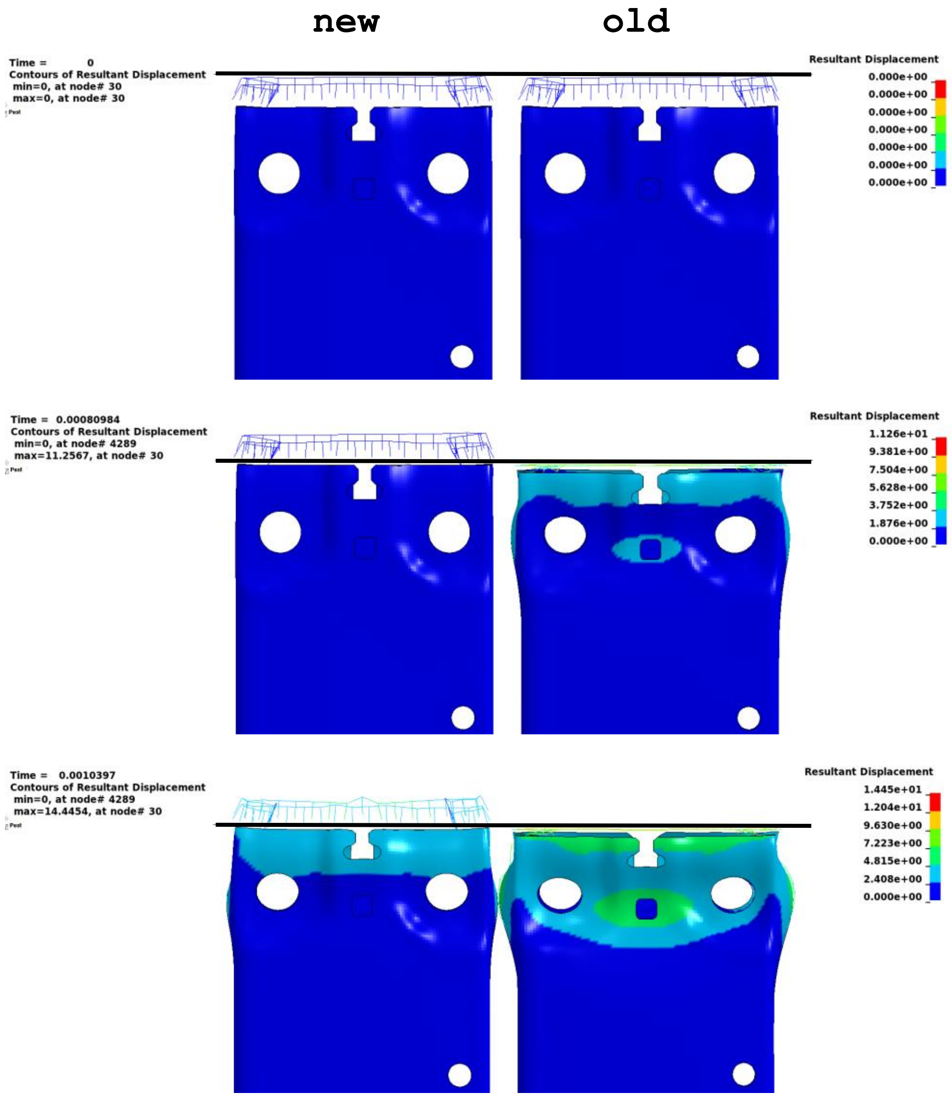
IGA with *RIGIDWALL_PLANAR

old:

- Contact enforced as constraints at control points (CPs)
- CPs usually not part of actual geometry
- **Less Accurate behavior**

new:

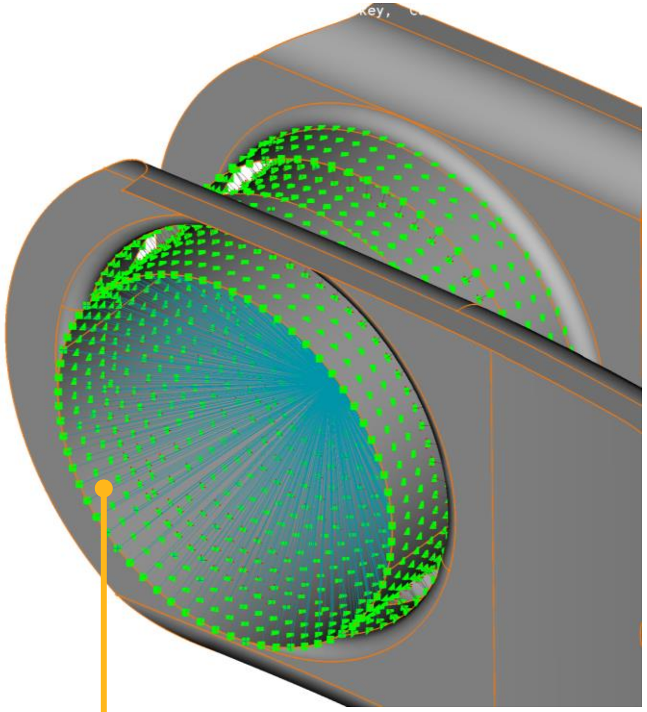
- Contact enforced via penalty constraint at interpolation nodes
- Interpolation nodes always part of actual geometry
- **CORRECT behavior captured**

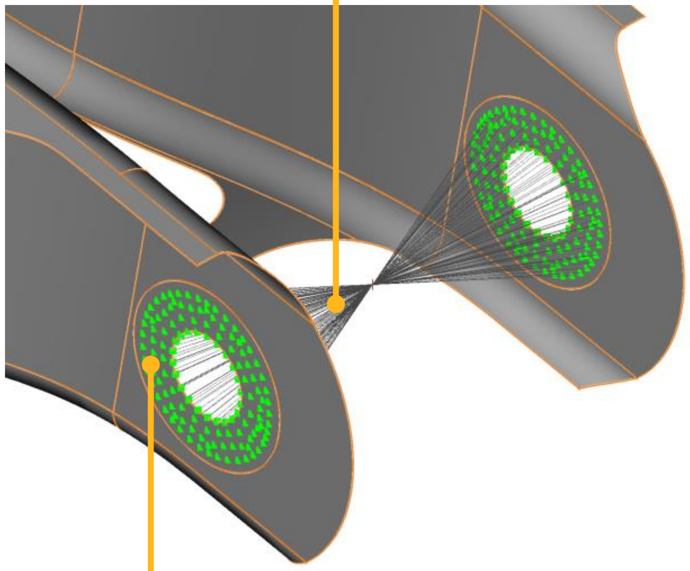


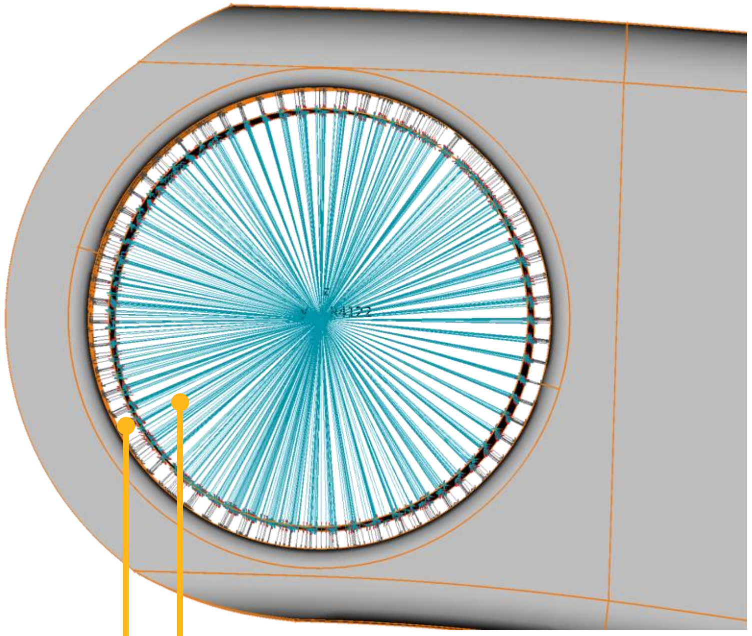
RBEs

- New Keywords also support RBEs

*CONSTRAINED_NODAL_RIGID_BODY (RBE2)







*IGA_POINT_UVW
parametric points on the isogeometric shell

*IGA_POINT_UVW
parametric points on the isogeometric shell

*CONSTRAINED_INTERPOLATION (RBE3)

*CONSTRAINED_NODAL_RIGID_BODY (RBE2)

Welds

Spotwelds

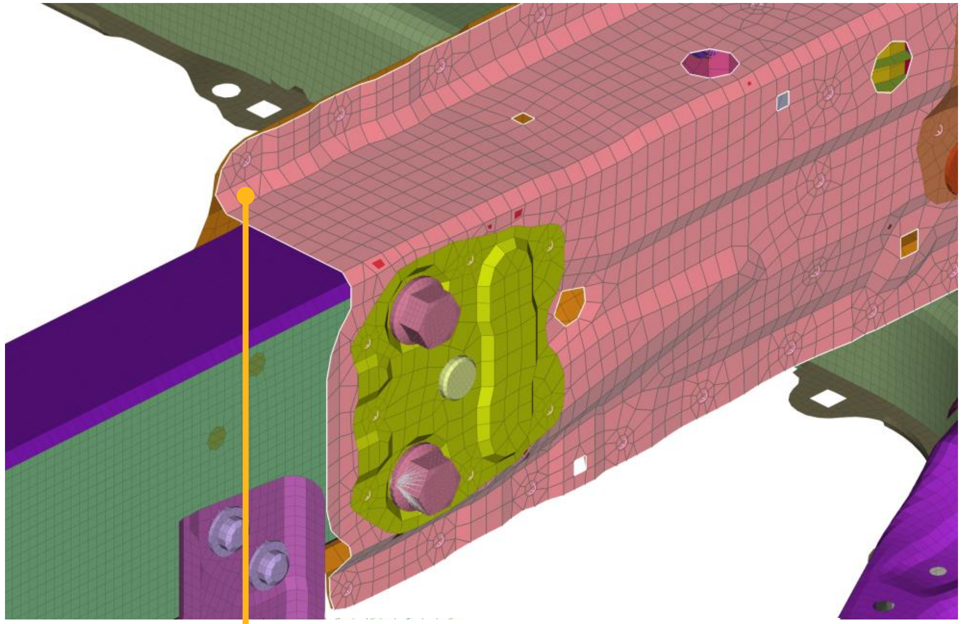
- Beams (penalty constrain)
- Solids and tied contact
- **SPR3 (New capability)**

Line welds *(several improvements)*

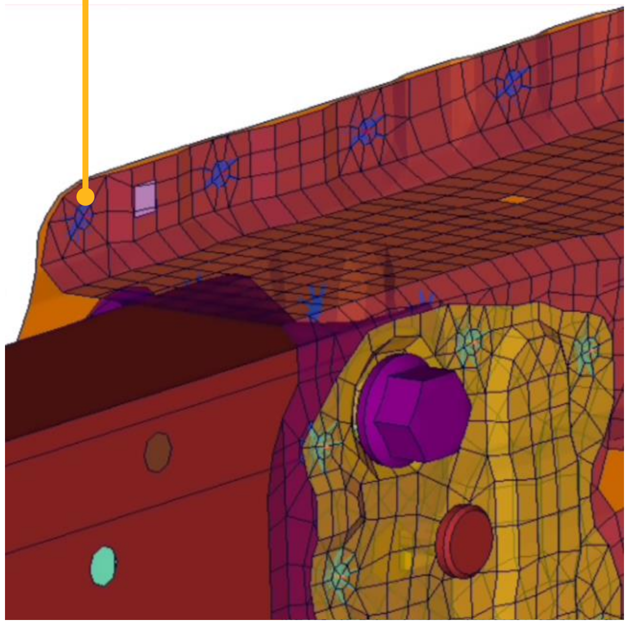
Failure

Heat affected zone *(several improvements)*

***CONSTRAINED_INTERPOLATION_SPOTWELD** (SPR3)



IGA no special meshing
Mesh independent formulation
No remeshing needed if position of the spotweld changes



FEA conformal mesh ("washers")
Mesh dependent solution
Remeshing needed if position of the spotweld changes

Continuum shells

NEW Formulation allows for prediction of ductile fracture governed by stress triaxiality.

Continuum shells

- Added thickness DOF with mass scaling
  - No time step penalty compared to solids
  - No significant change in rigid body response
- Linear through-thickness normal strains

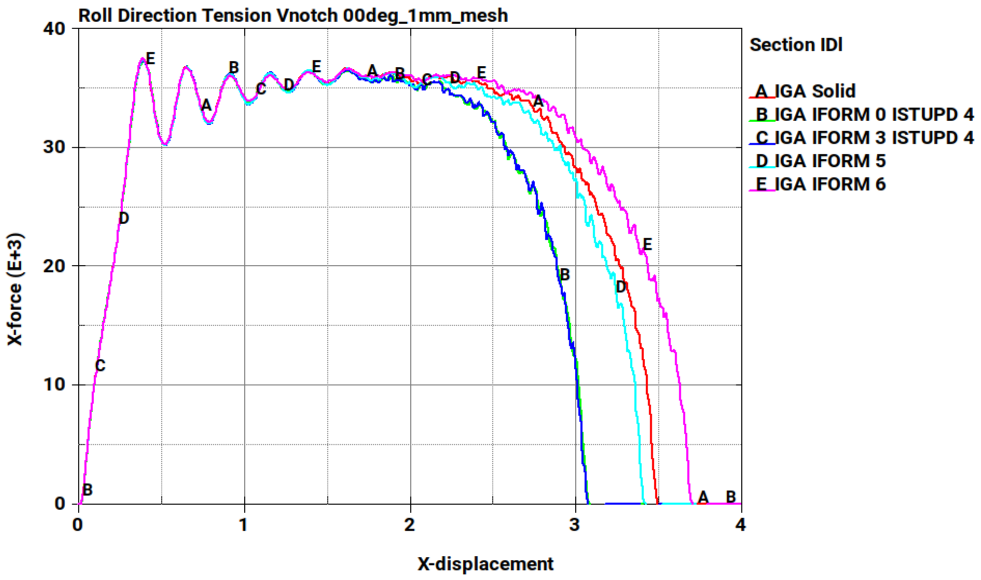
VS

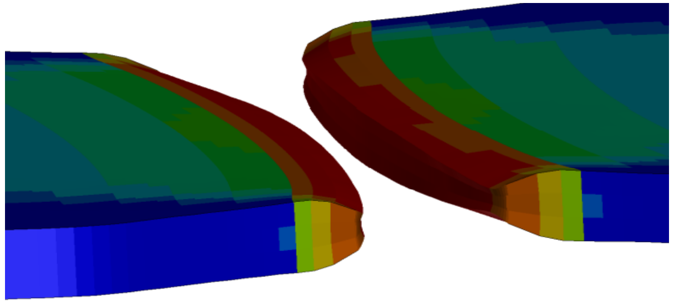
Shells

- Incomplete stress tensor - zero normal stress
- Inaccurate failure prediction

Solids

- Accurate but expensive





plastic strain in fracture zone

Materials

Modular Inelasticity

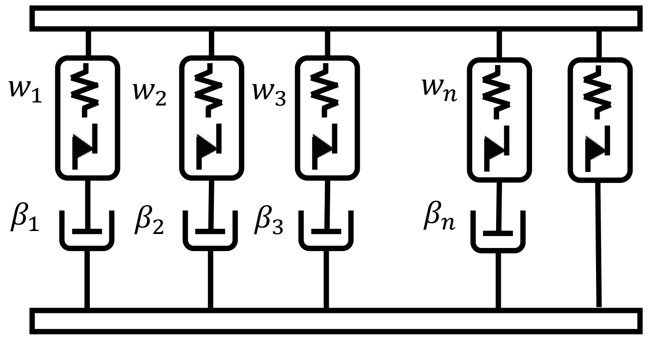
Norton-Bailey creep

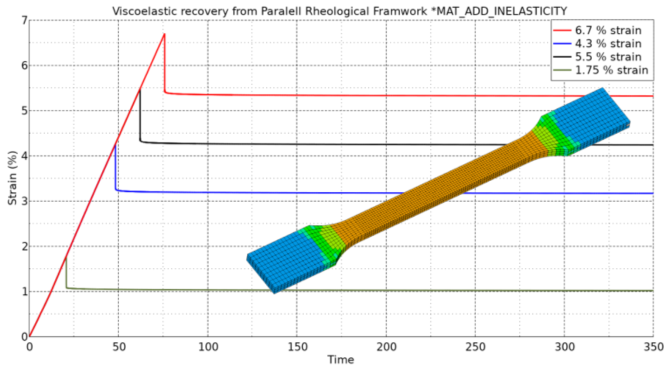
von Mises plasticity

→ **`*MAT_ELASTIC`**

Prony viscoelasticity

→ **`*MAT_PIECEWISE_LINEAR_PLASTICITY`**





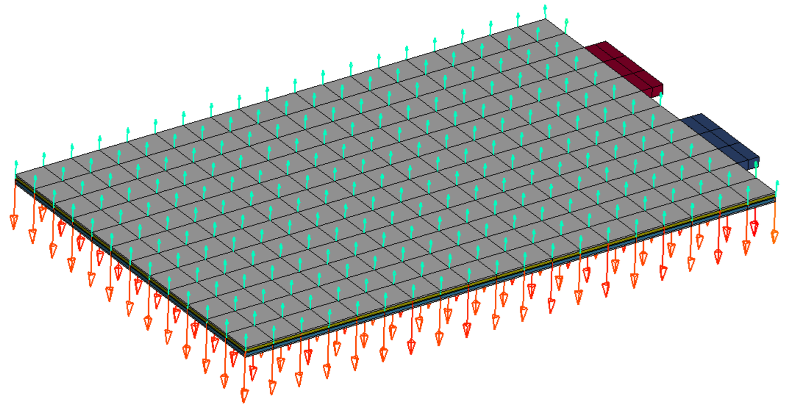
- *MAT_ADD_INELASTICITY **allows for amending inelasticity effects to any material model**
  - Add “missing” feature to a material model
  - Model a chain of polymeric networks
  - New features include
    - Orthotropic field overlay for thermal expansion and failure
    - Temperature shifts in hypo-viscoelasticity
    - New hyper-viscoelastic laws
    - Support of high order tetrahedrons
    - Norton-Bailey creep

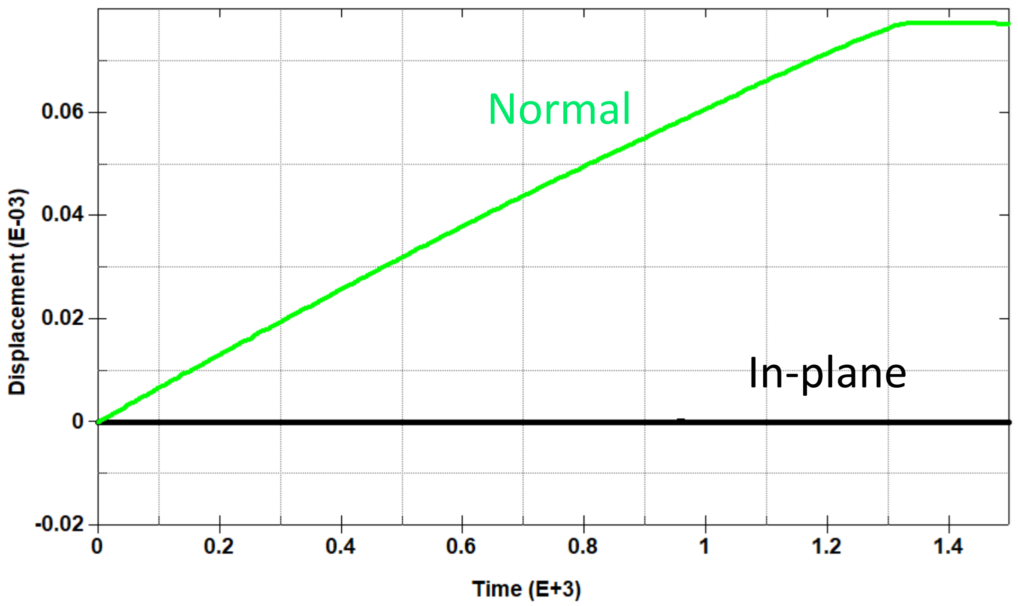
Norton-Bailey creep in uniaxial tension and release

State of charge expansion with orthotropy

- New orthotropy options for *MAT_ADD_SOC_EXPANSION
  - State of charge (SOC) from EM module induces structural displacements
  - Independent expansion coefficients MULT, MULTY, MULTZ for orthotropic materials
  - Or as curve input LCID, LCIDY, LCIDZ as function of current state of charge

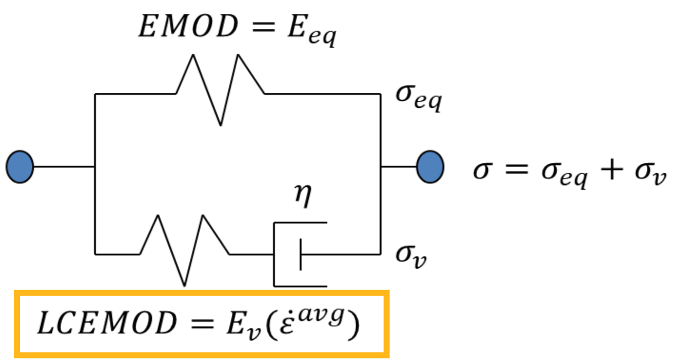
- Possible application: Battery modeling
  - Example: Unit cell with discharge
  - Consider swelling in thickness direction →



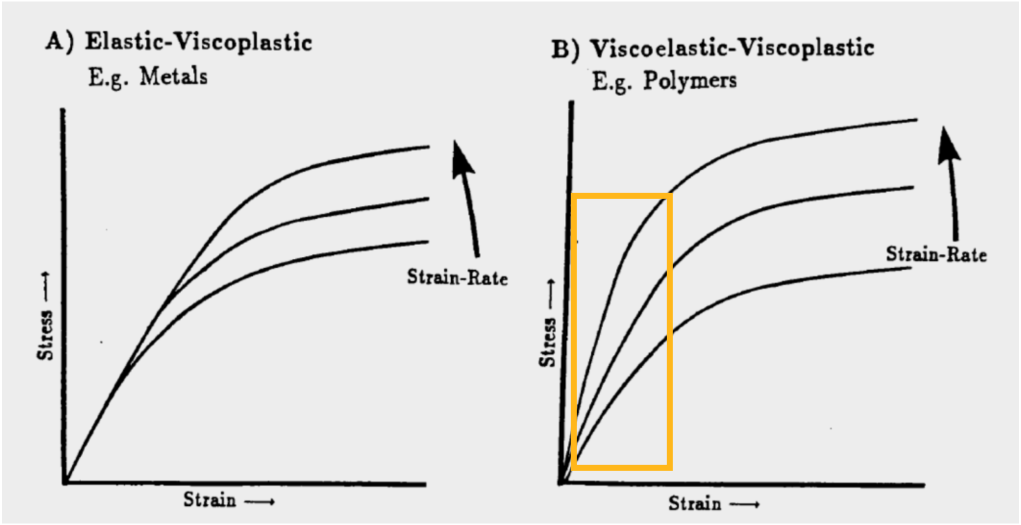


Plasticity plus non-linear viscoelasticity

- Enhancement for *MAT_123_RATE
  - Basis: vMises viscoplasticity with tabular input
  - Now supplemented with non-linear viscoelastic model
  - Youngs's modulus as function of (averaged) strain rate
  - Constant decay coefficient representing viscosity



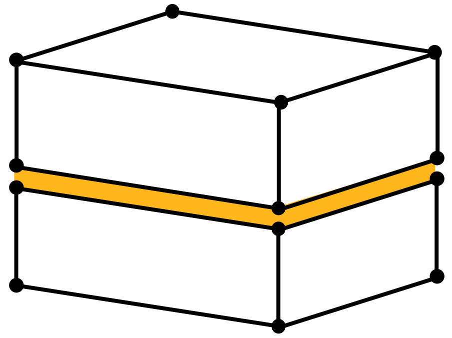
- Many polymers show viscoelastic-viscoplastic behavior
  - Easy material calibration with new approach



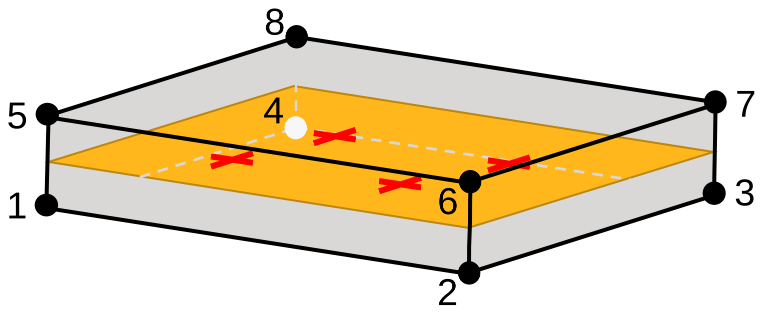
e.g. Figure from Diehl, T. "Modeling of elastic-viscoplastic behavior and its finite element implementation." (1988)

Enhancements for cohesive elements / materials

- Support of foam materials
  - *MAT_ADD_COHESIVE now works with *MAT_057 and *MAT_083
  - For thin layers (pads) of highly compressible foam



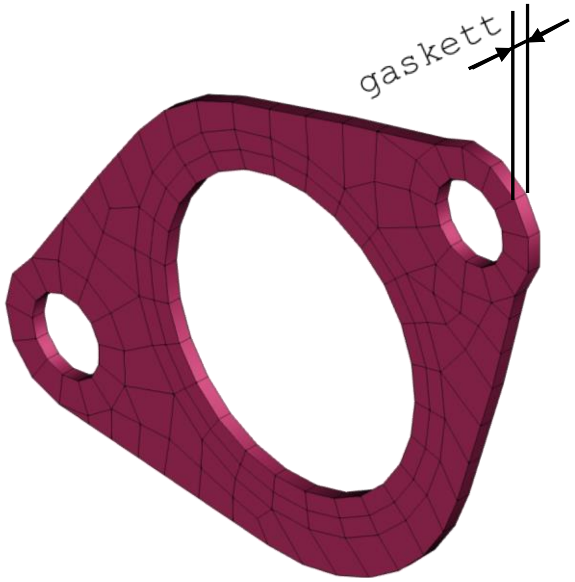
- Support ICOH=1 (*CONTROL_SOLID) for *MAT_169
  - This is automatic deletion of cohesive elements if neighboring shell or solid elements fail
  - Already available for genuine cohesive materials (138, 240, …), but now also for this “special” *MAT_ARUP_ADHESIVE

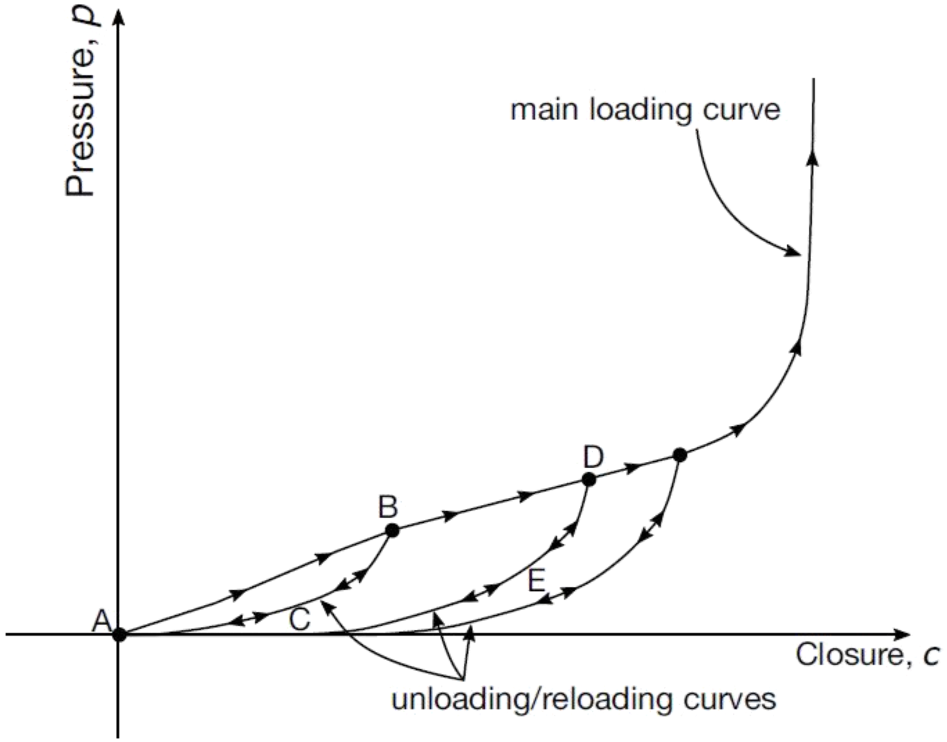


- Add new options to *MAT_240
  - Include connection partner properties: keyword option _FUNCTIONS
  - Flexible exponent for yield and damage initiation criterion (INICRT<0)

Gasket element

- Combine *MAT_COHESIVE_GASKET, cohesive elements (ELFORM=19/20) and GASKETT on *SECTION_SOLID to model gaskets
  - GASKETT corresponds to gasket thickness for membrane behavior
  - Out of plane behavior governed by loading and unloading curves



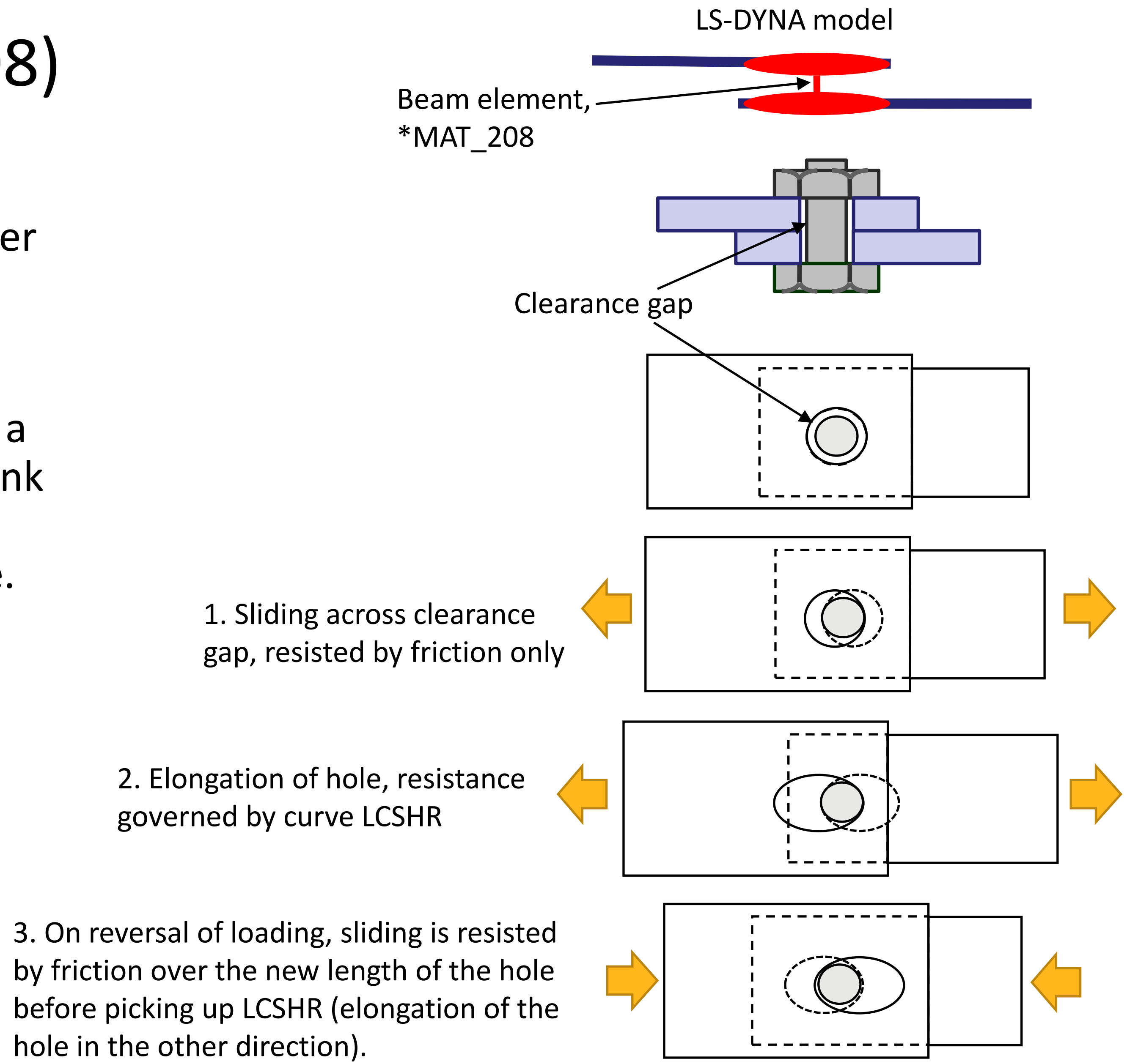


Bolt preload and subsequent loading of gasket used in interfacing manifold of exhaustion pipes to potential vehicle

*MAT_BOLT_BEAM (*MAT_208)

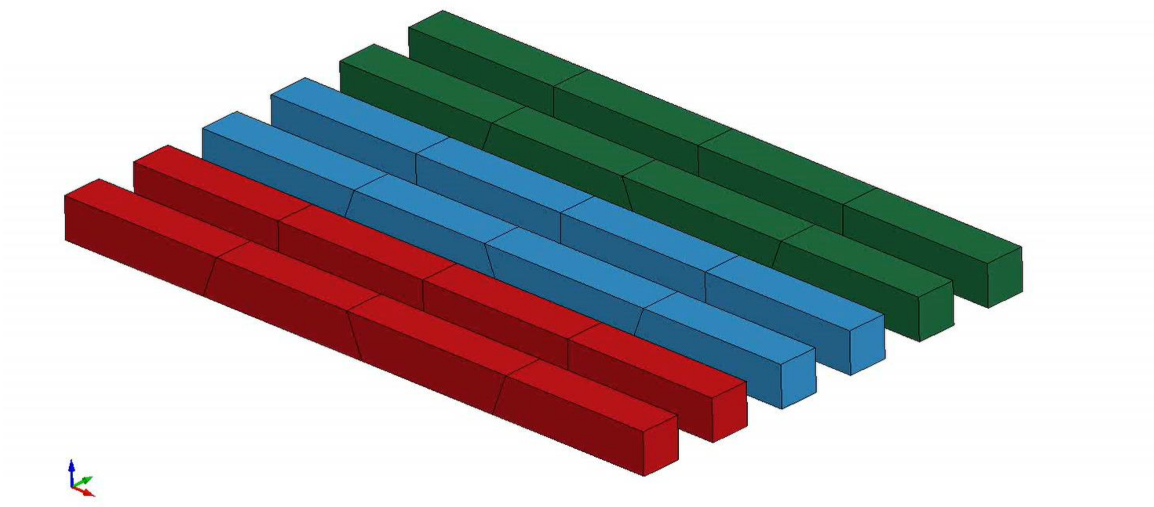
- Models the forces transmitted across a bolted joint, including preload, clearance gap and other features.

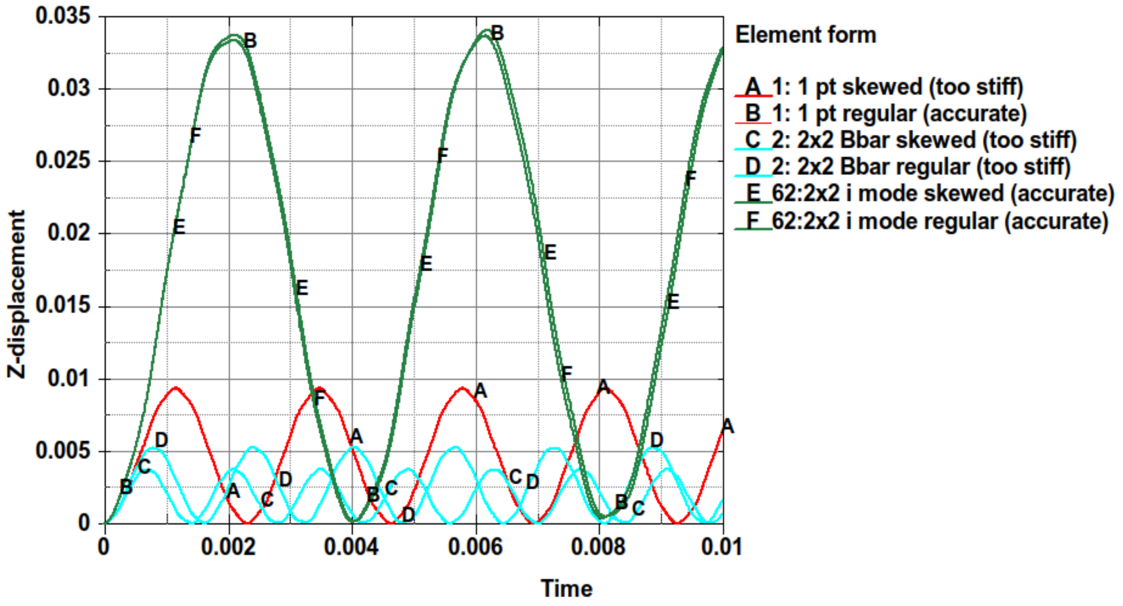
- New in R13: input parameter HOLSHR enables a shear deformation mode in which the bolt shank tears through the plates, enlongating the hole. Cyclic behavior accounts for the enlarging hole.



Solid element form 62

- Fully integrated brick element
- Incompatible modes by assumed strain
- Fast, accurate, and robust
- Accurate bending, even skewed





New options for Pushout Vent

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

Card 2	1	2	3	4	5	6	7	8
Variable	JT	IDS1	IDS2	IOPT1	PID1	IPD2	VANG	LCRED
Type	I	I	I	I	I	I	F	I
Default	0	none	none	none	none	none	0.	none

Optional card.

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

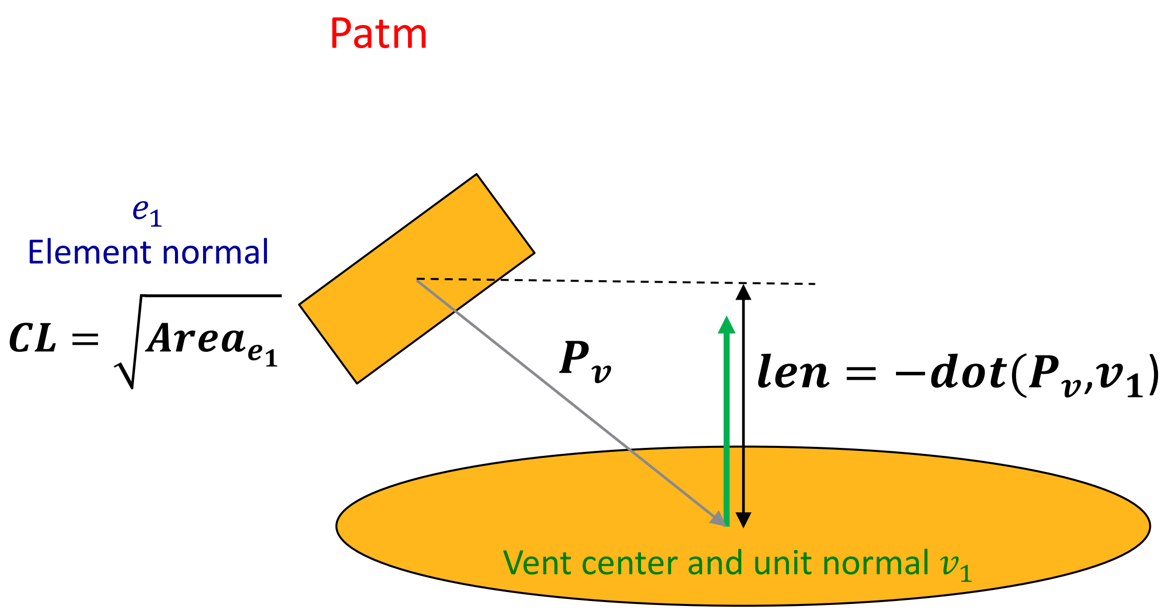
Pushout vent: IOPT=200

PSETPV: Part set ID: Internal airbag parts interact with this pushout vent.
SFPV: Scale factor for element characteristic length

If PSETPV is given, the ambient pressure is applied to the airbag element which has been push out from vent and has greater distance away from vent surface defined by the next slide.

Ansys

Pushout Vent



If $len > SFPV * CL$
Patm is applied to the e_1 surface facing the air.

Extension of *DEFINE_CPM_VENT

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

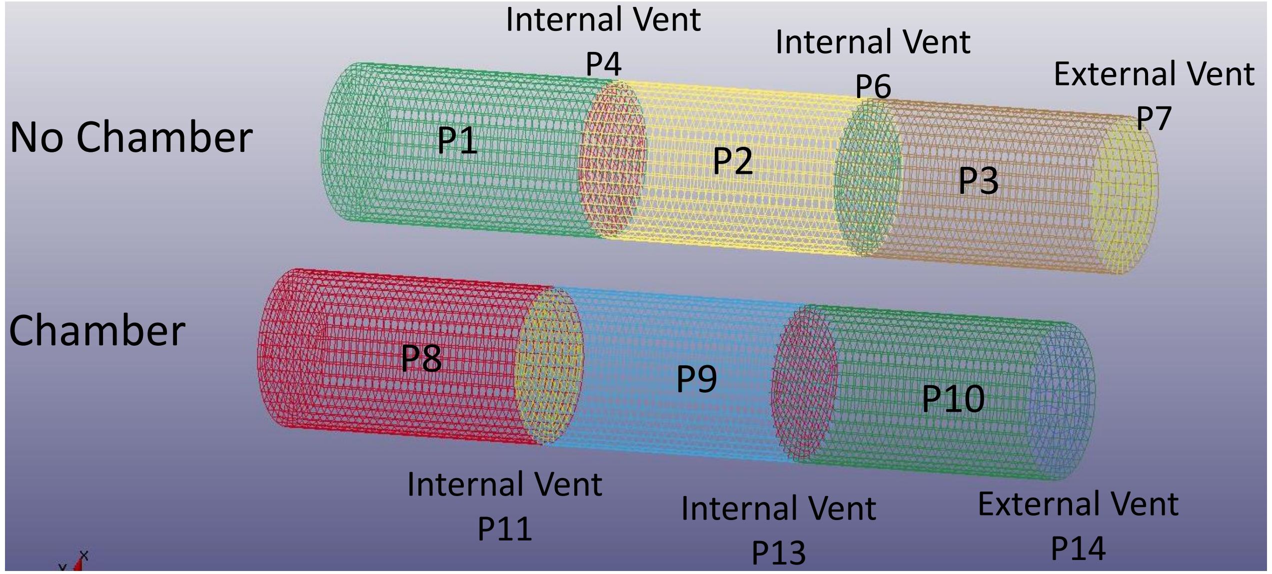
Card 2	1	2	3	4	5	6	7	8
Variable	JT	IDS1	IDS2	IOPT1	***PID1***	***PID2***	VANG	LCRED
Type	I	I	I	I	I	I	F	I
Default	0	none	none	none	none	none	0.	none

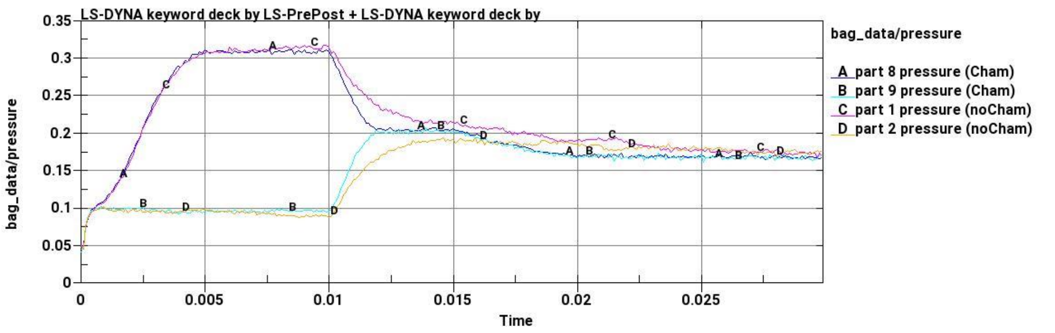
PID1 and PID2 are upstream and downstream parts for the internal vent. The part pressures will be used in the flow equation to determine the discharge probability density function. The feature was only applied to the bag with chamber definition.

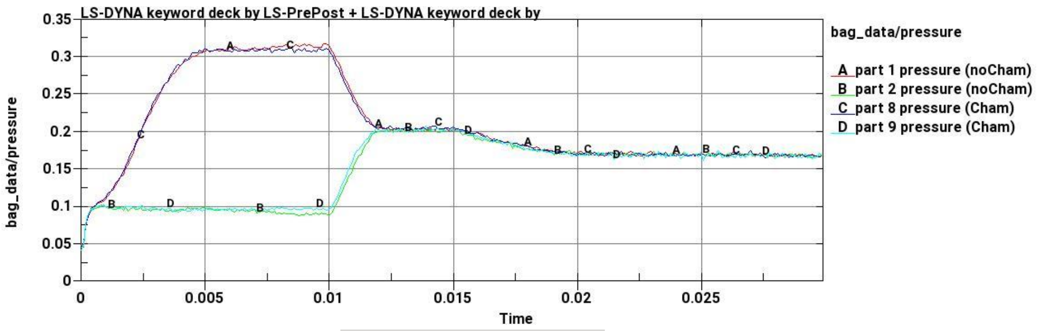
Now, we have removed the limitation and PID1/PID2 can also be used for vents (internal and external) without chamber definition.

1. If PID1/PID2 is external part, the part pressure will be used.
2. If PID1/PID2 is internal part, the part pressure+ (shell normal direction) will be used.
3. For external vent, user should define PID1=PID2

Internal vent with/wo chamber definition







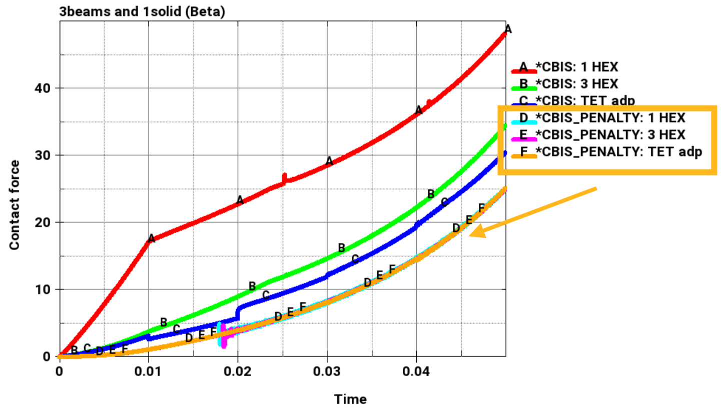
New Option

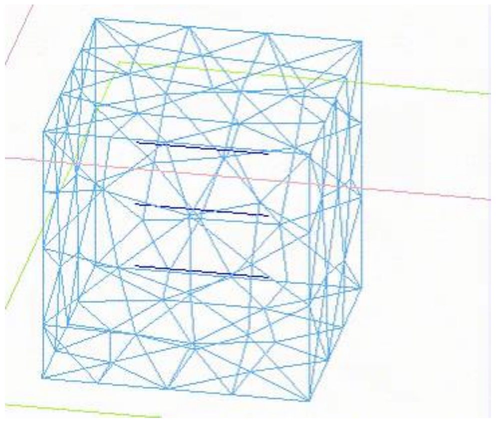
- With chamber defined, the vent rate through internal vents 11,13 and external vent 14 are calculated by flow equation. The part pressure for part 8 and 9 reach equilibrium after a short time.
- Without chamber, the mass can move bi-directional through internal vents 4 and 6. This takes much longer time for part 1 and 2 reach same pressure.

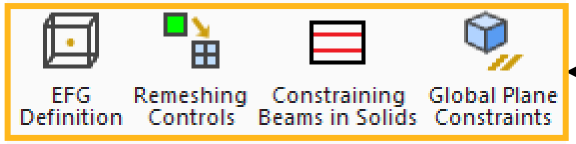
- With the new option, part pressure and vent rate match very well with/without chamber definition. The feature blocks the bidirectional flow of the internal vents.

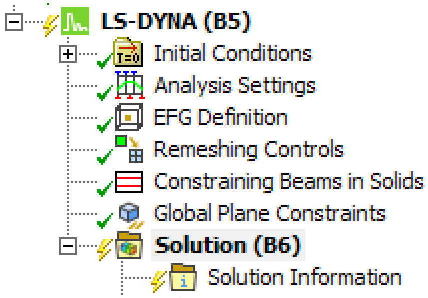
3D Adaptive FEM / EFG for Compression Molding

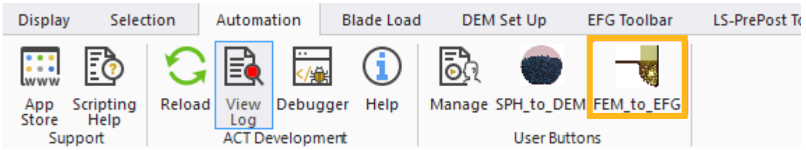
- New remapping capability (work with LST core team)
  - To transfer the internal variables of penalty-based non-conforming coupling formulation (*CONSTRAINED_BEAM_IN_SOLID_PENALTY) between fiber and matrix materials
  - Better numerical stability and accuracy compared to constrained-based coupling formulation
  - Rewrite remapping algorithm to directly transfer internal variables for non-adaptive parts
- ACT on Ansys Workbench (work with LST ACE team)
  - Capable to define all necessary keywords





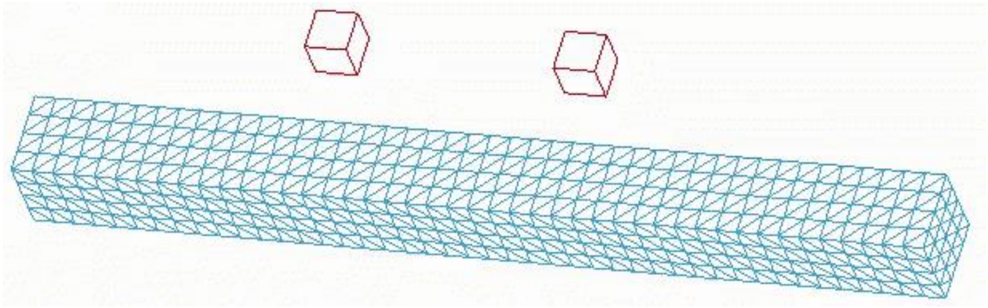


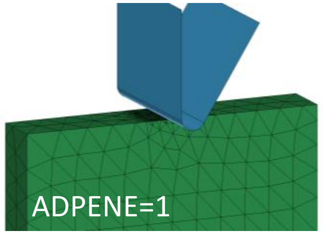


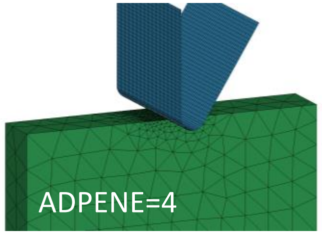


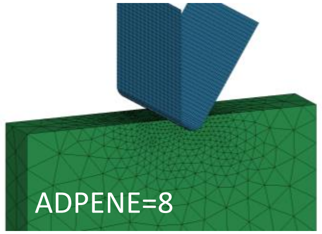
3D Adaptive FEM / EFG for Forging

- New feature in surface triangulation remeshing
  - Smooth transition from fine to coarse mesh when there is no contact curvature information available
  - The level of smoothing is defined through ADPENE in *CONTROL_ADAPTIVE
  - Better surface representation in large material deformation for forging simulation
- New option of defining moving box in *DEFINE_BOX_ADAPTIVE
  - Users are able to customize remeshing size within moving boxes









Enhancement of Smoothed Particle Galerkin (SPG) Features

- MPP for Momentum Consistent formulation in SPG method (MC-SPG)
  - Can handle large-scale models
  - Can handle coupled thermal mechanical analysis

- MMPs (multistage manufacturing processes) analysis by SPG
  - Particularly for mechanical joining process (first stage) and joint strength analysis (second stage)

- Small restart
  - To allow minor changes in a restart simulation for MMPs analysis

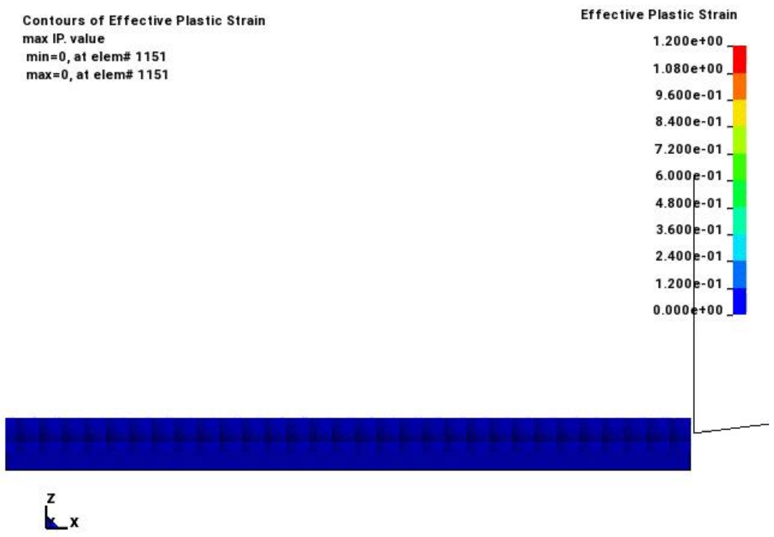
- New input format for *CONSTRAINED_IMMERSED_IN_SPG
  - Improvement of simulation setups

- Material support
  - mat5 / mat14 for wet ceramics, soil
  - mat72r3 for concrete
  - mat193 for silicon or pressure sensitive materials
  - mat59 for composite materials for aerodynamic applications
  - mat124 for different tension and compression strength materials

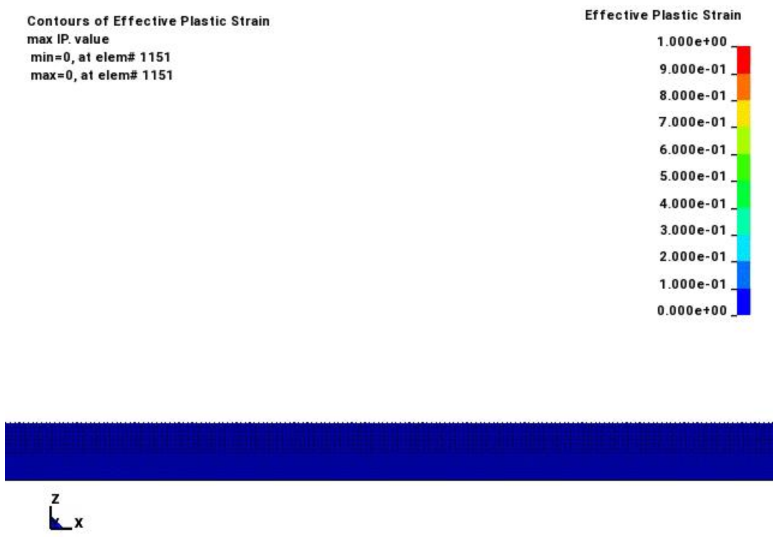
Ansys

Coupled Thermal Mechanical Analysis of 3D Orthogonal Cutting of Steel in Automotive Applications

- Coupled thermo-mechanical analysis with MC-SPG
  - Material failure and separation at tool tip without element erosion
  - Shear band forming and chip bending (important for force prediction)
  - Conservation of mass/momentum



rake angle = 0˚

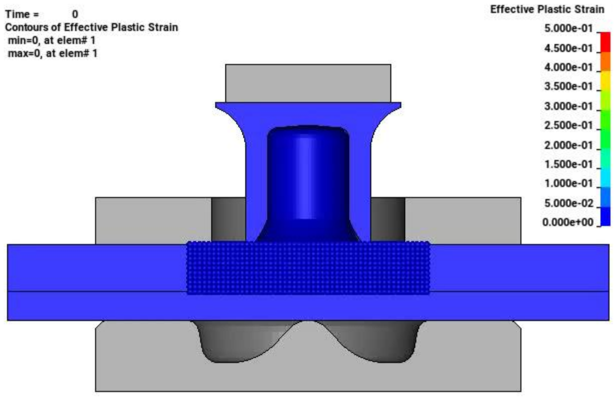


rake angle = 15˚

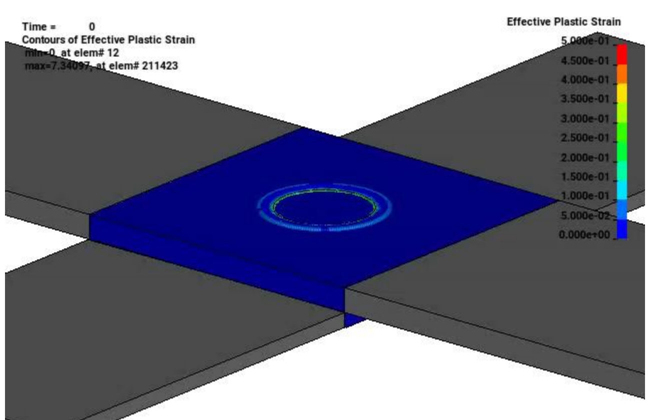
Ansys

Analysis of SPR Joining Process and Joint Strength in Automotive Applications

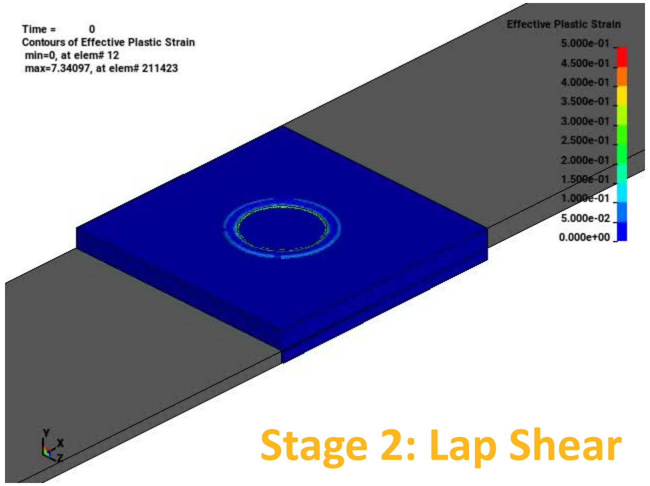
- Two-stage analysis
  - Stage 1: joining process, to form joint through mechanical locking
    - *INTERFACE_SPG_1
  - Stage 2: various strength analyses based on residual deformation from stage 1
    - *INTERFACE_SPG_2



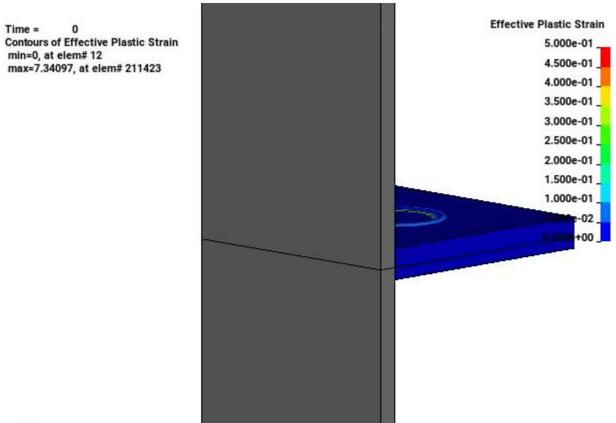
Stage 1: Joining process



Stage 2: Cross Tension



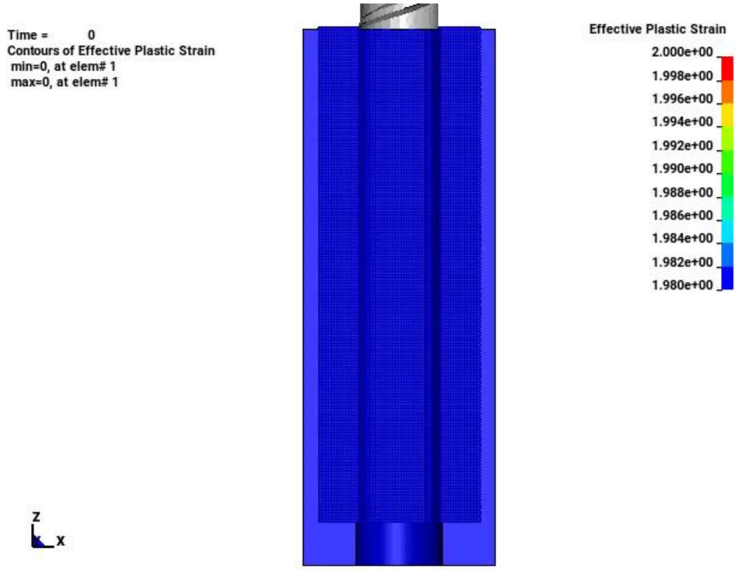
Stage 2: Lap Shear



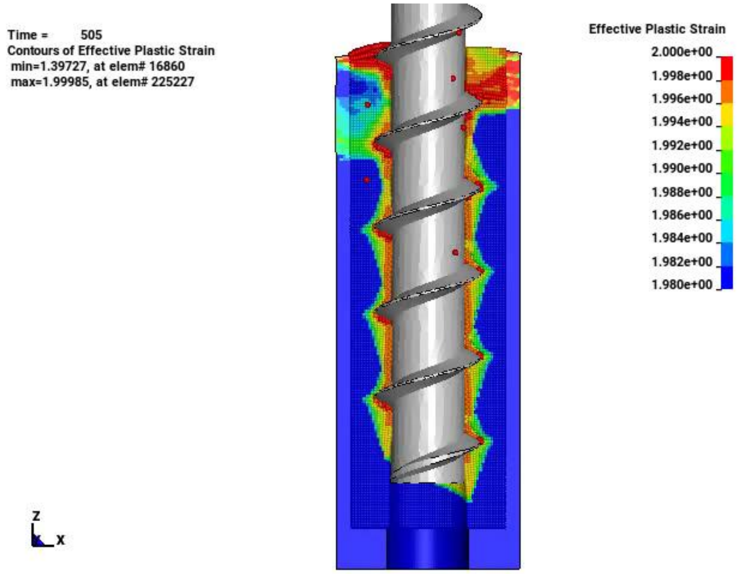
Stage 2: Coach Peel

MMPs Analysis of Concrete Drilling and Joint Strength

- Two-stage analysis
  - Stage 1: concrete drilling process
    - Capture new threads formed in concrete while concrete failure and separation occurring
  - Stage 2: anchor pullout to evaluate the strength of new formed threads



Drilling process

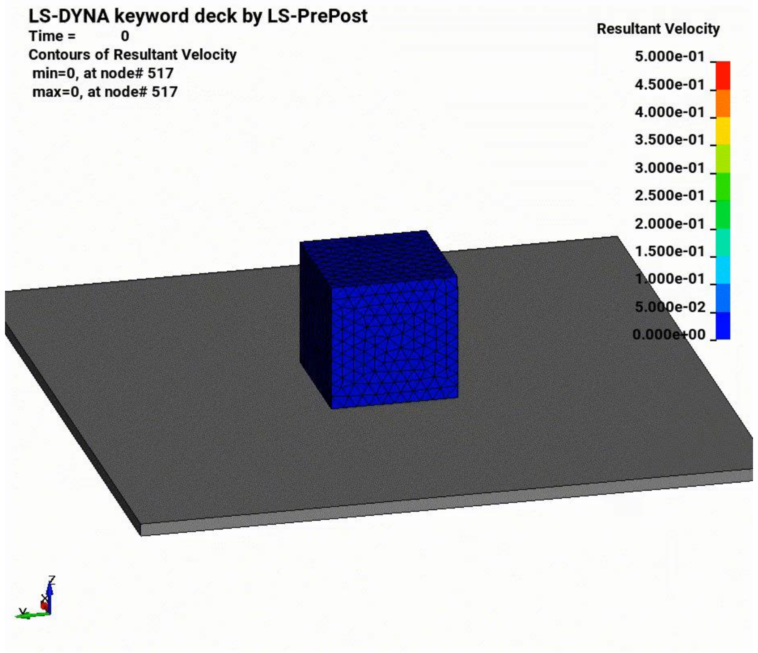


Anchor pullout

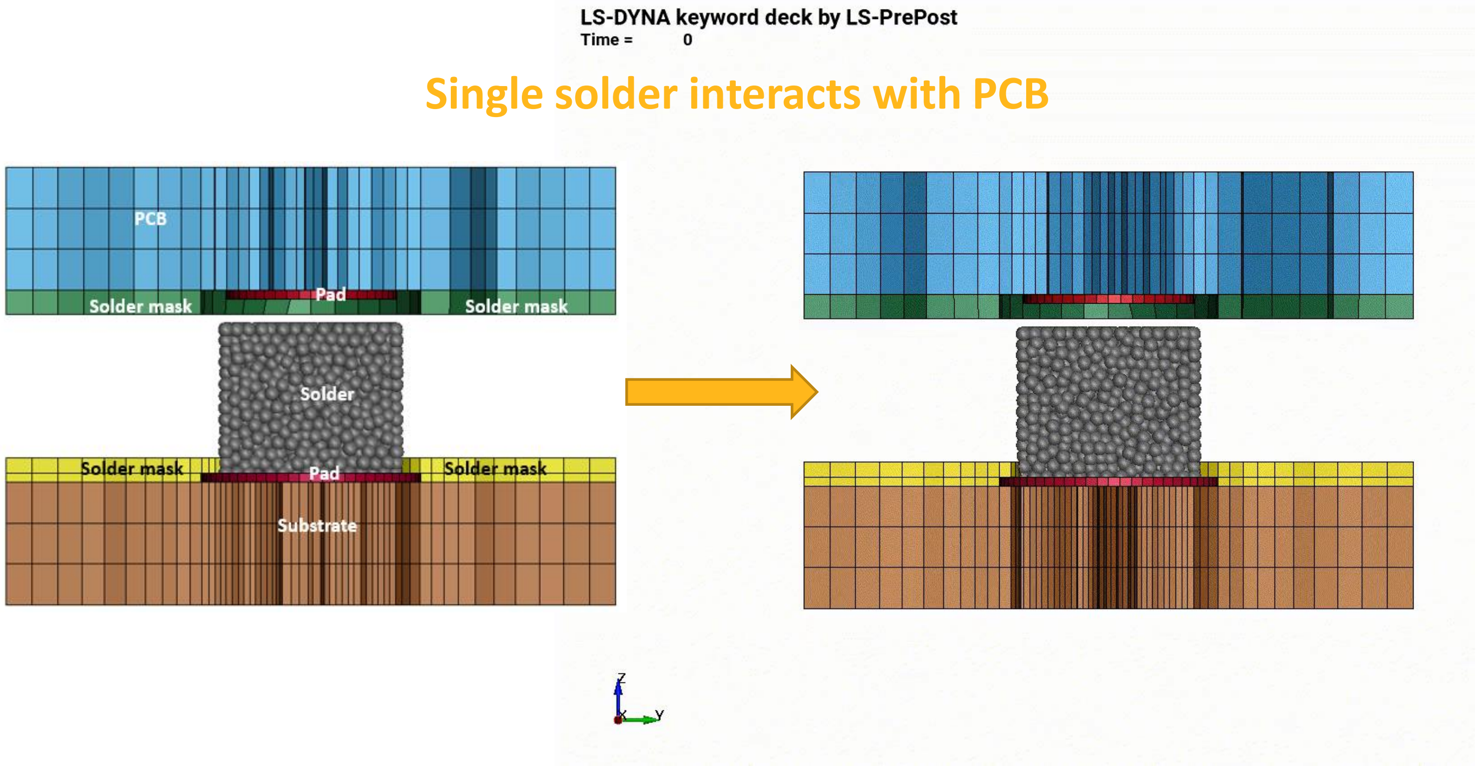
Fully Implicit Incompressible SPG (ISPG) Formulation

- Fully implicit ISPG formulation
  - A new Lagrangian Navier-Stoke solver
  - Can handle the surface tension and wall adhesion accurately and efficiently
  - Can simulate the solder reflow with complex models with the solder mask defined (SMD) pad and NSMD

Large deformation of droplet



Single solder interacts with PCB

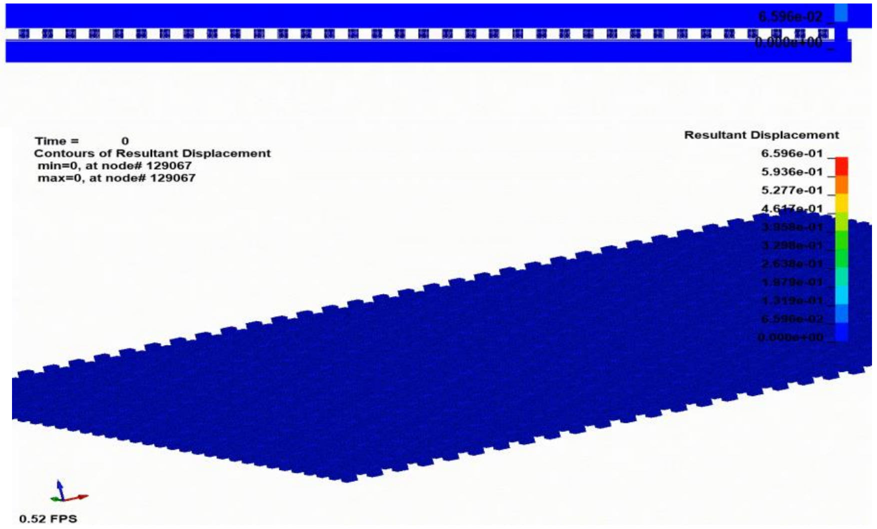


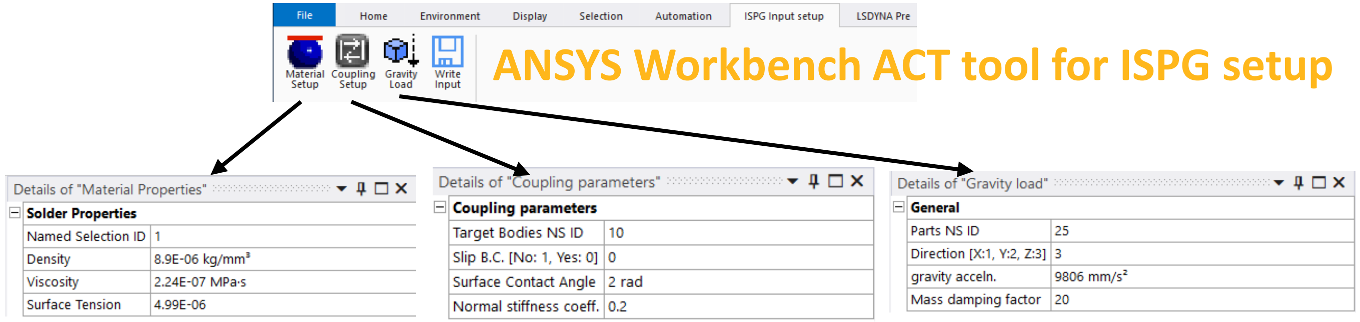
Fully Implicit ISPG Formulation

- Coupled with implicit thermal and structure solvers
  - for large scale thermal-mechanical PCB and solder reflow analysis (considering PCB warpage effect; SMP&MPP)

- ACT on Ansys Workbench available
  - Capable to define all necessary keywords

1225 solders (1.02M nodes) on PCB solved with MPP solver on 64 CPU cores (Run time – 7.5hrs)





ANSYS Workbench ACT tool for ISPG setup

Improvement of *DEFINE_DE_INJECTION

Card 1	1	2	3	4	5	6	7	8
Variable	PID	SID	XC	YC	ZC	XL	YL	CID
Type	I	I	F	F	F	F	F	I
Default	none	none	0.0	0.0	0.0	0.0	0.0	0

Card 2	1	2	3	4	5	6	7	8
Variable	RMASS	RMIN	RMAX	VX	VY	VZ	TBEG	TEND
Type	F	F	F	F	F	F	F	F
Default	none	none	RMIN	0.0	0.0	0.0	0.0	10^{20}

Card 3	1	2	3	4	5	6	7	8
Variable	IFUNC	NID	**IMULTI**	**LCVX**	**LCVY**	**LCVZ**		
Type	I	I	**I**	**I**	**I**	**I**		
Default	0	0	**1**	**0**	**0**	**0**		

Include when IMULTI>1

Card 3.1	1	2	3	4	5	6	7	8
Variable	**R1**	**P1**	**R2**	**P2**	**R3**	**P3**	**R4**	**P4**
Type	**F**	**F**	**F**	**F**	**F**	**F**	**F**	**F**
Default	**0.0**	**0.0**	**0.0**	**0.0**	**0.0**	**0.0**	**0.0**	**0.0**

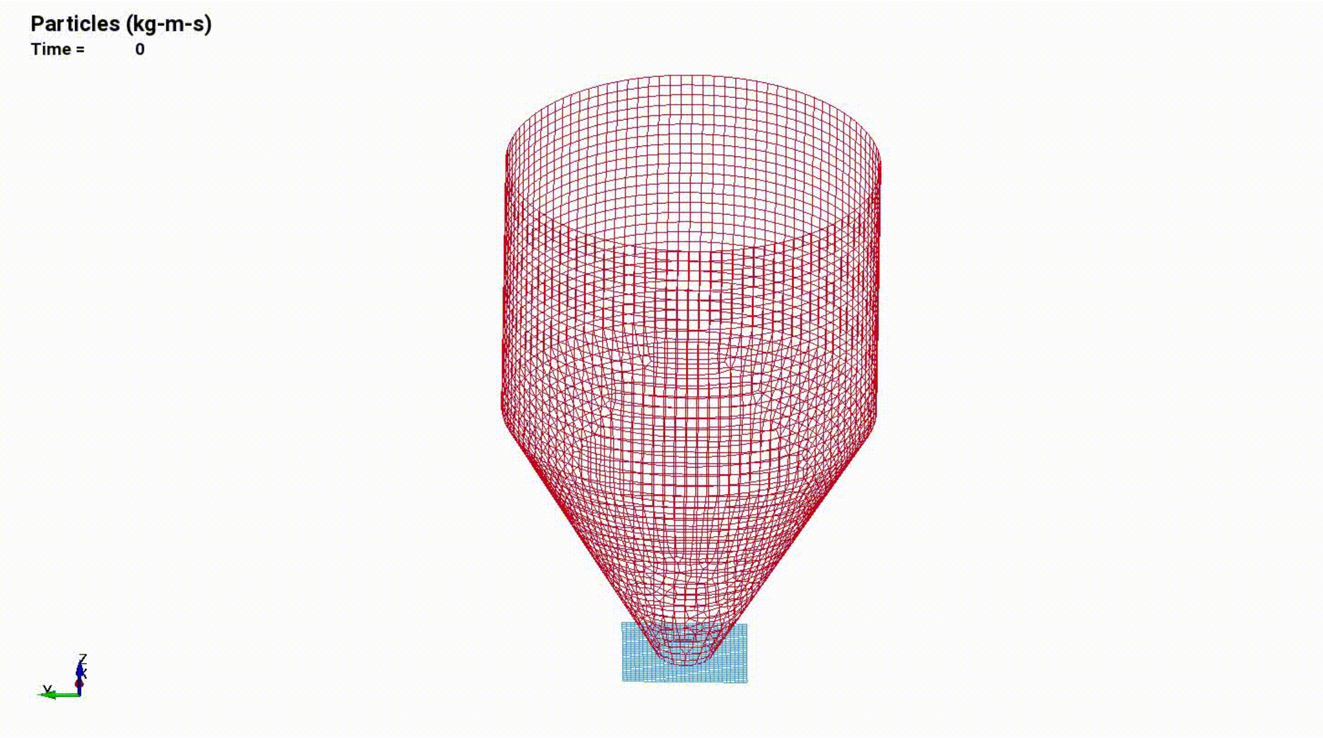
1. The new option IMULTI enables the capability of simultaneous injection of particles of different size with specified mass ratio.

2. The option of LCVX, LCVY and LCVZ allow that the particle injection rate can be varied through user defined curve instead of remaining as a constant.

Ansys

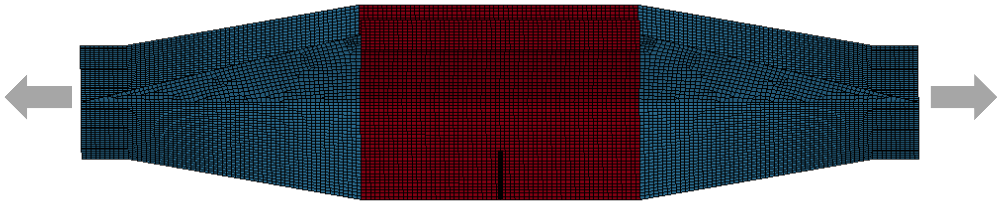
Example of DES injection with IMULTI=3

The injection consists of three different size DES particles with specified mass ratio - 22% 1.0mm particles, 28% 2.0mm particles and 50% 3.0mm particles

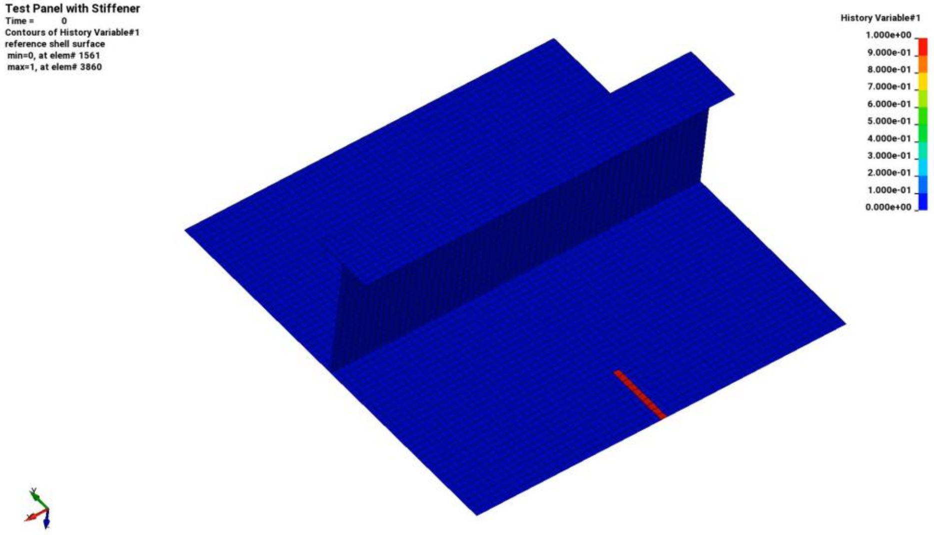


XFEM: Crack Propagation in Junction Structures

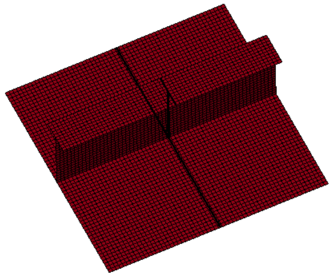
- Simulate crack propagation in T-junction and cross-junction shell structures
- SMP and MPP available



Stiffened Panel with Precrack



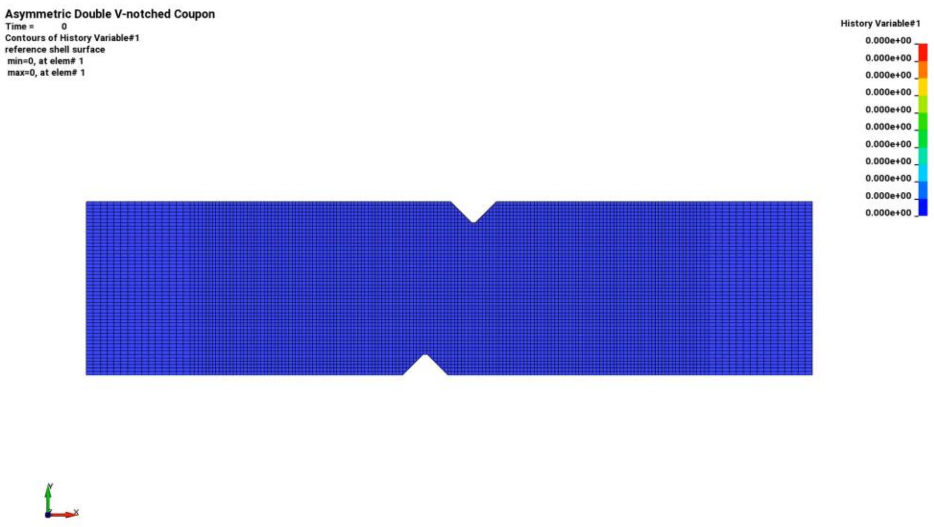
Crack propagation animation



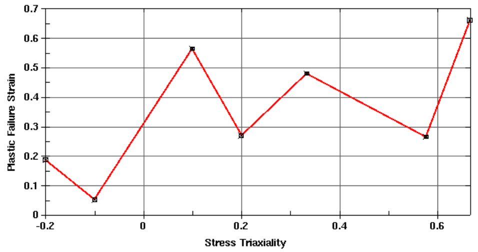
Crack pattern in initial configuration

XFEM: Stress Triaxiality Dependent Failure Strain

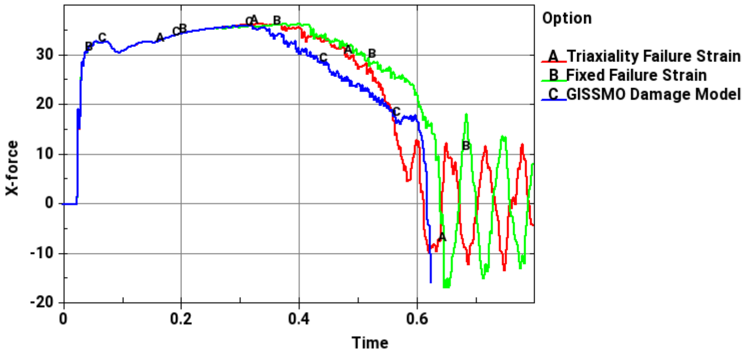
- Simulate failure in mixed modes and complex loading conditions
- Apply to standard material laws for ductile fracture
- *SECTION_SHELL_XFEM: FAILCR=-4, FS is the curve ID defining stress triaxiality dependent failure strain



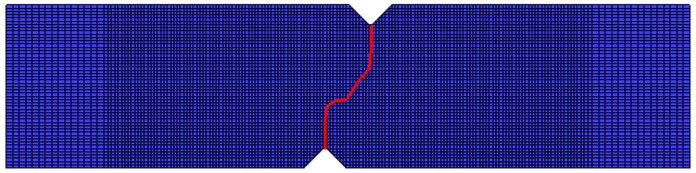
Crack propagation in asymmetric double V-notched coupon



Failure strain vs stress triaxiality



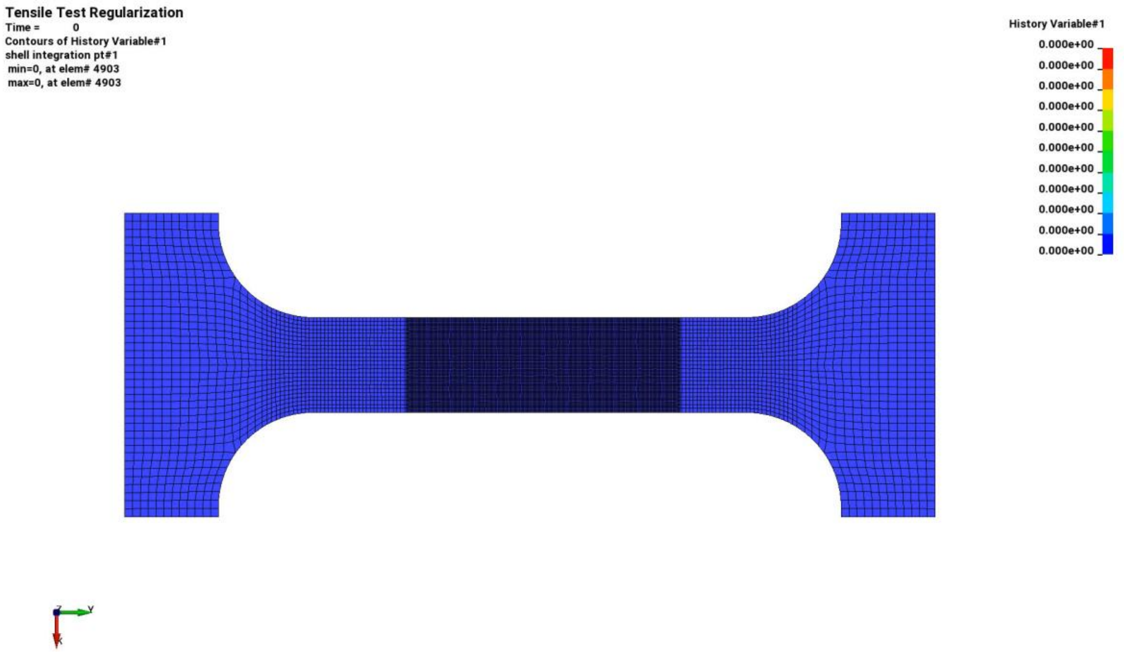
Comparison of section forces



Crack pattern in initial configuration

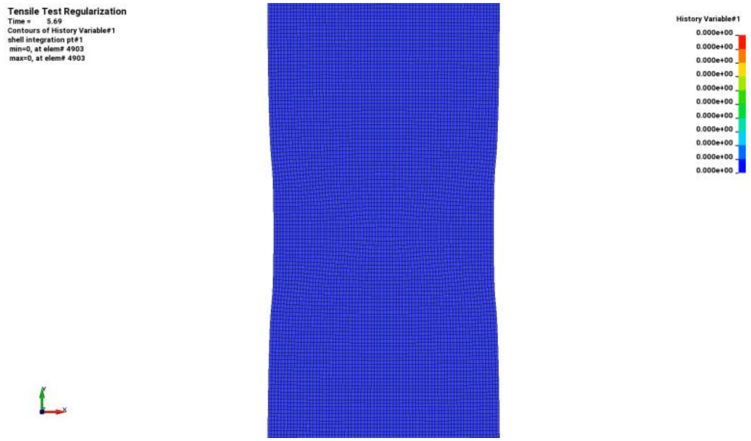
XFEM: Directional Center of EPS for Crack Propagation Direction

- Capture shear band failure
- *SECTION_SHELL_XFEM: PROPCR=3

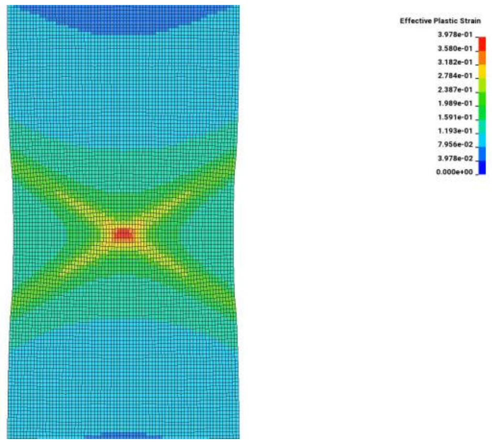


Crack propagation in tensile coupon
FAILCR=0 (GISSMO), PROPCR=33

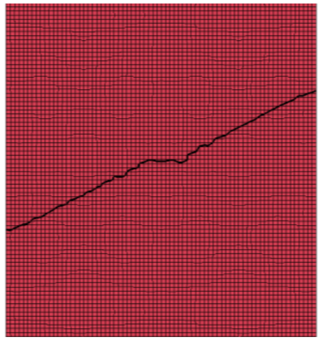
Zoomed Views

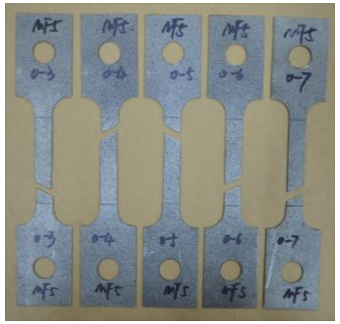


Failure Indicator



EPS

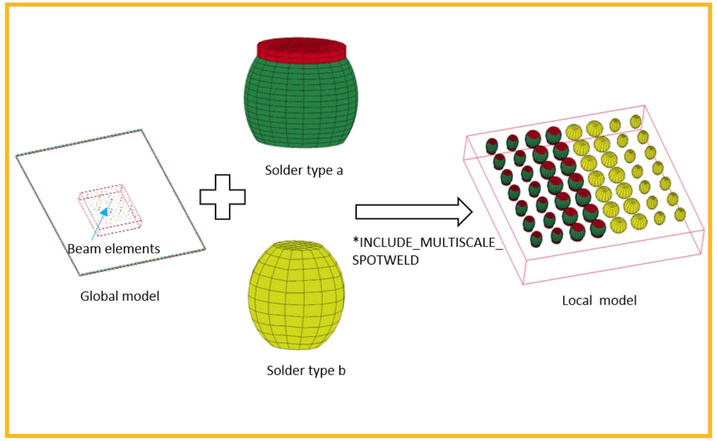




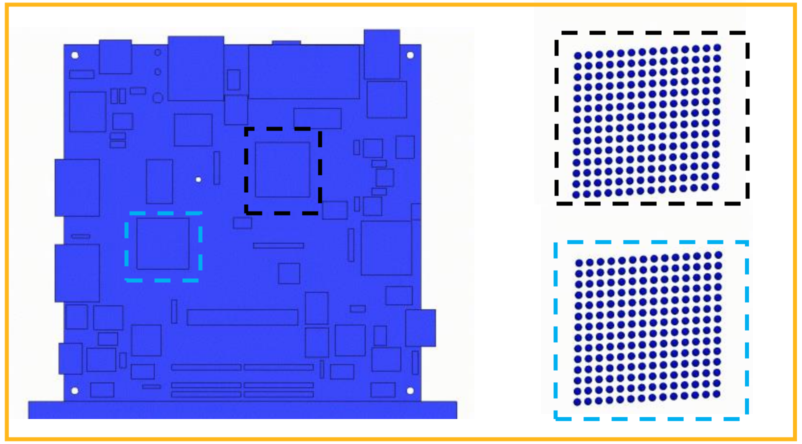
Crack pattern

Two-scale Co-simulation Approach in Electronics Applications

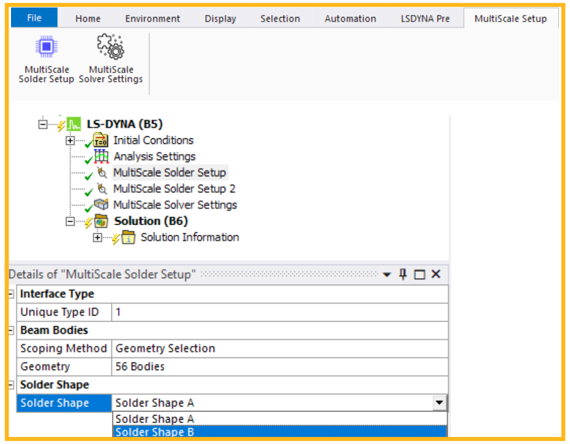
- New keywords: *INCLUDE_MULTISCALE, *DEFINE_MULTISCALE
  - The two-scale co-simulation couples the mesoscale model and the macroscale model using the non-matching discretization to co-simulate the structural response.
  - The beam element can be replaced by solid elements for the solder ball modeling.
  - This solder joint model will be modularized, duplicable, and numerically immersed in a meso-scale chip model automatically.
- ACT on Ansys Workbench (work with LST ACE team)



Automated process of replication solder joint



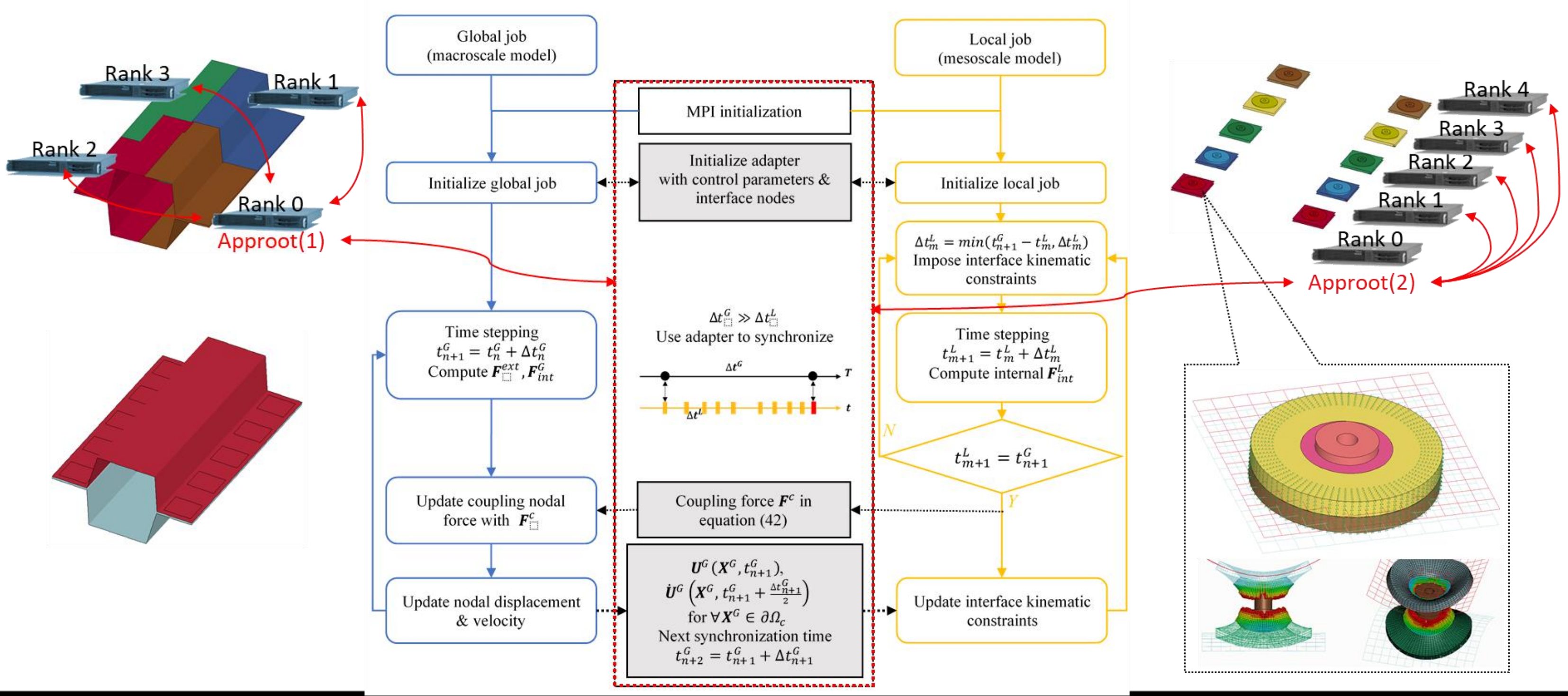
Trace mapped shell PCB with detailed solder joints model in drop test



ACT on Ansys Workbench

Two-scale Co-simulation with the Solid-in-shell Method

- New keywords: *INCLUDE_COSIM
  - The two-way coupling between macroscale shell structure and mesoscale joints
  - New subcycling with much smaller time step size in mesoscale and automatic synchronizing with macroscale
  - MPP-based data exchange at coupling interface across two scales in separated LS-DYNA jobs



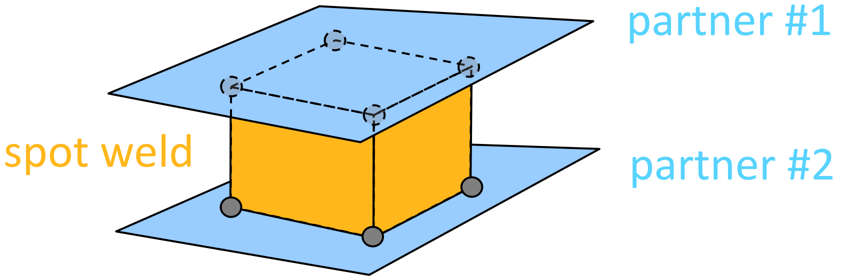
Contact Enhancements

Enhancements for connection methods

- Add velocity dependent hardening and failure to *MAT_068
  - Commonly used for 1-d connectors with large flexibility for all 6 d.o.f.s

- Add keyword option _UNIAXIAL for *MAT_SPOTWELD
  - Transverse stresses and transverse strains to be zero for solid spot welds
  - More stable than old approach with only transverse strains being zero (invoked with E<0.0)

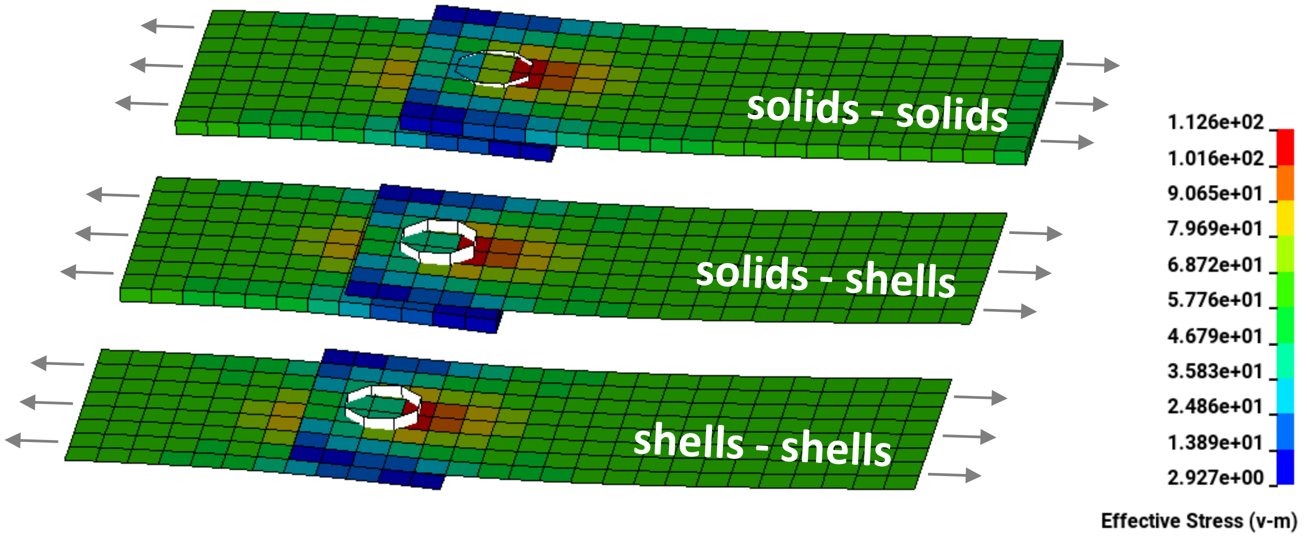
- Extend *DEFINE_CONNECTION_PROPERTIES
  - With PRUL=2/3, plastic yield and failure of connector is function of partner properties
  - Now also available for commonly used materials *MAT_036 and *MAT_251



Point wise connection of solid parts

- *CONSTRAINED_SPR2 now supports solid elements
  - Connection model for spotwelds, rivets, screws, etc.
  - SPR2 is an element-free interpolation method frequently used for shells-to-shells connections
  - Thick structures more and more often discretized with solid elements (hexa/tetra/pentahedral)

- Compares well with existing approach
  - Example: lap shear test



Improved orthotropic friction in segment-to-segment contact

- Input for both sides is expected
- But in R13, input for either side can be zeroed and the other side is used
- Orthotropic friction used between seatbelts and dummies
- Offset angles, hard to define for dummy segments, no longer needed
- Offset angles for seatbelt segments are sufficient

*CONTACT_OPTION1_{OPTION2}_... ***CONTACT**

ORTHO_FRICTION:

Additional cards for the ORTHO_FRICTION option:

*CONTACT_..._ORTHO_FRICTION_...

ORTHO_FRIC 1.

ORFR 1	1	2	3	4	5	6	7	8
Variable	FS1_S	FD1_S	DC1_S	VC1_S	LC1_S	OACS_S	LCFS	LCPS
Type	F	F	F	F	I	I	I	I
Default	0.	0.	0.	0.	0	0	0	0

ORTHO_FRIC 2.

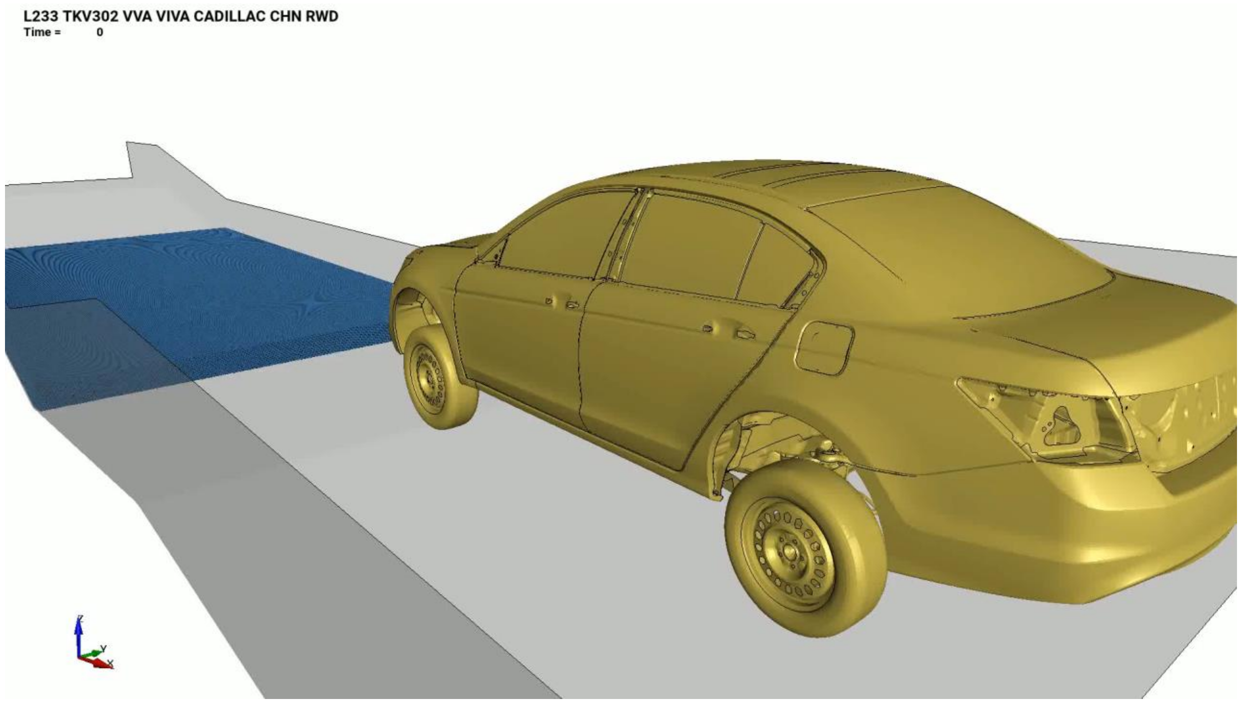
ORFR 2	1	2	3	4	5	6	7	8
Variable	FS2_S	FD2_S	DC2_S	VC2_S	LC2_S			
Type	F	F	F	F	I			
Default	0.	0.	0.	0.	0			

ORTHO_FRIC 3.

ORFR 3	1	2	3	4	5	6	7	8
Variable	FS1_M	FD1_M	DC1_M	VC1_M	LC1_M	OACS_M	LCFM	LCPM
Type	F	F	F	F	I	I	I	I
Default	0.	0.	0.	0.	0	0	0	0

Miscellaneous Mortar Contact Features

- 2D Mortar Contact
  - Supported in MPP
  - User defined friction
  - Frictional energy calculations
  - Interference using TDPEN
- 3D Mortar Contact
  - Linear stiffness (p vs d) activated by IMGAP.LT.0
  - Temperature dependent friction, through curve LCFST
  - Redecomposition/Full deck restart supported
  - User Tiebreak and Tied Weld for advanced lamination/delamination simulations



Redecomposition of mortar contact in wading simulation – used between tire and road to propel the vehicle

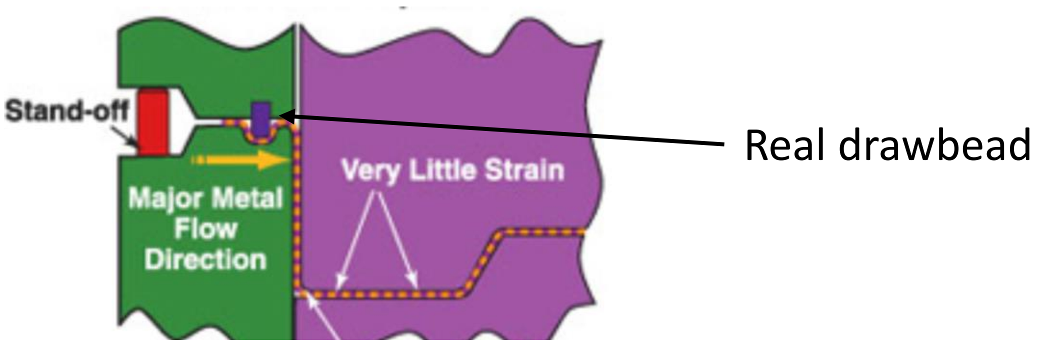
Stamping
Solver Enhancements

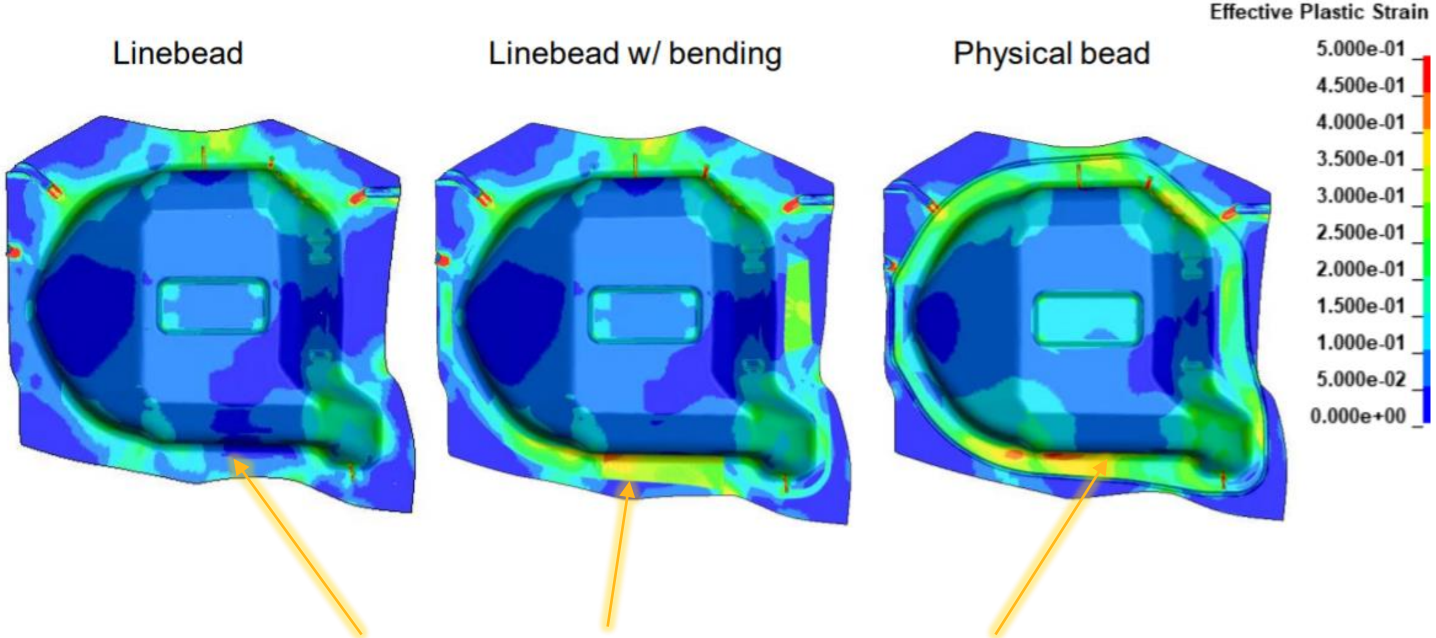
Stamping Solver Enhancements Overview

- Weakening effect for line bead
- Automatic contact normal check
- Add depth to trimming
- Multi-cuts or lancing during forming simulations
- Bug fix for In-core adaptivity
- Add differential hardening to MAT242
- Interference check: *control_forming_home_gap
- Speed up *include_stamp_parts
- New option to *CONTROL_FORMING_MAXID
- Improvement in trimming of sandwich parts
- improvement to *BOUNDARY_SPC_SYMMETRY_PLANE_SET

Weakening effect for line bead

- Drawbead is commonly used in deep drawing process to control material flow
- Drawbead has tight radius, which requires very fine mesh when the blank pass through the bead—end up with a big model
- Line bead or virtual bead is frequently used in simulation
- However, blank will not be stretched when it pass through line bead.
- Bead weakening effect is added to line
  - Allow blank to be stretched
  - Improve simulation accuracy
  - This is a joint project with Ford





With line bead, no obvious deformation in the bead region
With bead weakening effect, the material is stretched, which is more correct with real bead

Other new functions to LSDYNA

- Bug fix for In-core adaptivity
  - Many QAs have been done and bug found/fixed
- Add differential hardening to MAT242
  - Allow the yield surface to evolve during forming simulation
- Interference check: *control_forming_home_gap
  - Check interference between upper and lower tools, which is important for flanging simulation, where minor error in cam-steel orientation can cause interference
- Speed up *include_stamp_parts
  - Under BMW's request, we have speed up the speed of mapping of forming information to crash model
- New option to *CONTROL_FORMING_MAXID
  - Make sure no IDs conflict during multi-stage forming simulations

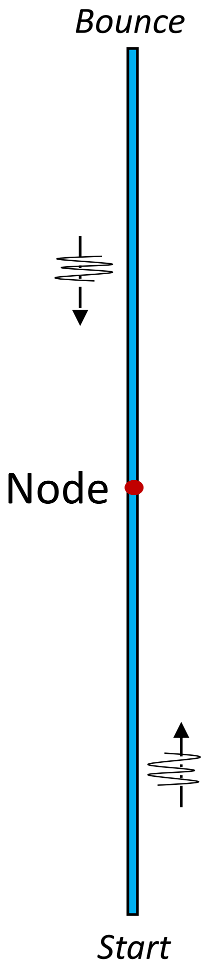
NVH and Fatigue Analysis

New features for NVH and fatigue analysis in R13

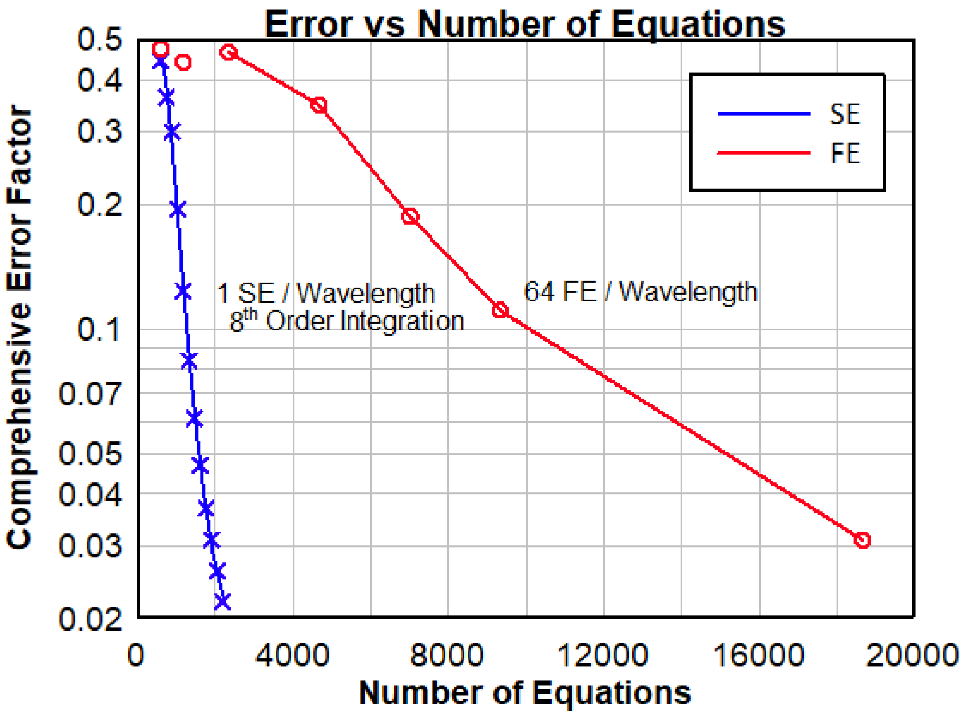
- Acoustic spectral element method
- SSD with coupled fluid-structure system
- New boundary / load conditions for acoustic analysis
- New acoustic material models
- Mode-based FEM acoustics
- SSD with multiple load cases
- Multi-directional response spectrum analysis
- Fatigue analysis based on modal dynamics
- Random vibration fatigue analysis based on IGA model
- D3MAX for stress and strain envelope

Acoustic spectral element method

- It is a sub-parametric FE
- The shape functions for the geometry are of a lower order than the interpolation functions for the element pressure
- The interpolation functions for the pressure employ Legendre polynomials of orders 2→15
- Element integration is with a Gauss-Lobatto-Legendre rule. Element dof are at those integration points.



Transient 1D Wave Propagation
(propagation of a 50KHz “chirp” in a 1m long column of air)

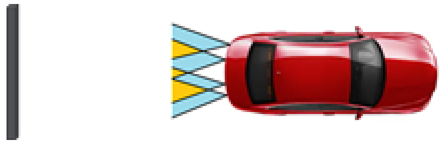


Ansys

Acoustic spectral element method – application 1

Acoustic SE is capable of high accuracy with manageable resource requirements, and so is well suited to high frequency and ultrasonic applications where the wavelengths are often short relative to the dimensions of interest:

- Ultrasonic sensors
  - Autonomous driving / parking
  - Fingerprint recognition



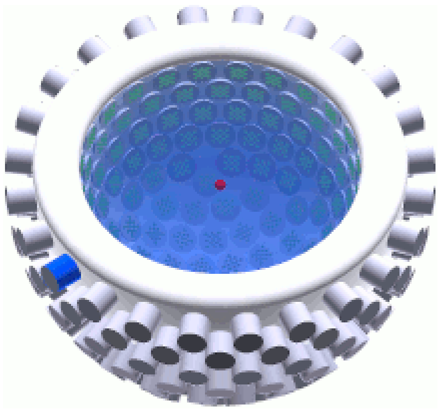
Acoustic spectral element method – application 2

- Medical imaging

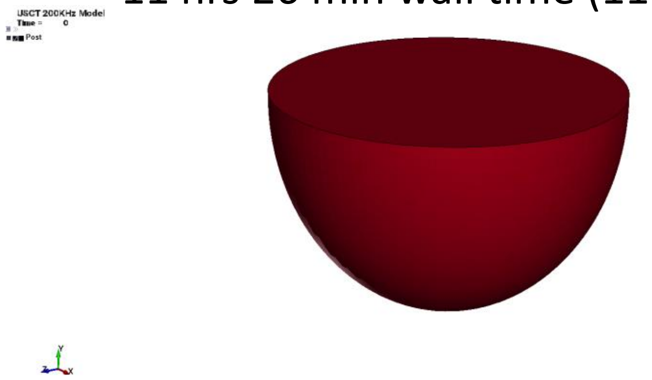
  USCT - Ultrasound Computer Tomography

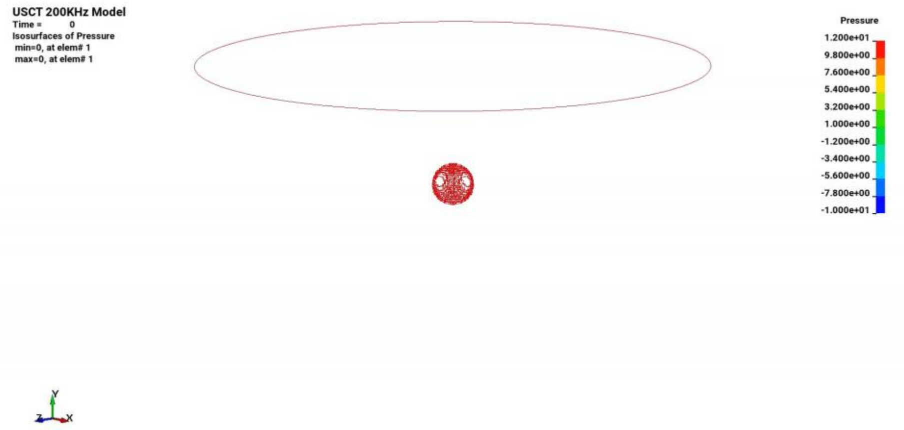
  - Tissue in water
  - Ultrasonic pulse from one transducer
  - Scattered signal measured on all others
  - Each transducer takes a turn

200 KHz pressure pulse



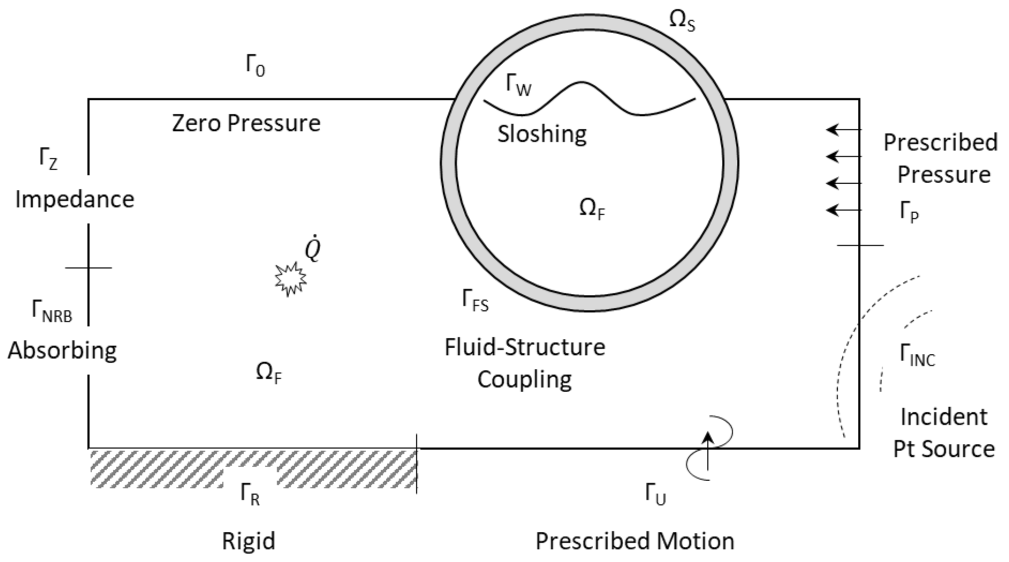
6,048,000 N=5 SE hexahedra
757,442,101 equations
Solution for 200 μs (57,967 steps)
11 hrs 26 min wall time (112 cores)





New boundary / load conditions for acoustic analysis

Feature	Keyword
Structural Coupling - Γ_{FS}	*boundary_acoustic_coupling_mismatch *boundary_acoustic_coupling_spectral
Weak Structural Coupling - Γ_{FS}	*interface_acoustic *boundary_acoustic_interface
Prescribed Boundary Motion - Γ_U	*boundary_acoustic_prescribed_motion
Prescribed Boundary Pressure - Γ_P	*boundary_acoustic_pressure *boundary_acoustic_pressure_spectral
Rigid Boundary - Γ_R	This is a natural condition
Impedance Boundary - Γ_Z	*boundary_acoustic_impedance *boundary_acoustic_complex *boundary_acoustic_mechanical
Absorbing Boundary - Γ_{NRB}	*boundary_acoustic_non_reflecting
Zero Pressure Boundary – Γ_0	*boundary_acoustic_free_surface
Linear Wave Boundary – Γ_W	*boundary_acoustic_free_surface
Internal Point Source – Q	*load_acoustic_source
Incident Wave Point Source – P_{inc}	*load_acoustic_source



New acoustic material models

*MAT_ACOUSTIC_COMPLEX

Complex, frequency-dependent density and bulk modulus

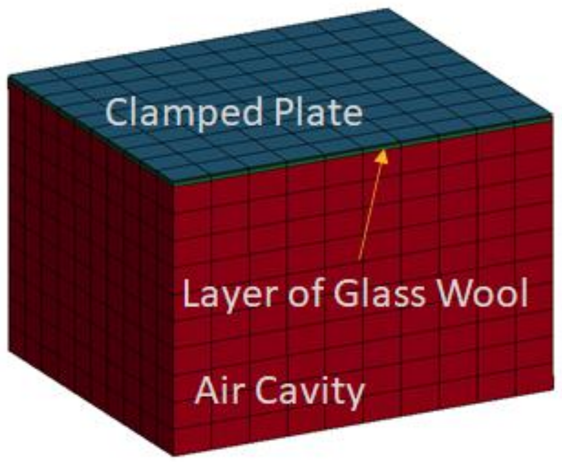
*MAT_ACOUSTIC_DAMP

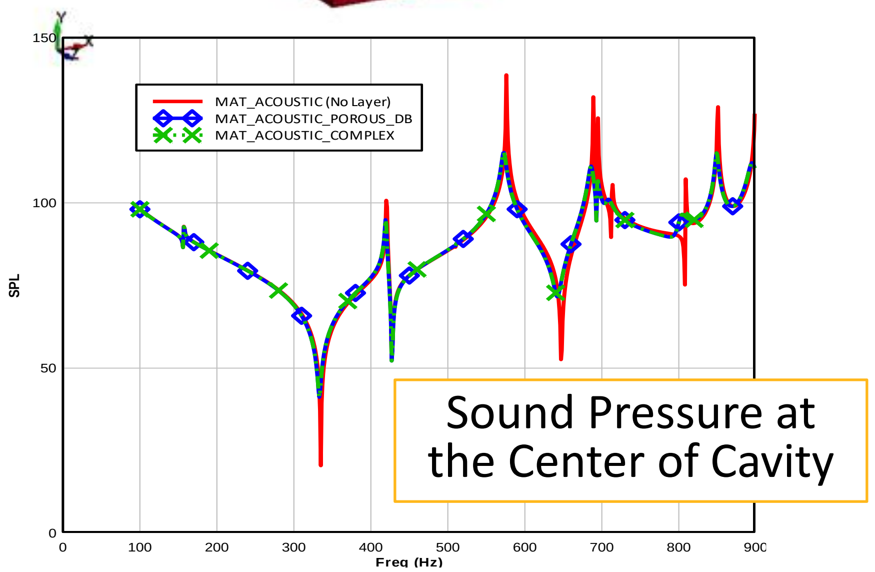
Fluid damping defined by volumetric drag coefficient for direct steady state vibration analysis, or viscous damping defined for explicit transient acoustic analysis

*MAT_ACOUSTIC_POROUS_DB

Regression model for porous acoustic media of Delany-Bazley and the coefficients of Allard-Champoux

J.-F. DEU FIGURE 3 WITH SOUND SOURCE, POROUS D-B LAYER





SSD with multiple load cases

*FREQUENCY_DOMAIN_SSD_SUBCASE

*DATABASE_FREQUENCY_..._SUBCASE

Benefits:

- Save extra keyword input reading
- Save extra initialization
- Save extra MPP decomposition
- Other savings

```
*FREQUENCY_DOMAIN_SSD_SUBCASE
$#   mdmin     mdmax     fnmin     fnmax    restmd    restdp    lcflag    relatv
         1       100        0.     2000.         0         0
$#   dampf     lcdam     lctyp    dmpmas    dmpstf
      0.01
$#                                                    nout     notyp      nova

$#  caseid     title                                                     nload
case1          The first loading case                                        2
$#     nid      ntyp       dof       vad       lc1       lc2       lc3       vid
```

subcase 1

```
       131         0         3         0       100       200
       180         0         3         0       101       201
$#  caseid     title                                                     nload
case2          The second loading case
$#     nid      ntyp       dof       vad       lc1       lc2       lc3       vid
```

subcase 2

```
       200         0         3         0       101       201
$#  caseid     title                                                     nload
case3          The third loading case
$#     nid      ntyp       dof       vad       lc1       lc2       lc3       vid
```

subcase 3

```
       258         0         3         0       102       202
*DATABASE_FREQUENCY_ASCII_NODOUT_SSD_SUBCASE
$#    fmin      fmax     nfreq    fspace    lcfreq
```

subcase 1
subcase 2
subcase 3

```
       10.      140.        14
       10.      140.       131
       10.      140.       261
*DATABASE_FREQUENCY_ASCII_ELOUT_SSD_SUBCASE
$#    fmin      fmax     nfreq    fspace    lcfreq
```

subcase 1
subcase 2
subcase 3

```
       10.      140.        14
       10.      140.       131
       10.      140.       261
*DATABASE_FREQUENCY_BINARY_D3SSD_SUBCASE
$#  binary
         1
$#    fmin      fmax     nfreq    fspace    lcfreq
```

subcase 1
subcase 2
subcase 3

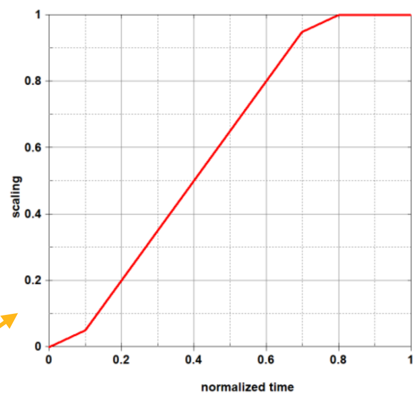
```
       10.      140.        14
       10.      140.        27
       10.      140.       131
```

Thermal

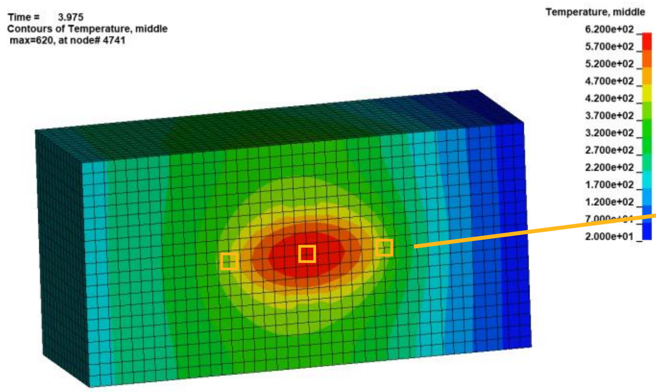
Modeling resistance spot welding

- New load curve option for temperature definition in the weld nugget
  - Use negative load curve to scale temperature between local temperature at birth time of the welding process and the given maximum temperature
  - Available for thermal-mechanical coupled simulation in *BOUNDARY_TEMPERATURE_RSW
  - Implemented as thermal load in a purely mechanical simulation in *LOAD_THERMAL_RSW
  - Necessary to simulate process for non-uniformly preheated parts

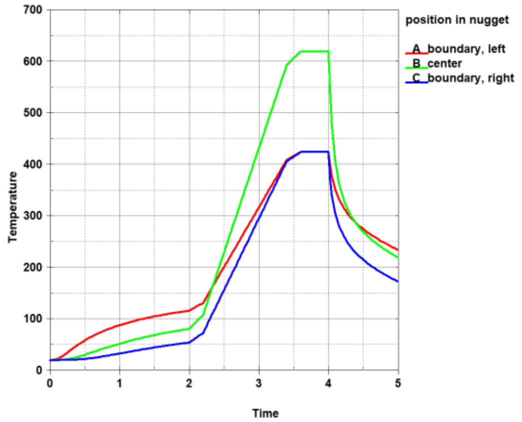
- Example process
  - Initial temp. T=20°C
  - Heating with T=200°C from the left during the whole process
  - Spot Welding
    - Birth time t=2s
    - Deactivated at t=4s
    - T_{ctr}=620°C, T_{bnd}=420°C



Scaling load curve



Temperature distribution



Temperature evolution for marked nodes in the nugget

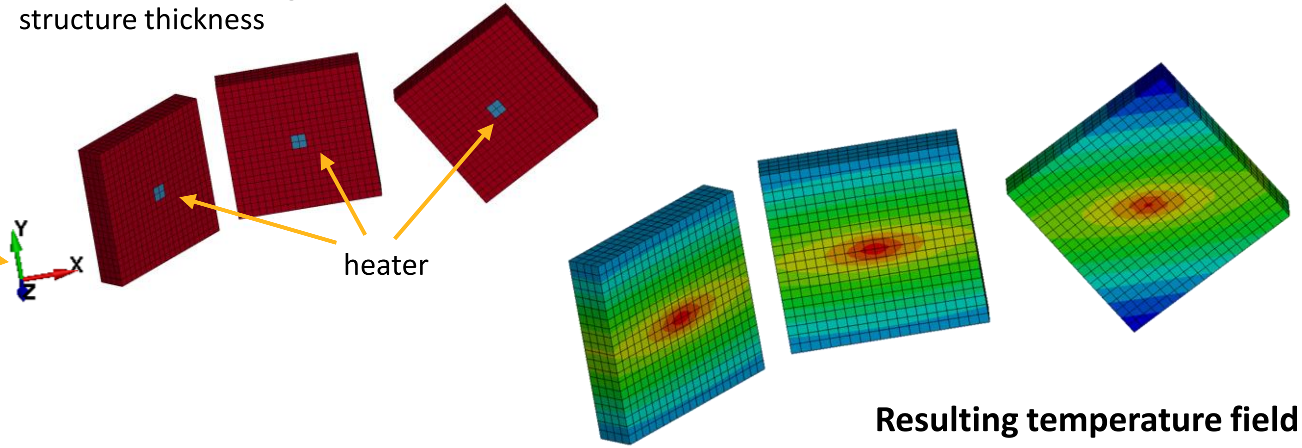
Anisotropic thermal materials

- Consolidation of anisotropy option
  - Added anisotropy option (AOPT=3) to define a material coordinate system with first direction being orthogonal to the element normal and to a given vector
  - All anisotropy options (AOPT between 1 and 4) now available for all materials with anisotropic thermal conductivity
  - Affects materials *MAT_T02, *MAT_T04, *MAT_T08,*MAT_T11-15, *MAT_T17

```
*MAT_THERMAL_ORTHOTROPIC
$#     tmid       tro     tgrlc    tgmult      aopt
          2       1.0         0       0.0         3
$#       hc        k1        k2        k3
        1.0      10.0       1.0       1.0
$#       xp        yp        zp        a1        a2        a3
        0.0       0.0       0.0       0.0       0.0       0.0
$#       d1        d2        d3
        0.0       1.0       0.0
```

Setup

Element normals aligned with
structure thickness



Resulting temperature field

*ALE_STRUCTURED_FSI

*ALE_STRUCTURED_FSI							
SLAVE	MASTER	SSTYP	MSTYP				MCOUP
		PFAC			IFLIP		

SLAVE: Structure ID
MASTER: 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)

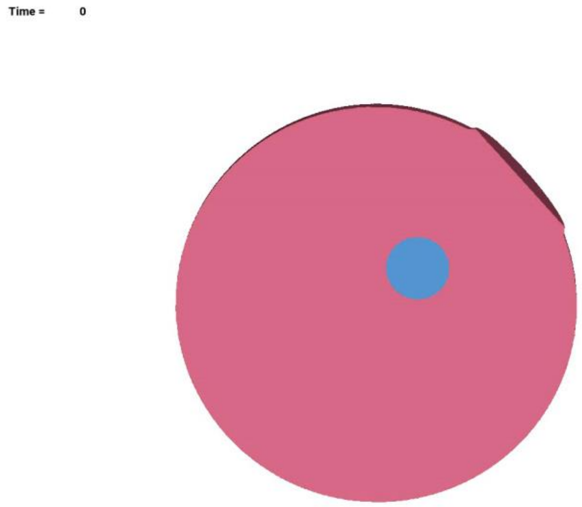
Automate Everything:

- All parameters are internally calculated, automatically chosen.
- PFAC: Penalty stiffness is the only one users need to pick.
- Automated Leakage control
- Eroding option always on
- Edges automatically generated and on

Better Performance:

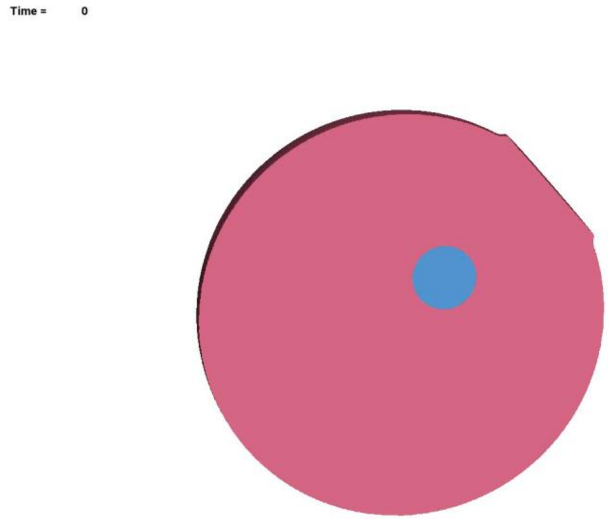
- Enhanced Leakage Control
- Stable, Faster
- MPP efficiency greatly improved; Nonblocking, Groupable MPP

*ALE_STRUCTURED_FSI: Leakage Prevention



***ALE_STRUCTURED_FSI:** came with a much better leakage prevention

- Much more accurate at capturing fluid interface
- Not only process structure segments, but also edges, nodes.
- Enhanced leakage control algorithm
  - Better estimation of spring stretch
  - Innovative algorithm to achieve energy balance
  - New algorithm is order-free → more stable
- New MPP implementation (order-free calculation) ➔ more stable



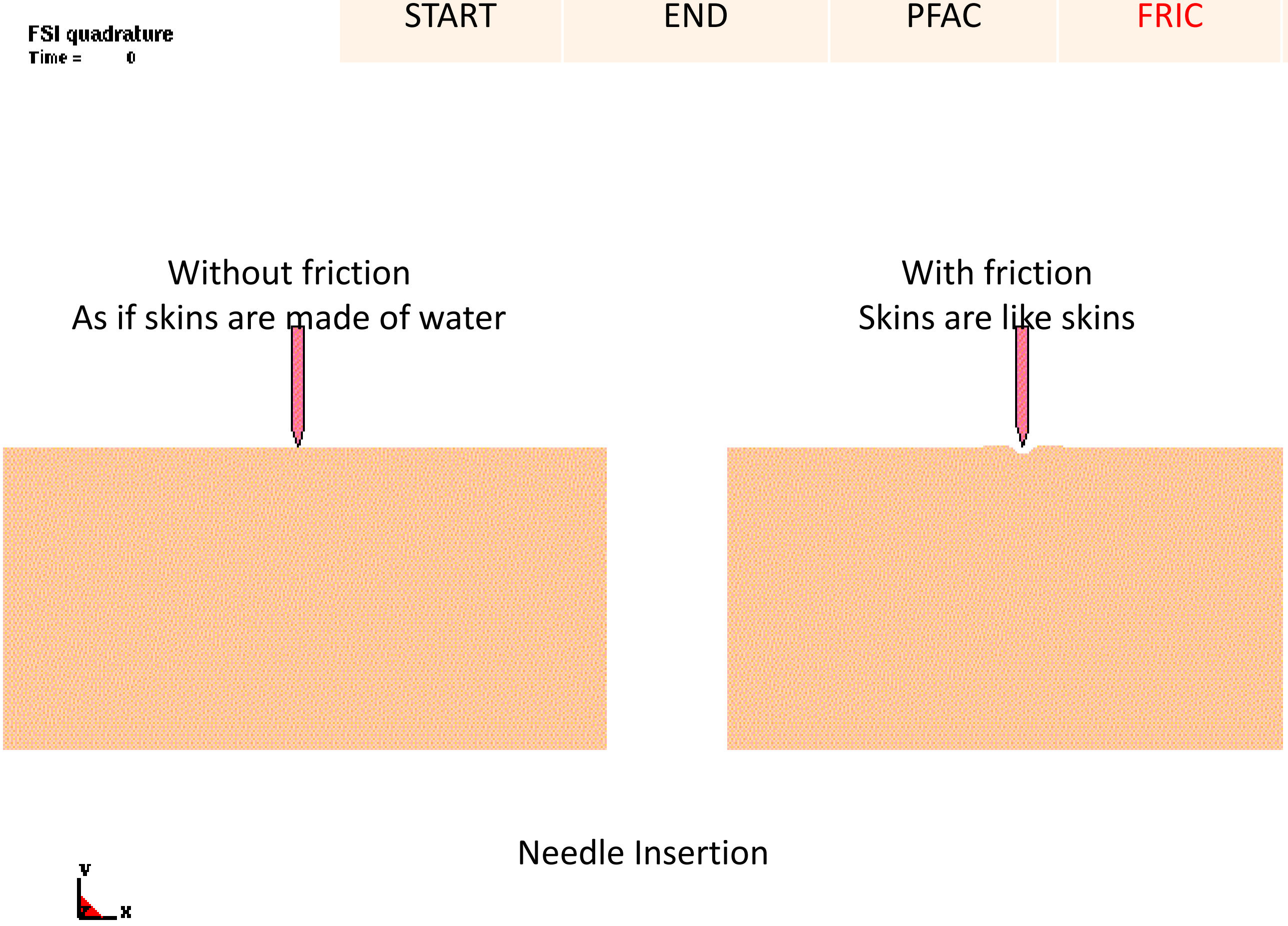
However, at times, still, leakages do occur, mostly

- Highly pressurized + Light material (gas, air) → Blast, detonation
- A little unbalanced penalty spring forces would cause high pressure gradient at places → Air always manages to find the low-pressure place and goes there.

A recent (Nov 2020) algorithm enhancement → big improvement

*ALE_STRUCTURED_FSI: Variable Friction in FSI

*ALE_STRUCTURED_MESH_FSI							
SLAVE	MASTER	SSTYP	MSTYP				MCOUP
START	END	PFAC	FRIC			FLIP	



Needle Insertion

FRIC:
= “0.1” (constant friction coeff)
= “-N” (variable friction, N table ID)

Table:
Coupling pressure versus load curve

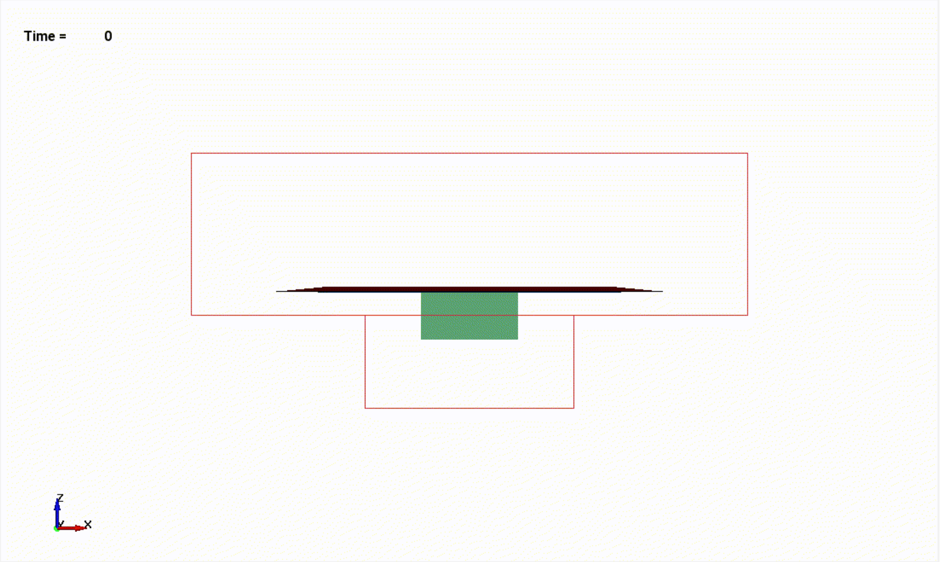
Load Curve:
Relative velocity versus fric coeff

```
*DEFINE_TABLE
     2
          0.00          3
        1.00e2          4
        1.00e3          5
        1.00e4          6
*DEFINE_CURVE
     3
         0.000        0.600
         1.0e9        0.600
*DEFINE_CURVE
     4
         0.000        0.500
         1.0e9        0.500
*DEFINE_CURVE
     5
         0.000        0.300
         1.0e9        0.300
*DEFINE_CURVE
     6
         0.000        0.100
         1.0e9        0.100
```

*ALE_STRUCTURED_MESH_MOTION: COVER_LAG

*ALE_STRUCTURED_MESH_MOTION					
MSHID	OPTION	SID	SIDTYPE	NODCEN	FRCPAD
1	COVER_LAG	101	0		

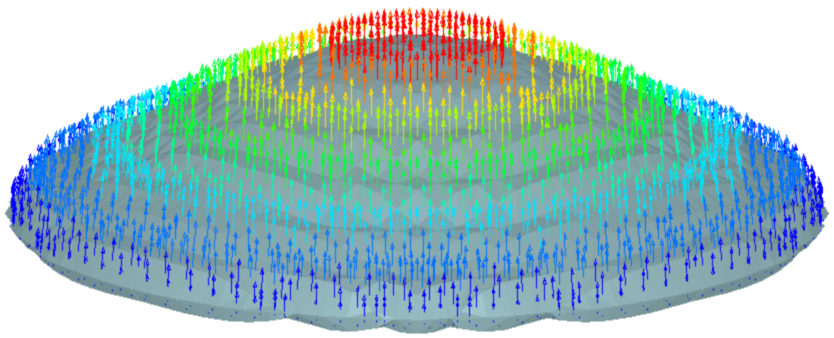
COVER_LAG: : to make the mesh follow the motion of a Lagrangian structure and expand/contract so that the Lagrangian structure is fully covered in the S-ALE mesh. It is most useful to model airbag deployment.



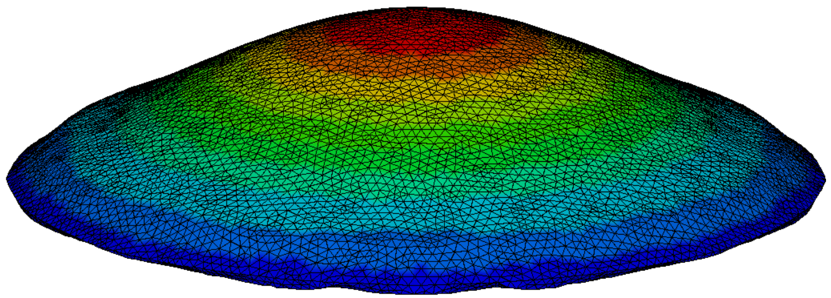
https://ftp.lstc.com/anonymous/outgoing/hao/sale/models/meshmotion/airbag1/

ALE: Map ALE hexahedra to Lagrangian tetrahedra

*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.



ALE material after the 1st run



Tetrahedral mesh at the beginning of the 2nd run

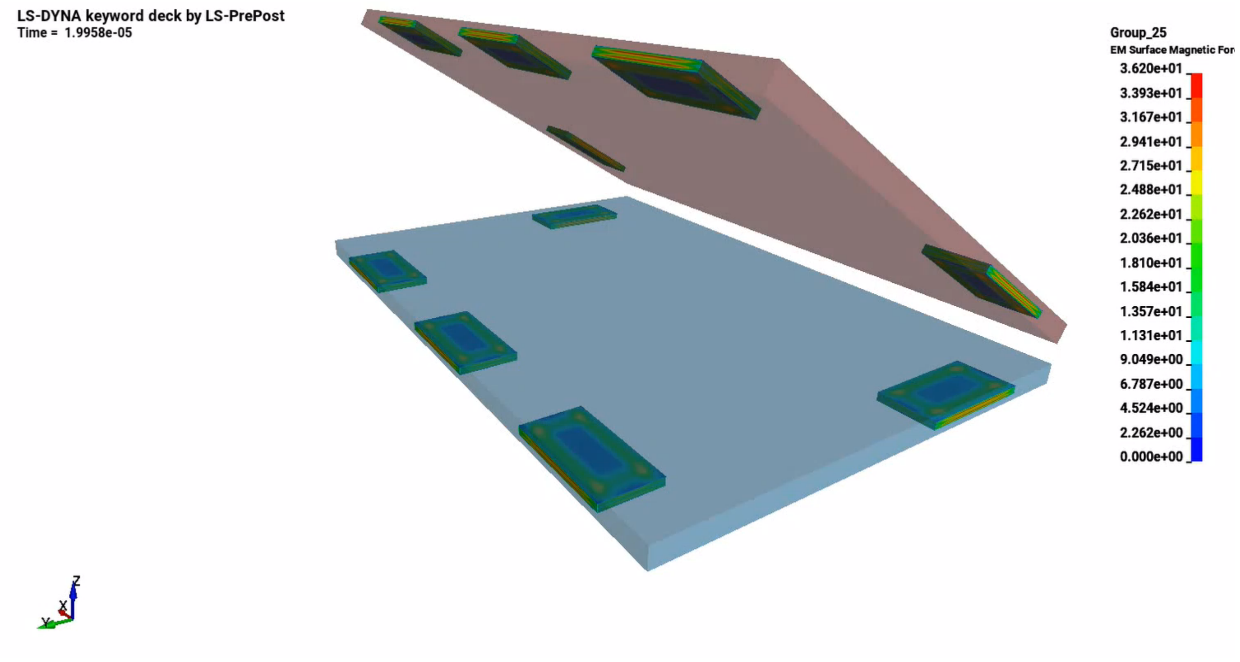
Electro-Magnetic Solver

Electro-Magnetic Solver New Enhancements

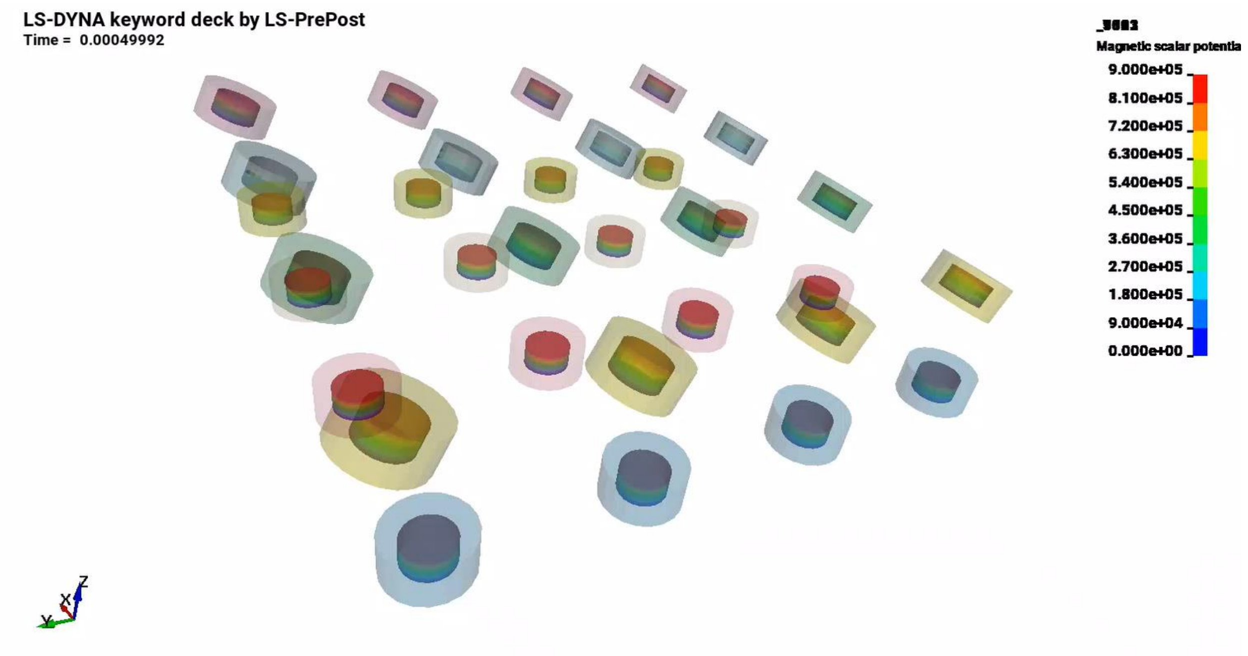
Magnets	• Addition of magnetic material capabilities (B-H curve) • Addition of capabilities to simulate permanent magnets • Addition of capabilities to calculate forces coming from magnets.
Inductive heating	• Added support of monolithic solver for higher timesteps. • Added support of 2D axisymmetric solver. • Added support of source circuits (stranded circuits). • Permeability of material can be function of temperature.
Battery modelling	• Erosion of elements can be combined with batmac feature for nail penetration. • ECM (Randles model) can be replaced by user defined model (chemistry model) • NREL's 4 equations model for thermal runaway has been implemented.
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.
Others	• Introduction of mortar contact to handle FEM current transfer in contact applications. • Added coupling of EM solver with explicit thermal solver. • User defined boundary conditions and laws defining resistivity can be implemented.

Magnet latching

- Thanks to the EM's solver robust FEM/BEM method, no air mesh is needed which means all dofs can be left free for magnet snapping and latching applications.



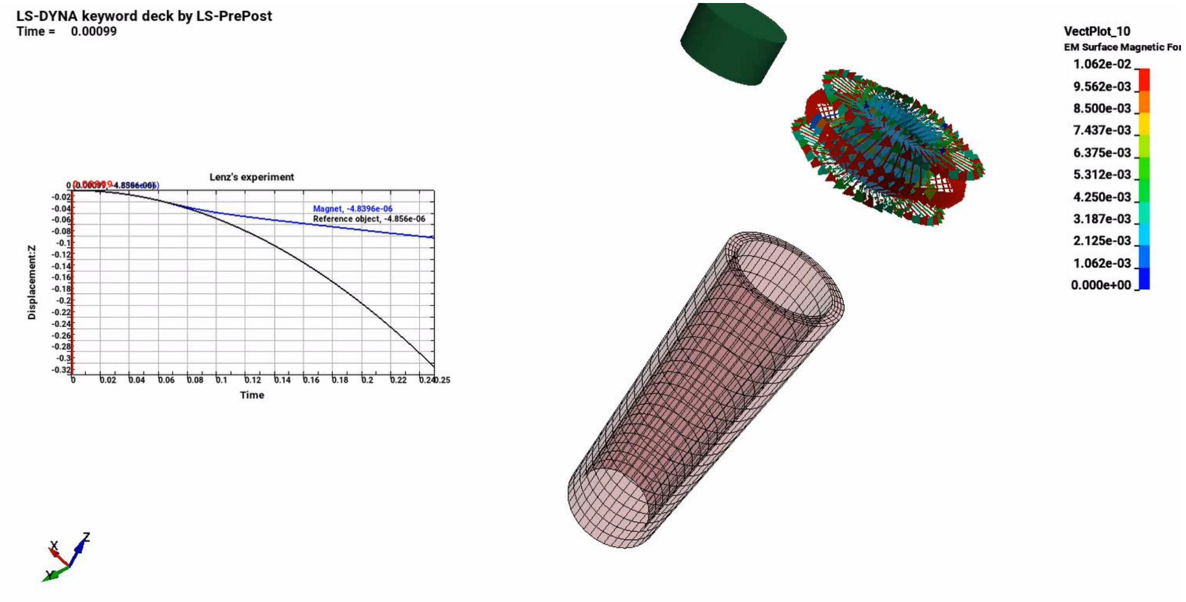
Magnet latching on electronic device



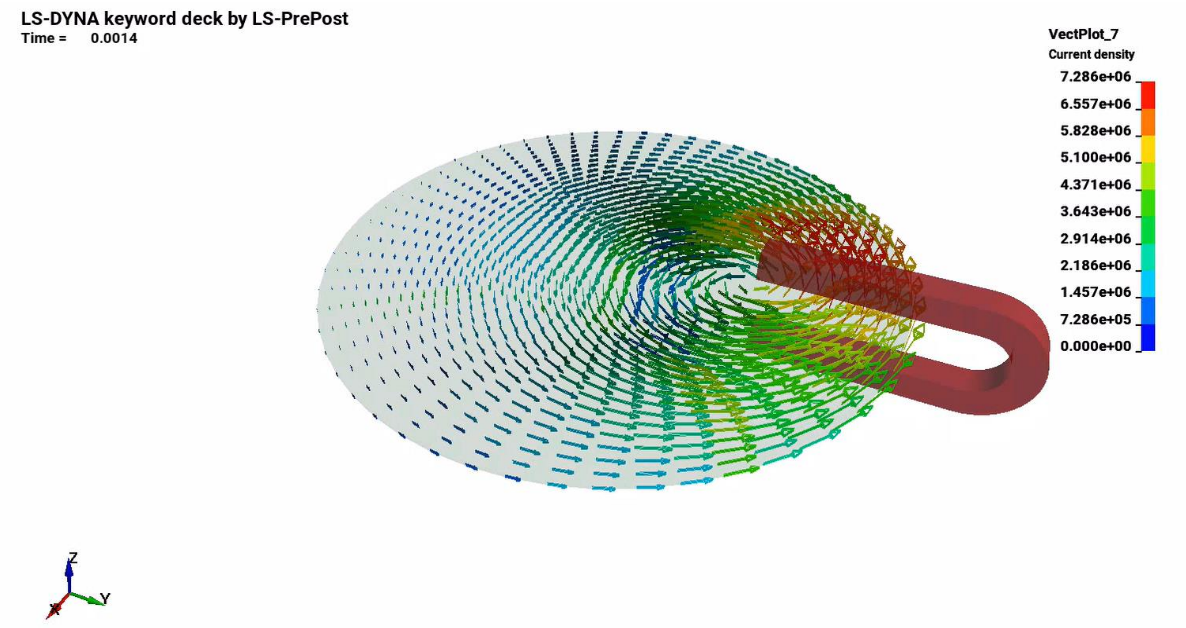
32 magnets snapping

Electromagnetic brakes and motors

- The addition of magnets in LS-DYNA allows the simulation of electromagnetic brakes or motors. Coupling with the structural solver or the thermal solver to consider motion or Joule losses is straightforward.



Lenz experiment for eddy current brakes

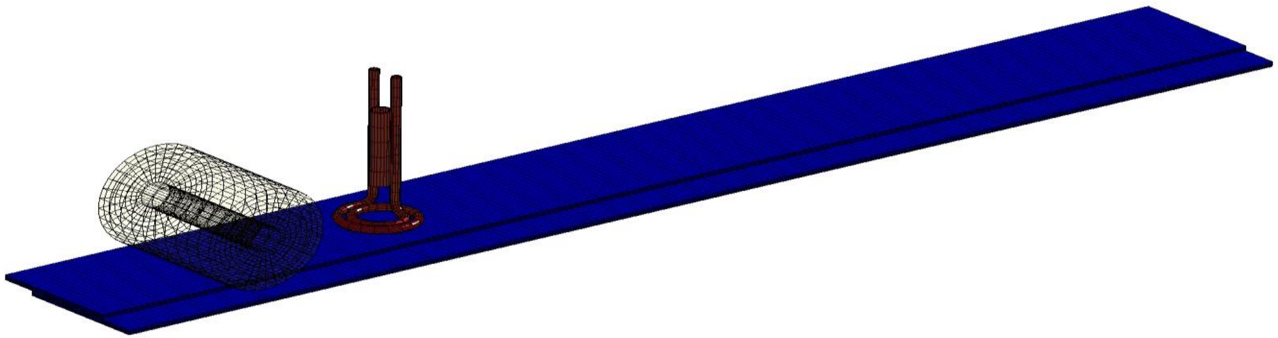


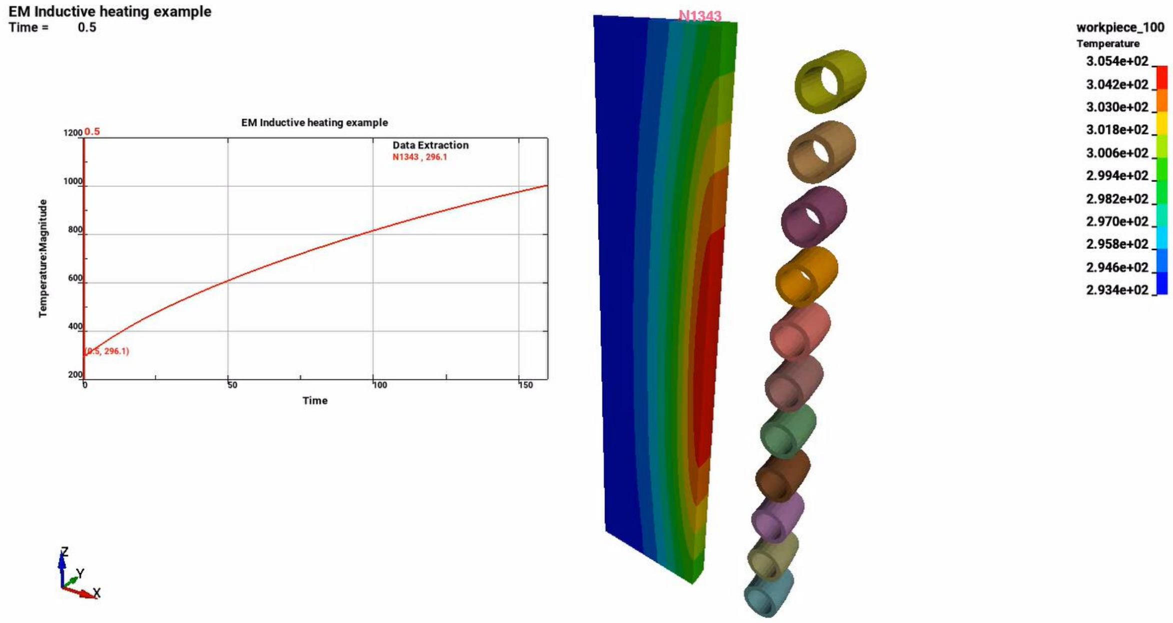
Arago's disk experiment for electromagnetic motor

Inductive heating

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

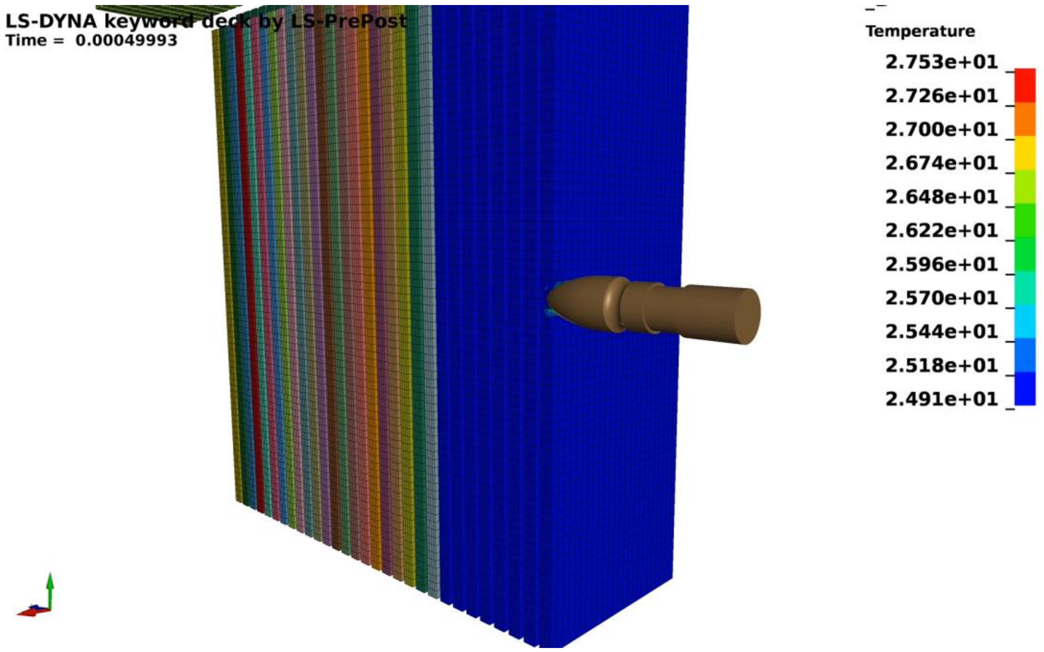
- Support of features such as 2D axisymmetric solver, or stranded coils to further expand the Inductive heating solver’s capabilities.





Battery improvements

- Extension of Equivalent circuit model (ECM) following release of EM solver battery module in R12 : support of erosion for nail penetration problems and addition of user defined models.
- Addition of thermal runaway models in the thermal solver (NREL's four equation model).



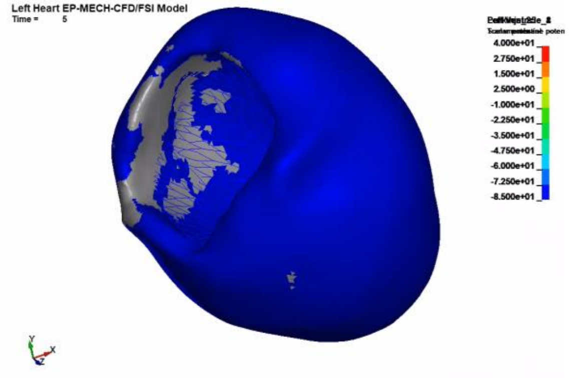
Nail Penetration case using batmac model

- NREL's four-equation model
  - *T. D. Hatchard, D. D. MacNeil, A. Basu, and J. R. Dahn, "Thermal Model of Cylindrical and Prismatic Lithium-Ion Cells", J. of the Electrochemical Society, 2001.*
  - *Gi-Heon Kim, Ahmad Pesaran, Robert Spotnitz, "A Three-dimensional thermal abuse model for lithium-ion cells", J. of Power Resources, 2007.*

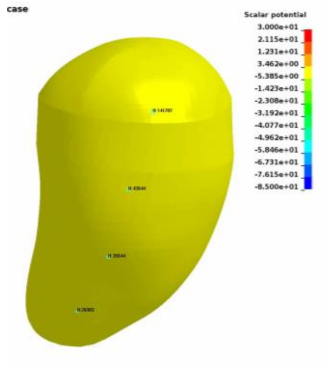
Introduction of Electrophysiology (EP) Models

A Path towards patient specific heart modelling :

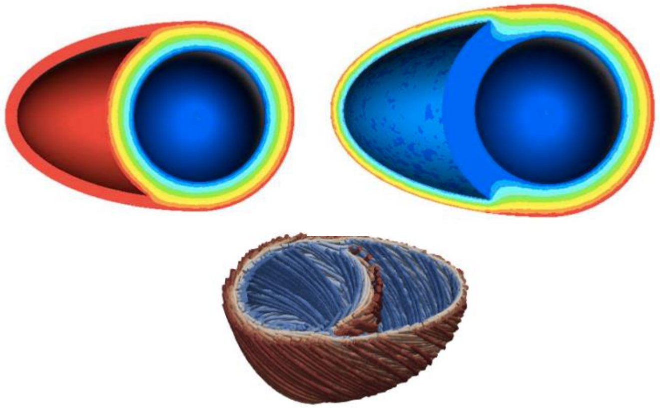
- EP models : Monodomain, Bidomain, bath + coupling between the 3 possible
- Ionic cell models: FitzHugh-Nagumo, Fenton-Karma, ten-Tussher, user defined, soon Tomek
- Current and voltage stimuli
- Automatic generation of Purkinje network which can be coupled to mono/bi domain models
- Automatic generation of fiber orientations, used by EP and mechanics
- Coupling EP-mechanics through Ca ion concentration or Transmembrane potential



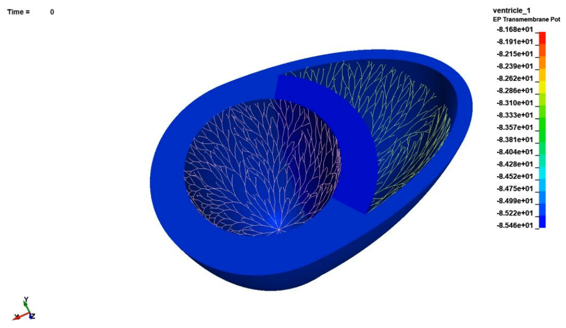
Grey zones



Arrhythmia and spiral waves



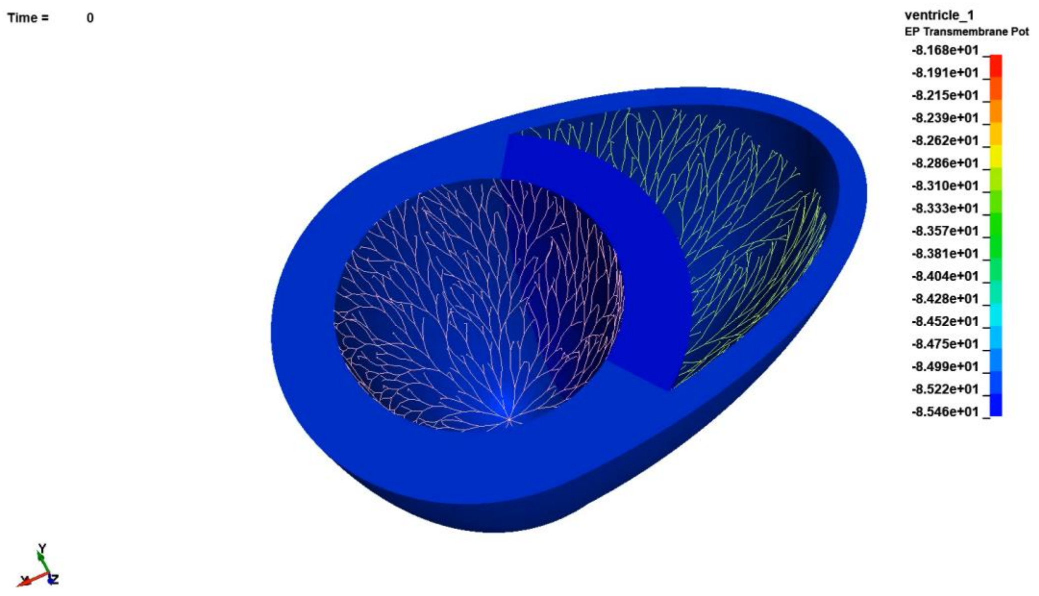
Fiber Orientation

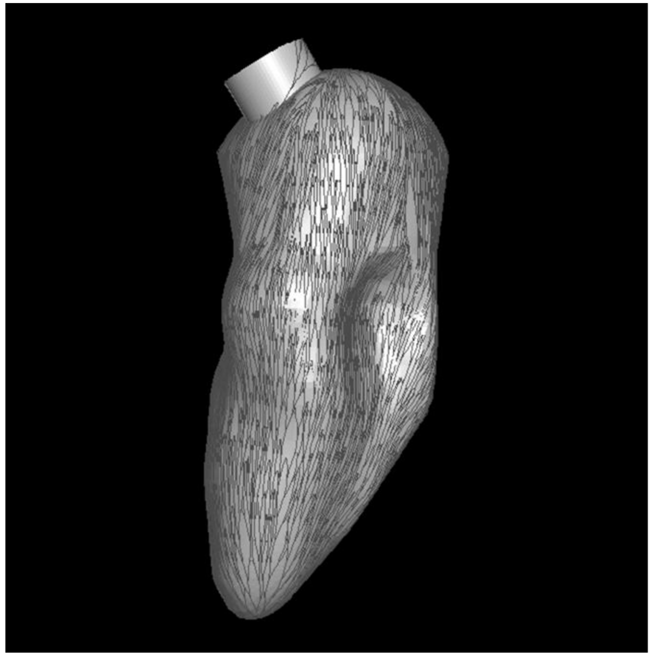


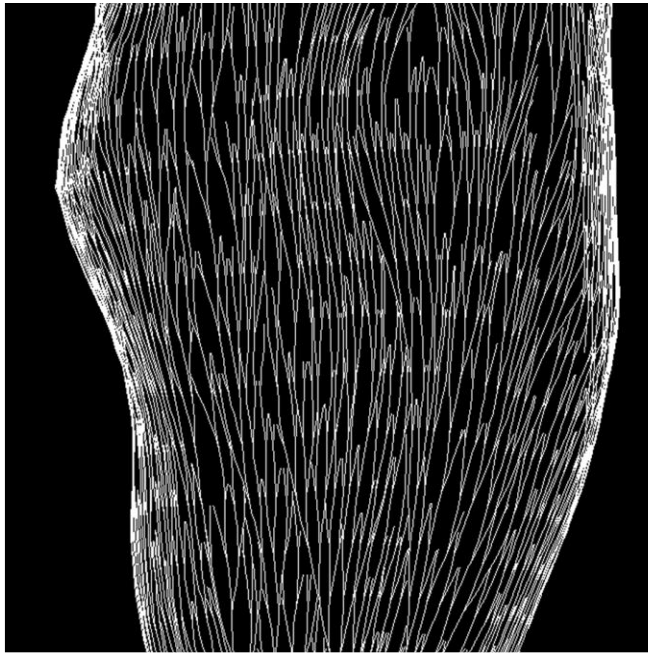
Purkinje network on endocardium

EP: Monodomain + Purkinje network

- Automatic build of Purkinje network on a surface (for example inner surface of the ventricles)
- Purkinje network coupled with monodomain.
- Fast propagation in the network compared to the walls => very different propagation.
- Important for patient specific studies and to get closer to realistic heart model.







ICFD Incompressible flow solver

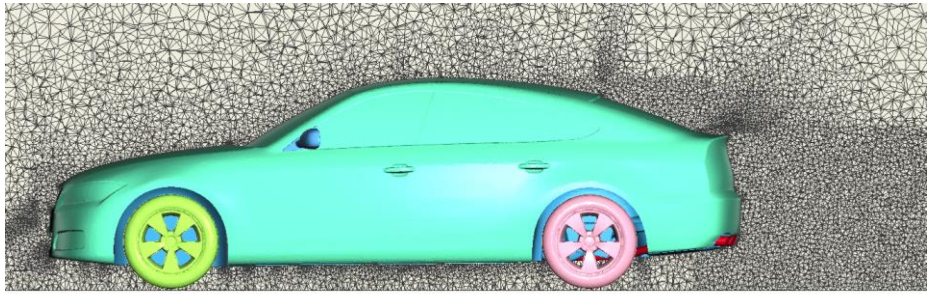
ICFD Solver Enhancements

Meshing	• A new version of the automatic volume mesher is released. • Added a new strategy for mesh motion in FSI models aimed at rotating problems.
Free Surface	• Level set as a function of height and multiple spherical shapes. • Wave generation: one and two parameter Pierson-Moskowitz spectrum. • New boundary condition Navier. • General improvements for FSI cases.
Fluid Structure Interaction	• Solid surfaces can "slide" over fluid surfaces. • Weak FSI with implicit mechanics (no sub-stepping). • Prevent flow through non-manifold surfaces intersections in embedded shells. • Output fluid loads to run a mechanical model only solution using CFD results. • Backflow stabilization for pressure boundary conditions.
Multi-Species	• One species transport model.
Others	• Initialization of a general CFD/FSI model using a steady state solution. • Reduced assembly of FEM matrices to save storage. • Prescribed local roughness height. • Thermal time step can differ from the fluid time step.

Meshing improvements

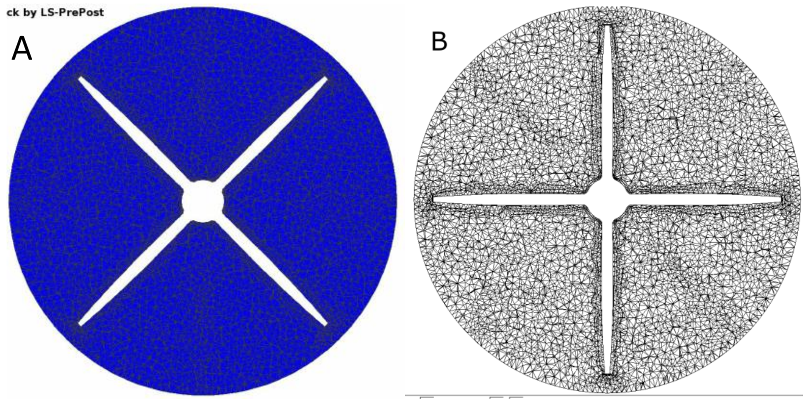
- See *ICFD_CONTROL_MESH. The new library version provides improve element quality, robustness and performance. It is not activated by default to allow consistency with previous version.

Approx. 20% faster mesh generation



- See ICFD_CONTROL_MESH_MOVE : New strategy for mesh motion in FSI models aimed at rotating (sliding) problems

A) old mesh motion, B) new option that preserves element quality.

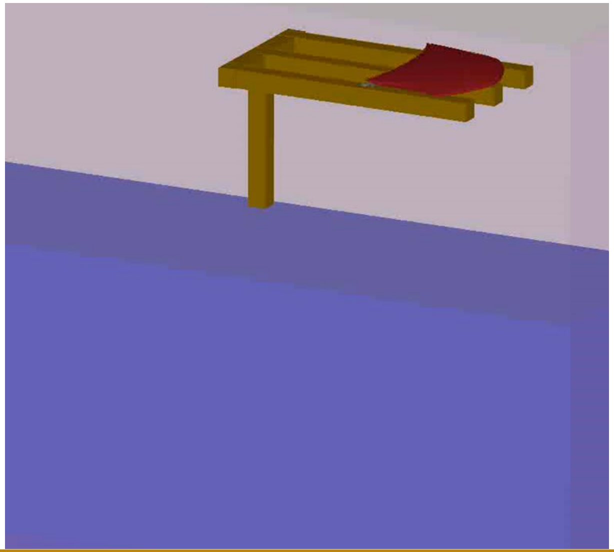


Ansys

Free Surface

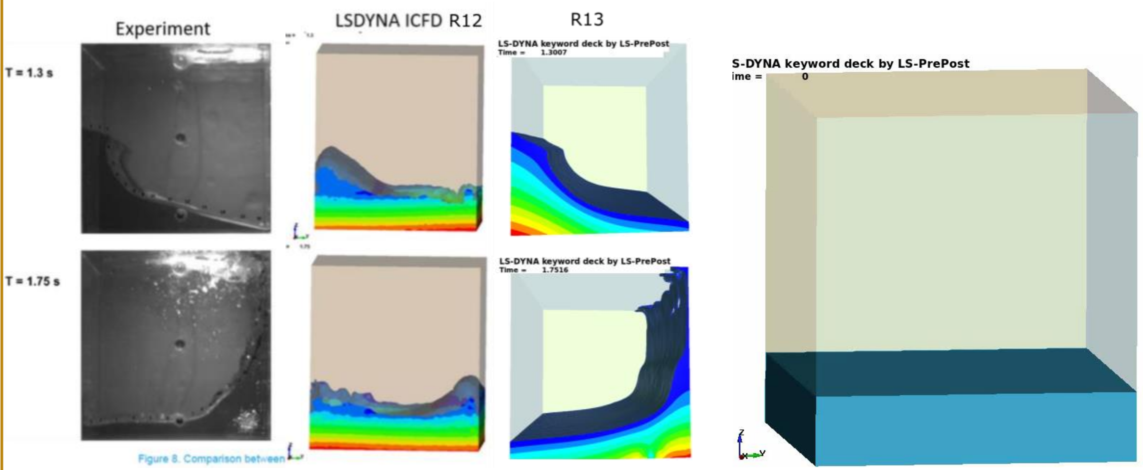
- Rodip : Improvement in robustness for thin structures (hood) interacting with free surface.

Vehicle hood submerged in paint. FSI simulation.



- Sloshing/Dam breaking : Improvements in free surface behavior and shape for high fidelity solutions.

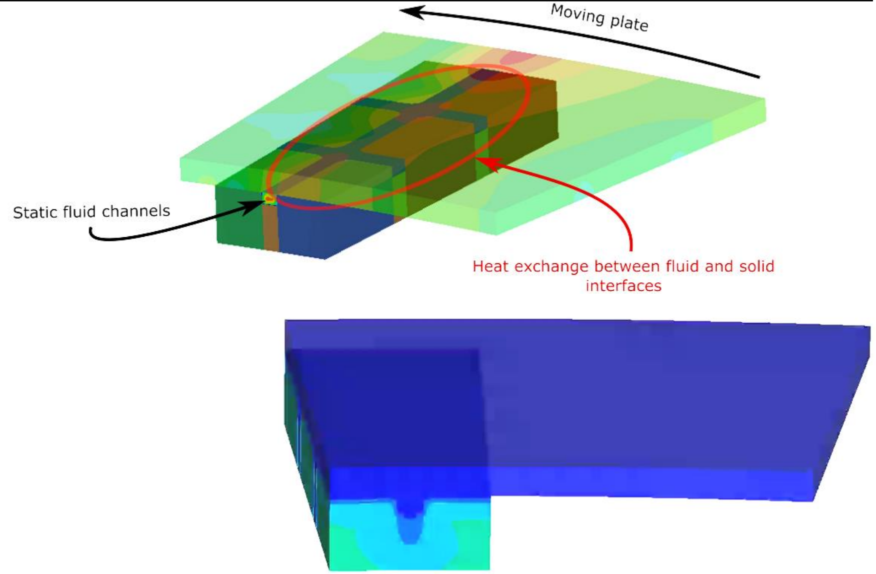
Sloshing simulation. Experiment vs R12 vs R13.



Fluid Structure Interaction

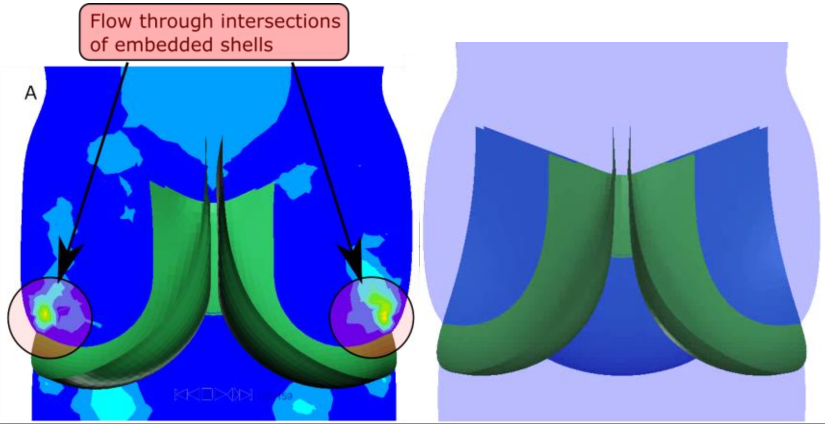
- See ICFD_BOUNDARY_FSI_FIXED. Clutch problem. Hot metal plate is allowed to slide over and exchange heat with static fluid channels.

Clutch model where heat is generated due to friction between the plate and the pad which is cooled by fluid channels.



- The intersection of embedded shells results in non-watertight meshes where fluid can spill through the mesh. The new approach prevents this effect isolating the mesh effectively and preventing any fluid escapement. See *ICFD_CONTROL_EMBEDSHELL.

Heart valve leaflets. Contour of fluid velocity. A) flow through intersections. B) Intersections are sealed.

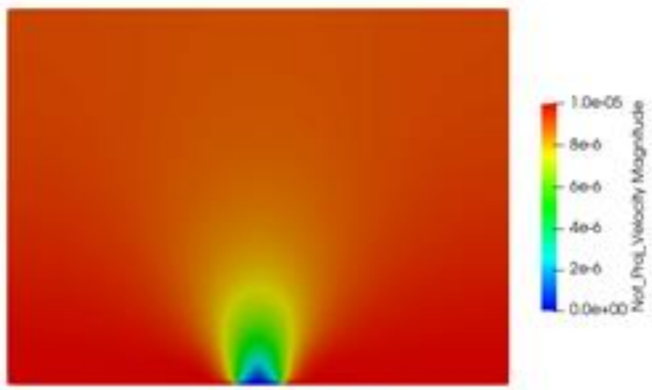


Multi-Species transport

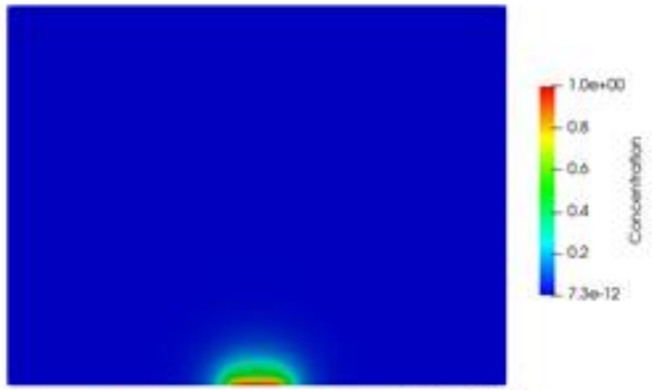
- Monolithic solver (all multi-physics equations coupled in one solver),
- Flexible Porous media flow solver,
- Multi-species transport (one species so far),
- Free-Surface Flows (for saturated/unsaturated regions),
- Chemical reactions and Electro-chemically charged species (planned),
- Coupling with LS-DYNA structural solver for potentially modeling the needle nailing.

See *ICFD_MODEL_SPECIES_TRANSPORT,

*ICFD_PRESCRIBED_SPTRANSP_CONC



Velocity field



Concentration field

Ansys

Miscellaneous Enhancements

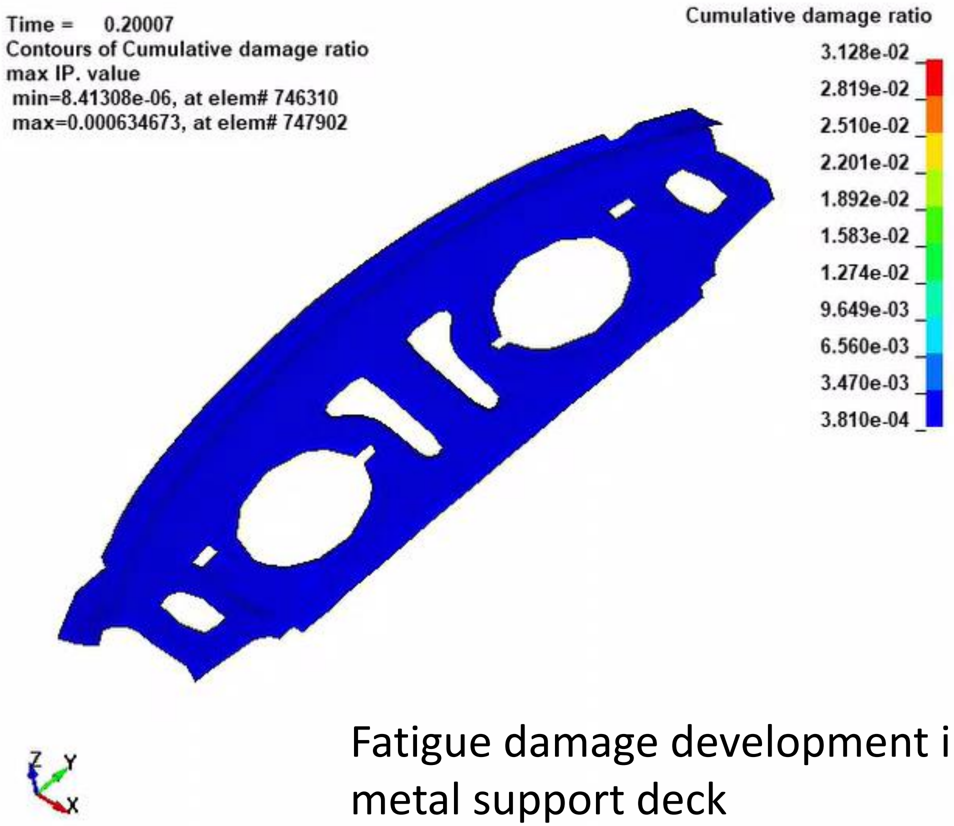
Fatigue analysis based on modal dynamics

*FATIGUE_MODAL_DYNAMIC

- Use coefficients from modal dynamic analysis to reconstruct stress history
- Saving in hard drive space since no elout or d3plot is needed
- Saving in CPU cost since no stress output is needed in transient simulation

	CPU cost (sec, 1 core)
FATIGUE_MODAL_DYNAMIC	255
FATIGUE_ELOUT	659[1]
FATIGUE_D3PLOT	462[2]

[1] with 7G binout files
[2] with 15G d3plot files

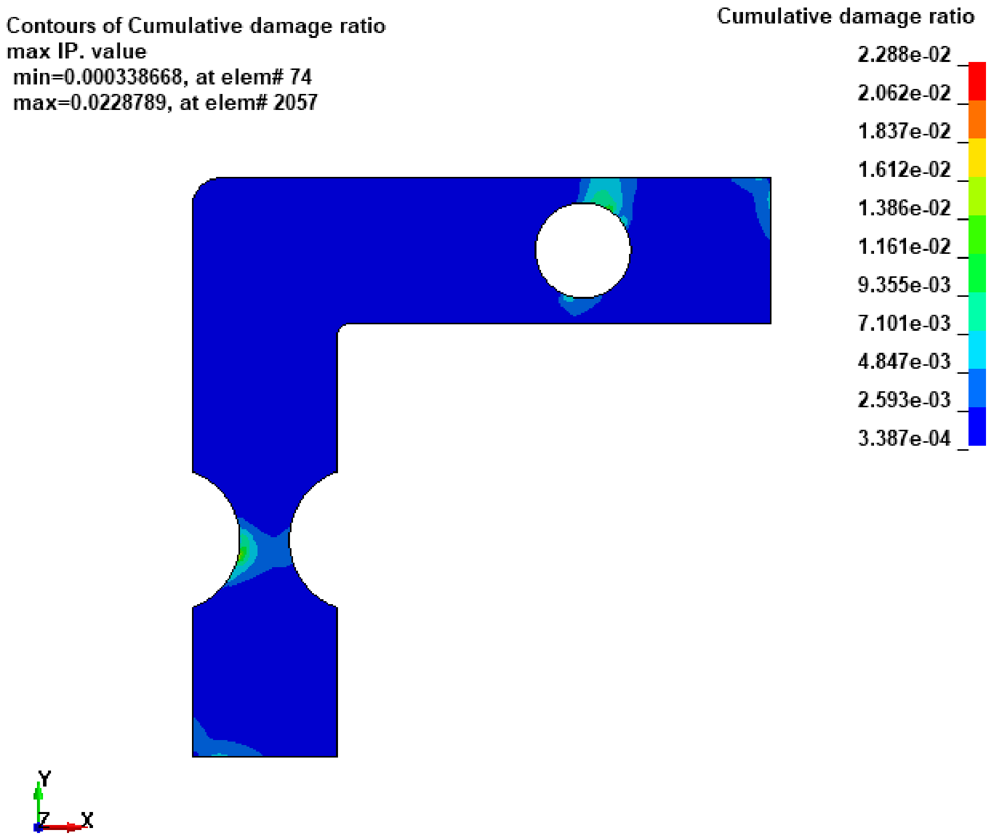


Fatigue damage development in a metal support deck

Random vibration fatigue analysis based on IGA model

*FREQUENCY_DOMAIN_RANDOM_VIBRATION_FATIGUE

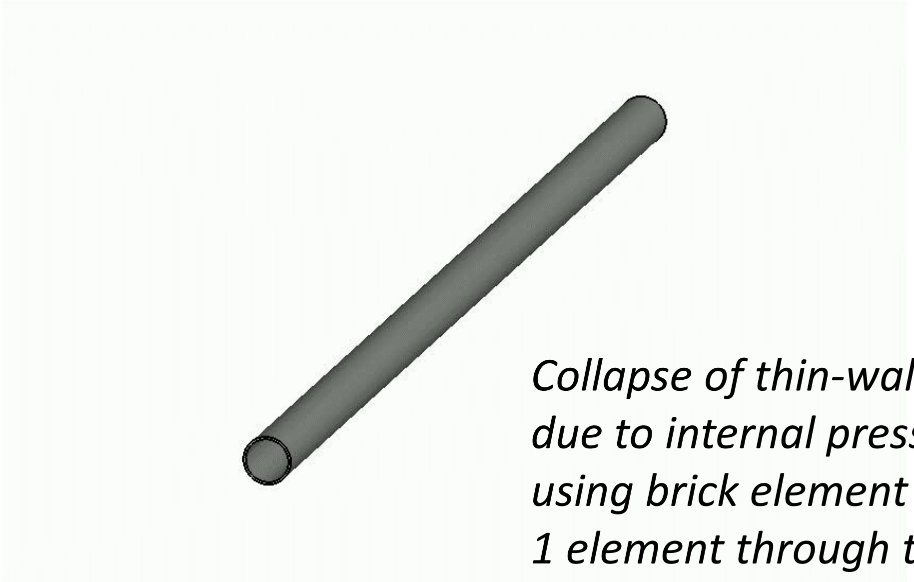
- IGA runs modal analysis and computes modal stresses
- Modal stresses from IGA are mapped to interpolation finite elements (can be skipped in the future)
- Random vibration fatigue analysis is performed using stress results on interpolation elements.
- Fatigue results are more accurate than that given by FEM approach since the stresses are more accurate.



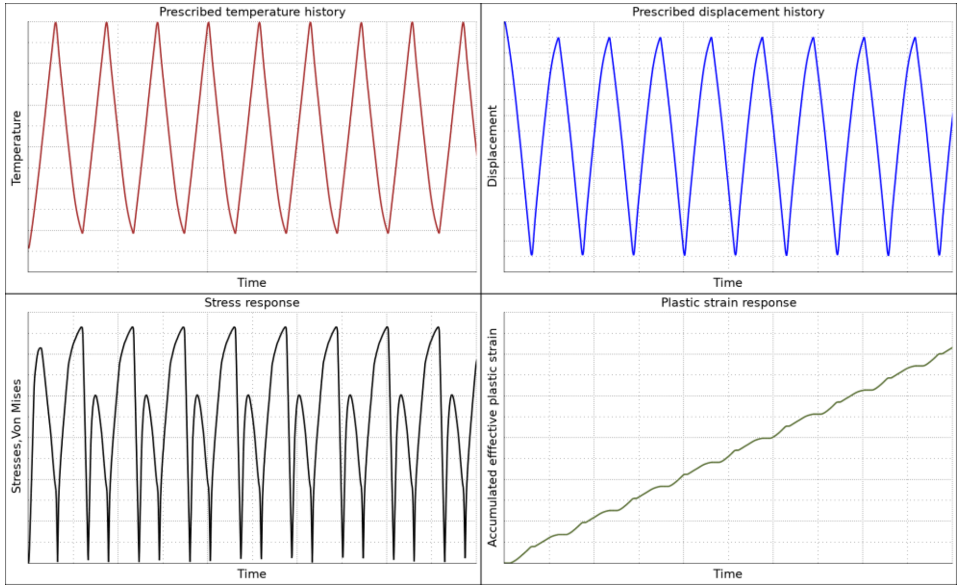
A metal bracket (originally modelled by IGA) subject to base acceleration PSD

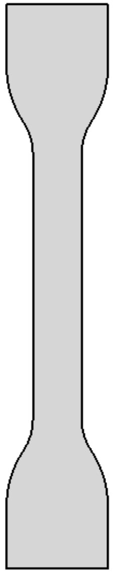
Element/Material/Implicit

- ELFORM=-18 on *SECTION_SOLID invokes an enhanced assumed strain element for coarse mesh accuracy
  - 13 incompatible modes to avoid shear and volumetric locking
  - Primarily intended for implicit analyses of thin components
- Material 153 with thermal effects
  - Elastic moduli as functions of temperature
  - Hardening and back stress parameters as functions of temperature
  - Material parameter fit accounting for temperature
  - Enhanced accuracy for implicit simulations



Collapse of thin-walled tube due to internal pressure – using brick element -18 and 1 element through the thickness

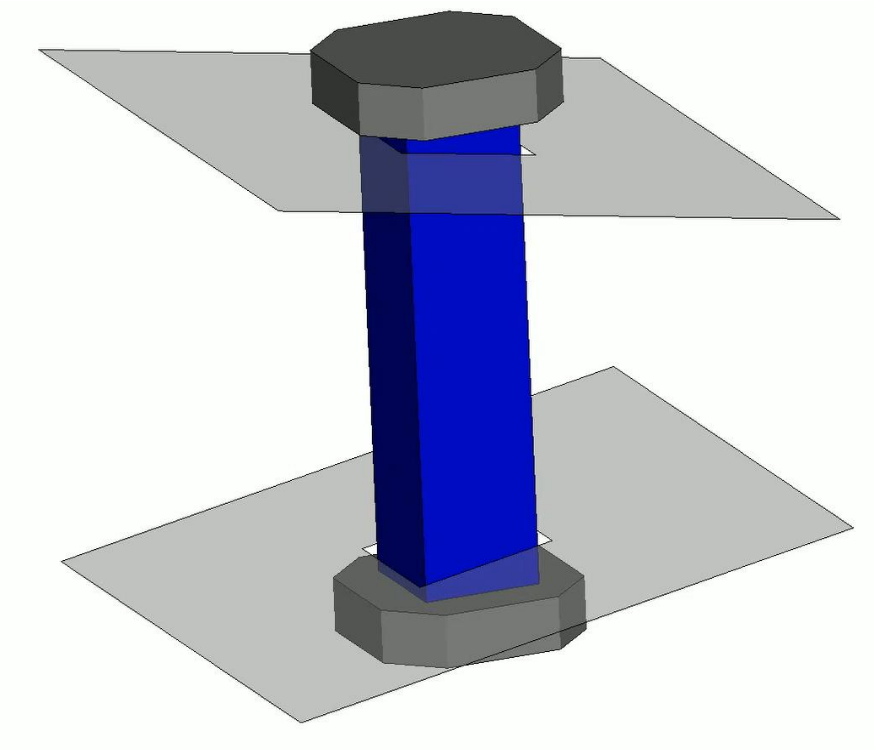




Response for combining cyclic loading and temperature of specimen using material 153

Element/Material/Implicit

- Improved/new tangents
  - Shell types 4,16,… (nonsymmetric tangent support)
  - Materials 133,187,… (consistent tangents)
- Exponential damage function in DIEM for accurate post-instability response
- Plastic Energy Density output in *DEFINE_MATERIAL_HISTORIES
- COHOFF on *SECTION_SOLID can be used for relative location of cohesive interfaces
- IZSHEAR=2 on *INITIAL_STRESS_SECTION supported for all cross section types
  - Bolt preload can also start from nonzero stress state
- Rigid body stoppers supported in implicit by way of Lagrangian Multipliers
- Accurate tied contact supported for explicit when IACC=1 on *CONTROL_ACCURACY
  - For compatibility between implicit and explicit in switching and sequential simulations



*Preloading of bolt between two non-flat plates with complete shank included in the *DATABASE_CROSS_SECTION_SET*

Multi-dimensional tables: new input format

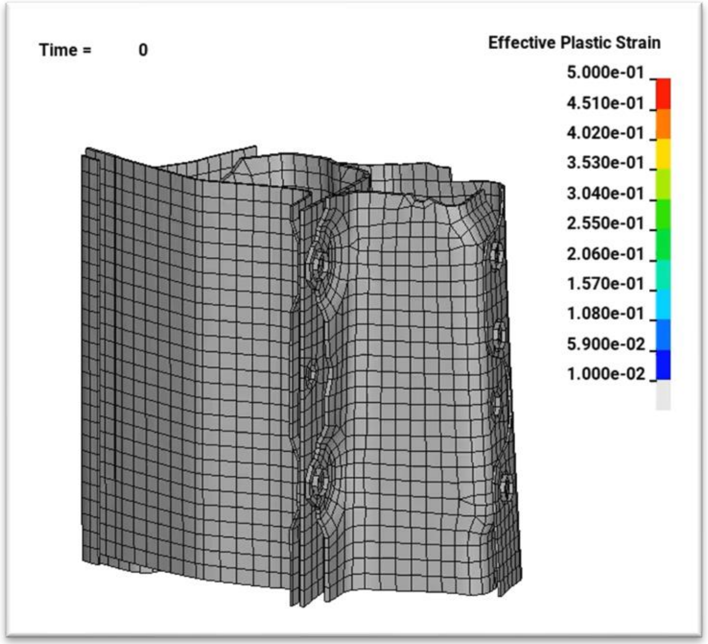
- New keyword *DEFINE_TABLE_COMPACT
  - Allows clearly arranged (Excel like) tabular input of data
  - Can replace current staggered format of several *DEFINE_TABLE_XD's
  - Mostly supported in material models at the moment

- Example for 4-dimensional table
  - Stress as function of strain, strain rate, temperature, and hardness
  - Could be used this way in *MAT_TAILORED_PROPERTIES (*MAT_251)

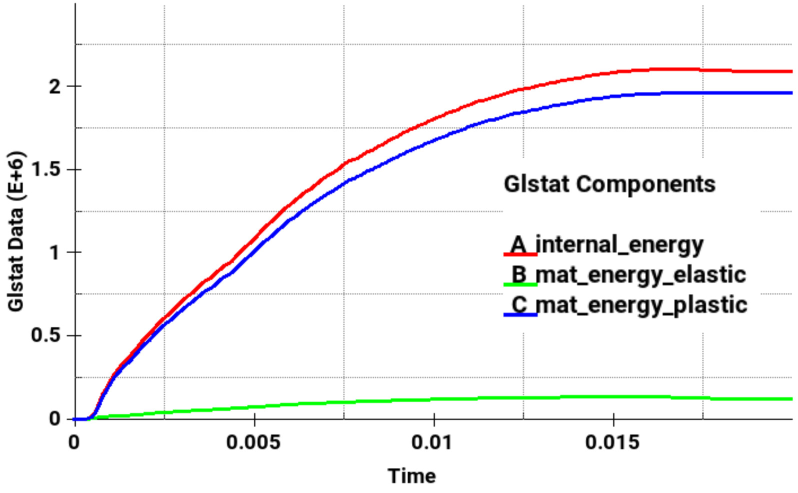
```
*DEFINE_TABLE_COMPACT
$     tbid    numvar     lcint
     10000         4
$    value      var1      var2      var3      var4
$      sig       eps      rate      temp      hard
     360.0       0.0       0.0      20.0      80.0
     570.0       0.3       0.0      20.0      80.0
     780.0       1.0       0.0      20.0      80.0
     470.0       0.0     100.0      20.0      80.0
     680.0       0.3     100.0      20.0      80.0
     860.0       1.0     100.0      20.0      80.0
     180.0       0.0       0.0     400.0      80.0
     285.0       0.3       0.0     400.0      80.0
     390.0       1.0       0.0     400.0      80.0
     235.0       0.0     100.0     400.0      80.0
     340.0       0.3     100.0     400.0      80.0
     430.0       1.0     100.0     400.0      80.0
     540.0       0.0       0.0      20.0     200.0
     855.0       0.3       0.0      20.0     200.0
    1170.0       1.0       0.0      20.0     200.0
     705.0       0.0     100.0      20.0     200.0
    1020.0       0.3     100.0      20.0     200.0
    1290.0       1.0     100.0      20.0     200.0
     270.0       0.0       0.0     400.0     200.0
     427.5       0.3       0.0     400.0     200.0
     585.0       1.0       0.0     400.0     200.0
     352.5       0.0     100.0     400.0     200.0
     510.0       0.3     100.0     400.0     200.0
     645.0       1.0     100.0     400.0     200.0
```

Detailed output of internal energy

- New parameter MATEN on *CONTROL_ENERGY
  - Internal energy is split into elastic, plastic, and damage portions
  - Reported in matsum and glstat as mat_energy_elastic, mat_energy_plastic, and mat_energy_damage
  - Currently supported for choice of material models: {3, 4, 15, 19, 24, 63, 81, 82, 98, 104, 105, 106, 107, 123, 124, 188, 224, 225, 240, 251} + DIEM / GISSMO

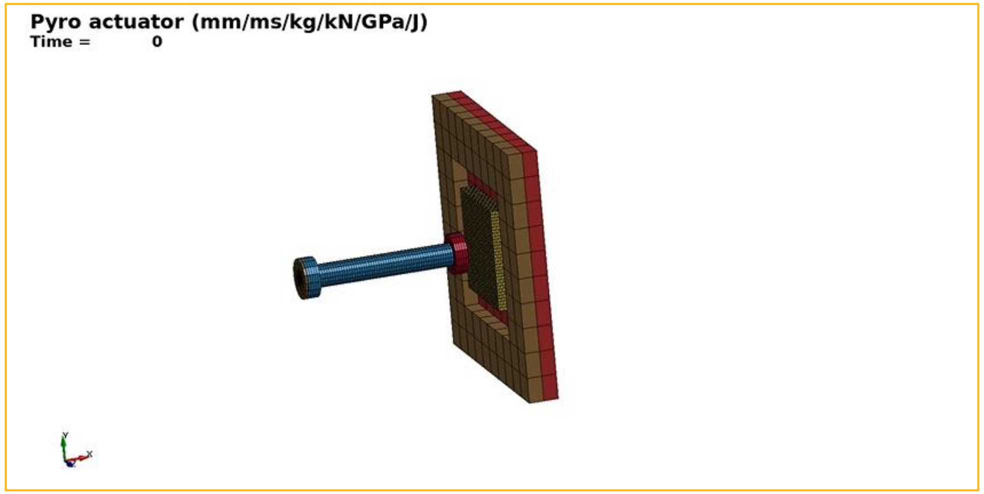


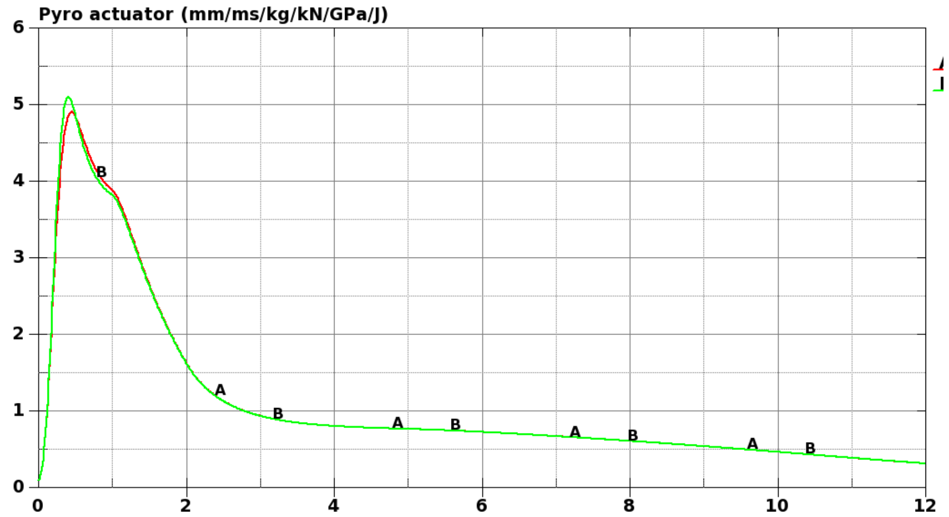
- Improved understanding of structural behavior
  - e.g. energy consumption in crash parts

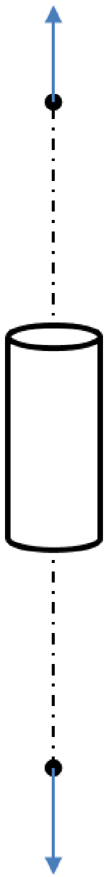


*LOAD_PYRO_ACTUATOR/*DATABASE_PYRO

- Calculates nodal/segment forces from a pyrotechnic piston actuator without the need to model the chamber/piston
- Application: active hood lifters for pedestrian protection
- Input: nodes/segments, mass flow curve, chamber cross section area, etc







Example of actuator and resulting pressure. Chamber is defined by two nodes.

*DEFINE_PRESSURE_TUBE

- New solution method options
  - MTD=1: Discontinuous Galerkin (DG) for Acoustic Equations
    - Less diffusive than CG
  - MTD=2: DG for Isothermal Euler Equations
    - More accurate for large amplitudes
    - Handles shocks and rarefaction waves

Discontinuous Galerkin (MTD=1): Alternatively, the linear system

$$\frac{\partial p}{\partial t} + \frac{\partial \ln A}{\partial t} p + \frac{p_0}{A}\frac{\partial y}{\partial x} = -\mathrm{DAMP} \cdot (p - p_0)$$

$$\frac{\partial y}{\partial t} + A\frac{c_0^2}{p_0}\frac{\partial p}{\partial x} = 0$$

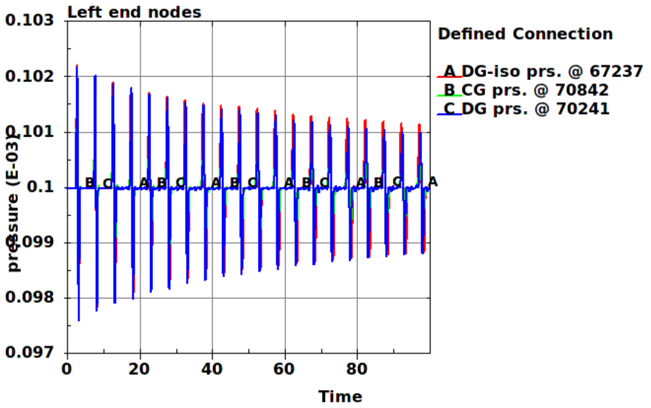
can be solved with the Discontinuous Galerkin method, using piecewise linear basis functions on each element, Lax-Friedrich flux, and a MUSCL flux limiter to limit spatial oscillations. Time integration is done with Heun's method which is a 2nd order TVD Runge Kutta method. VISC>0 is in this case the smoothing factor from the MUSCL limiter, with VISC=0 indicating no smoothing. The DG method gives less diffusion (smearing) than the CG method.

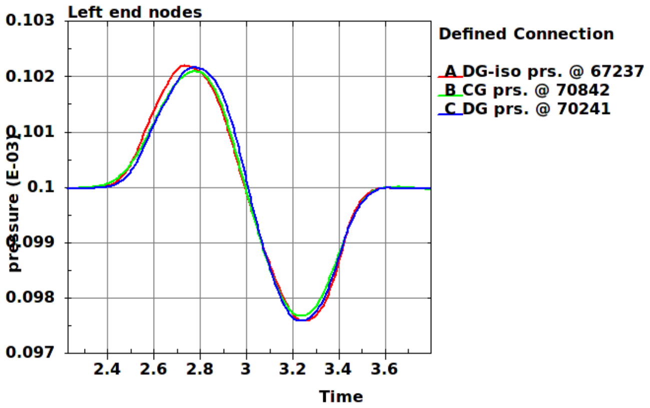
Discontinuous Galerkin (MTD=2): This option uses DG to solve the damped non-linear isothermal Euler equations

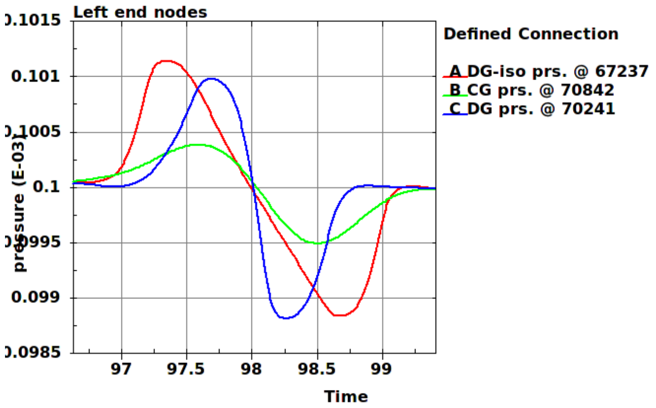
$$\frac{\partial}{\partial t}(pA) + \frac{\partial}{\partial x}(puA) = -\mathrm{DAMP} \cdot (p - p_0)A$$

$$\frac{\partial}{\partial t}(puA) + \frac{\partial}{\partial x}(pu^2 A + c_0^2 pA) = c_0^2 p \frac{\partial A}{\partial x}$$

VISC has the same meaning as for MTD=1. This option may be beneficial for scenarios with large pressure/velocity perturbations, where the acoustic approximation is poor.







Ansys