New Features in LS-DYNA R10.0

LSTC & DYNAmore, December 2017

- Release R10.0 published in June 2017
- This presentation about major changes since R9.2
- Comprehensive list of enhancements and corrections in
  http://www.dynasupport.com/news/ls-dyna-r10.0.0-r10.118302-released

LS-DYNA Versions

- Version numbering scheme
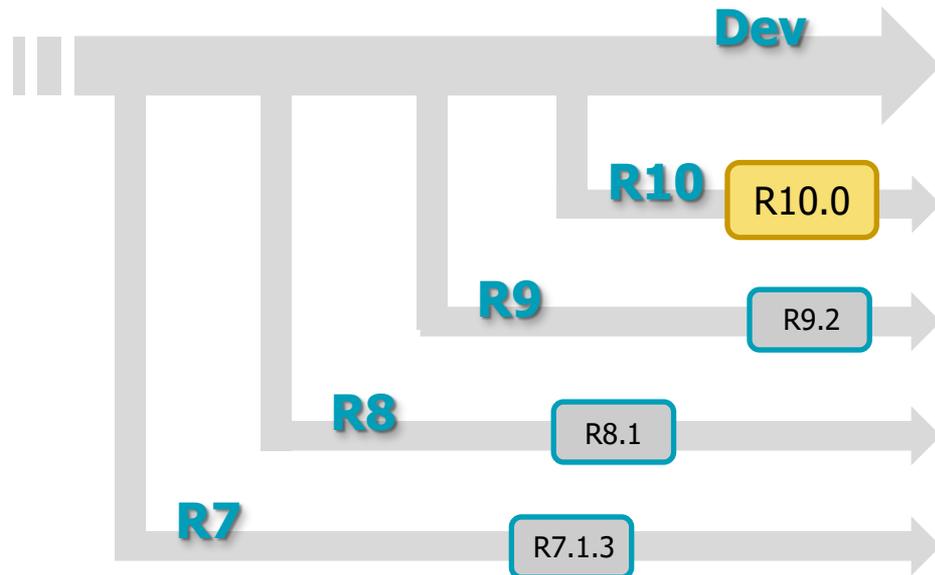 - Major branches called R6, R7, R8, R9, R10 …
  - Official releases such as R6.1.2, R7.1.3, R9.2

- Robust releases
 - Releases R7.1.3 / R9.2
  - Recommended production versions

- Latest official versions
  - Release R8.1 from February 2016
  - Release R9.2 from August 2017
 - Release R10.0 from June 2017: New features shown in this presentation

Overview

- Element Technology
- Contact
- Material models
- Implicit analyses
- Frequency Domain
- Forming applications
- Miscellaneous

- Meshfree methods: SPH
- ALE / S-ALE
- Fluid solver - CFD / FSI
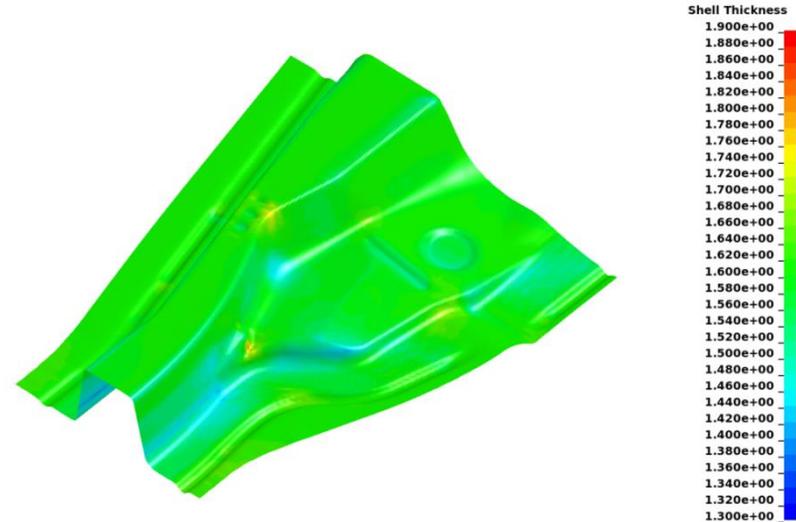- Electromagnetics

- Bug fixes
- Conclusion
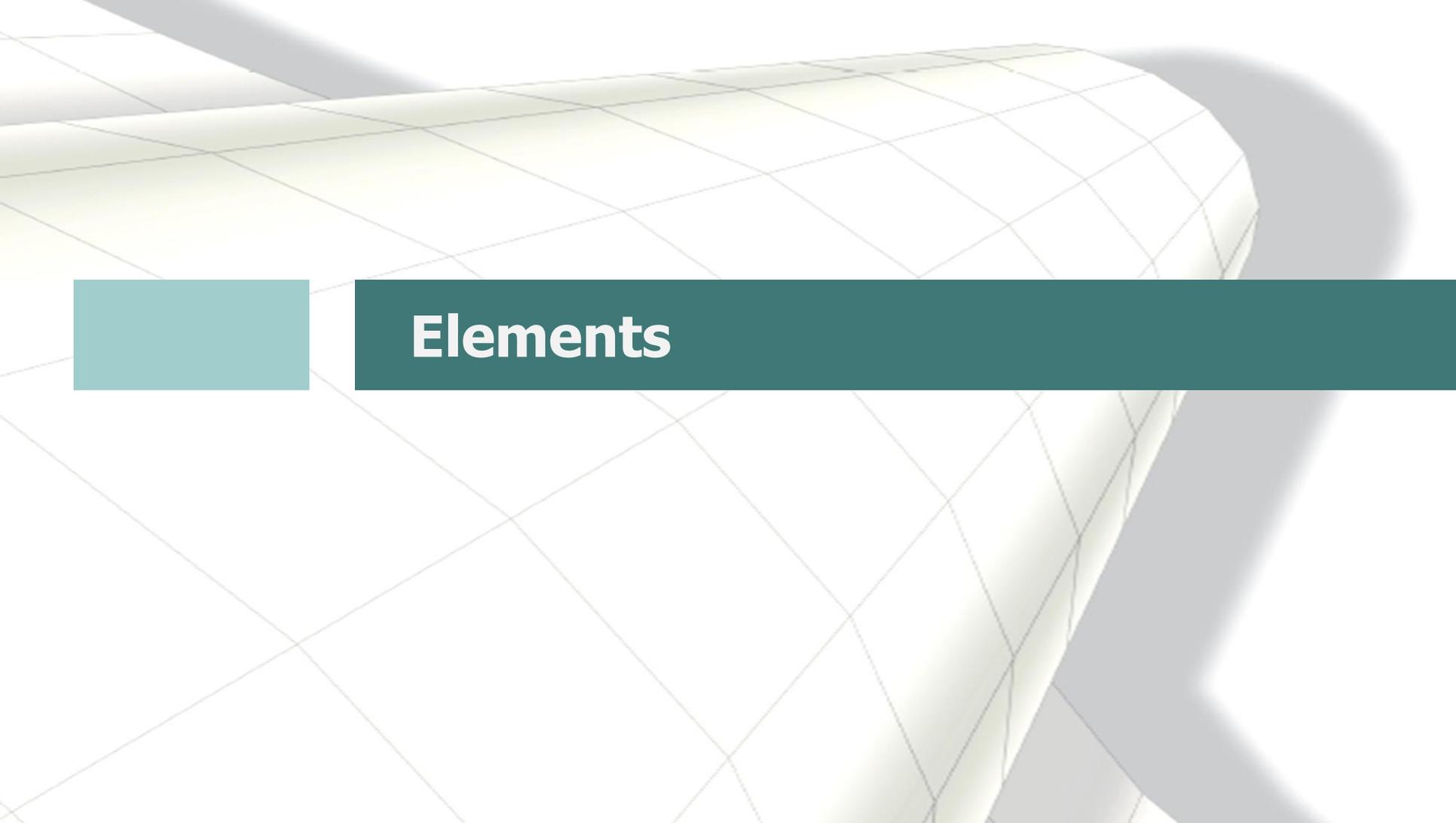
Elements

Isogeometric analysis (IGA)

- Many enhancements and bug fixes for trimmed NURBS
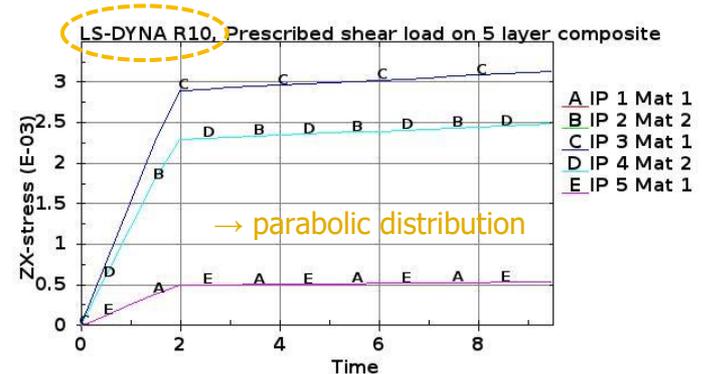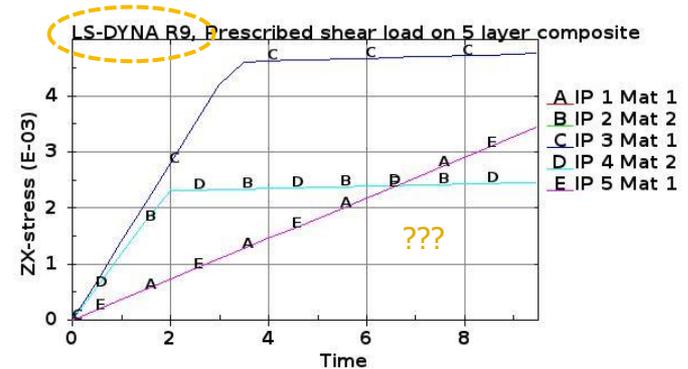

physical space

parametric space
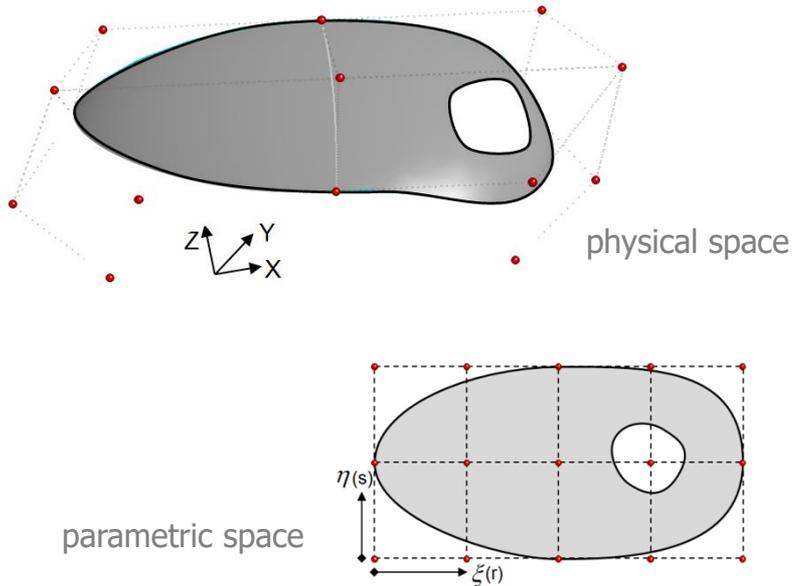
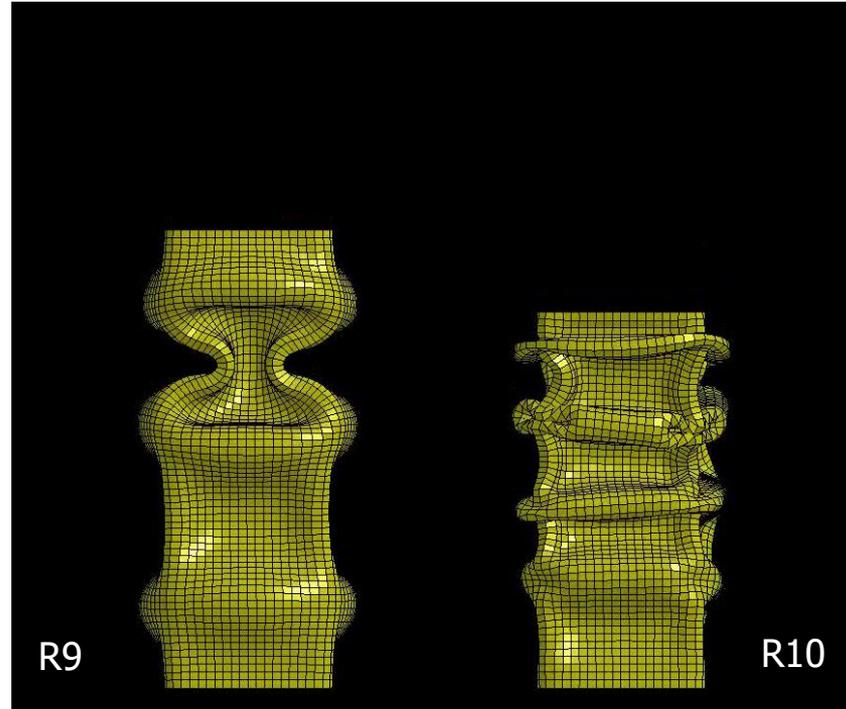- Added post-processing of strains and thickness change

Thick Shell Enhancements

- Use of thick shells in composite modeling has lead to new developments, in particular the transverse shear treatment is affected

  - SHRFAC added as option to *PART_COMPOSITE_... to provide a general transverse shear stiffness profile

    - TSHEAR=1 (constant stress profile) fixed in the process

  - Enable recalculation of shear stiffness based on a yielding of individual integration points; yielding in one layer will result in lower elastic stiffness in others due to a coupling effect

    - applies to materials 3, 18, 24, 123 and 165

LS-DYNA R9, Prescribed shear load on 5 layer composite

A IP 1 Mat 1
B IP 2 Mat 2
C IP 3 Mat 1
D IP 4 Mat 2
E IP 5 Mat 1

???

LS-DYNA R10, Prescribed shear load on 5 layer composite

A IP 1 Mat 1
B IP 2 Mat 2
C IP 3 Mat 1
D IP 4 Mat 2
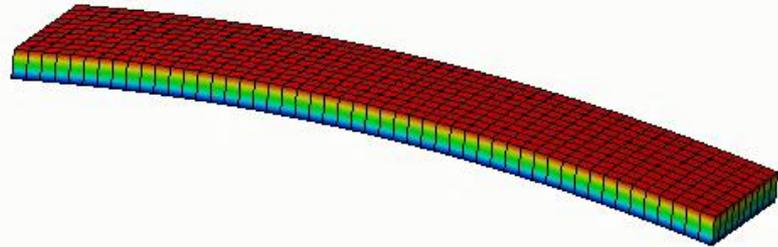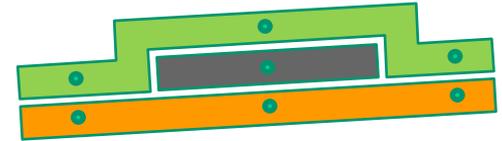E IP 5 Mat 1

→ parabolic distribution

Thick Shell Enhancements

- Example: Tube buckling
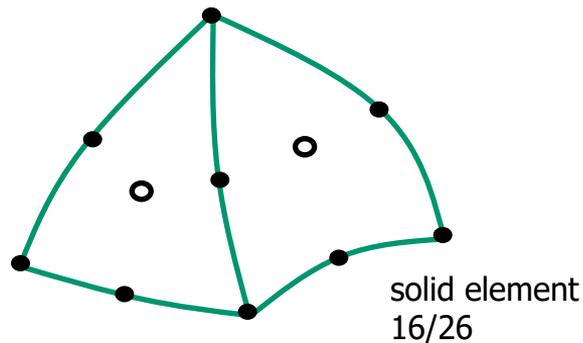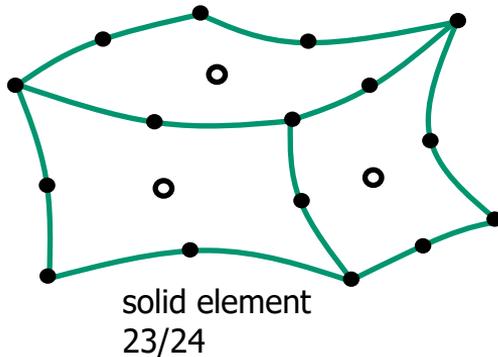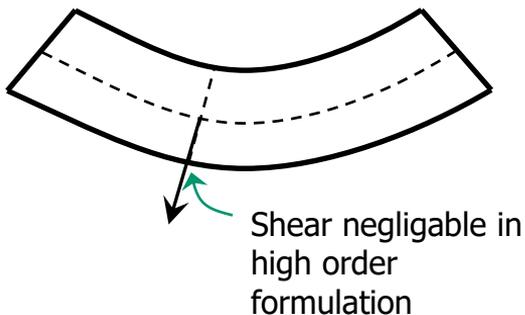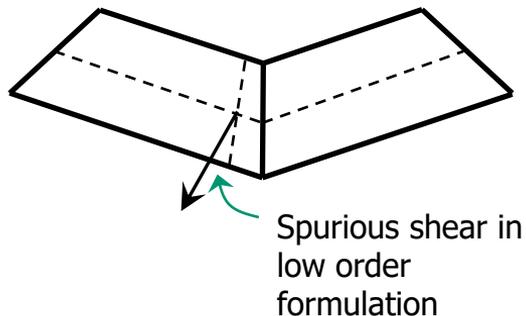
More Thick Shell Enhancements

- Hourglass and Poisson effect treatment revised for thick shell type 5 to render a more realistic response

- Improved thick shell type 5 when mixing materials in the layers
  - resulting in better z-strain distribution and accurate stress tensor

- Time step calculation improved for better stability
  - in particular for triangular and type 3 and 5 thick shells

- Materials 54, 55 and 56 supported for thick shell types 1, 2, and 6

- Thick shells supported in Selective Mass Scaling

- Anisotropic option on *MAT_ADD_THERMAL_EXPANSION supported for thick shells
  - works for layered composites

Thick and Thin Shell Enhancements

- Permit zero layer thickness in *PART/ELEMENT_(T)SHELL_COMPOSITE(_LONG)
    - Number of integration points can remain constant despite different number of physical plies, which is important when number of plies vary between structures and facilitates post-processing
- Some improvements for thin shell composites

    - A less conservative time step calculation approach results in larger used time steps
    - Correction of rotational inertia for layers using mixed materials
- *ELEMENT_SHELL_SOURCE_SINK supported for Belytschko-Wong-Chang elements

- Thermal expansion supported with thick thermal shells

High Order Solids

Spurious shear in low order formulation

Shear negligable in high order formulation

solid element 23/24

solid element 16/26

- 20-noded (#23) and 27-noded solids (#24) available before R10

- Element recoded to increase speed (explicit)

- Consistent mass matrix implemented (implicit)

- Automatically apply boundary conditions for H8TOH27 option

- *SET_NODE_GENERAL supported for high order solids

Contact in High Order Solids

- **Segment based contact (SOFT=2) supported with 27-noded solids**

- **Mortar Contact supported with 27-noded solids**

  - Kinematics performed on child-segments, assuming linear or bilinear shapes

  - Kinetics assumes quadratic shape functions, and contact pressure is assembled to all nodes consistently to mother segment

  - Accurate for reasonably flat surfaces

  - Will not capture the curvature of cylindrical axes, etc.

child segments

mother segments
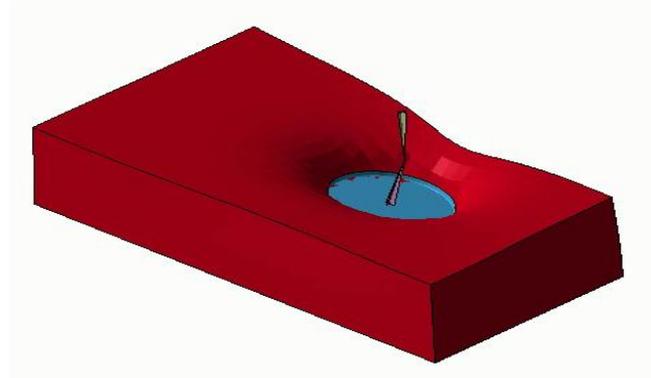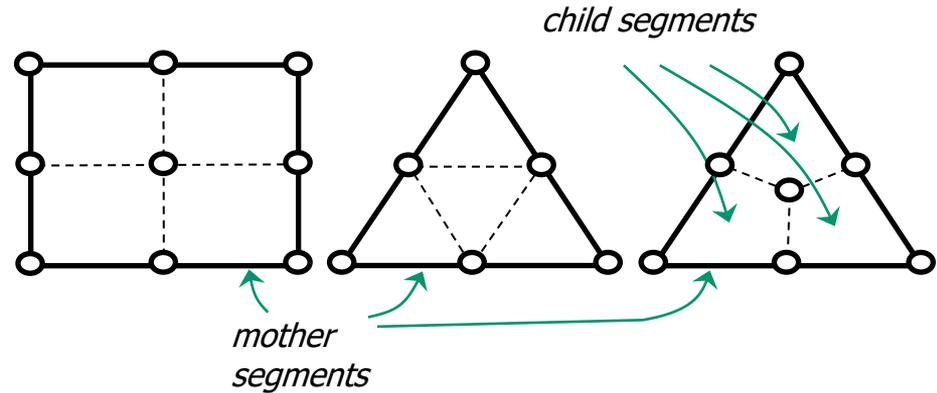
Cohesive Solid Elements

- ELFORM=19-22 on *SECTION_SOLID

 - Used for planar connections such as adhesives, composite interlayers, etc.
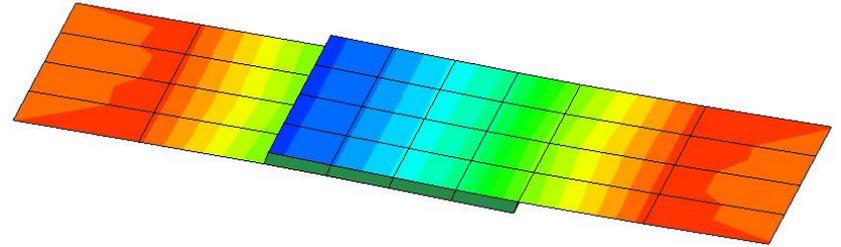
- New in R10

  - Increase accuracy of ELFORM=19 and 21 by using double precision for local coordinates and other internal variables (also ported back to R9.2)

  - Alternative mass calculation for critical time step estimate of cohesive elements to improve their robustness, defined via ICOH on *CONTROL_SOLID

  - *MAT_ADD_COHESIVE now works in implicit analysis

  - *MAT_USER_DEFINED_MATERIAL_MODELS for cohesive elements (umatXXc) can now be used in implicit analysis and a few more input variables are provided: temperature, element size, implicit rejection flag, integration point identifier, and total number of integration points
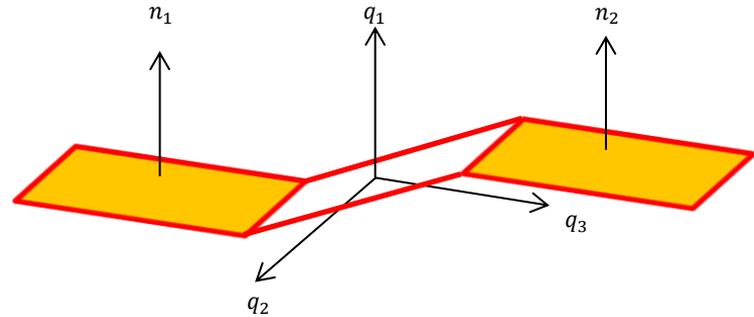
Cohesive Shell Elements

- New mid-layer coordinate system ELFORM = -29 (suitable for pure shear)
- *MAT_240 now supports ELFORM = +/-29
- Fixed absence of part mass in d3hsp

ELFORM = +29 (q_3 from nodes)
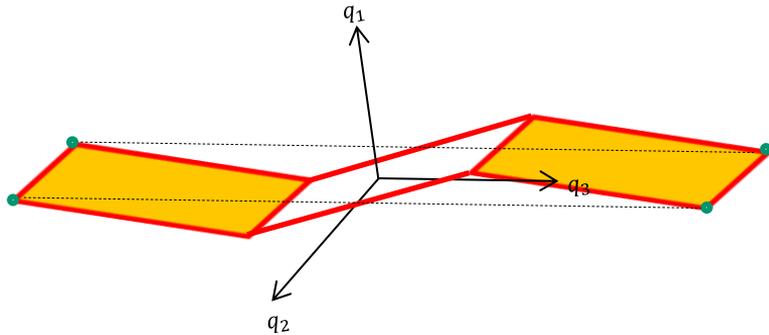
ELFORM = -29 (q_1 from normals)

Constrained Beam in Solid

■ R-adaptivity supported in R10

   ■ Goal is to model the manufacturing process of Fiber Reinforced Plastic,
   in particular the shear force between fiber and the matrix

   ■ The matrix undergoes large deformation; remeshing needed

   ■ Constrained Beam in Solid needs to dump out and read back history variables
   related to debonding force calculations during each restart

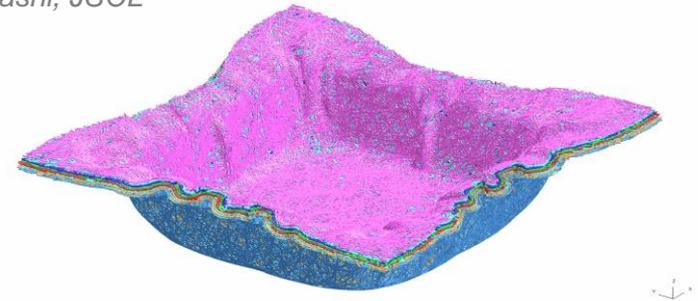   ■ Application: Fiber reinforced molding

Courtesy of Shinya Hayashi, JSOL
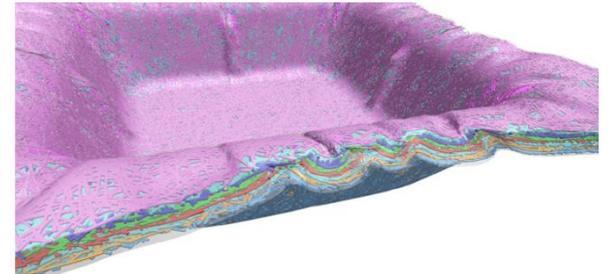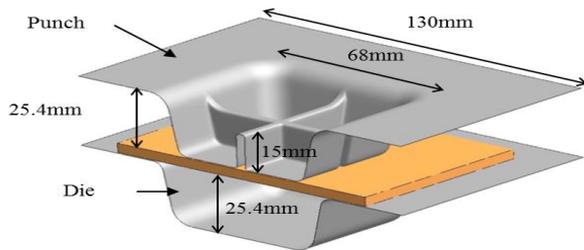
Constrained Beam in Solid

■ Application: Fiber reinforced molding

Weld line

Fiber orientation

Matrix rich region
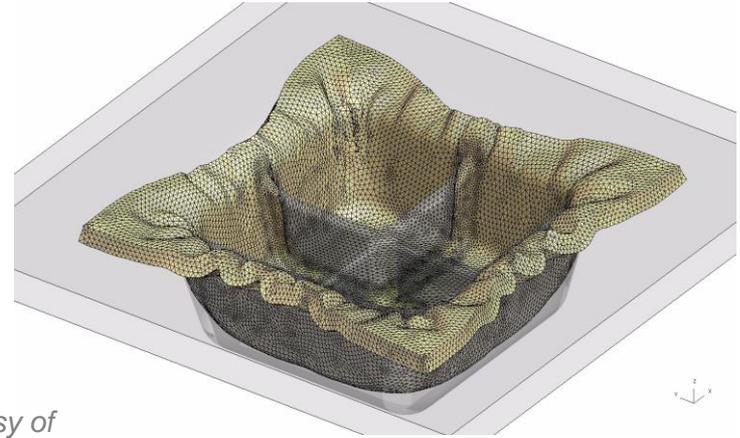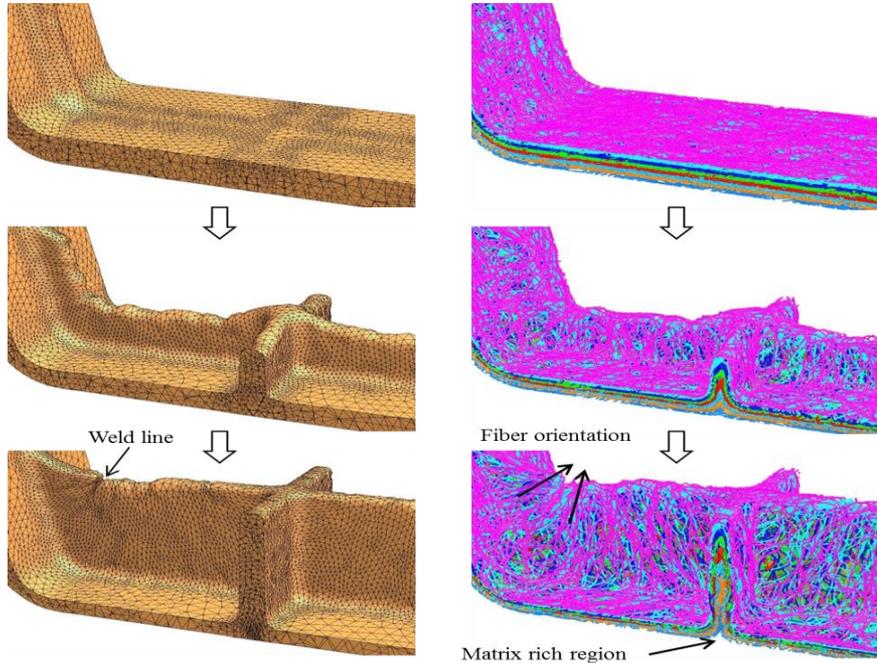
Courtesy of
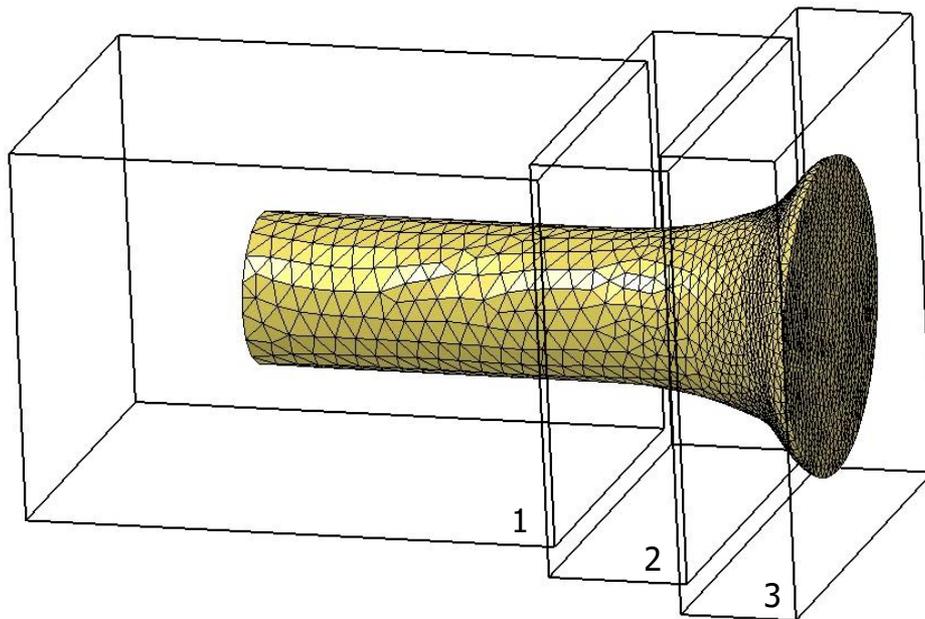*Shinya Hayashi, JSOL*

Local adaptive refinement for solids

- 3D r-adaptivity (ADPOPT=7) with type 13 tets now supports *DEFINE_BOX_ADAPTIVE

  - Different mesh size limits in specific regions can be defined

  - Replaces *CONTROL_REMESHING settings

- Example: Taylor bar impact with 3 different boxes

  - Box 1: RMIN=0.8, RMAX=1.5
  - Box 2: RMIN=0.6, RMAX=1.2
  - Box 3: RMIN=0.4, RMAX=0.9

General Element Enhancements

- *SECTION_BEAM_AISC provides predefined length conversion factors for specific unit systems

- Hughes-Liu beam supports *INTEGRATION_BEAM such that each integration point can have different thermal expansion coefficient from *MAT_ADD_THERMAL_EXPANSION

- User defined elements (ELFORM=101 to 105 on *SECTION_SHELL/SOLID) support *MAT_ADD_EROSION

- *ELEMENT_BEAM_SOURCE provides a pull-out force vs time through the parameter FPULL

- MPP now supports *ELEMENT_MASS_MATRIX_NODE_(SET)

- Make *TERMINATION_DELETED_SOLIDS work with hex spot weld failures
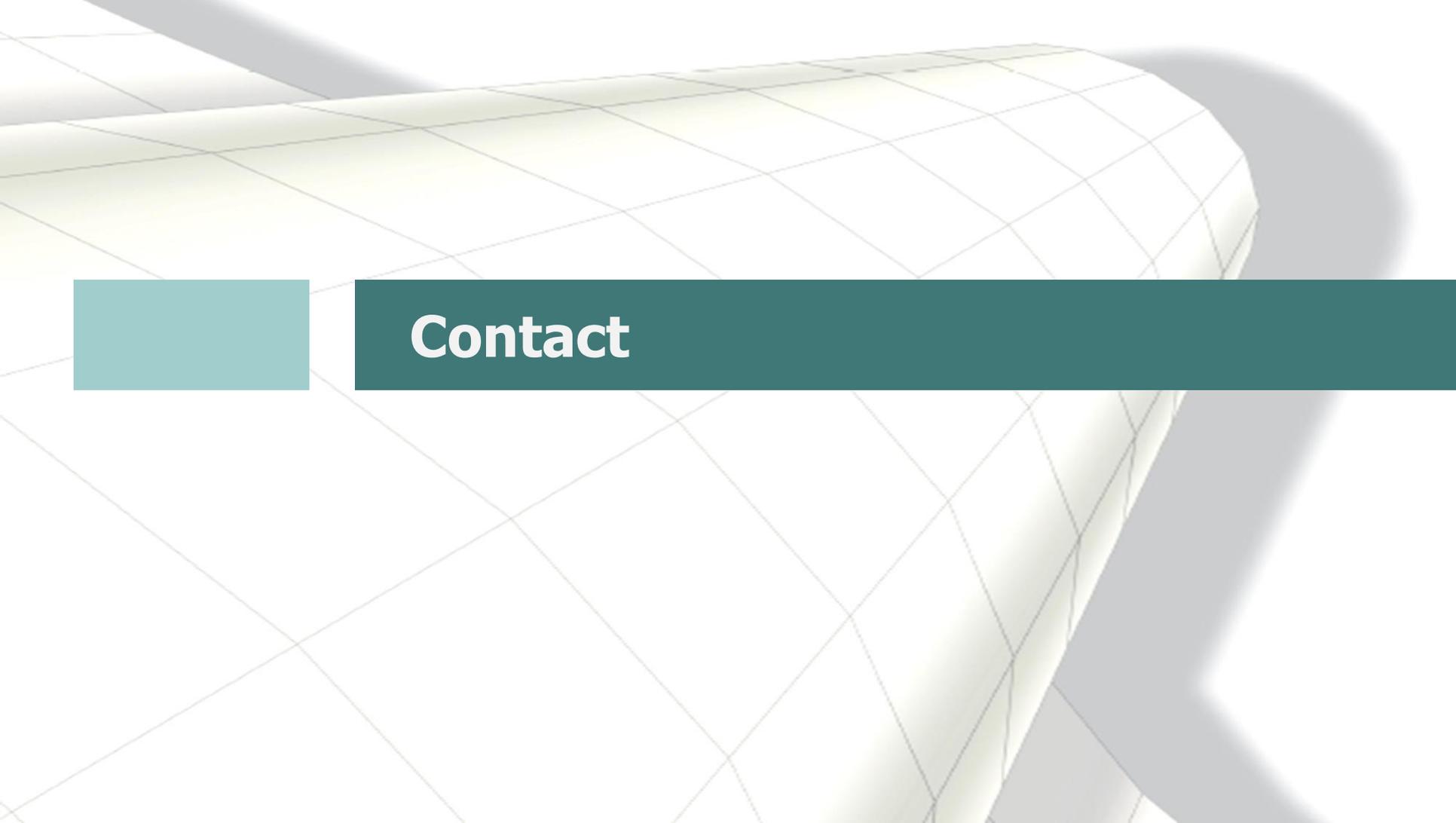
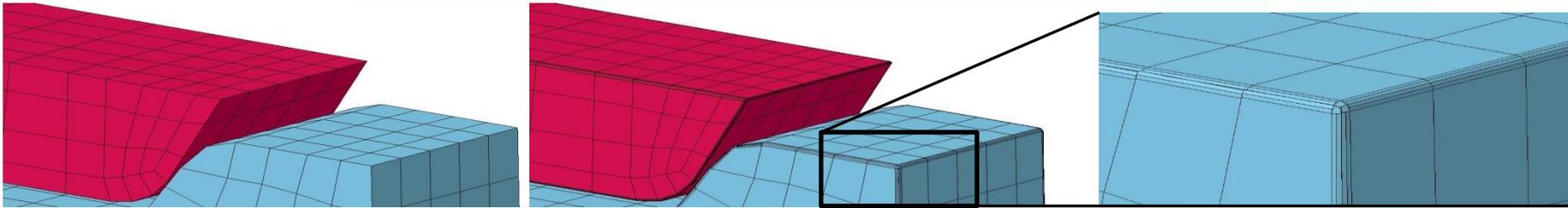# Contact

SOFT=2 contact enhancements

- Added support for the SRNDE parameter on optional card E, choice of different shell edge treatments

- When *PART_CONTACT is used, the friction coefficient used in contact is the average between the slave and master coefficients (as in SOFT=0/1)

- Box option (SBOXID, MBOXID) now supported in eroding contact

- Improve efficiency of segment based contact for SMP/Hybrid, significantly impacting simulations that use a large number of cores

- VC parameter supported in segment based contact (SOFT=2), use preferrably with FNLSCL>0 and DNLSCL=0 to provide a smoother pressure distribution

- Added support for *ELEMENT_SOURCE_SINK

Mortar Contact Edge Treatments – Shells and Beams
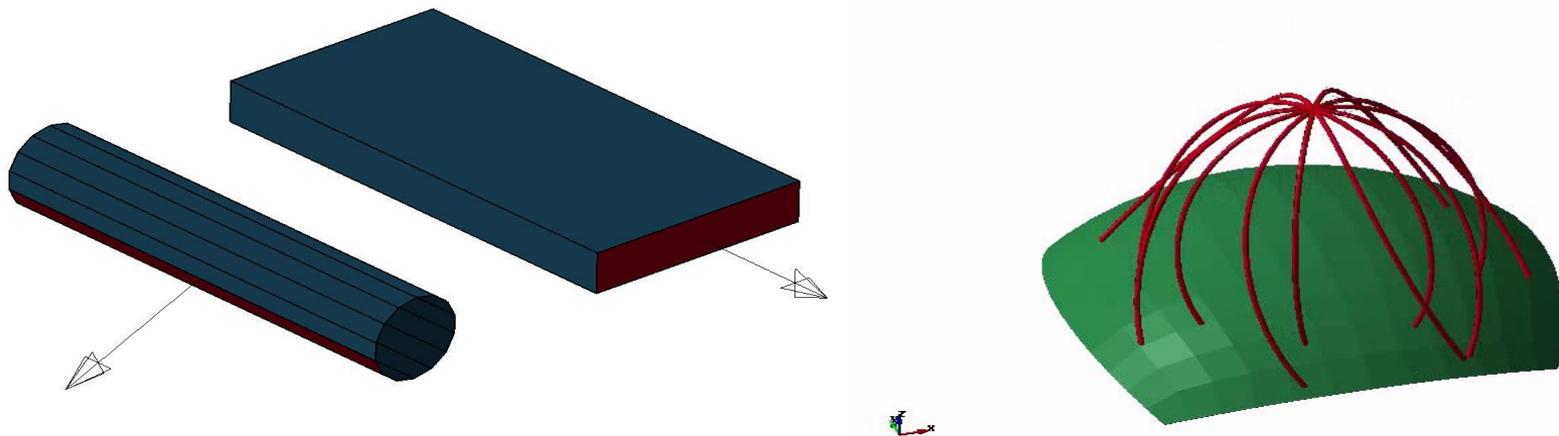
- SRNDE=2 mandatory, shell edges are flat and presumably consistent with geometry
- Beam lateral surfaces are discretized into segments
- In R10 any free end point is treated as a flat surface

Mortar Contact Edge Treatments - Solids
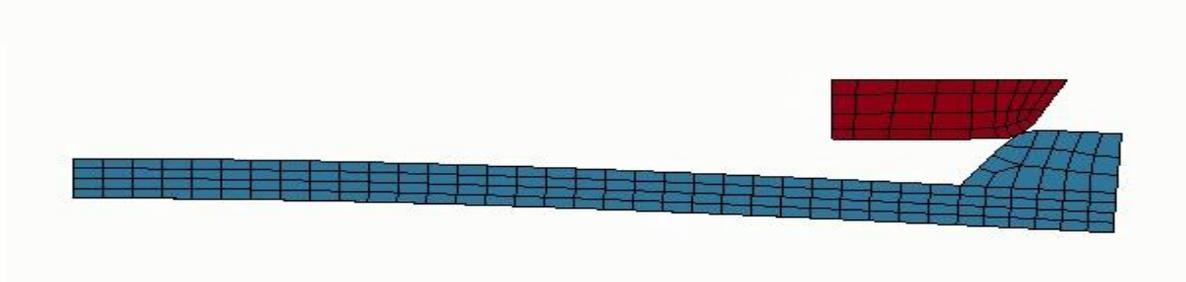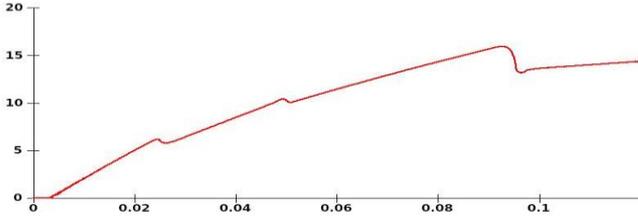
- Mortar Contact creates internal contact segments to deal with edges
- In R10, high order solids are supported

Mortar Contact – Explicit Analysis

- A nonzero bucket sort frequency implemented in R10
- Mortar Contact will never be as fast as the traditional SOFT contacts
    - Goal is to not fall too far behind and provide a decent alternative in those cases when called for
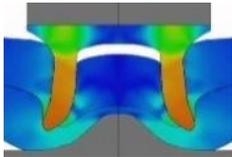
Problem	SOFT=0		SOFT=2		MORTAR	
SPR detachment (24 cores, MPP single)	1.13	2.02	1.00	1.00	1.89	6.27
					2.68	11.38
B-pillar bend (8 cores, MPP single)	1.13	1.47	1.00	1.00	2.32	10.03
					7.48	45.76

R10 total time | R10 contact time | R9 total time | R9 contact time

Miscellaneous Mortar Contact Features

- *CONTACT_2D_..._MORTAR
  - Only for implicit and SMP
- As of R10, PENMAX and SLDTHK governs the maximum penetration depth solid element offsets
  - Avoid paradoxic behavior in models with mixed solids and shells
- Contact gaps are output to the INTFOR file
- Force transducers supported
  - Parts or part sets mandatory
- Tiebreak options 2, 4 and 7 supported (and ..._TIED_WELD)
- Dynamic friction (FD and DC) supported for explicit and dynamic analyses
- Characteristic size of solid elements revised, to provide a better estimation of contact stiffness and geometry details
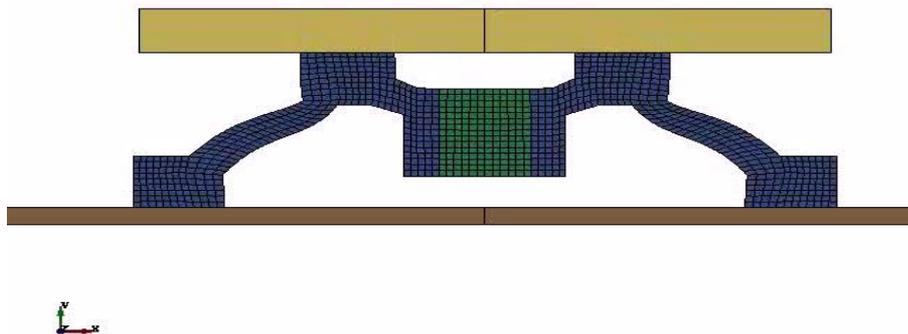
User-defined tiebreak models

■ New option _USER for *CONTACT_AUTOMATIC_..._TIEBREAK

    ■ Allows implementation of alternative models to standard OPTIONs

   ■ Available for SMP and MPP
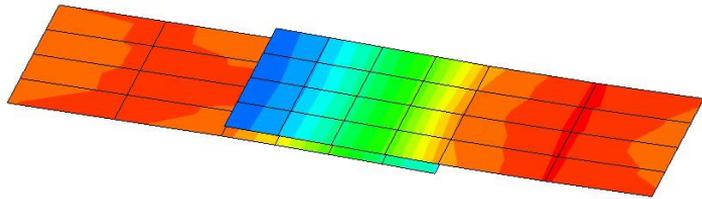
stress distribution
In lap shear test

```
 subroutine utb101(sig_n,sig_t,disp_n,disp_t,vel_n,vel_t,cn,ct,
     . uparm,uhis,idcon,idsn,idms,areasn,areams,time,dt2,ncycle,crv,
     . nnpcrv,temp,ifail,ioffset)
c
c     User subroutine for tiebreak contact: OPTION=101
c
c     Purpose: To define normal and tangential stresses and possible
c              failure in a contact with tiebreak connection
c
c     Variables:
c
c     sig_n,sig_t   = normal and tangential stress (output)
c     disp_n,disp_t = normal and tangential displacement (input)
c     vel_n,vel_t   = normal and tangential relative velocity (input)
c     cn,ct         = normal and tangential stiffness (input)
c     uparm         = user defined tiebreak parameters (input)
c     uhis          = user defined tiebreak history variables
c                        (input/output)
c     ...
```

Miscellaneous Contact Enhancements

- New values for user friction element history (*USER_INTERFACE_FRICTION): material directions, relative velocity components and yield stress

- *CONTACT_AUTOMATIC_GENERAL with adaptivity supported in MPP

- *CONTACT_INTERIOR can be multiply defined, and there can be multiple part sets in each one

- *CONTACT_TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET can have normals updated incrementally through TIEDID=1 in SMP

- ...
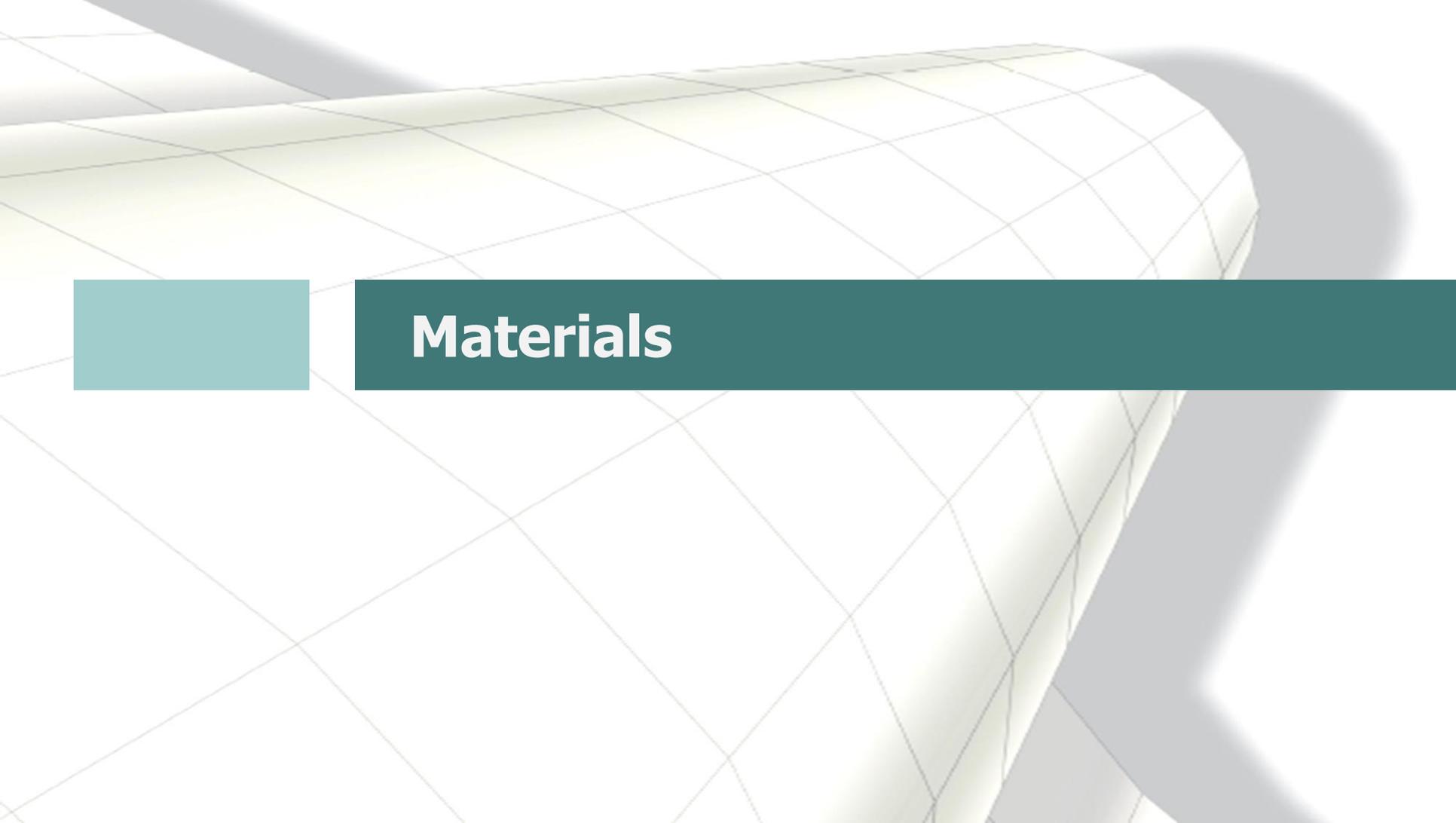
Materials

New micromechanical material model

- *MAT_4A_MICROMEC / *MAT_215
- Mainly for short-fiber reinforced plastics
- Distinguishes between fiber and matrix properties
  - Fiber: (transversal) isotropic elastic
  - Matrix: isotropic elasto-viscoplastic (like *MAT_024)
- Takes into account fiber orientation and density distribution
  - Information from injection molding simulation can be used directly
- Micromechanical approach based on Mori Tanaka Mean-Field Theory
- Damage and Failure model similar to DIEM
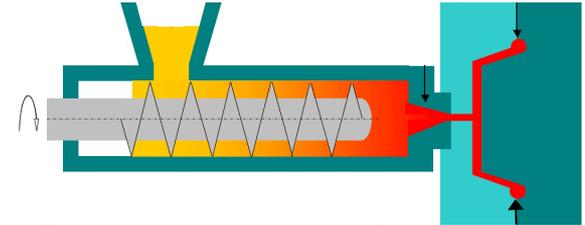
[www.genplastkenya.com]
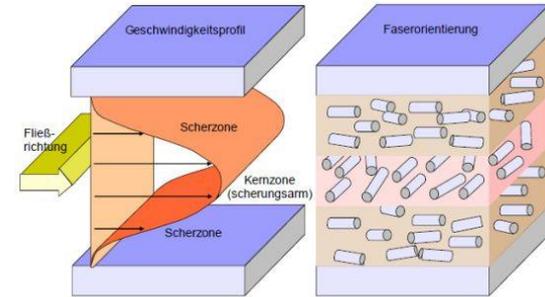
[wiki.polymerservice-merseburg.de]

Generalized damage model

- ***MAT_ADD_GENERALIZED_DAMAGE (MAGD)**
  - General damage model as add-on for other material models
  - Intention: non-isotropic damage
  - Up to 3 history variables as damage driving quantities
 - Very flexible due to input via *DEFINE_FUNCTIONs

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ 0 \\ \sigma_{12} \\ \sigma_{23} \\ \sigma_{31} \end{bmatrix} = \begin{bmatrix} D_{11} & D_{12} & 0 & D_{14} & 0 & 0 \\ D_{21} & D_{22} & 0 & D_{24} & 0 & 0 \\ 0 & 0 & D_{33} & 0 & 0 & 0 \\ D_{41} & D_{42} & 0 & D_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & D_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & D_{66} \end{bmatrix} \begin{bmatrix} \tilde{\sigma}_{11} \\ \tilde{\sigma}_{22} \\ 0 \\ \tilde{\sigma}_{12} \\ \tilde{\sigma}_{23} \\ \tilde{\sigma}_{31} \end{bmatrix}$$

- **New in R10**
  - Support of solid elements ------------>
    (only shells in R9)
  - Option for pre-defined damage tensors
    (lowers CPU cost)
  - Optional damage limitation curve
    (as function of triaxiality and Lode parameter)
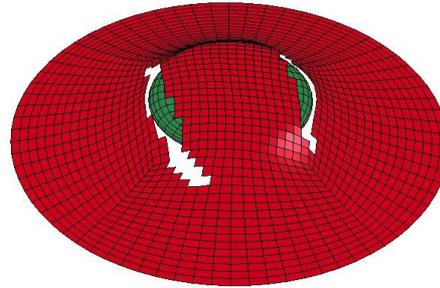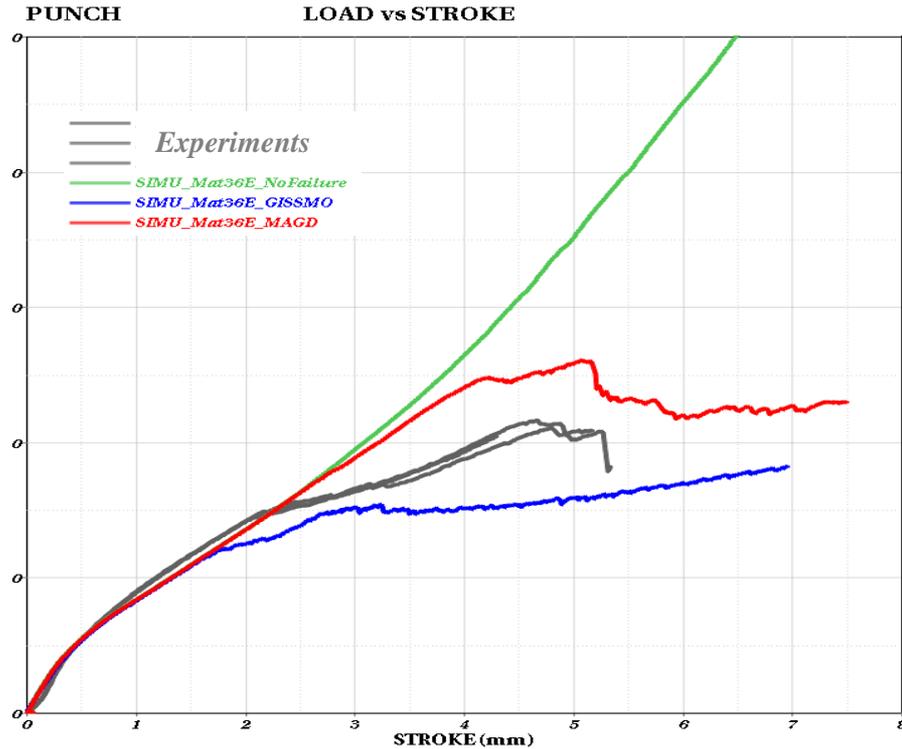

shell elements
type 16, NIP=5
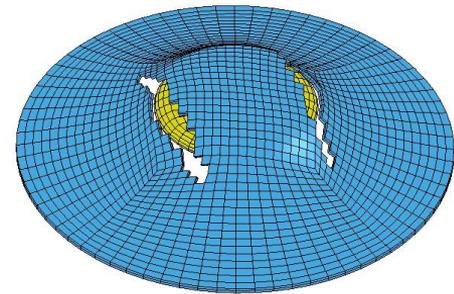
solid elements
type -1, 2 over thickness
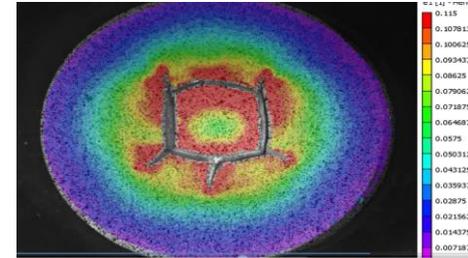
Extended Barlat Model

- New *MAT_EXTENDED_3-PARAMETER_BARLAT (*MAT_036E)
  - 10-inputs to define a single material for strength/stiffness:
    - 5 Strain Hardening load curves
    - 5 R-Value load curves (or constants)
  - Novel shear and biaxial R-value definitions
  - Yield surface adapted to fit more input data points, smoother interpolation

	σ^0	σ^{45}	σ^{90}	σ^s	σ^{bi}	R0	R45	R90	RSHR	RBIAX
MAT_036, HR=3	YES	X	X	X	X	YES	YES	YES	X	X
MAT_036, HR=7	YES	YES	YES	X	X	YES	YES	YES	X	X
MAT_036E	YES	YES	YES	YES	YES	YES	YES	YES	YES	YES
Test Mode	T_0	T_{45}	T_{90}	Shear	Biaxial	T_0	T_{45}	T_{90}	Shear	Biaxial

Example: MAT_036E + MAGD

PUNCH   **LOAD vs STROKE**

Experiments

SIMU_Mat36E_NoFailure
SIMU_Mat36E_GISSMO
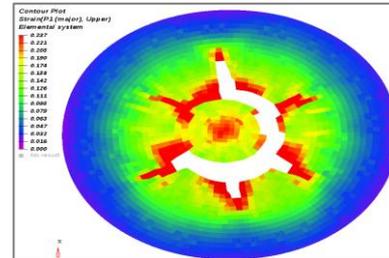SIMU_Mat36E_MAGD

STROKE (mm)
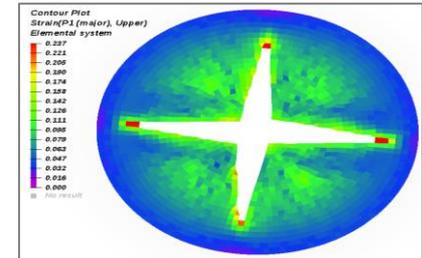
Biaxial punch test

Courtesy of Honda

MAGD   *Gissmo*

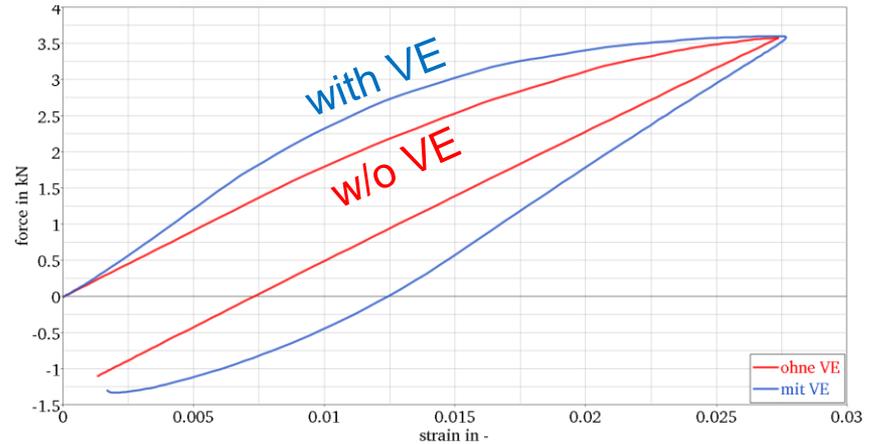# SAMP for Polymers

- ***MAT_SAMP-1**
  - Semi-Analytical Model for Polymers
  - Uses an isotropic C-1 smooth yield surface

- **New in R10**
  - Further speed-up of algorithm
 - Viscoelastic option allows definition of rate-dependent Young's modulus

$$\dot{\sigma}(t) = -\beta \cdot \sigma(t) + E(\dot{\varepsilon}(t)) \cdot \dot{\varepsilon}(t)$$

 - Presented by Paul Du Bois in Salzburg 2017

Glass

- *MAT_GLASS (*MAT_280)
  - Brittle smeared fixed crack model for shell elements (plane stress), added in version R9

- New in R10
  - Optional damage model for stress and stiffness softening
  - Possible distinction between tensile and compressive damage
  - Might be preferrable for compression dominated problems
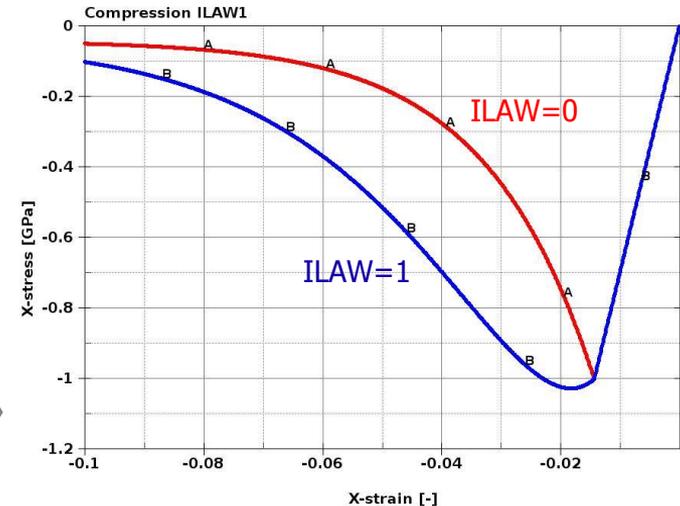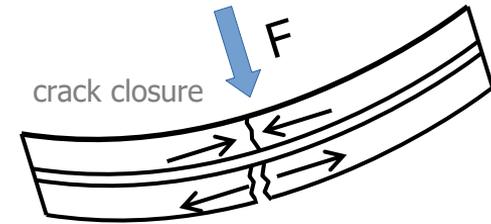
Stress-strain behavior for compressive loading and IMOD=1 (damage)

crack closure

F

Compression ILAW1

ILAW=0

ILAW=1

Spotwelds

- New options on *MAT_SPOTWELD available for DMGOPT=10,11,12

  - FMODE: Allows ductile or brittle shear failure by way of initiating damage (Damage initiated by the failure option only if EFAIL=0)

  - FFCAP: Failure function limit
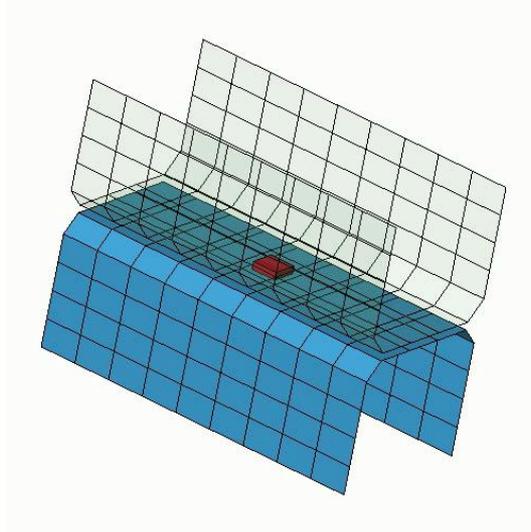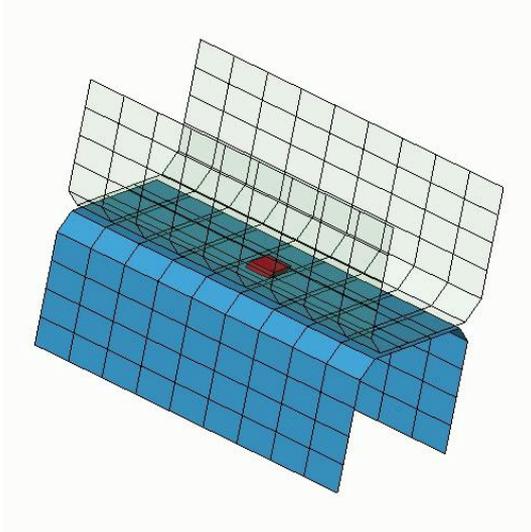
# Stochastic variations

- Keyword *DEFINE_STOCHASTIC_VARIATION
  - Allows random distribution of yield stress and/or failure strain for certain materials (24, 81, …)
  - New in R10: Option IRNG to choose between deterministic (pseudo-) random number generator (IRNG=0, default) and non-deterministic (true) random number generator (IRNG=1).

IRNG=0: same distribution in 4 different runs

IRNG=1: different distribution in 4 different runs
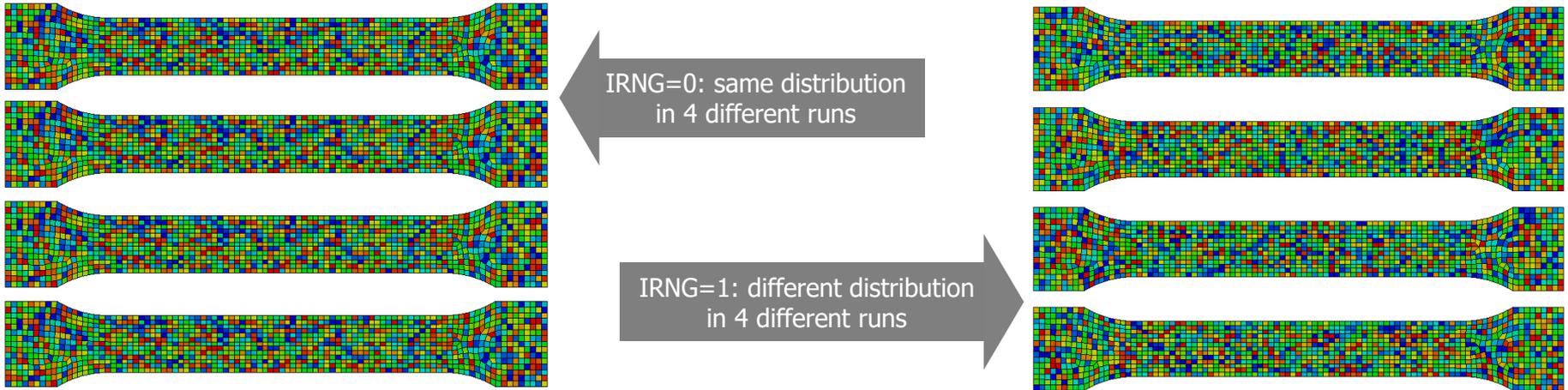
Miscellaneous Material Updates

- Material 54 (*MAT_ENHANCED_COMPOSITE_DAMAGE)
  - New transversal isotropic option for solid elements
  - Parameter TI=1 invokes equal treatment of
    all cutting planes around the axis of main fiber direction
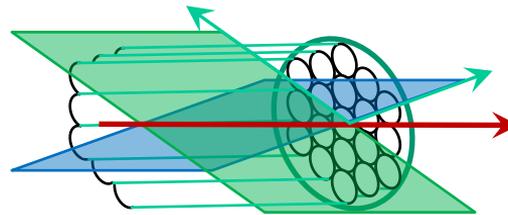
- Material 157 (*MAT_ANISOTROPIC_ELASTIC_PLASTIC)
  - Added orthotropic failure criteria (Tsai-Wu/Hill) with locally varying strength limits
  - Allow strain-rate dependent strength values using *DEFINE_CURVE

- Material 261/262 (*MAT_LAMINATED_FRACTURE_DAIMLER_PINHO/CAMANHO)
  - Fracture toughness values as function of characteristic element length and strain rate

Miscellaneous Material Updates

- Material 254 (*MAT_GENERALIZED_PHASE_CHANGE)
  - Can now be used with shell elements and thermal thick shells
- Add shell implementation to *MAT_277 (*MAT_ADHESIVE_CURING_VISCOELASTIC).
- Add *MAT_293 (*MAT_COMPRF)
  - Non-orthogonal material model for carbon fiber prepreg forming simulation
  - Available for shell elements
- Enable *MAT_ADD_COHESIVE to be used in implicit analysis
- *MAT_ADD_FATIGUE: Add new form of Basquin equation to define material's SN curve
- Add option to define element size dependent parameters EN and SN for *MAT_120
- Add optional in-plane failure strain to *MAT_169 (*MAT_ARUP_ADHESIVE)
- …
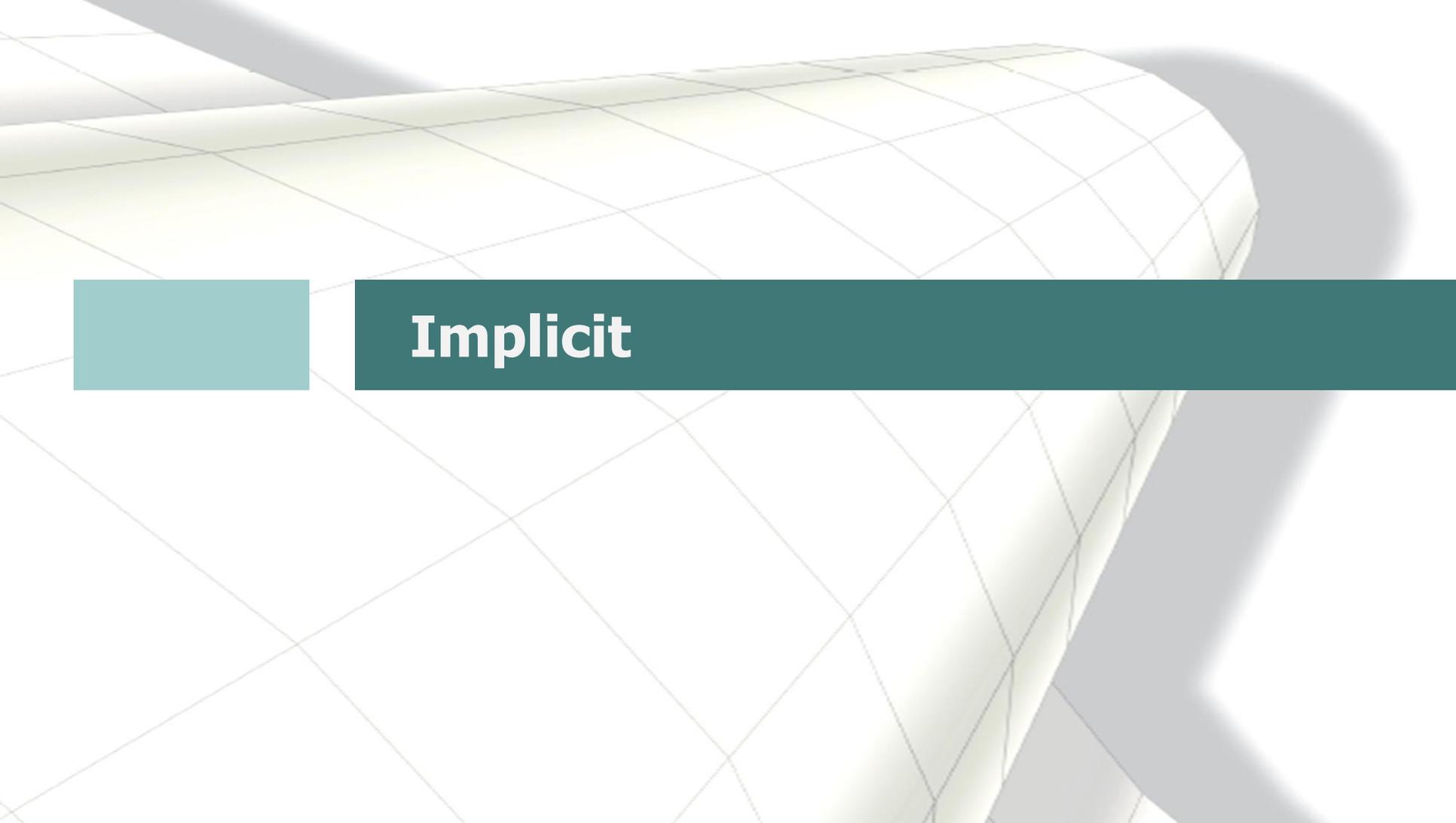
Implicit

Rigid Bodies with Prescribed Motion

■ Enforcing the motion of a rigid body into a stress free deformable structure provides a problem

■ Line search convergence based on relative measures breaks down when the initial values are exactly zero (or close thereto), and the forces from the initial contact are significant

■ As a result, the simulation never gets off the ground, the implicit solver simply does not allow the body to go in contact, and if it does then this is credited to luck and coincidence rather than anything else

■ As of R10, prescribed motion on rigid bodies induce a fictitious force that properly resolves this issue
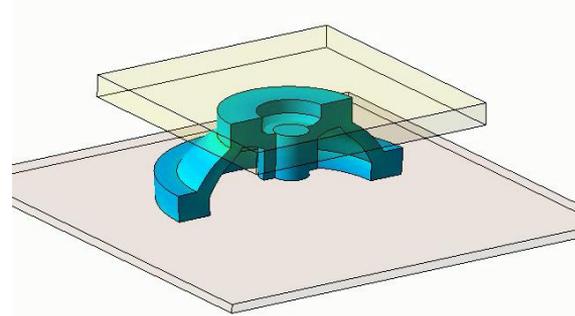
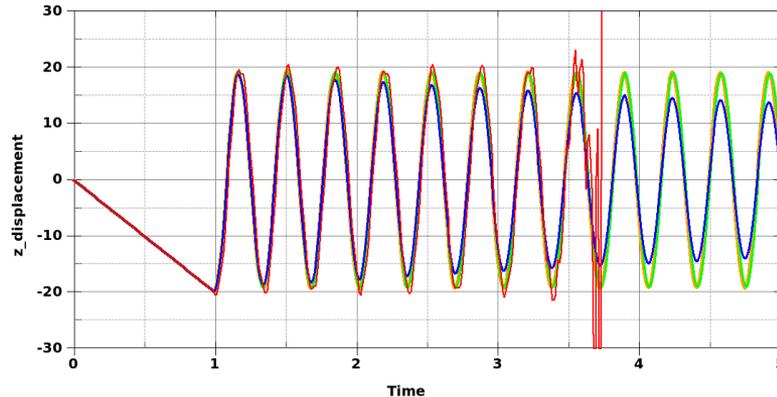# Strict Tolerance Option for Increased Accuracy

- Currently, DCTOL is the main important tolerance parameter for nonlinear implicit
  - Convergence is based on relative reduction of Euclidian norm of displacement vector
  - Default values will in most situations provide good results
- Experienced users with demand for high accuracy is benefitted from a more flexible input
  - DCTOL, RCTOL and ABSTOL can be used to tighten the tolerances

- New in R10: Maximum tolerances introduced as a mean to explicitly control the accuracy
  - DMTOL, EMTOL and RMTOL are relative measures in max norm
  - NTTOL, RTTOL, NRTOL and RRTOL are absolute measures in max norm
- Tolerance settings are found on *CONTROL_IMPLICIT_SOLUTION where DNORM<0 will activate the new ones

HHT Time Integration

- Newmark Time Integration is unstable for large or fast rotational motion
  - Bathe 3-point integration was introduced in R9
- ALPHA<0 on *CONTROL_IMPLICIT_DYNAMICS activates HHT (Hilber-Hughes-Taylor)
  - Now available in R10
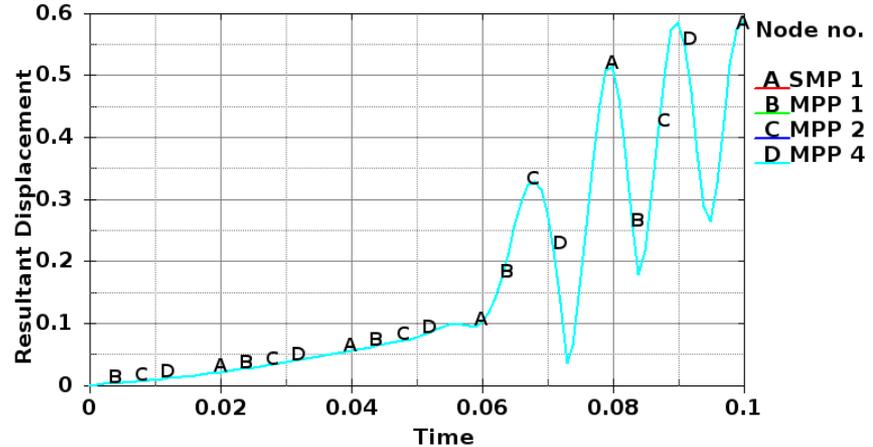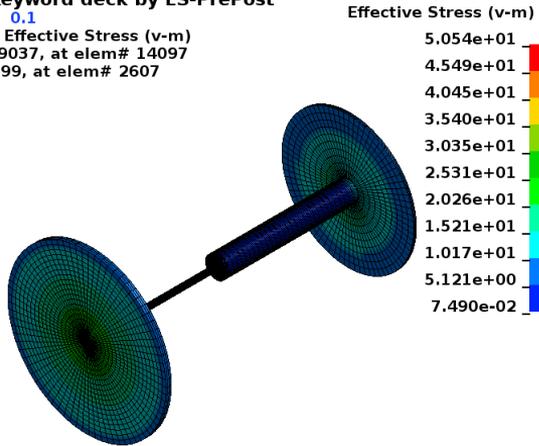 - Stabilizes the overall scheme

Default Newmark:
GAMMA = 0.5, BETA = 0.25
getting unstable

Damped Newmark:
GAMMA = 0.55, BETA = 0.30
considerable damping

Bathe: ALPHA = 0.5
HHT: ALPHA = -0.05
both stable and
energy conserving

Example: ruler under vibration

Rotational Dynamics Analysis

CPU #	SMP 1	MPP 1	MPP 2	MPP 4
Elapsed Time	49	42	24	15

■ MPP supported in R10

■ Rotation axis can be defined through *DEFINE_VECTOR_NODES to allow movement of the axis
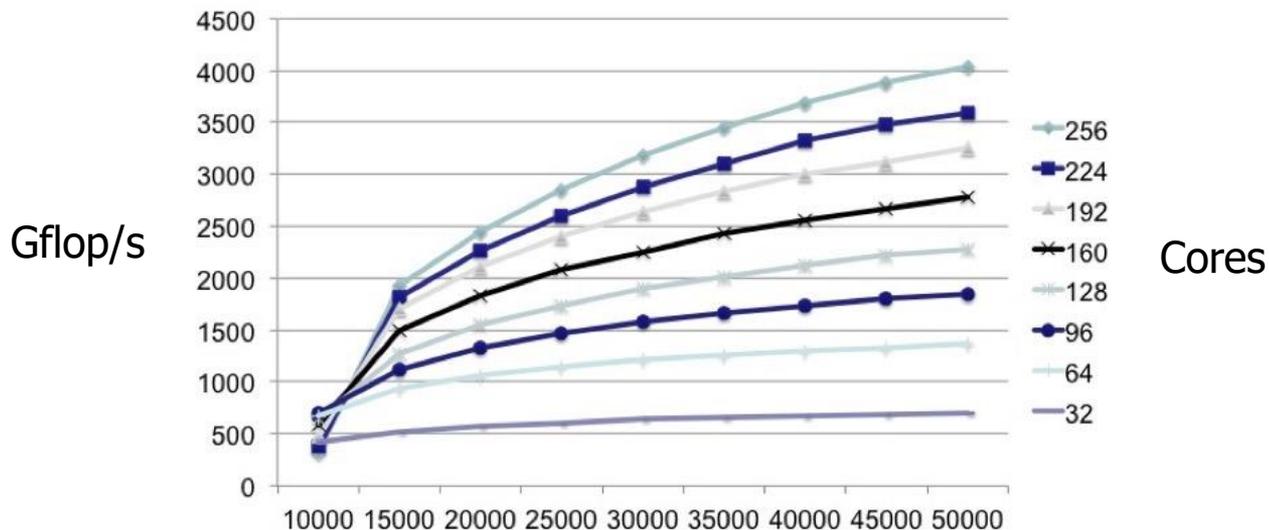
Miscellaneous Implicit Enhancements

- A d3plot from last converged step is always written
  - Regardless of d3plot output frequency
- Solid element type 13 supported for all materials in implicit
  - Nodal volume averaging treated in the element rather than averaging strains in the materials
- When ISTUPD>0, the thickness update is incorporated in the solution process to render force continuity between steps
  - Particularly important for forming with contact
- More feature support of implicit accuracy (IACC=1 on *CONTROL_ACCURACY)
  - High order shells, 1D seatbelts, Hughes-Liu and Spotweld beams
  - Joint stiffnesses

LS-DYNA Implicit Linear Solver

■ Solving large sparse linear systems is the bottleneck for implicit calculations

  ■ Multifrontal method is the default as it is robust and reliable

  ■ Super linear growth in operations dominates overall run time

  ■ Super linear growth in storage creates the peak memory requirement

■ LSTC is responding to this challenge in two ways

  ■ Increasing scalability to enable thousands for cores to be brought to bear

  ■ Research into reducing the storage and operations required

LS-DYNA Implicit Linear Solver

- New scalable arithmetic kernels enable LS-DYNA implicit calculations to exploit the power of thousands of processor cores.


*Size of simulated dense frontal matrices, measurements taken on Cray XC40*
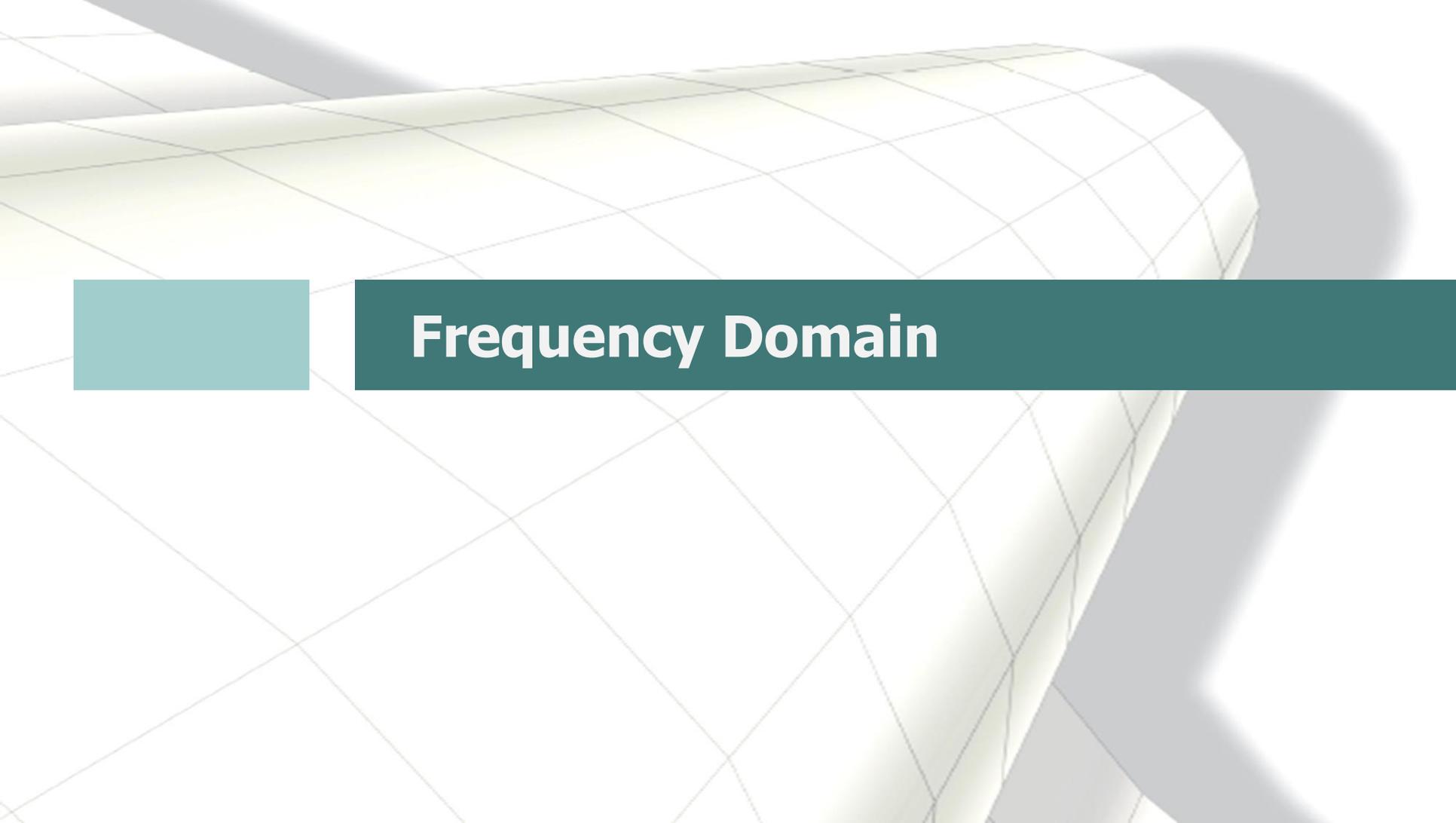
Frequency Domain

Miscellaneous Enhancements

- New global control keyword *CONTROL_FREQUENCY_DOMAIN
  - e.g. REFGEO (reference geometry) and MPN ("large mass method")
- New features for *FREQUENCY_DOMAIN_FRF/SSD
  - New loading types such as torque, large mass method, …
- New acoustic options: *FREQUENCY_DOMAIN_ACOUSTIC_BEM
  - Incident waves, fringe plot, output of acoustic intensity, …
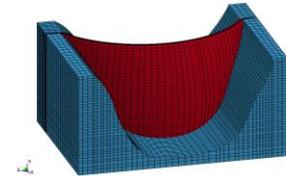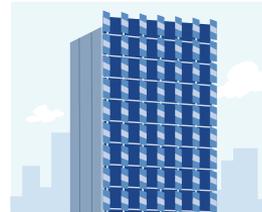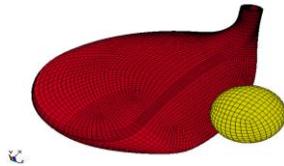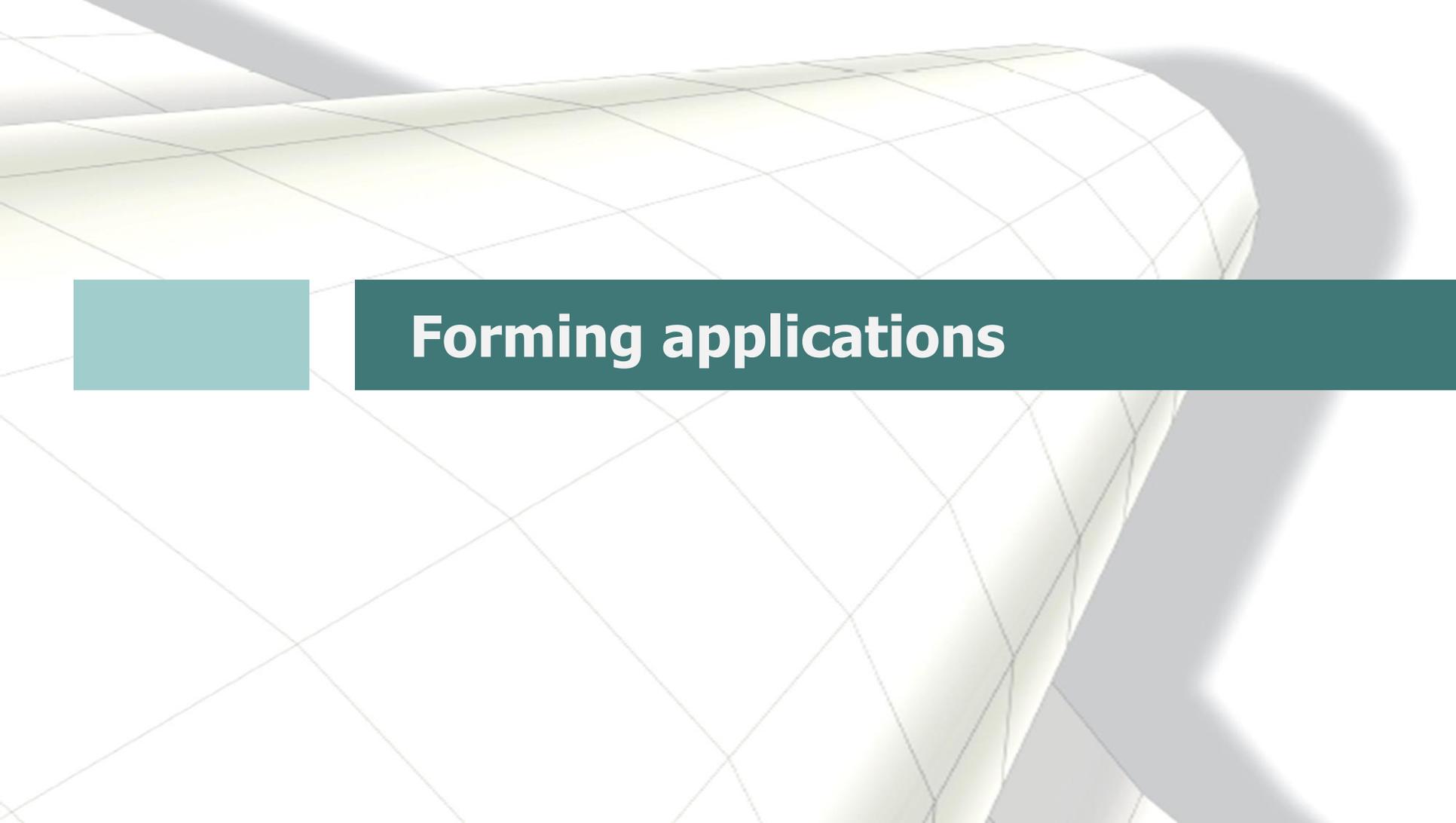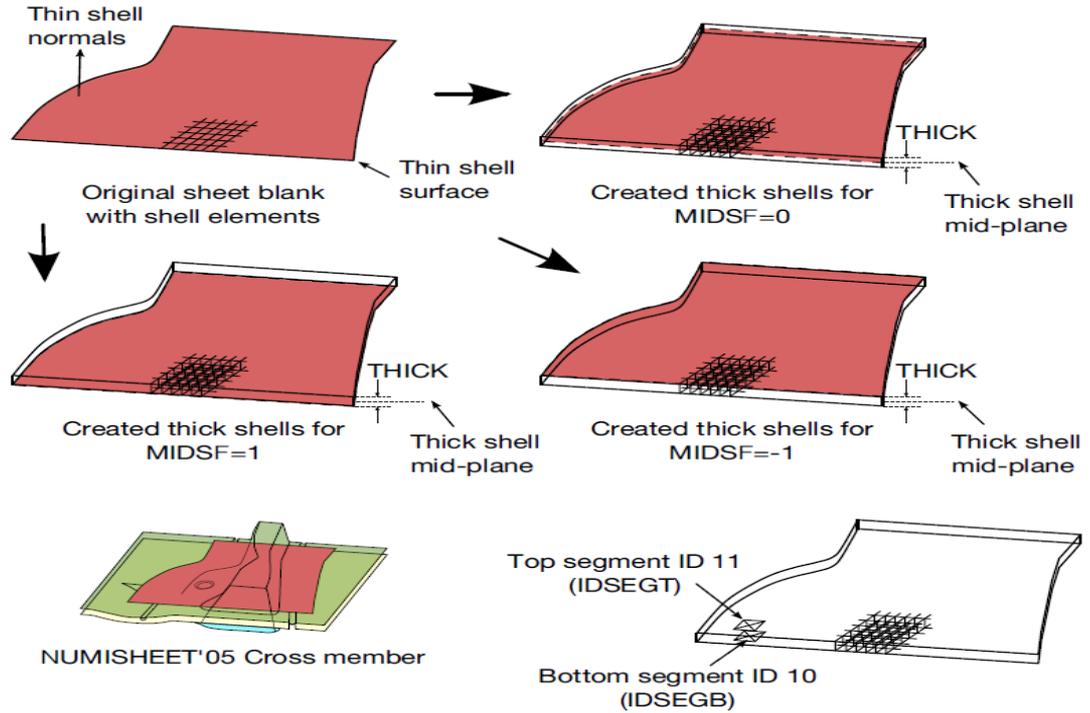
Forming applications

*CONTROL_FORMING_SHELL_TO_TSHELL

- Automatic conversion of shells to thick shells
- Specify
  - Part ID of thin shells
  - Thickness of thick shells
  - Mid surface location (MIDSF)
  - Top and bottom surface segment set IDs
- Supports adaptivity

Thin shell normals

Original sheet blank with shell elements — Thin shell surface

Created thick shells for MIDSF=0 — THICK — Thick shell mid-plane

Created thick shells for MIDSF=1 — THICK — Thick shell mid-plane

Created thick shells for MIDSF=-1 — THICK — Thick shell mid-plane

NUMISHEET'05 Cross member

Top segment ID 11 (IDSEGT)

Bottom segment ID 10 (IDSEGB)

One Step Forming

- New options for *CONTROL_FORMING_ONESTEP_...
  - TRIA: old default where all quadrilateral elements are split into two triangles
  - QUAD: new option to keep quadrilateral elements for better results
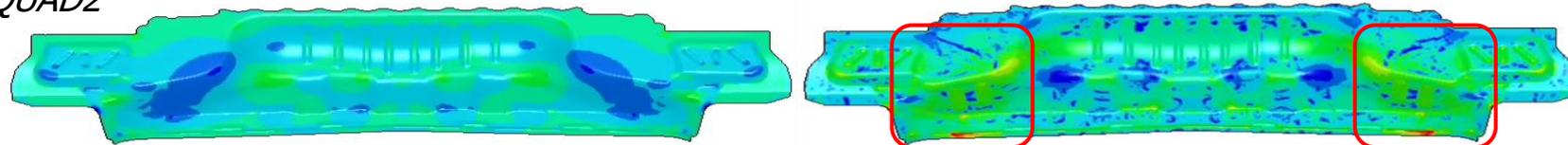 - QUAD2: similar to QUAD, but further improvement (new default in R10)

TRIA
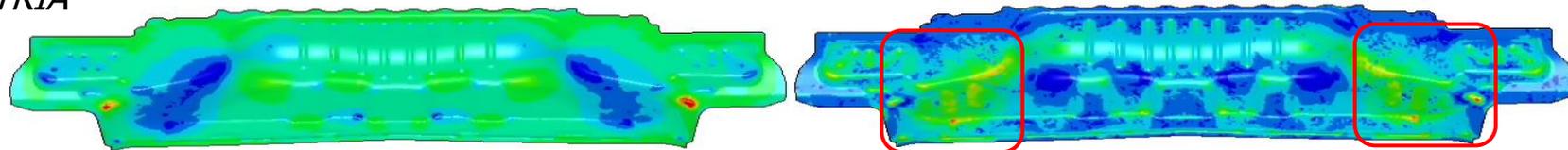
thinning

plastic strain

QUAD2

Improved Guide Pin Contact (SOFT=6)

- Purpose is to eliminate missing contact between blank edge and guide pins during gravity loading and forming

- New feature allows definition of blank PID as SSID, without the need to define a node set for the expected area of blank edge contact with the pin

- Available only for *CONTACT_FORMING_NODES_TO_SURFACE

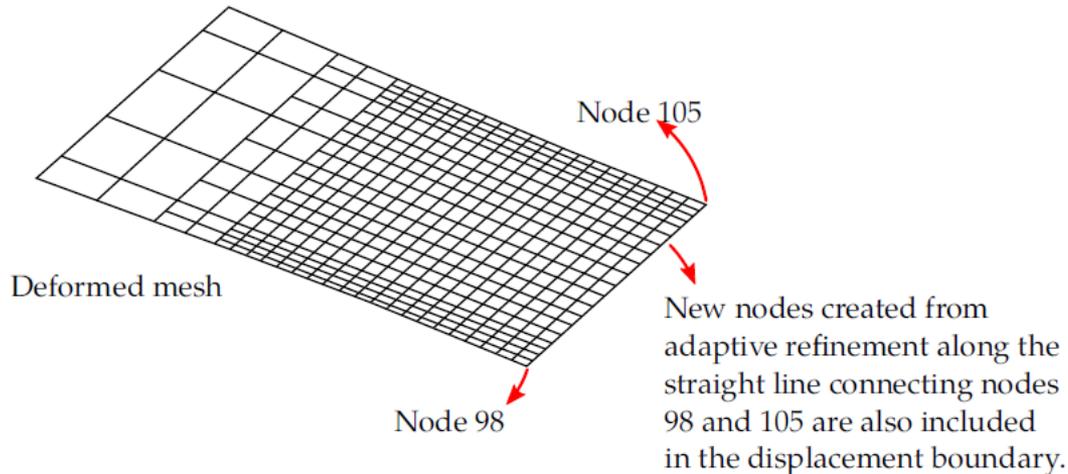Guide pin is very narrow, smaller than blank element edges

Blank edge mesh is very coarse
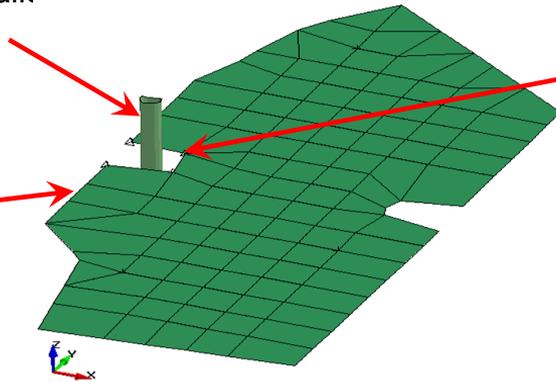
Previously a node set must be defined along the blank edge in the area of expected contact with the gage pin.

Boundary Conditions in h-adaptivity

- Add a new option SET_LINE to *BOUNDARY_PRESCRIBED_MOTION
- This option allows a node set to be generated including existing nodes and new nodes created from h-adaptive mesh refinement along the straight line connecting two specified nodes to be included in prescribed boundary conditions.

Node 105

Deformed mesh

Node 98

New nodes created from adaptive refinement along the straight line connecting nodes 98 and 105 are also included in the displacement boundary.

Miscellaneous Forming Updates

- Extend *INCLUDE_AUTO_OFFSET to solid and beam elements (draw beads)
- Add a new option VECTOR to *CONTROL_FORMING_BESTFIT to output deviation vector
- Add a new keyword *INTERFACE_WELDLINE_DEVELOPMENT
    - obtain initial weld line from the final part and the final weld line position
- Several enhancements for *ELEMENT_LANCING, e.g.
    - allow several parts to be cut during lancing
    - automatically merge nearly closed lancing curves
- Add a new keyword to perform trimming after lancing: *DEFINE_LANCE_SEED_POINT_COORDINATES
- Enhancements for sandwich parts (*CONTROL_ADAPTIVE, *DEFINE_CURVE_TRIM)
    - refine elements along trimming curve to make sure no slave nodes are cut by trimming curves
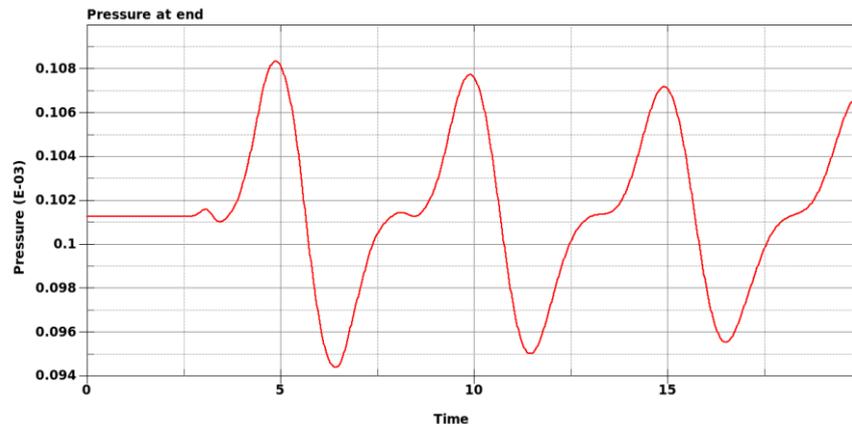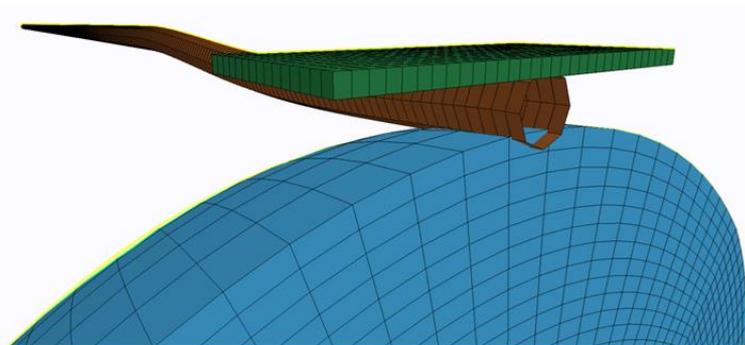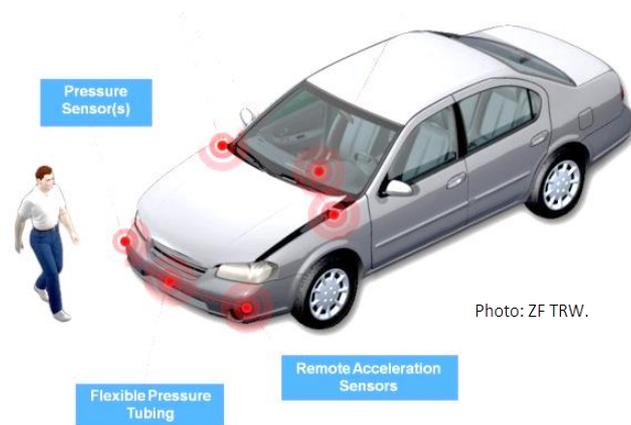   - allow mesh adaptivity, trimming of thick shells, and multi-layers of solids
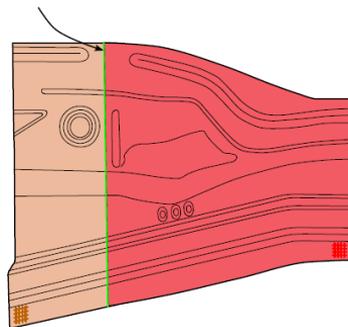
Miscellaneous

Pressure Tube

■ Models pressure waves in gas filled tube

- tubular beam elements, 1D acoustic solver
- pressure generated by area change (area given by contact penetration)
- independent of tube mechanics
- Keyword *DEFINE_PRESSURE_TUBE

Photo: ZF TRW.

Pressure Sensor(s)

Flexible Pressure Tubing

Remote Acceleration Sensors

Pressure at end

*AIRBAG_PARTICLE

■ Add new feature to allow user to define local coordinates of jetting of particles through internal vents

■ Support *SENSOR_CONTROL for CPM airbag

■ Allow solid parts in definition of internal part set

  ■ The solid volume will be excluded from the airbag volume

■ Allow additional internal part set for shells

  ■ The shell part should form a closed volume and its volume will be excluded from the airbag volume

Inflator With diffuser
Time = 4.9995

10 deg

30 deg
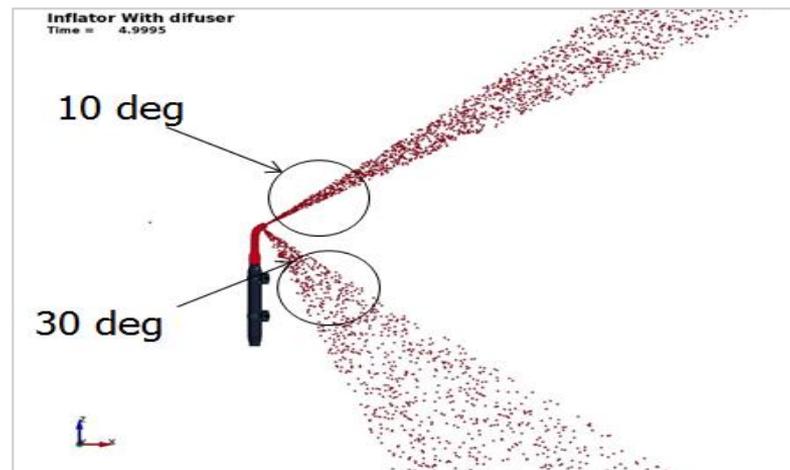
*PARTICLE_BLAST

- Consider erosion of shell elements
- Support interface force file output for gas particle-structure coupling
- Add option to define detonation point using a node (e.g. for moving bomb)
- Add interface force file output
- Add built-in smoothing function for particle structure interaction
- Add support for solids when modeling irregular shaped charge with HECTYPE=0/1
- Consider reflecting plane as infinite

LS-DYNA keyword deck by LS-PrePost
Time = 0.0020598
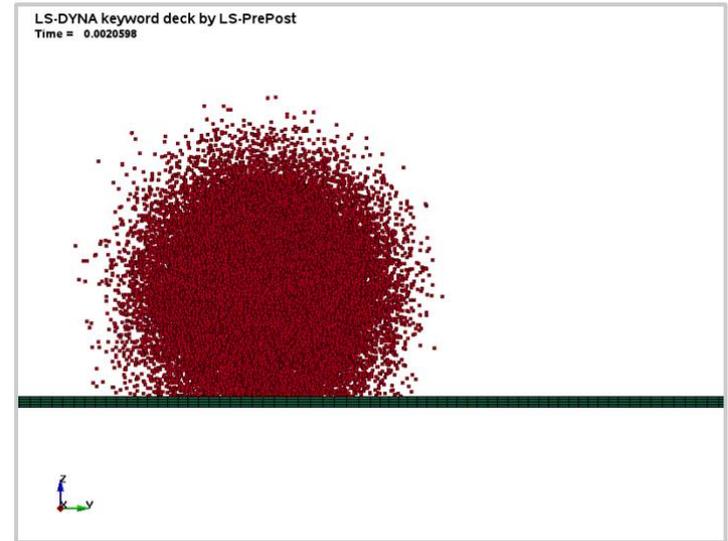
Miscellaneous

- The SPR models (*CONSTRAINED_SPR2, *CONSTRAINED_INTERPOLATION_SPOTWELD) now support the SPOTDEL option of *CONTROL_CONTACT
  - That means if shell elements involved in the SPR domain fail, the SPR gets deactivated.
- Enable *PART_STACKED_ELEMENTS to pile up shell element layers.
  - Before, it was necessary that solid element layers were placed between shell element layers. Now, shell element layers can follow each other directly.
- Allow *PART_STACKED_ELEMENTS to be used in adaptive refinement simulations
- Add variable IZSHEAR in *INITIAL_STRESS_SECTION to initialize shear stress
- Add _SET option to *INITIAL_STRESS_SOLID for element sets
- Added new option *INTERFACE_SPRINGBACK_EXCLUDE
 - To exclude selected portions from the generated dynain file

Meshfree methods: SPH

Pure Thermal Coupling between SPH and Solid

- Updated temperature transfer when converting solid elements into SPH particles

- New pure thermal coupling between SPH part and solid parts

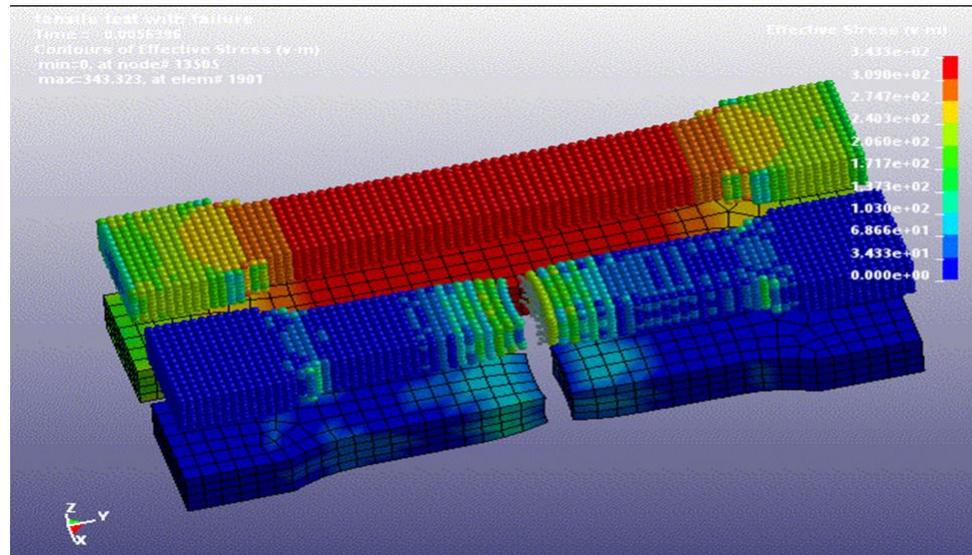- Added a thermal coupling conductivity parameter

Example:
High velocity impact with solid plate

Heat from contact friction and the conversion of mechanical work into heat
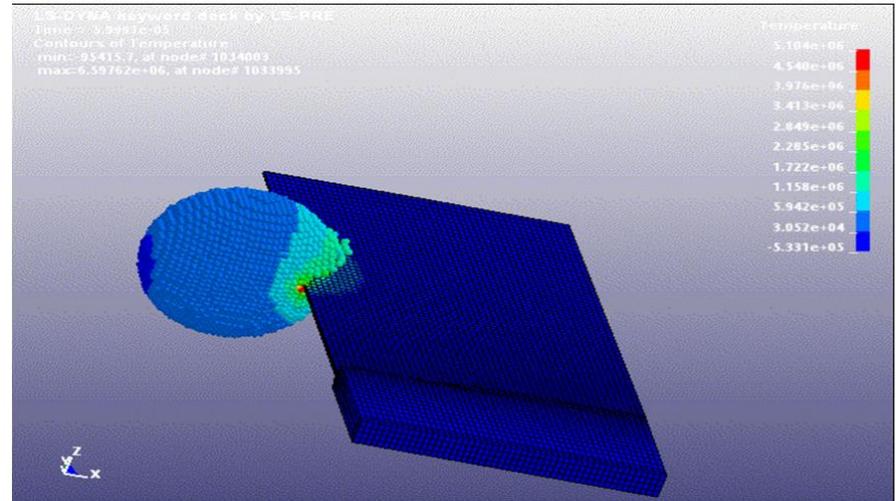
Failure with SPH

- Enable real deactivation of SPH particles via IERODE=2 on *CONTROL_SPH
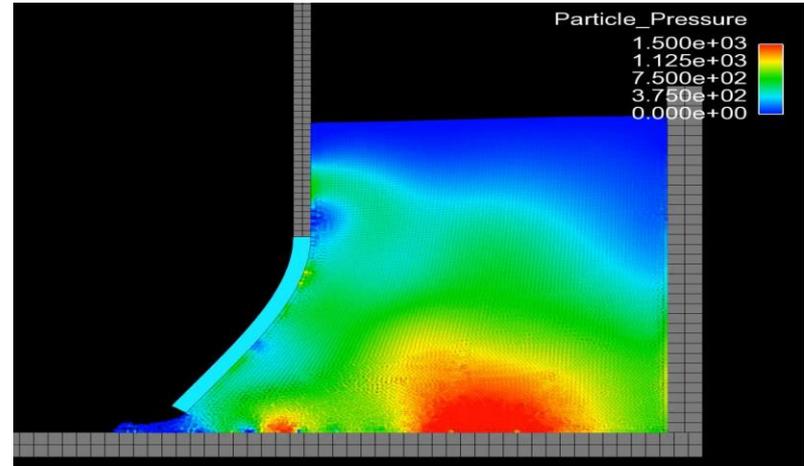- Add support for *MAT_ADD_EROSION, including GISSMO and DIEM

SPH Pressure Smoothing

■ Enhanced fluid flow formulations (FORMs 15/16) with pressure smoothing

  ■ Density smoothing by a kernel results in a smoother pressure field

 ■ Density smoothing is now material sensitive
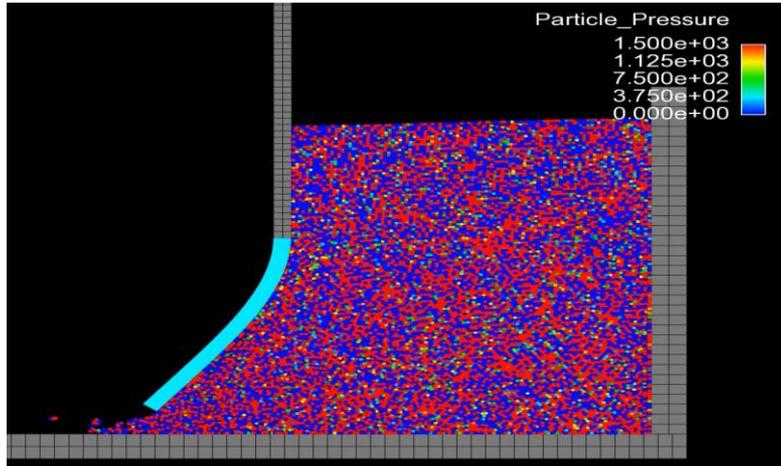
IFORM=0

IFORM=15

Miscellaneous SPH Updates

- ITHK on *CONTROL_SPH gives SPH particles a thickness to be employed by contacts
  - Affects the automatic and 2D nodes to surface contacts
  - Activated only if SST or OFFD are set to zero on the contact cards
- SOFT=1 on 2D nodes to surface contact and SPH to SPH coupling supported
  - PEN=0.1 default
- Non reflecting boundary conditions for SPH through *BOUNDARY_SPH_NON_REFLECTING
- Improved total Lagrangian forms 7 and 8 to scale with increased NCPUs
- Support formulations 0 and 1 for axisymmetric simulations
- Improved tracer particles output
- Added parameter OFFD to *CONTACT_2D_NODE_TO_SOLID to specify contact offset
- Added Equation of State 19 (*EOS_MURNAGHAN) for use with forms 15 and 16
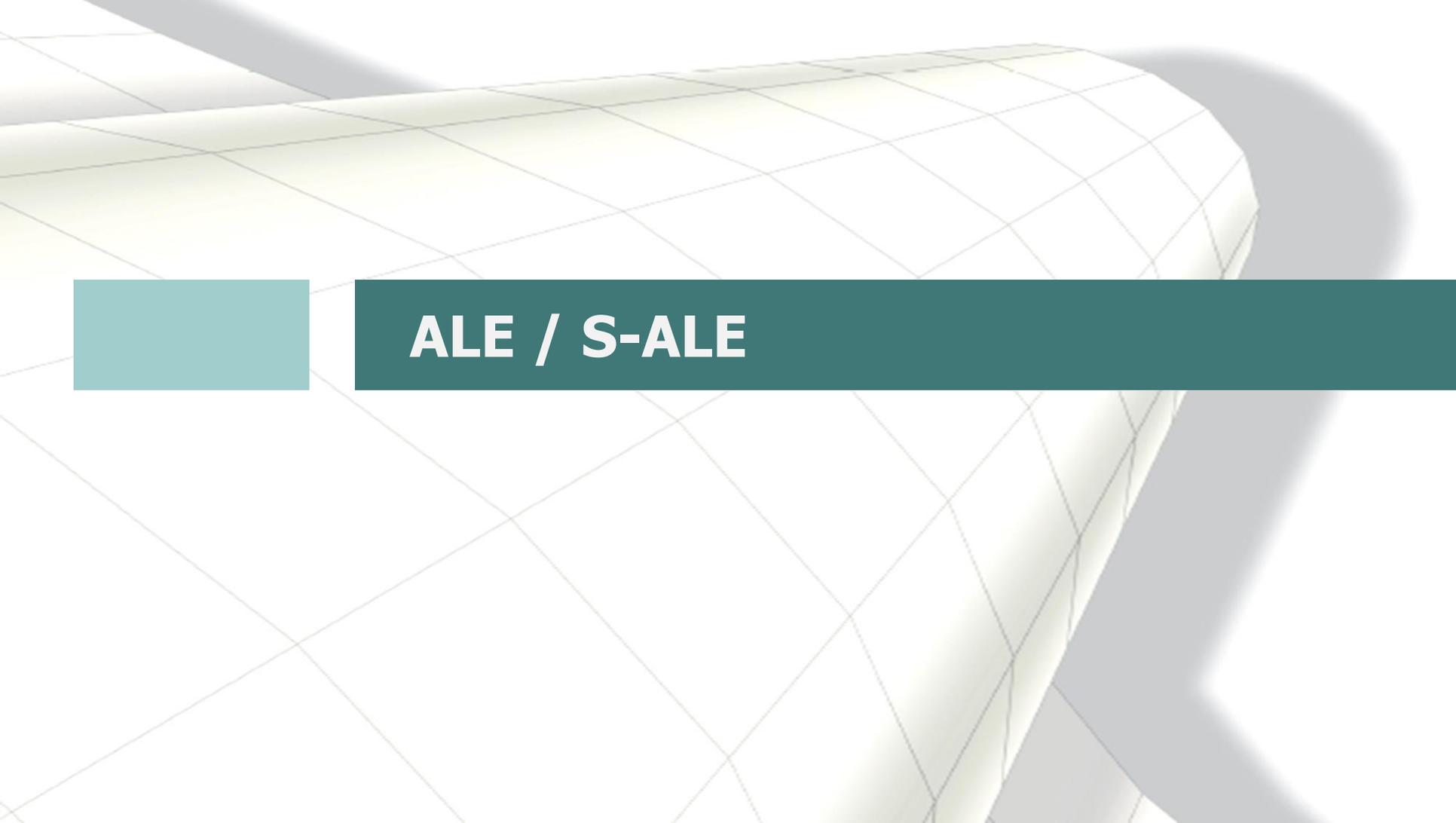
# ALE / S-ALE

ALE to S-ALE Converter

- Purpose is to automatically convert existing ALE model using rectilinear mesh to S-ALE
- ALE mesh info extracted, tilted mesh supported
- All ALE related input cards converted, no human intervene needed
- To use, insert an empty *ALE_STRUCTURED_MESH keyword into the existing ALE model
- CPIDX = -1 generates the cards and runs S-ALE solver

*ALE_STRUCTURED_MESH					
MSHID	PID	NBID	EBID		
1	optional	optional	optional		
CPIDX	CPIDY	CPIDZ	NID0	LCSID	
-1					

Miscellaneous ALE Updates

■ *LOAD_BLAST_SEGMENT: Automatically generate the ALE ambient elements attached to a segment or segment set.

■ *BOUNDARY_AMBIENT_EOS: implement *DEFINE_CURVE_FUNCTION for the internal energy and relative volume curves.

■ *ALE_FSI_TO_LOAD_NODE: Implement a mapping of the FSI accelerations (penalty forces/masses) computed by *CONSTRAINED_LAGRANGE_IN_SOLID (ctype=4) between different meshes.

■ *DATABASE_ALE: Add VAR=15 to output the kinetic energy by elements and ALE groups.

■ Structured ALE (S-ALE) solver:
   ■ S-ALE progressive mesh generation via RATIO in *ALE_STRUCTURED_MESH_CONTROL_POINTS.
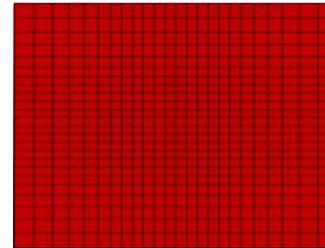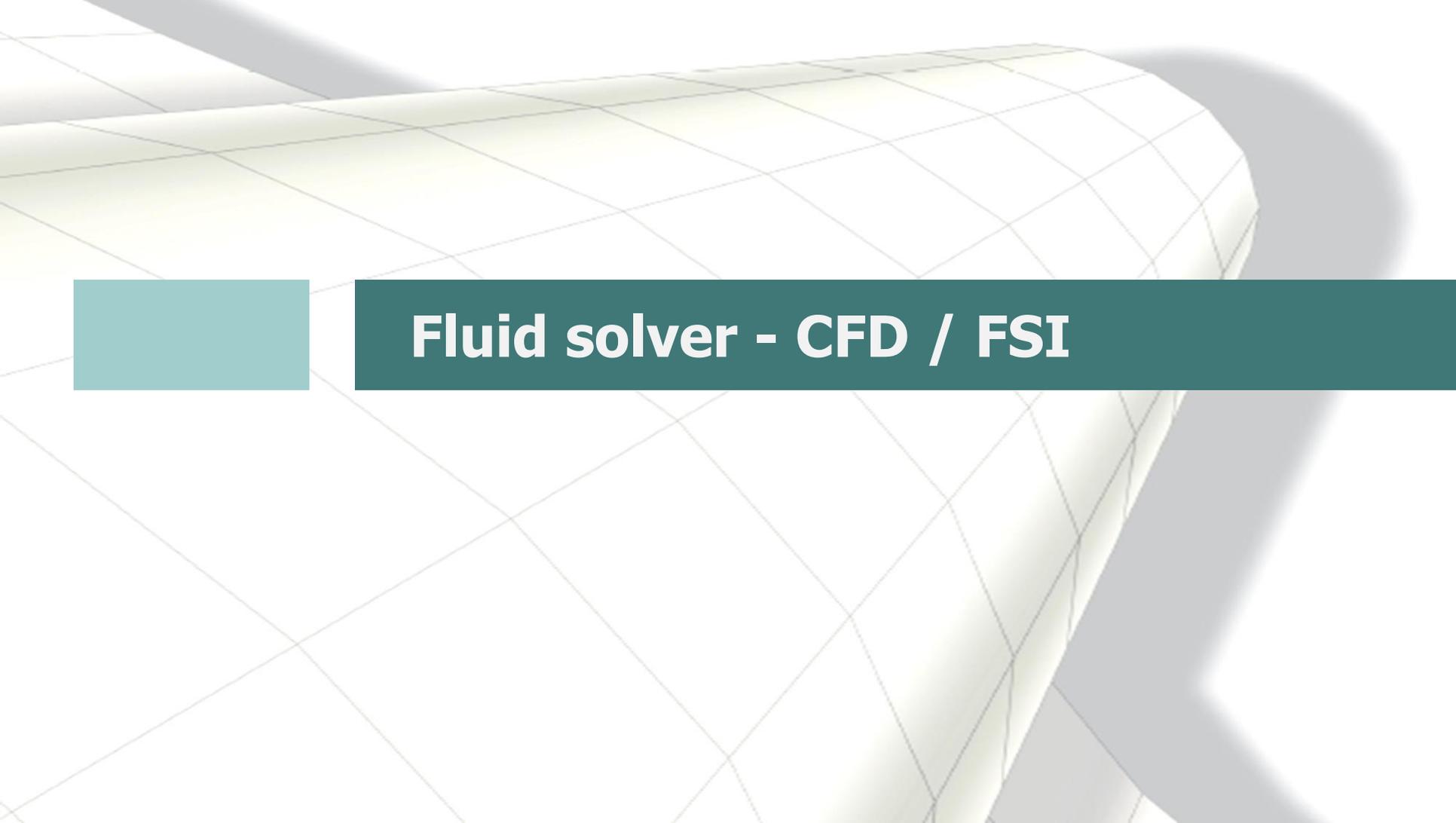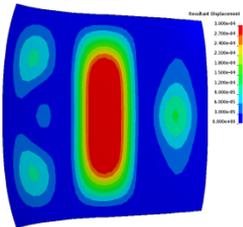
Fluid solver - CFD / FSI
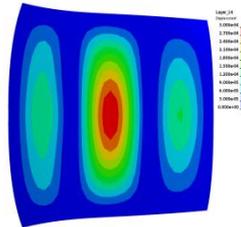
From full FSI to pure mechanical

- Fluid-Structure Interaction: three different options to solve the same problem

  - Solve Full Navier–Stokes with FSI non linear coupling. (1)

  - Solve Potential flow with a non-linear step at the end. (2)

  - Solve the structural analysis alone using the output from Navier-Stokes and the *LOAD_SEGMENT automatically generated input deck.
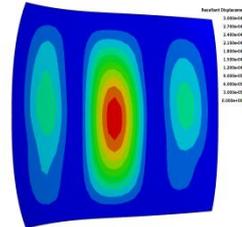   Use new *ICFD_DATABASE_DRAG to write the files. (3)
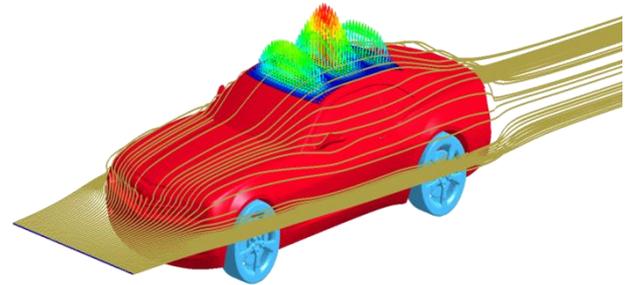
Roof vibration analysis

(1): 24 hours

(2): 20 minutes

(3): 2 seconds

From full FSI to pure thermal

■ Conjugate heat radiation: three different options to solve the same problem

■ Solve Full Navier–Stokes with thermal non linear coupling. (1)

■ Solve Navier Stokes using the steady state or potential flow solver and continue with conjugate heat transfer analysis once steady state has been reached. (2)

■ Solve the thermal analysis alone using the output from Navier-Stokes and the *BOUNDARY_CONVECTION_SET automatically generated input deck.
Use new *ICFD_DATABASE_HTC to write the files. (3)
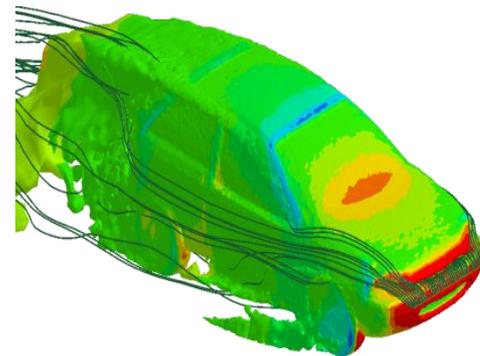

Engine heat radiation during drive
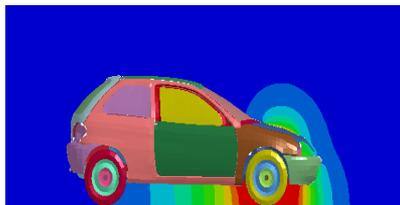
(1): 48 hours

(2): 2 hours

Miscellaneous ICFD updates

- Added Steady state solver
  - See *ICFD_CONTROL_GENERAL and ICFD_CONTROL_STEADY
- Added wave damping capabilities
  - See *ICFD_DEFINE_WAVE_DAMPING
- Added Windkessel boundary conditions for blood flow
  - See *ICFD_BOUNDARY_WINDKESSEL
- Two way coupling with DEM particles
- Option to shut off Navier Stokes solve after a certain time for conjugate heat transfer analysis
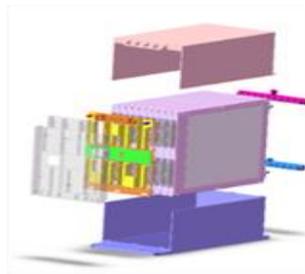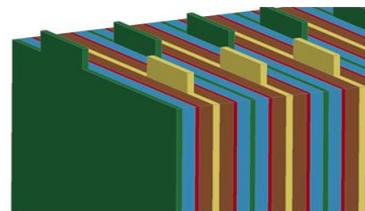 - See *ICFD_CONTROL_TIME
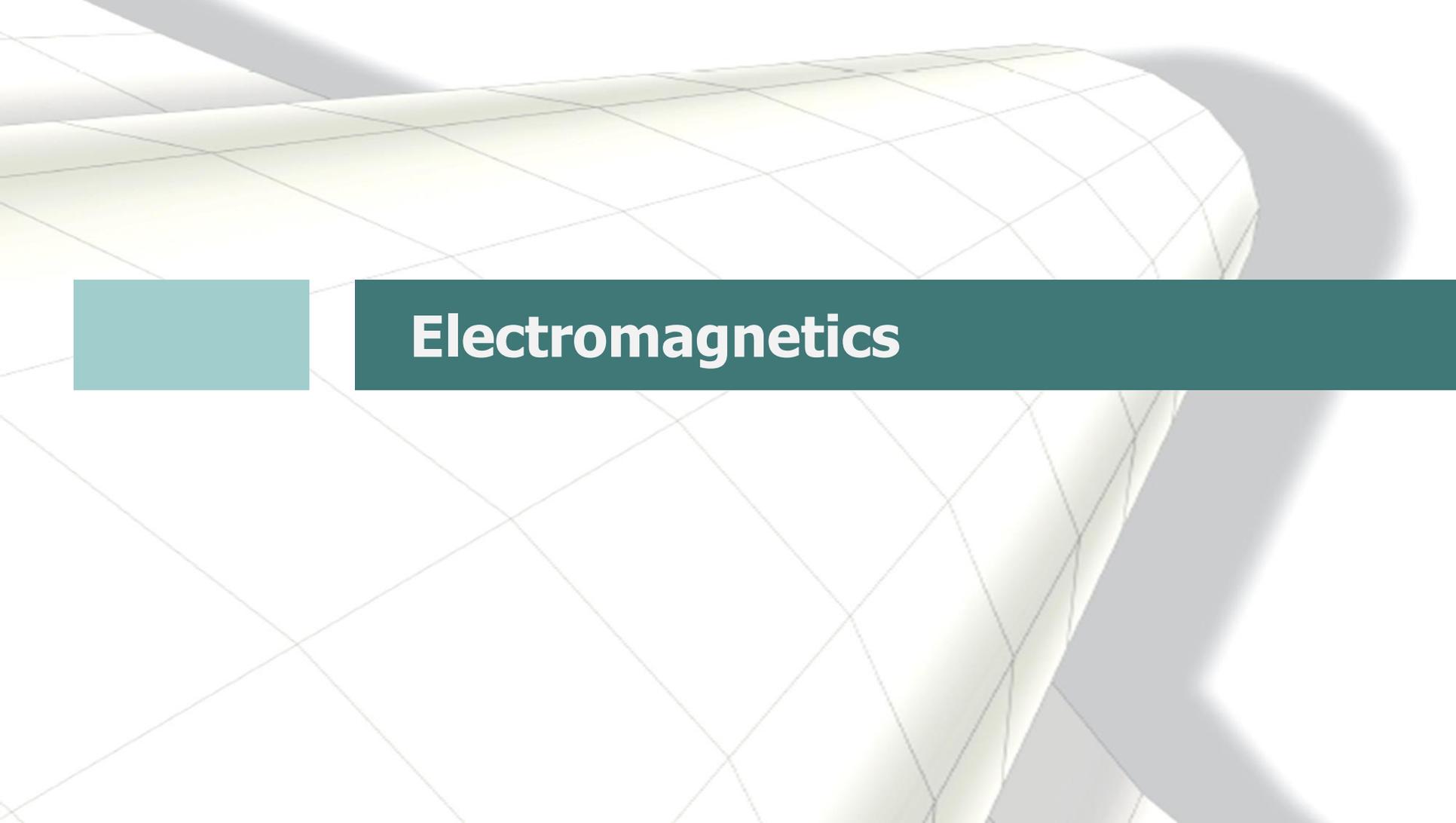
Electromagnetics

Battery modeling

- Randles circuit model for Li-Ion cells and modules
- Internal and external short circuits
- Cells can be modeled using solid or thick shell elements
- New Battery Packaging application in LS-Prepost 4.5 for easy model setup
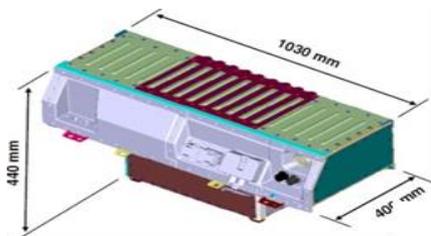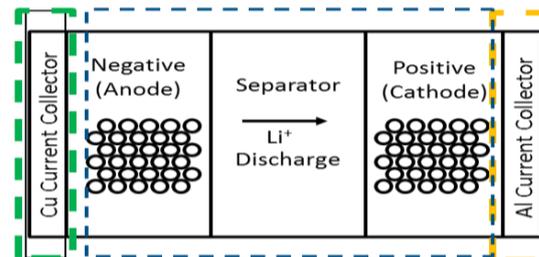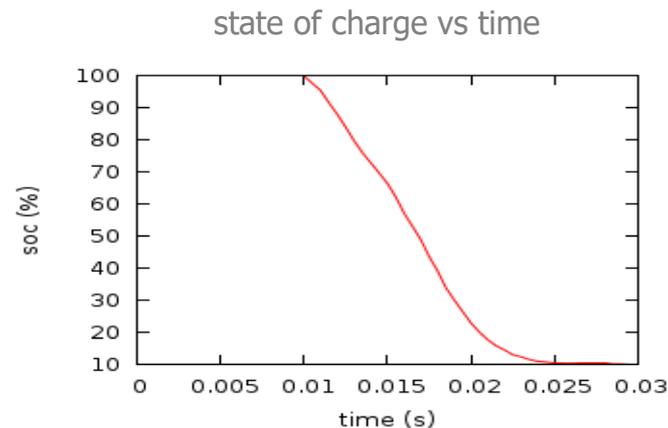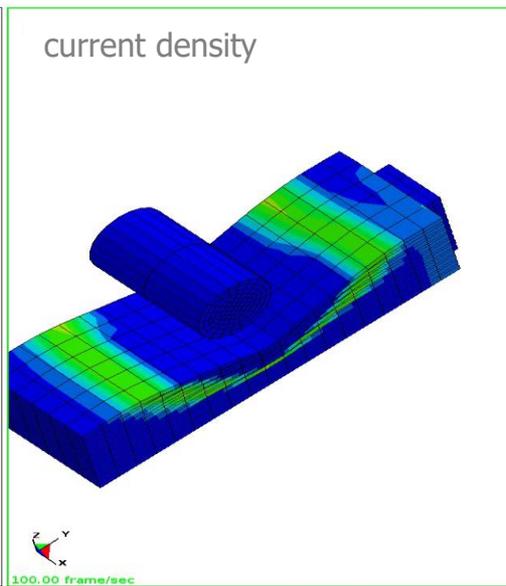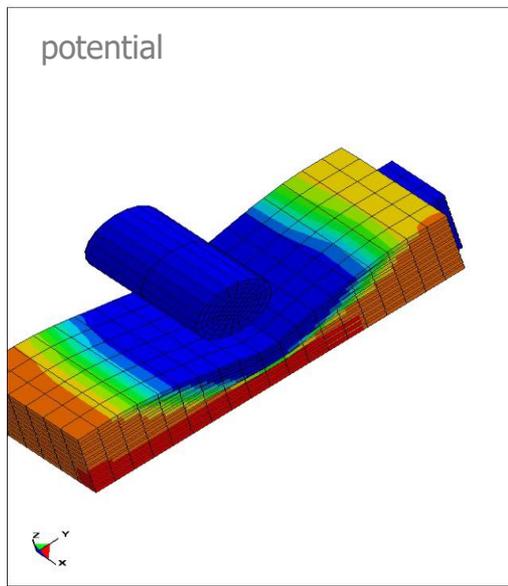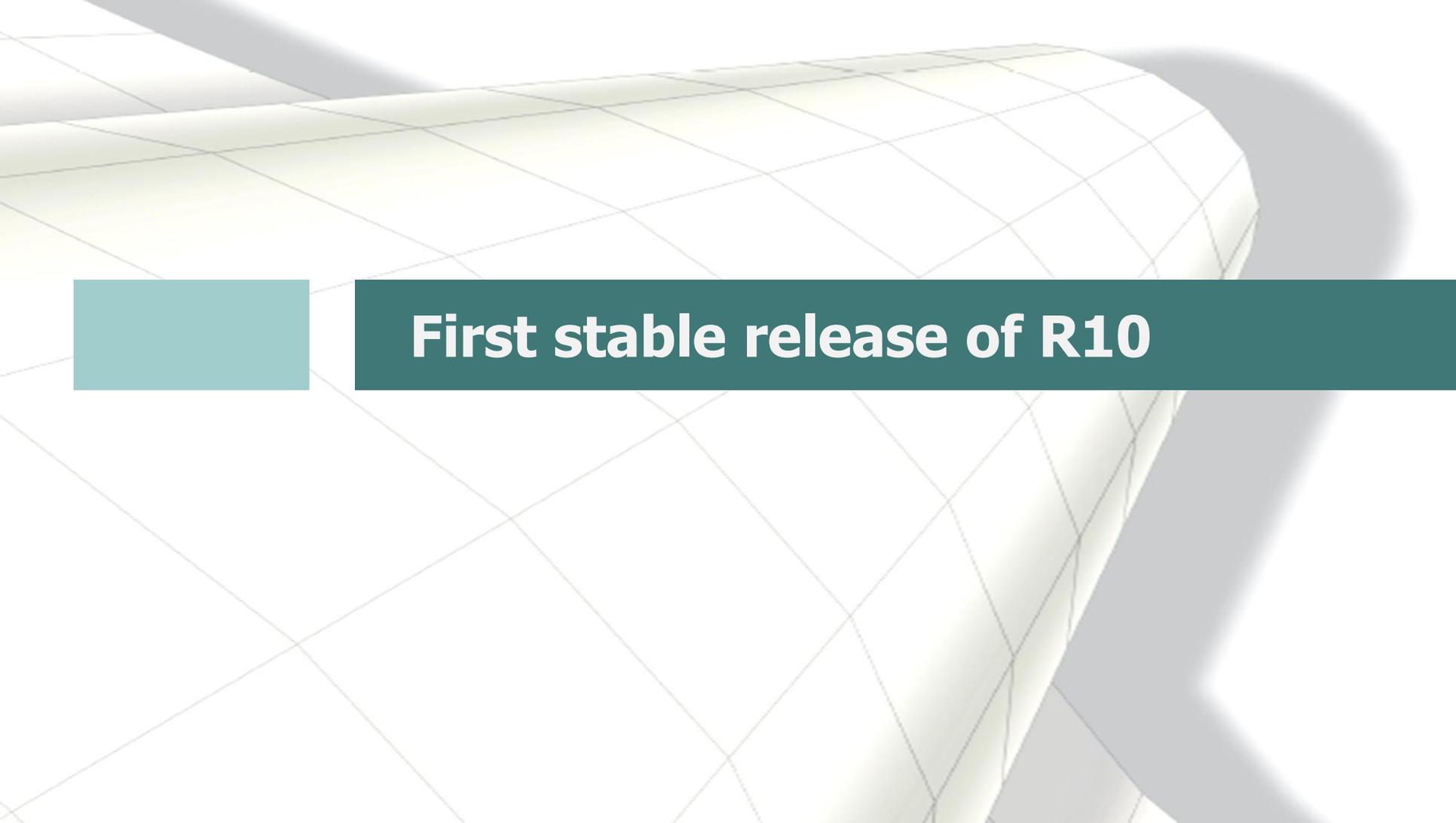

pack

module

cell

unit cell

Battery modeling

- Example: 10 cells module crushed by a cylinder using Composite Tshells
- Internal short circuit leads to discharge

First stable release of R10

Bug fixes

- A wide range of code corrections, inter alia,
    - Fixed rcforc output for MPP 2D automatic contact
    - Fixed force transducers (SOFT=2) when segments are involved with multiple, 2-surface force transducers
    - Fixed MPP SOFT=2 contact for implicit solutions
    - Fixed the default hourglass control for shell form 16, it was defaulting to type 5 hourglass control rather than 8
    - Correct the strain calculation for tet formulation 13, this did not affect the stress response, only output of strains
    - Enhanced termination process when the implicit solver deterimined an early termination
    - Remove *LOAD_SEGMENT_FILE file size limit (it used to be 200M)
    - Fix incorrect operation of TDEL for materials 81, 82, and 124
    - Enable *MAT_ADD_EROSION to be safely used with material models that have more than 119 history variables, e.g., *MAT_157
    - *DEFINE_HAZ_PROPERTIES: Distance of shell from the weld center is treated consistently under MPP and the shell material's yield stress is scaled properly
    - et cetera ...

... find more in release notes:

http://www.dynasupport.com/news/ ls-dyna-r10.0.0-r10.118302-released

Conclusion: LS-DYNA R10.0

- Newest release contains variety of new features
- Recommended for multiphysics and implicit analyses or if new options are needed
- R10 Keyword User's Manual can be downloaded from www.dynamore.de/en/downloads/manuals
- This presentation will be available on www.dynamore.de soon