LS-DYNA:
Status and Development Plan

John Hallquist, Yun Huang, Iñaki Çaldichoury, Jason Wang

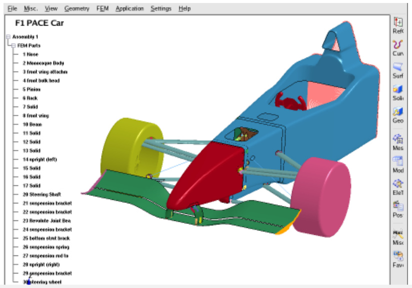
13th International LS-DYNA User's Conference
June 8-10, 2014

LSTC
Livermore Software
Technology Corp.

Outline

- Introduction
- Developments
  - Recent enhancements – John Hallquist
  - Linear solver – Yun Huang
  - LS-PrePost: ICFD & EM – Iñaki Çaldichoury
  - Particle methods – Jason Wang
- Conclusions

LSTC Products



LS-PrePost

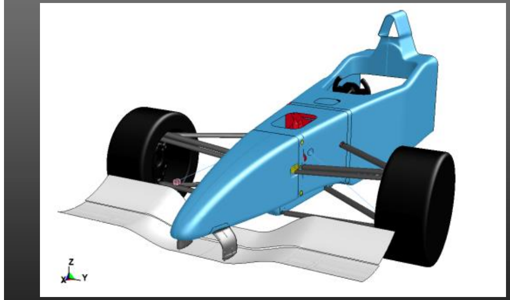
Surface

2D Interpolator

Accuracy

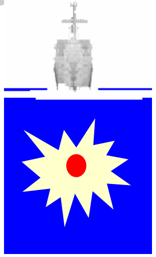
Sensitivity

LS-OPT/LS-TaSC



LS-DYNA

Dummies & Barriers



USA

No additional license cost

LS-DYNA Application Areas

Development costs are spread across many industries

Automotive

- Crash and safety
- NVH
- Durability
- FSI

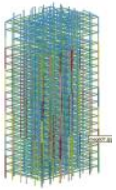
Aerospace

- Bird strike
- Containment
- Crash

Manufacturing

- Stamping
- Forging

Consumer Products

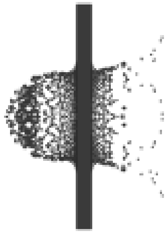


Structural

- Earthquake safety
- Concrete structures
- Homeland security

Electronics

- Drop analysis
- Package analysis
- Thermal



Defense

- Weapons design
- Blast response
- Penetration
- Underwater Shock Analysis

LS-DYNA - One Code, One Price, Strategy

“Combine the multi-physics capabilities into one parallel scalable code for solving highly nonlinear transient problems to enable the solution of coupled multi-physics, multi-scale, and multi-stage problems ”

Explicit/Implicit ✓

Heat Transfer ✓

ALE & Mesh Free ✓
i.e., EFG, SPH, Airbag Particle

User Interface ✓
Elements, Materials, Loads

Acoustics, Frequency Response, Modal Methods ✓

Discrete Element Method ✓

LS-DYNA R7 release includes

Incompressible Fluids ✓

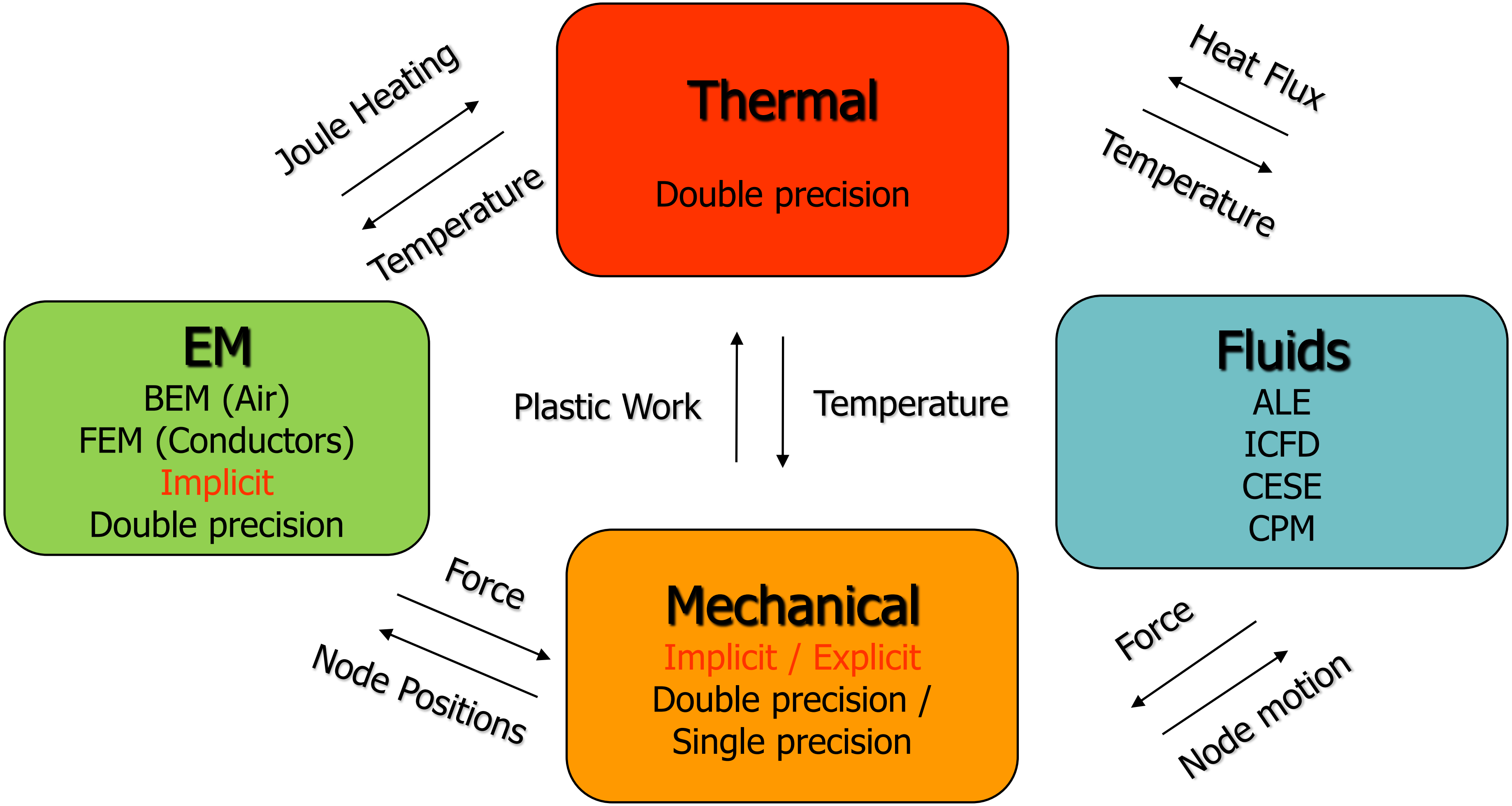
CESE Compressible Fluids ✓

Electromagnetics ✓

Future: Control systems

Accommodates Coupled Simulations

Multiple field equations are strongly coupled



Thermal

Double precision

EM

BEM (Air)

FEM (Conductors)

Implicit

Double precision

Fluids

ALE

ICFD

CESE

CPM

Mechanical

Implicit / Explicit

Double precision /

Single precision

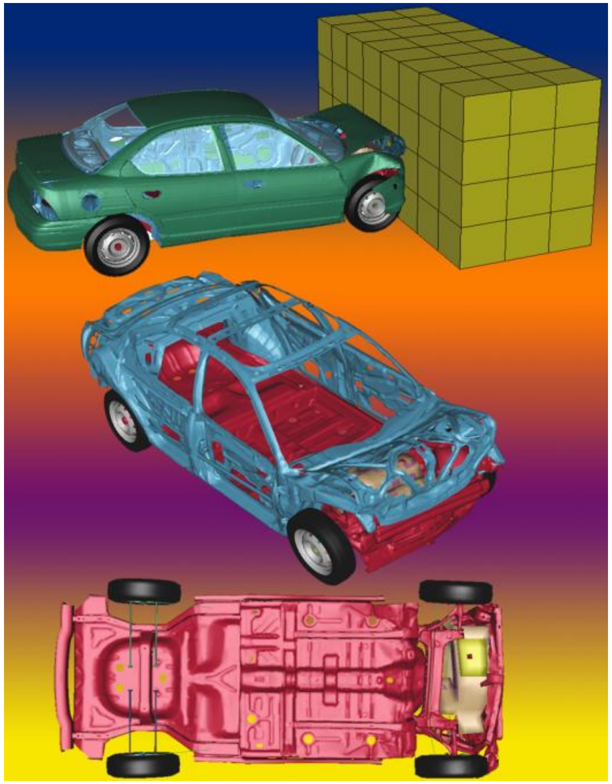
One Code for Multiple Solutions

One Model

+

LS-DYNA

- Multi-physics
    - Structure + Fluid + EM + Heat Transfer + , ..
- Multi-stage
    - Implicit + Explicit ….
- Multi-scale
    - Failure predictions, i.e., spot welds
- Multi-formulations
    - linear + nonlinear +



Many Results

Manufacturing, Durability, NVH, Crash, FSI

Developments

- Recent Enhancements
  - John Hallquist
- Linear Solver
  - Yun Huang
- LS-PrePost: ICFD & EM
  - Iñaki Çaldichoury
- Particle Methods
  - Jason Wang

FEA Solvers

- Element Technology
- Contact
- Connection
- Material
- Forming
- Crash/Safety

Subcycling and multi-scale

Subcycling

- Partitions elements in groups based on their characteristic time step size
- Each partition is then integrated independently using a time step for the partition with the exception of special treatment at the shared element interfaces.
- Up to seven sub models are automatically generated each integrated in steps of 1, 2, 4, 8, 16, 32, and 64 times the smallest characteristic time step of the entire model.

Subcycling and multi-scale

Multi-scale

- User may manually designate parts to be integrated at specific time steps.
- Special treatment at the shared element interfaces.
- Approach is intended for detailed modeling of critical components in a large simulation model
- Different time scales save CPU time.

Subcycling and multi-scale

Advantages of new approach:

- Both methods are combined with mass scaling to avoid future sorting
- Permits element partitions with different time step sizes to be uniformly distributed across processors at the beginning of the simulation
  - Improved load balancing
- No additional user input than minor modifications to the subcycling control keywords
- Eliminates the complications related to multiple models running simultaneously on the same or separate cores
  - Complex input

Disadvantage: Stability issues related to subcycling

Subcycling and multi-scale

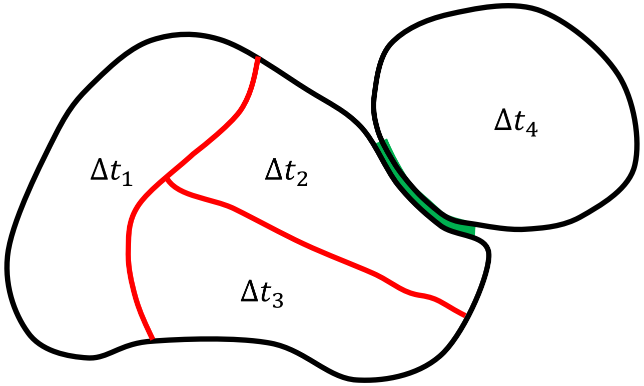
*CONTROL_SUBCYCLE_{K}_{L}

*CONTROL_SUBCYCLE_MASS_SCALED_PART_{SET}

B-pillar refined

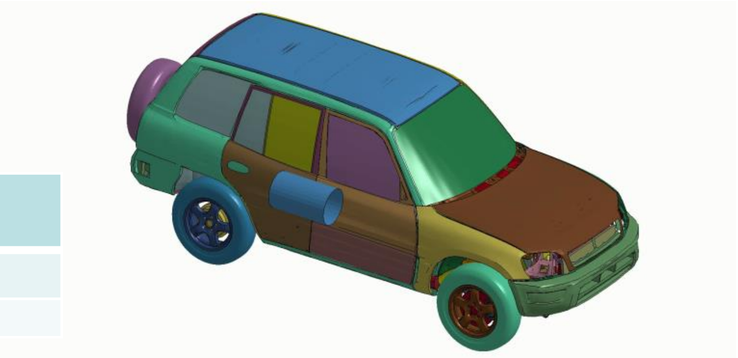
DT2MS=-1e-3

DT(B-pillar)=1.3e-4



Using different time steps in different subdomains, interface between subdomains by constraints indicated by red and due to contact by green.

Rav4 CPU timings (s)	Subcycling, K=64, L=4	Multi-scale, L=4	No subcycling
Contacts	133	133	288
Elements	194	206	636



Implicit rotational dynamics

Rotational dynamics: the study of vibration of rotating parts (turbine blades, propellers in aircraft and rotating disks in hard disk drives etc.).
The **deformation of rotating components** can cause damage as the rotational velocity increases.
A **resonant vibration** can lead to premature fatigue failure in rotating components, bearings and support structures.

The goal of the Rotational Dynamics in LS-DYNA is to study the above vibration-related phenomenon by considering the spin softening and gyroscopic effects.

Implicit rotational dynamics

Progress:

- A *CONTROL_IMPLICIT_ROTATIONAL_DYNAMICS card is added to LS-DYNA to do Rotational Dynamics analysis.
- Four types of elements: beam, shell, thick shell and solid, are available for the rotational dynamics studies.
- Vibration and modal analysis are verified using theoretical results or tested compared to 3rd party code results.
- Campbell diagram plotting

- Developments are ongoing:
  - Please contact Liping Li (liping@lstc.com) with feedback and requests for additional features.

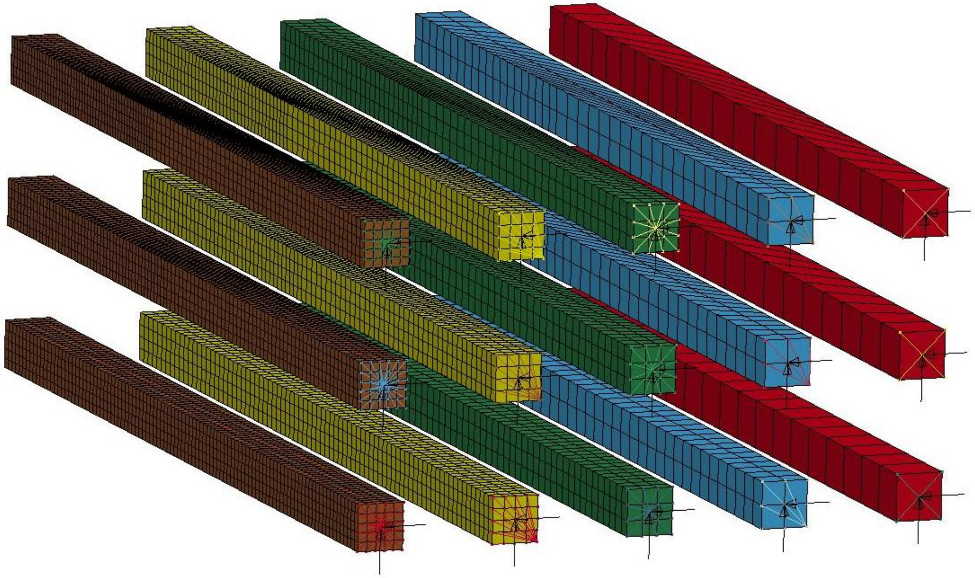
Cosserat point element

R7.1

- Brick element using Cosserat Point Theory
  - 8-node hexahedron,
    - 1-point solid element with hourglass type 10.
  - 10-node tetrahedron element
    - solid element type 16 with hourglass type 10.
  - Hourglass is based on a total strain formulation
  - Provides hight accuracy and insignificant mesh sensitivity

Cosserat point element

R7.1

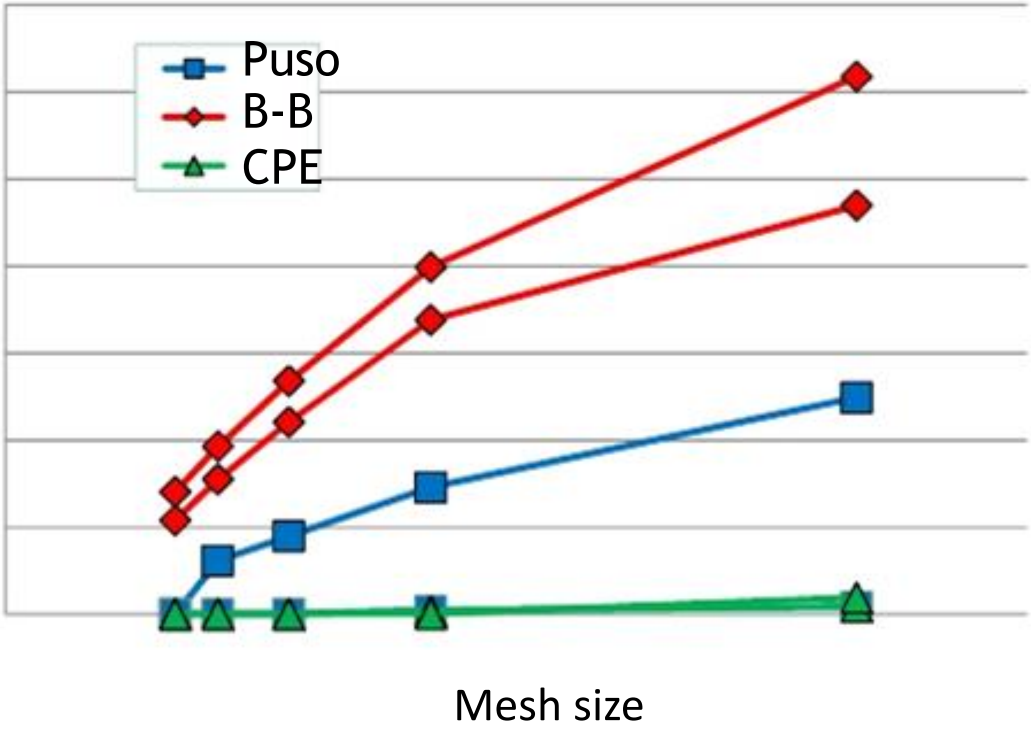


- Tip loaded cantilever beam
    - 5 mesh size levels (H=10, 5, 3.33, 2.5, 2 mm)
    - 3 distortion levels (a=-20, 0, 20 mm)
    - 2 load cases (horizontal (H) and vertical (V))
- Analytical tip displacement 0.21310 mm

Cosserat point hexahedron

R7.1

Cosserat	Belytschko -Bindeman	Puso
1.7%	61.8%	24.8%
0.8%	46.8%	14.7%
0.6%	40.0%	14.5%
0.3%	39.8%	9.2%
0.2%	33.9%	8.5%
0.2%	27.0%	6.2%
0.1%	24.6%	5.3%
0.1%	22.3%	3.6%
0.1%	19.0%	0.9%
0.1%	15.4%	0.3%



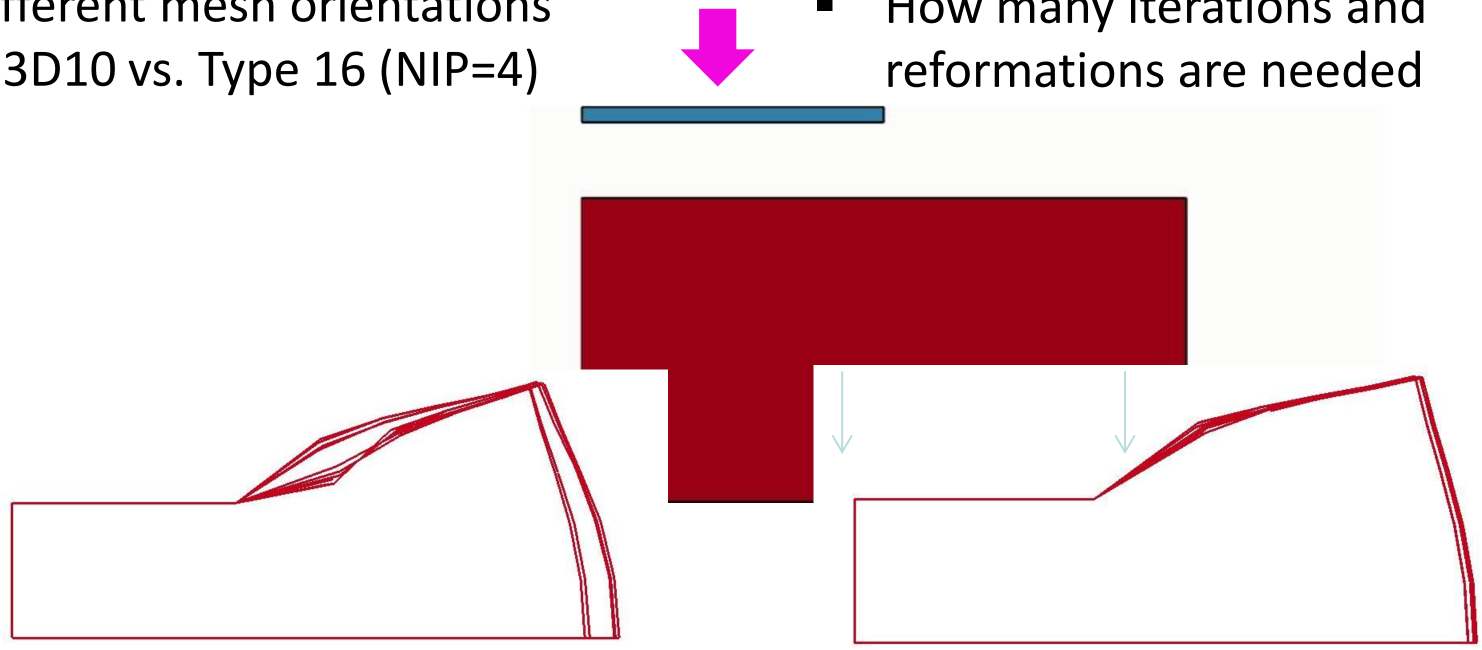
- Worst errors for three hourglass formulations

Cosserat 10-noded tetrahedron

R7.1

- Plane strain
- Implicit with extremely tight convergence criterion
- Hyperelastic rubber
  (PR=0.4997)
- 5 different mesh orientations
- CPE3D10 vs. Type 16 (NIP=4)

- Three basic checks
  - Sensitivity of results with respect to mesh orientation
  - How far can the block be compressed
  - How many iterations and reformations are needed



Fully integrated tetrahedron

CPE tetrahedron

Cosserat 10-noded tetrahedron

R7.1

Check #2 and #3 - Robustness and Convergence

- CPE3D10

Max%comp.	Vol%error	Iter/Ref
56.5	0.5	900/57
61.5	0.6	883/55
51.5	0.4	883/56
40	0.3	858/50
51	0.5	882/56

- Type 16

Max%comp	Vol%error	Iter/Ref
29	0.5	1562/110
35.5	1.6	983/61
32	0.5	1237/84
32.5	0.8	1031/66
35	0.6	1162/77

Single point pentahedron

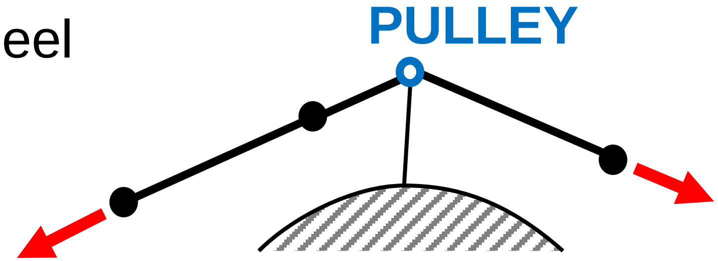
R7.1

- Implemented as element type 115
- Supports Flanagan-Belytschko viscous and stiffness hourglass types
- More robust than the 2 point integrated pentahedron element
- Degenerated single point hexahedron elements are sorted to type 115
- Supported for implicit calculations

*ELEMENT_BEAM_PULLEY

R7.1

- General framework for **pulley mechanism**:
  rope / cable / belt / chain runs over a wheel
  → beam elements run over pulley node



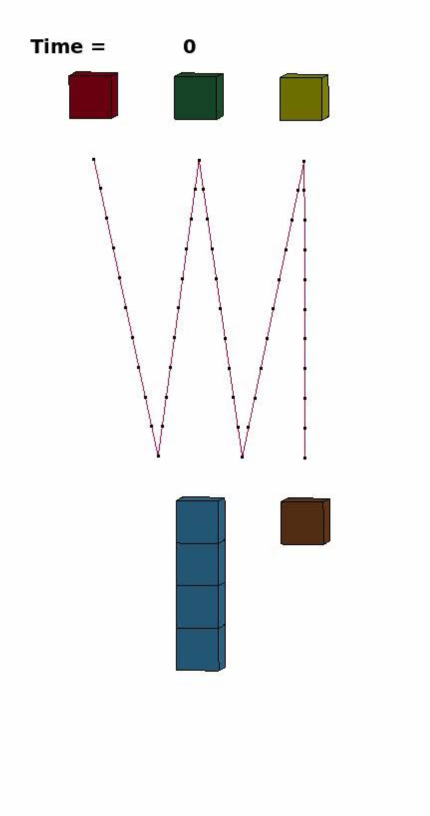
- Available for **truss beam elements**
- Available for ***MAT_ELASTIC** and ***MAT_MUSCLE**,
  more materials could be implemented
- Automatic detection of adjacent beam elements

```
*ELEMENT_BEAM_PULLEY
$       PUID         BID1         BID2         PNID    …
         101            0            0        20001
```

*ELEMENT_BEAM_PULLEY

R7.1

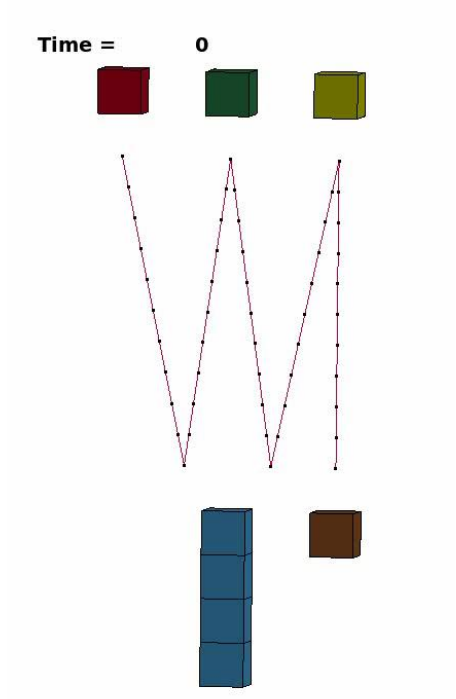
- **Increase accuracy for slipping and swapping** by tightening slip condition tolerances, correcting velocity of swapped node, and changing internal precision from single to double for selected pulley variables



Example:
wire rope under
gravity load and
prescribed motion
→ velocity vectors

OLD: R7.0.0

NEW: R7.1.0



less noise!
(same range)

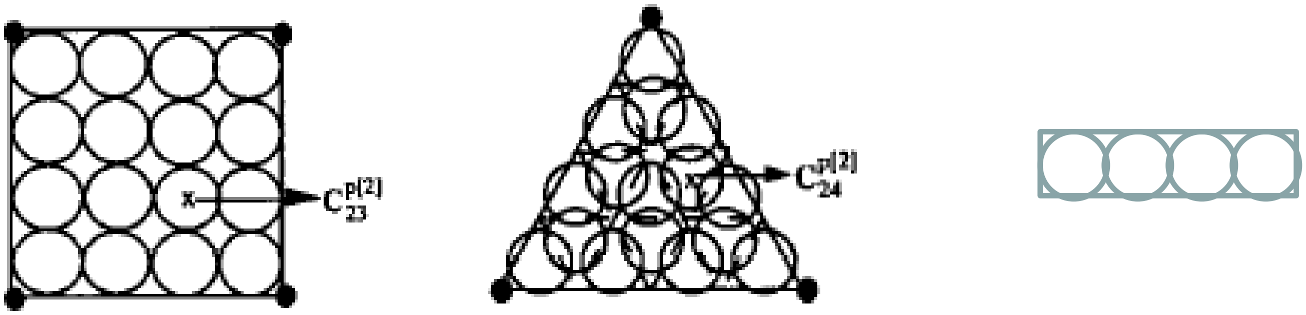
FEA Solvers

- Element Technology
- **Contact**
- Connection
- Material
- Forming
- Crash/Safety

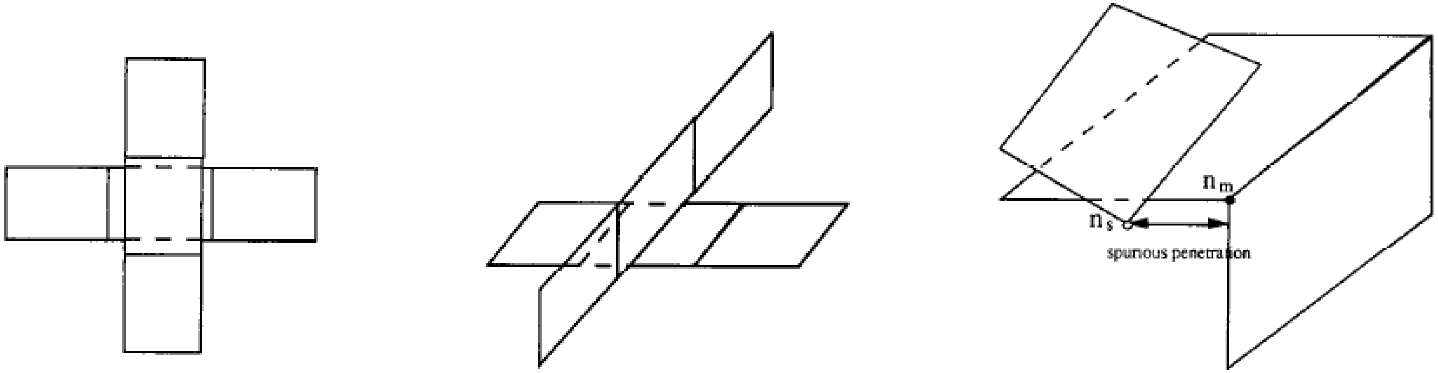
New Option for Segment_based Cont

R7.1

- Based on Splitting Pinball Method, Belytschko and Yeh, 1993

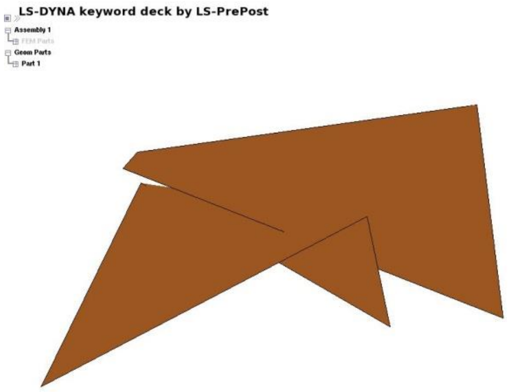


- Able to treat numerous types of contact in a consistent way, including those posing difficulties for node-to-segment contact .

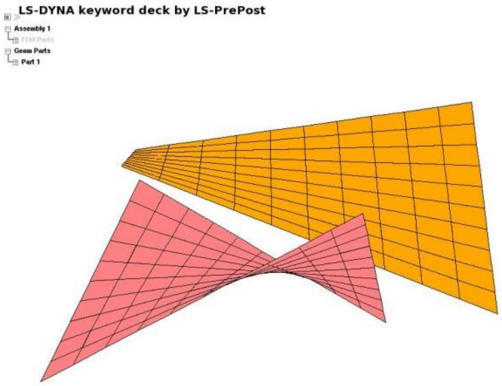


New option for segment_based contact

- This new option for computational airbag folding
  - is based on the penetration into the bilinear patch
  - conducts unified treatment of various contact types, including edge contact and node-to-surface contact etc..
- Is activated by setting “soft=2” and “depth=45”
- An intersection report is printed for the new option which provides information on interpenetrations



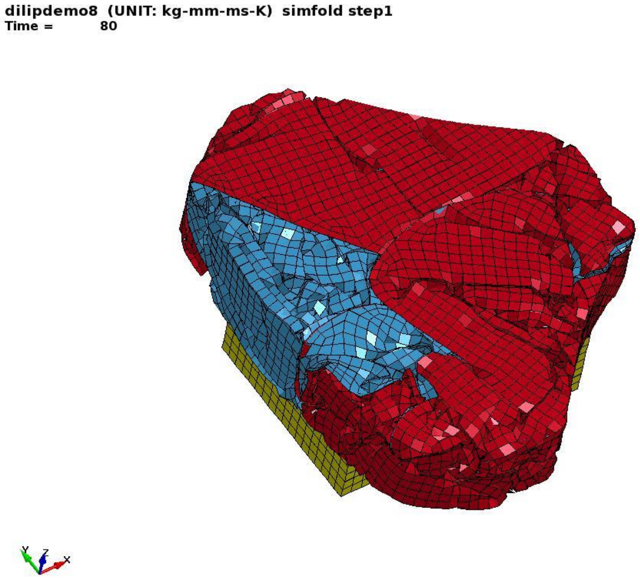
Intersection for node-to-projected plane

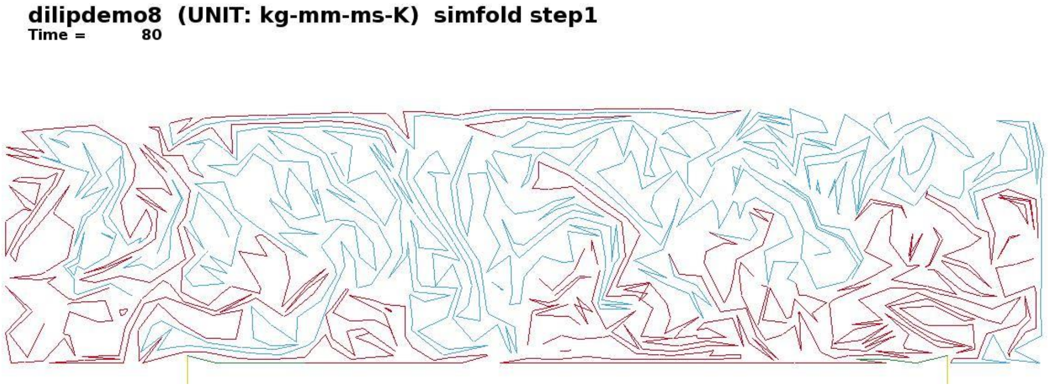


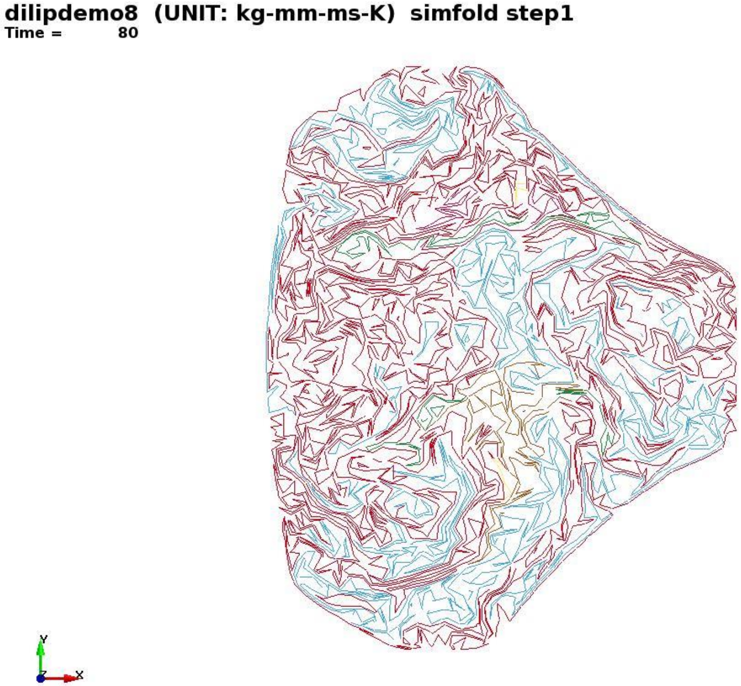
Intersection-free for node-to-bilinear patch

New option for segment_based contact

R7.1 – Folded intersection-free bag

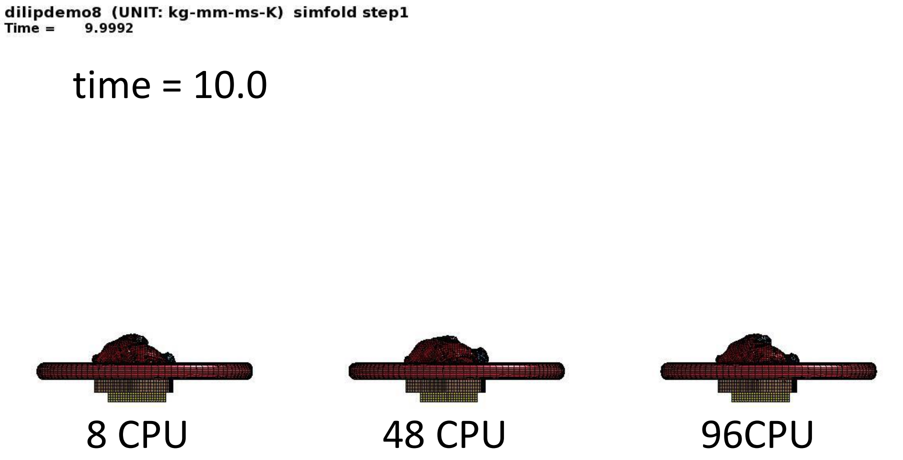






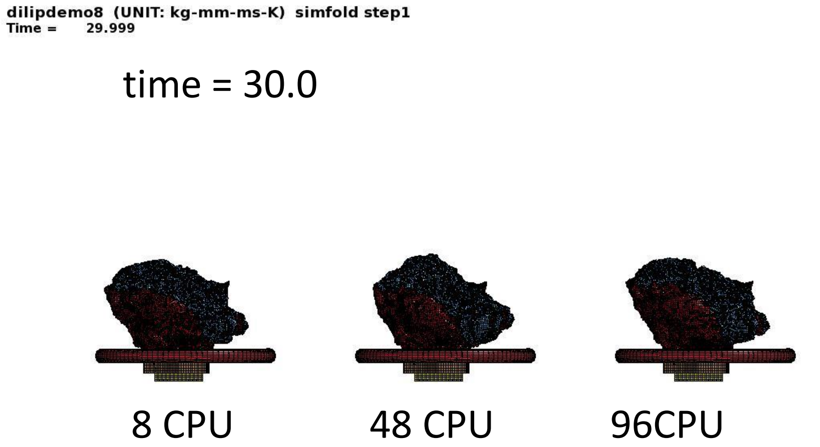
New option for segment_based contact

R7.1 – Deployment of the folded bag using various number of CPUs



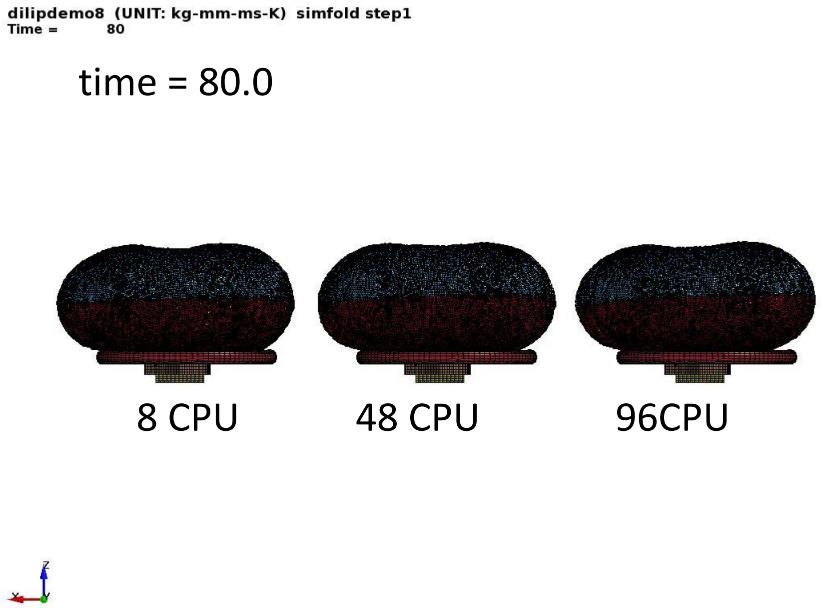
time = 10.0

8 CPU 48 CPU 96CPU



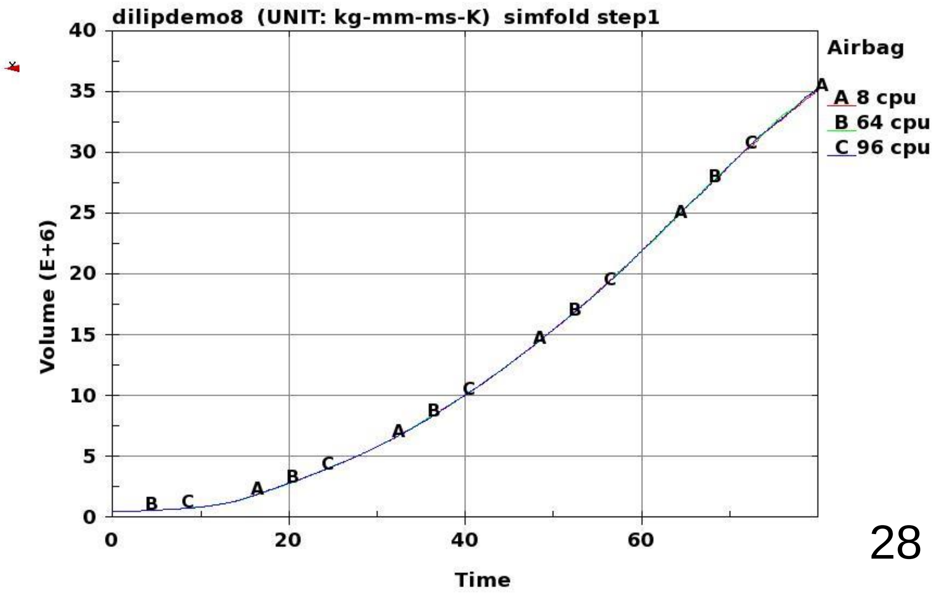
time = 30.0

8 CPU 48 CPU 96CPU



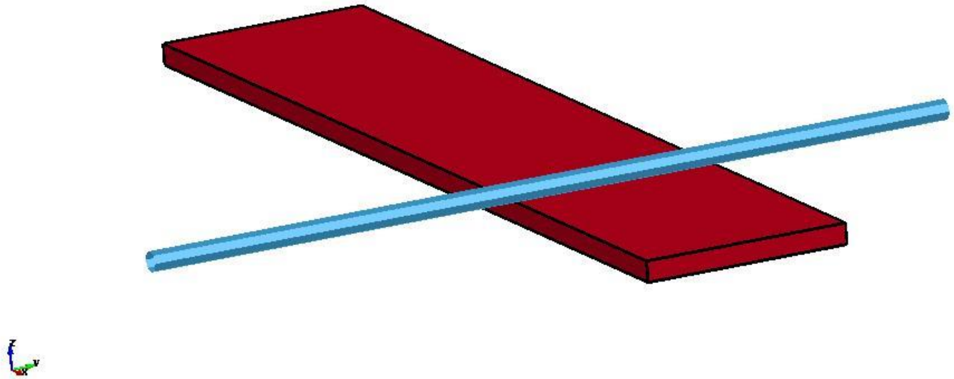
time = 80.0

8 CPU 48 CPU 96CPU



Mortar Contact

R7.1



- Recommended for implicit simulations
- Beam contact with lateral surface area supported in AUTOMATIC_..._MORTAR contact
- In addition, bucket sort is made more efficient for large scale applications with the advent of R7.1.0

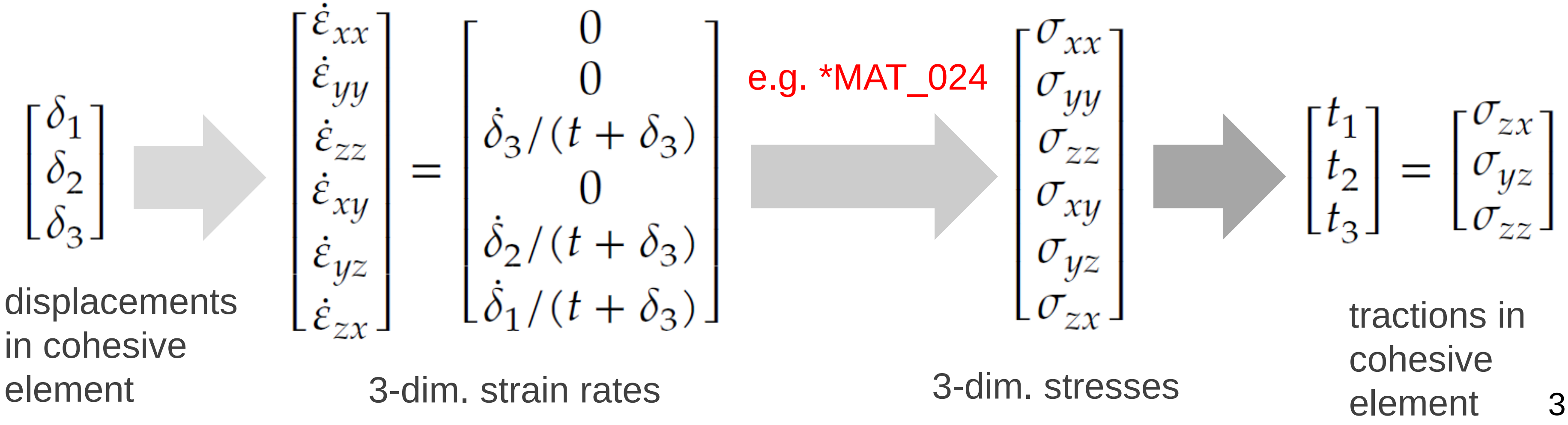
FEA Solvers

- Element Technology
- Contact
- **Connection**
- Material
- Forming
- Crash/Safety

*MAT_ADD_COHESIVE

R7.1

- Offers the possibility to use a selection of 3-dim. material models in conjunction with cohesive elements
- Cohesive elements (ELFORM = 19 and 20 of *SECTION_SOLID) can only be used with a small subset of materials (138, 184, 185, 186, 240). With this additional keyword, more material models can be used (mat-1, 3, 4, 6, 15, 24, 41-50, 81, 82, 89, 96, 98, 103, 104, 105, 106, 107, 115, 120, 123, 124, 141, 168, 173, 187, 188, 193, 224, 225, 252, and 255).
- Assumptions of inhibited lateral expansion and in-plane shearing



$$\begin{bmatrix}\delta_1\\ \delta_2\\ \delta_3\end{bmatrix}$$

displacements in cohesive element

$$\begin{bmatrix}\dot{\varepsilon}_{xx}\\ \dot{\varepsilon}_{yy}\\ \dot{\varepsilon}_{zz}\\ \dot{\varepsilon}_{xy}\\ \dot{\varepsilon}_{yz}\\ \dot{\varepsilon}_{zx}\end{bmatrix} = \begin{bmatrix}0\\ 0\\ \dot{\delta}_3/(t+\delta_3)\\ 0\\ \dot{\delta}_2/(t+\delta_3)\\ \dot{\delta}_1/(t+\delta_3)\end{bmatrix}$$

3-dim. strain rates

e.g. *MAT_024

$$\begin{bmatrix}\sigma_{xx}\\ \sigma_{yy}\\ \sigma_{zz}\\ \sigma_{xy}\\ \sigma_{yz}\\ \sigma_{zx}\end{bmatrix}$$

3-dim. stresses

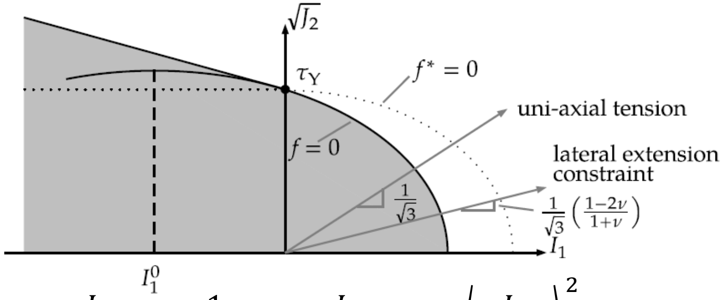
$$\begin{bmatrix}t_1\\ t_2\\ t_3\end{bmatrix} = \begin{bmatrix}\sigma_{zx}\\ \sigma_{yz}\\ \sigma_{zz}\end{bmatrix}$$

tractions in cohesive element

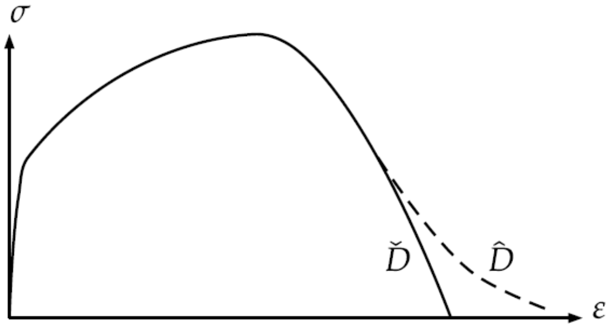
*MAT_TOUGHENED_ADHESIVE_POLYMER

R7.1- Material model 252 for crash optimized high-strength adhesives under combined shear & tensile loading

- Drucker-Prager-Cap type plasticity + rate dependence + damage + failure
- well suited for combination with *MAT_ADD_COHESIVE

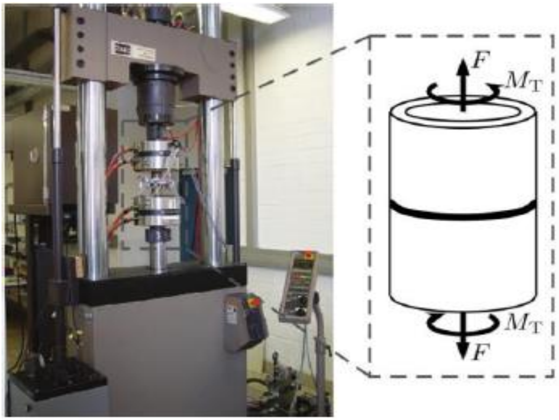


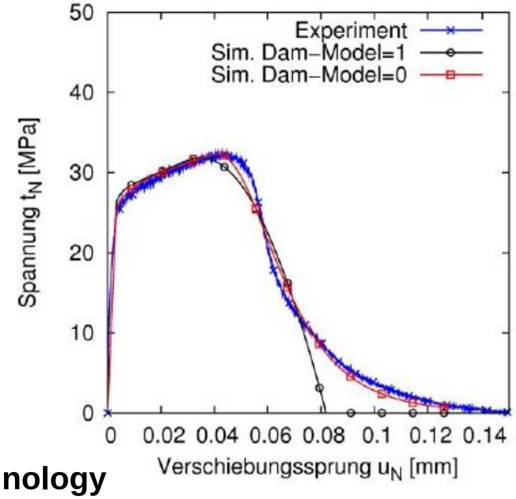
$$f := \frac{J_2}{(1-D)^2} + \frac{1}{\sqrt{3}} a_1 \tau_0 \frac{I_1}{1-D} + \frac{a_2}{3} \left\langle \frac{I_1}{1-D} \right\rangle^2 - \tau_Y^2 = 0$$



damage

yield surface of Drucker-Prager-Cap



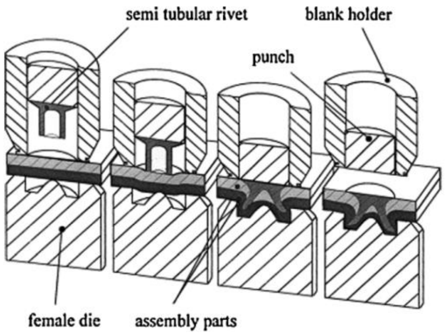


***Research Association for Automotive Technology**

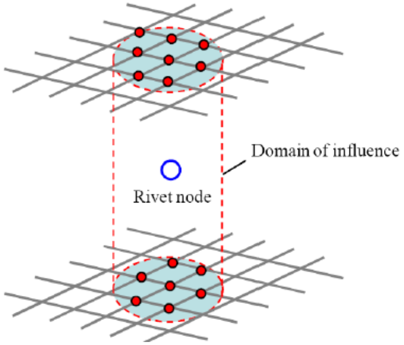
*CONSTRAINED_INTERPOLATION_SPOTWELD

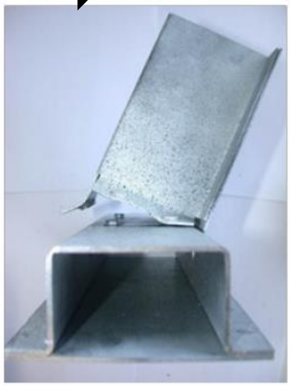
R7.1

- Model for self-piercing rivets, based on paper by M. Bier, 2013
- Replaces *CONSTRAINED_SPR3
- The algorithm does a normal projection from the two sheets to the spotweld node and locates all nodes within the user-defined diameter of influence

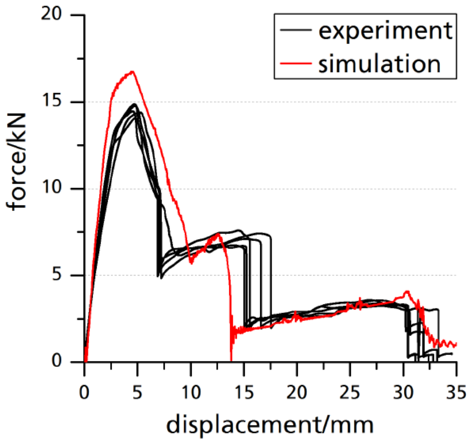


“element-free“ modeling technique





T-joint impact analysis

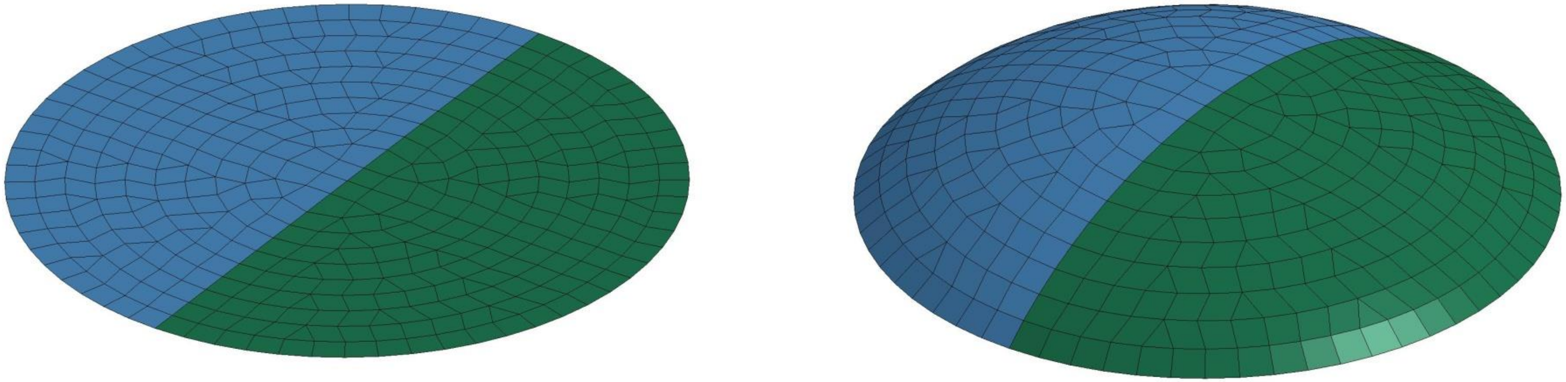


FEA Solvers

- Element Technology
- Contact
- Connection
- **Material**
- Forming
- Crash/Safety

Material 34 bending stiffness for implicit

R7.1

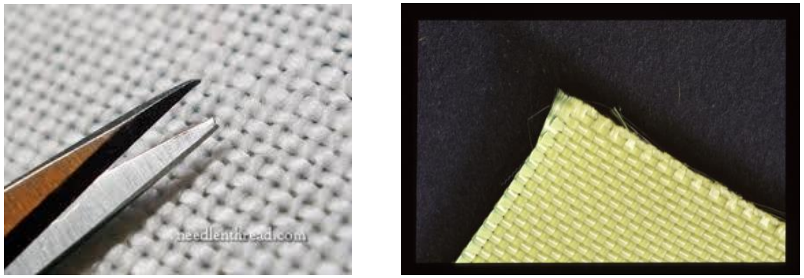


- Specify coating properties of fabric
- Pressurized coated membrane shown
- Solved implicit statics, not possible without bending stiffness

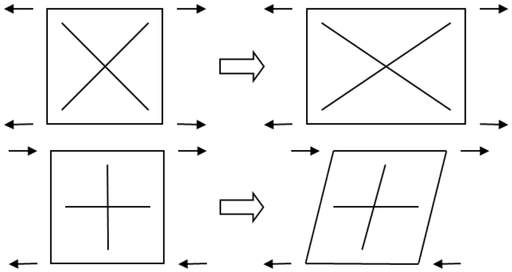
*MAT_DRY_FABRIC

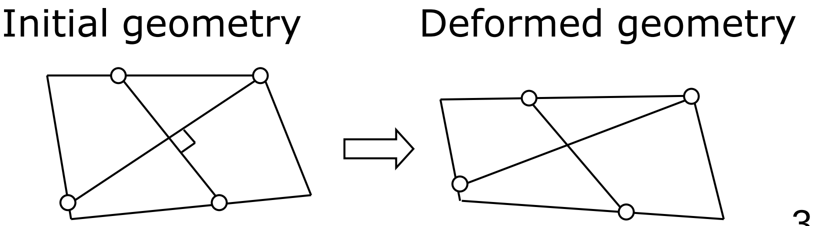
R7.1

- *MAT_DRY_FABRIC (*MAT_214) is a shell element material used to model high strength woven fabric (e.g., Kevlar® 49) with transverse orthotropic behavior.
- candidate materials for use in structural systems where high energy absorption is required, like materials used in propulsion engine containment system, body armor and personal protections
- When fibers scissor, the stress update becomes less accurate and single and double precision solutions were inconsistent.



- To address this, *MAT_214 track a-fiber and b-fiber directions independently.

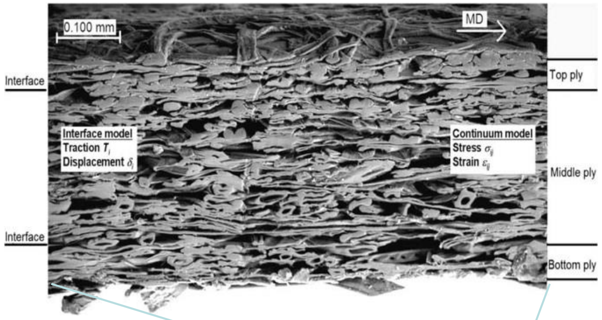


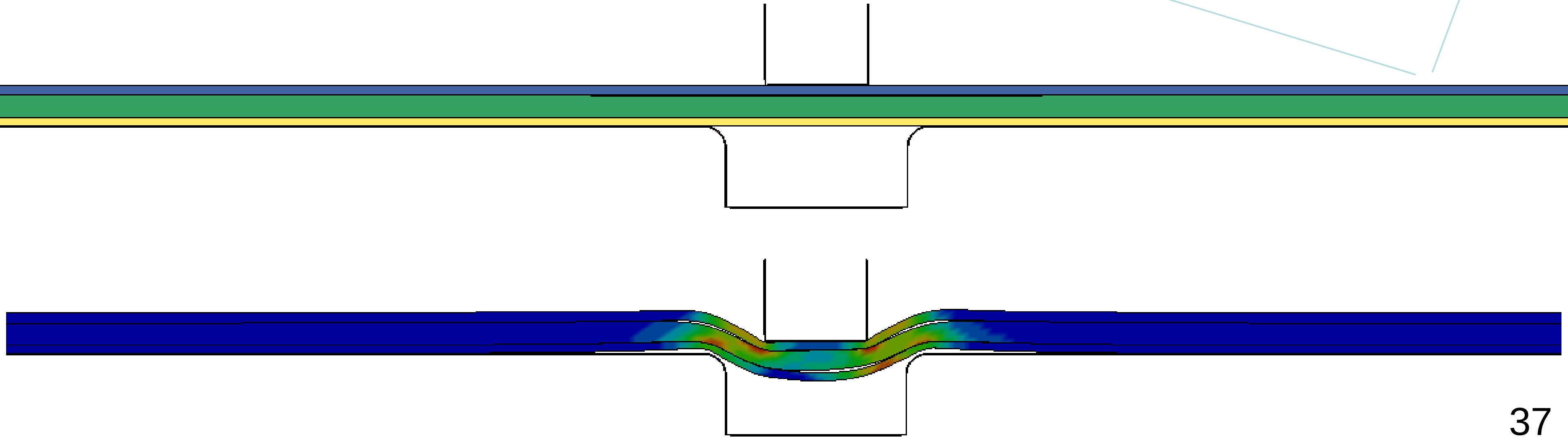


*MAT_PAPER (MAT_274) for paperboard

R7.1

- Orthotropic elastoplastic model based on Xia (2002) and Nygards (2009)
- Used for modeling of paperboard, a strongly heterogeneous material, creasing simulation with delamination of individual plies
- Available for solid and shell elements
- Has shown to reproduce experimental data well



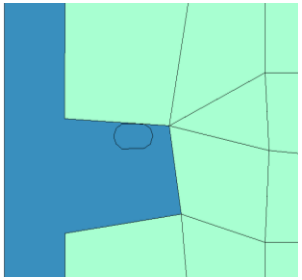


FEA Solvers

- Element Technology
- Contact
- Connection
- Material
- Forming
- Crash/Safety

LS-DYNA in Stamping Simulation

- Lancing to alleviate metal thinning during forming
  - Progressive lancing: blank was gradually cut
  - Instant cut: blank was cut instantly



- New contact for guide pins and blank edge
  - Eliminate missing contact,
  - ignore blank thickness
- Blanksize developments to optimize initial blank size
  - Iterative method
  - Allow several stages, including forming, trimming, flanging, etc.

LS-DYNA in Stamping Simulation

- Two new features in springback compensation
  - After springback compensation, if there is small change to the part, then the die will be modified based on the change
  - Local smoothing in springback compensation.
- More friendly control of d3plot output:
  - Based on the punch position to its home distance
- Arbitrary polygon can be used in defining adaptive box
- Automatic close of open-ended trim curve loops to make trimming more robust
- MAT 122 was extended to 3D elements
  - Anisotropy in both elastic and plastic deformation
  - Applicable to composite materials

FEA Solvers

- Element Technology
- Contact
- Connection
- Material
- Forming
- Crash/Safety

Energy-based pretensioner

R7.1

- Pull-in or belt load history of pretensioners vary with the size of the dummy, or if pretensioners are activated at different times.
- Different pretensioner models are needed for different crash scenarios when pull-in or belt load options are used to model the pretensioners.
- A pretension-energy based option is now available. This allows a single/unique pretensioner model to be used for various scenarios.

MAT_ADD_EROSION

R7.1

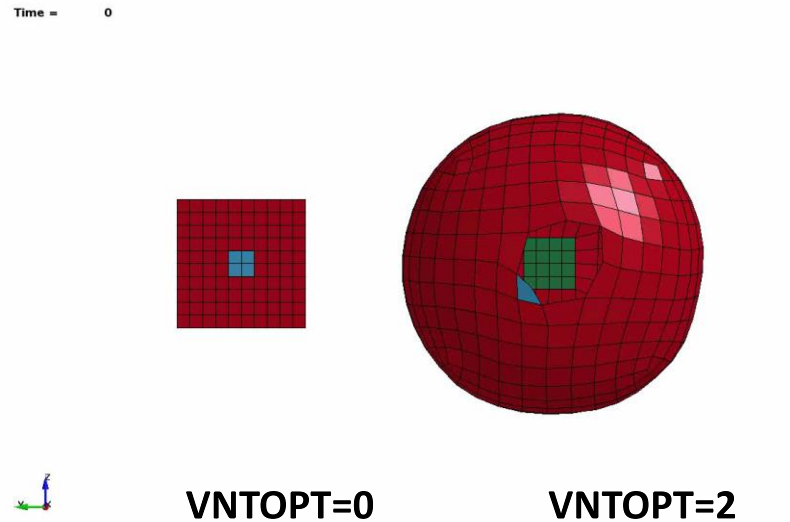
- MAT_ADD_EROSION application is extended to MAT_34, and
  - Shell formulation 18, 20, 21 ,23 , 24 and 54
  - Beam formulation 7 and 8
- Treatment of failed elements in an airbag model

  CARD 6 of *AIRBAG_HYBRID

  VNTOPT: bag venting option

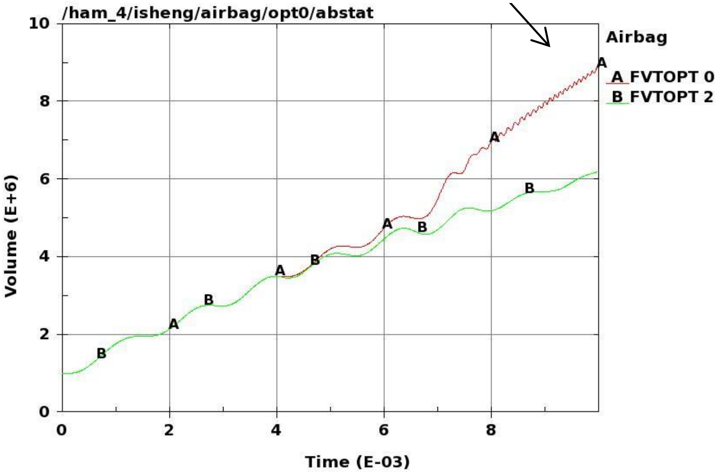
  EQ. 2: the areas of failed elements at failure times are added to the venting area defined by A23.

OPT	PVENT	NGAS	LCEFR	LCIDM0	VNTOPT



VNTOPT=0 **VNTOPT=2**

Wrongly calculated volume results in erroneous results



Volume History

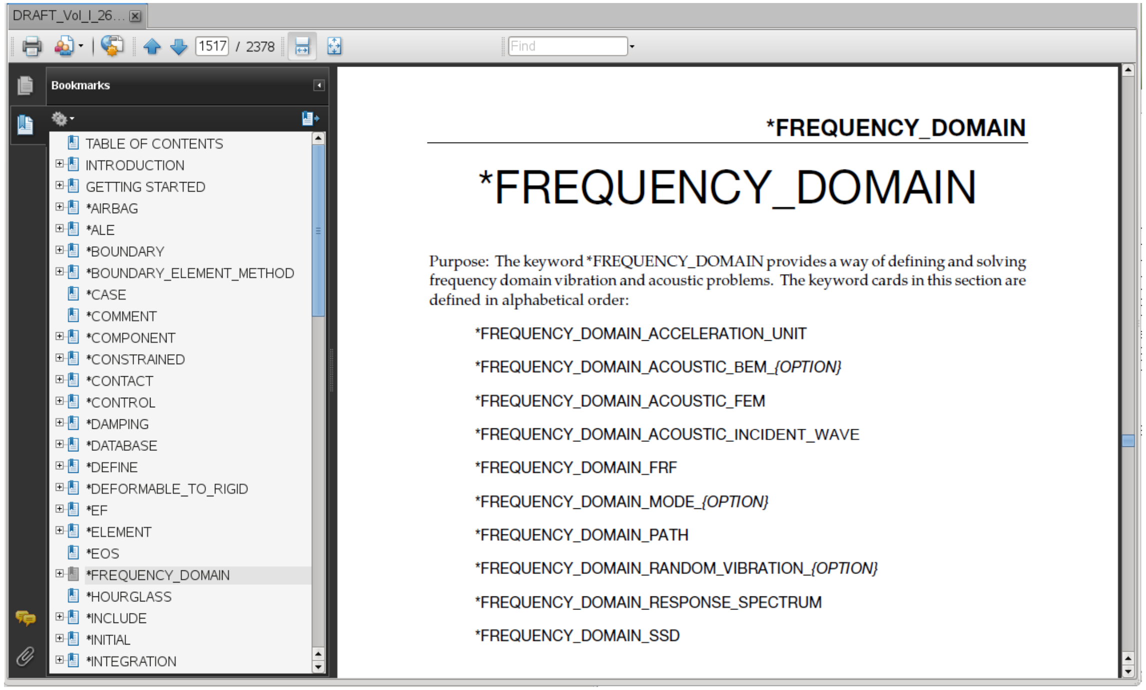
Thank You!

Linear Solvers

Yun Huang

Frequency domain features

- FRF
- SSD
- Random vibration
- Random fatigue
- Response spectrum analysis
- BEM Acoustics
- FEM Acoustics



*FREQUENCY_DOMAIN

*FREQUENCY_DOMAIN

Purpose: The keyword *FREQUENCY_DOMAIN provides a way of defining and solving frequency domain vibration and acoustic problems. The keyword cards in this section are defined in alphabetical order:

*FREQUENCY_DOMAIN_ACCELERATION_UNIT

FREQUENCY_DOMAIN_ACOUSTIC_BEM_*{OPTION}

*FREQUENCY_DOMAIN_ACOUSTIC_FEM

*FREQUENCY_DOMAIN_ACOUSTIC_INCIDENT_WAVE

*FREQUENCY_DOMAIN_FRF

FREQUENCY_DOMAIN_MODE_*{OPTION}

*FREQUENCY_DOMAIN_PATH

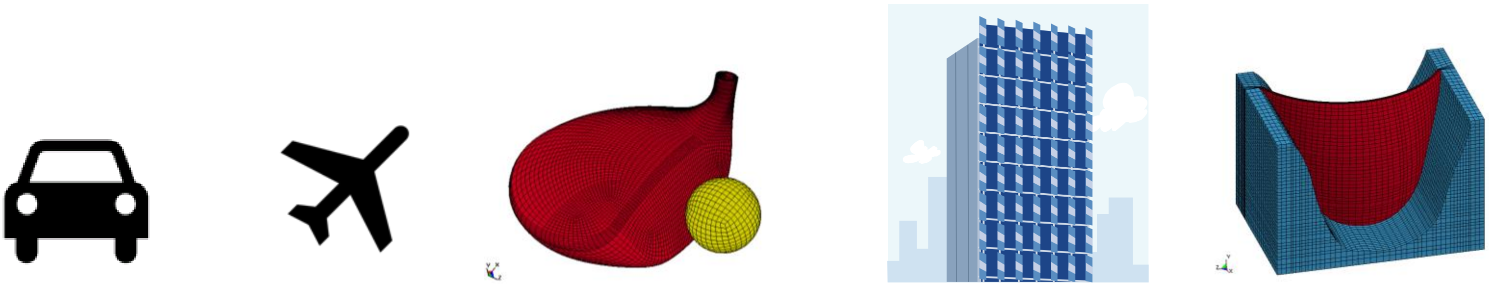
FREQUENCY_DOMAIN_RANDOM_VIBRATION_*{OPTION}

*FREQUENCY_DOMAIN_RESPONSE_SPECTRUM

*FREQUENCY_DOMAIN_SSD

Application

- NVH of automotive and air craft
- Acoustic design and analysis
- Defense industry
- Fatigue of machines and engines
- Civil and hydraulic engineering
- Earthquake engineering



FRF

What is FRF?

Load

F(ω)

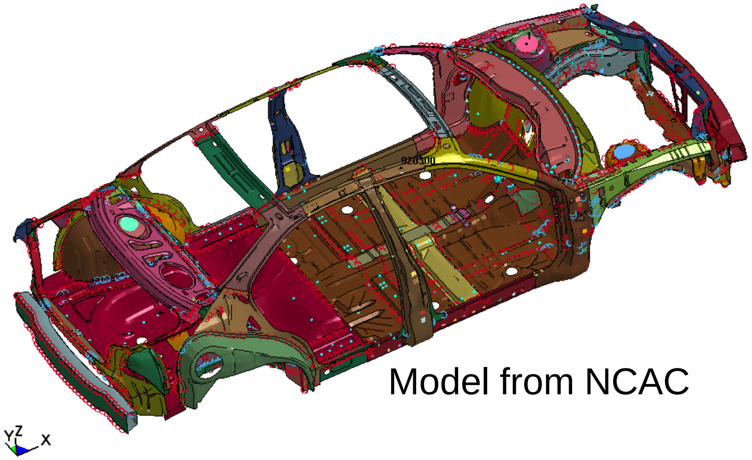
Transfer function H(ω)

Response

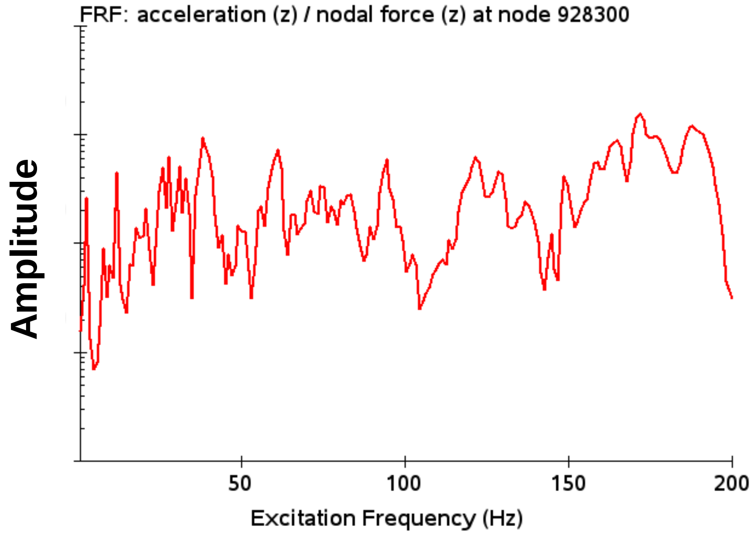
X(ω)

$$H(\omega) = \frac{X(\omega)}{F(\omega)}$$

- **FRF, as a transfer function, expresses structural response to applied load as a function of frequency**
- **Property of structure system**
- **Dependent on frequency**



Model from NCAC



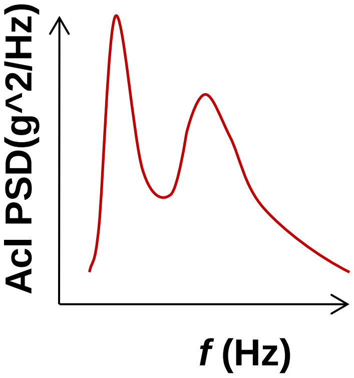
Random vibration

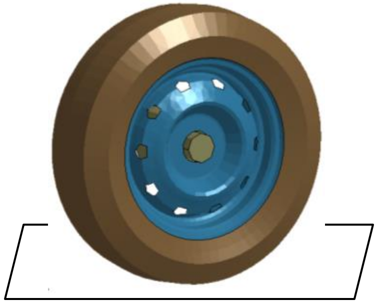
Why do we need random vibration analysis?

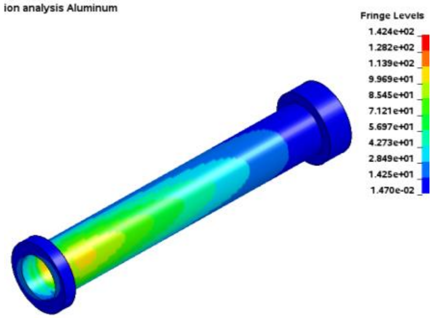
- The loading on a structure is not known in a definite sense
- Many vibration environments are not related to a specific driving frequency
- Examples:
  - Wind-turbine
  - Air flow over a wing or past a car body
  - Acoustic input from jet engine exhaust
  - Earthquake ground motion
  - Wheels running over a rough road
  - Ocean wave loads on offshore platforms

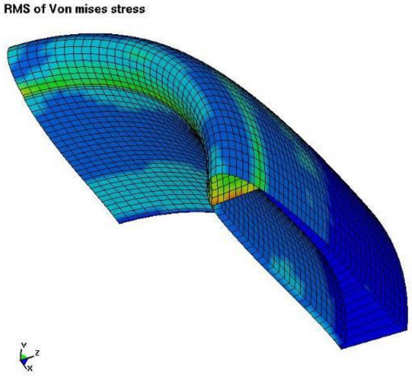
Input: PSD (Power Spectral Density), or SPL (Sound Pressure Level)

Output: D3PSD, D3RMS, NODOUT_PSD, ELOUT_PSD



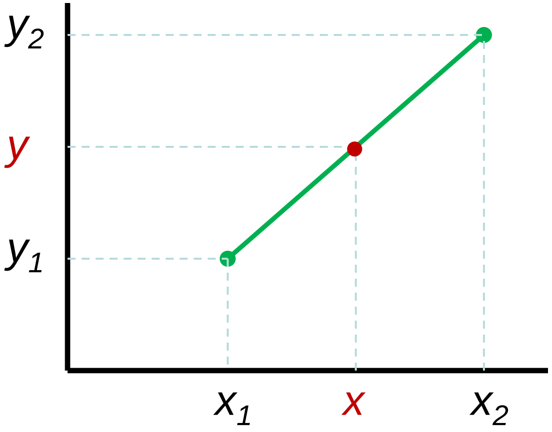




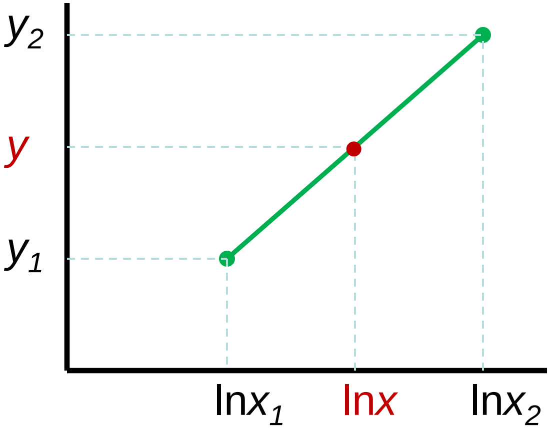


New interpolation options on PSD curve

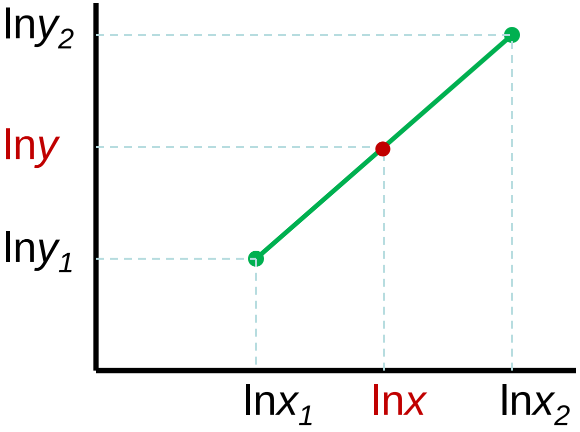
Three types of interpolation for PSD curves



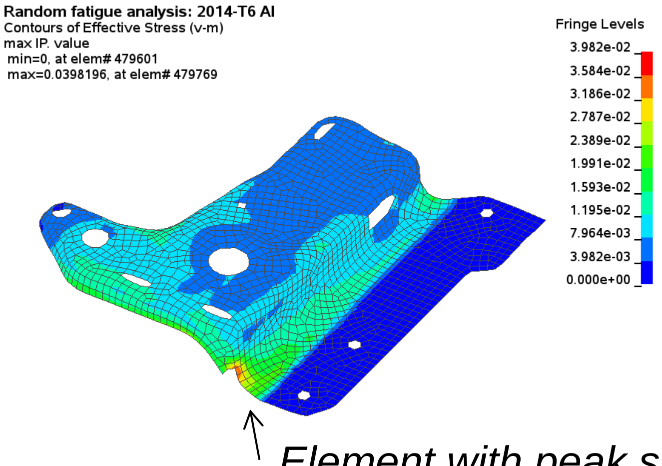
linear-linear



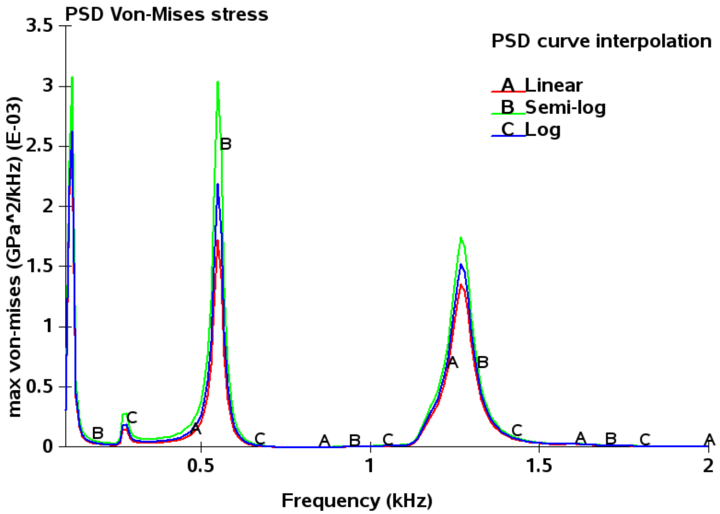
linear-log (semi-log)



log-log



Element with peak stress

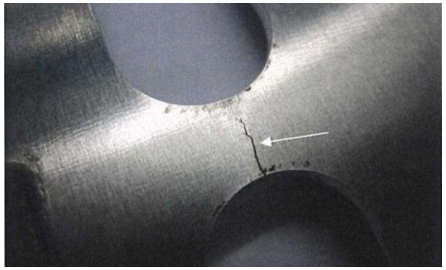


Livermore Software Technology Corp.

Random vibration fatigue

*FREQUENCY_DOMAIN_RANDOM_VIBRATION_*FATIGUE**

- Calculate fatigue life of structures under random vibration
- Based on S-N fatigue curve
- Based on probability distribution & Miner's Rule of Cumulative Damage Ratio



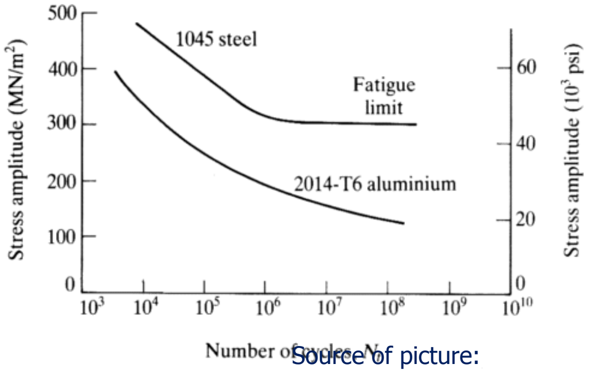
*MAT_ADD_FATIGUE

Card 1	1	2	3	4	5	6	7	8
Variable	MID	LCID	LTYPE	A	B	STHRES	SNLIMT	
Type	I	I	I	F	F	F	I	
Default	none	-1	0	0.0	0.0	none	0	

$$N \cdot S^m = a$$

$$\log(S) = a - b \cdot \log(N)$$

$$R = \sum_i \frac{n_i}{N_i}$$



Source of picture: http://www.efunda.com

SSD

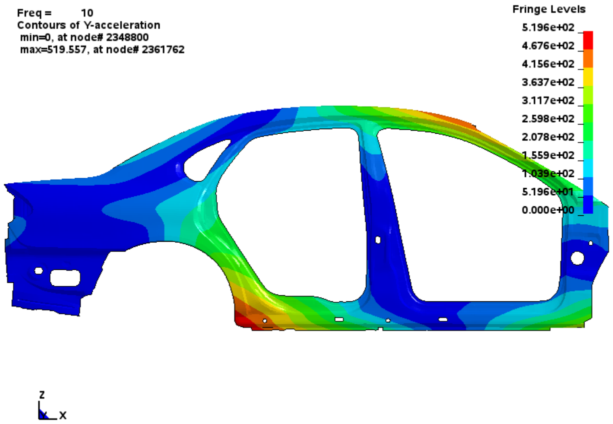
Why do we need SSD (Steady State Dynamics)?

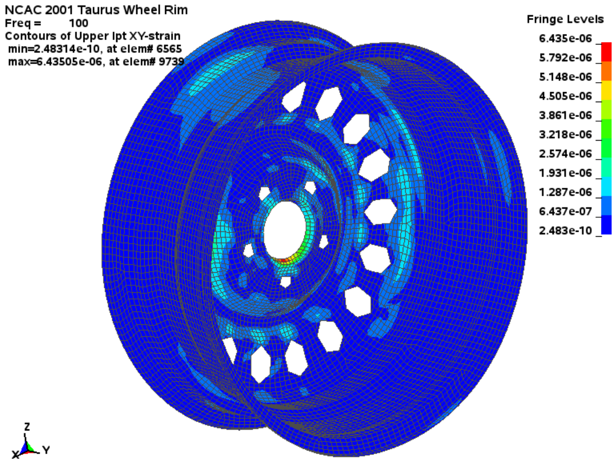
- Harmonic excitation is often encountered in engineering systems. E.g. it is commonly produced by the unbalance in rotating machinery.
- The load may also come from periodic load, e.g. in fatigue test.
- The excitation may also come from uneven base, e.g. the force on tires running on a zig-zag road

$$F\ (t) = F_0 \sin(\omega t + \phi)$$

Input: frequency load spectrum (complex variable)

Output: D3SSD, NODOUT_SSD, ELOUT_SSD

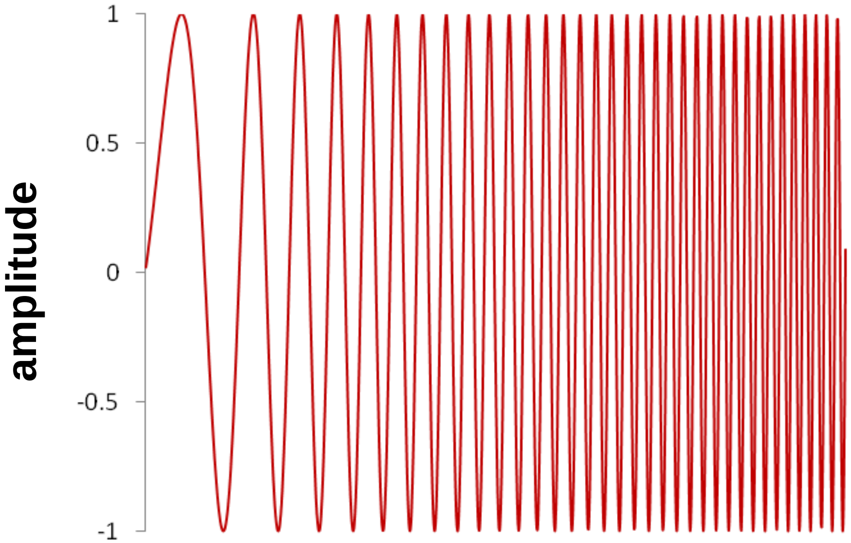




SSD Fatigue (ongoing development)

FREQUENCY_DOMAIN_SSD_*FATIGUE

- Calculate fatigue life of structures under steady state vibration (e.g. sine sweep)
- Based on S-N fatigue curve
- Based on Miner's Rule of Cumulative Damage Ratio
- Rain-flow counting algorithm for each frequency for one period



Sine sweep

$$R = \sum_i \frac{n_i}{N_i}$$

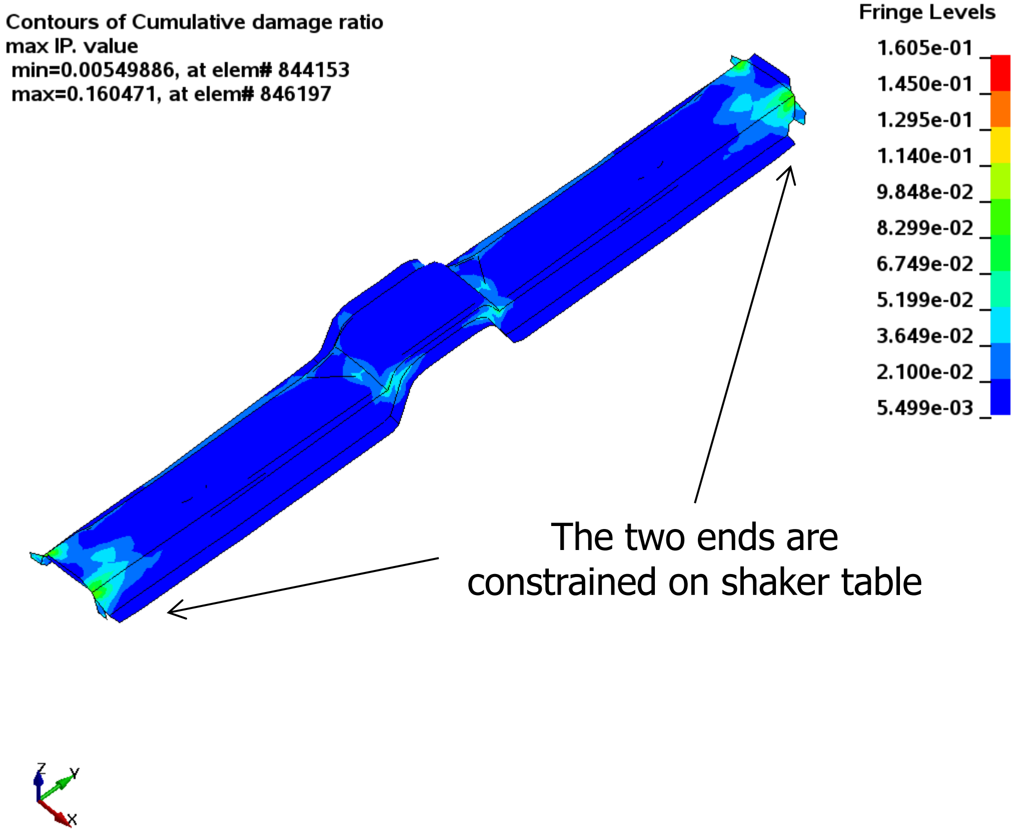
Example of SSD Fatigue

Loading condition

Freq (Hz)	Acl (g)	Duration (min)
16	0.5	10
20	0.5	10
25	0.5	10
31.5	0.5	10
…	…	…
2000	0.5	10

SN fatigue curve

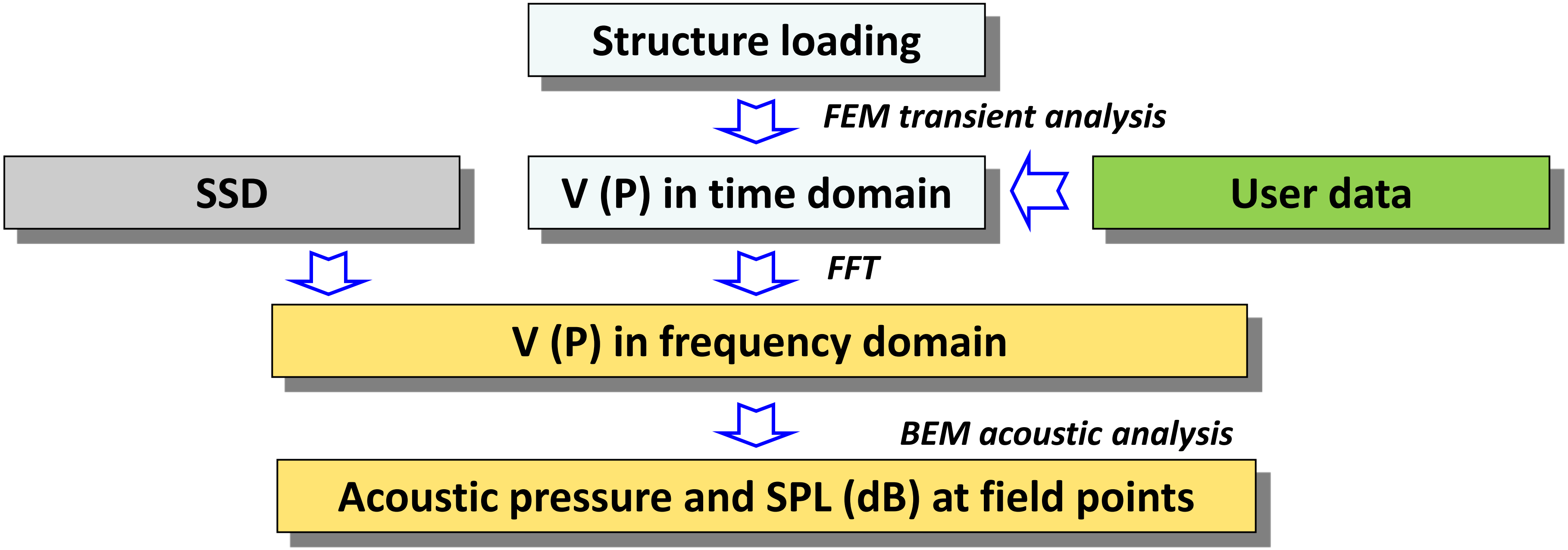
σ (MPa)	N
100	8×10^{4}
10	8×10^{5}
1.	8×10^{6}
0.1	8×10^{7}
0.01	8×10^{8}



The two ends are constrained on shaker table

BEM Acoustics

vibro-acoustics



Structure loading

FEM transient analysis

SSD

V (P) in time domain

User data

FFT

V (P) in frequency domain

BEM acoustic analysis

Acoustic pressure and SPL (dB) at field points

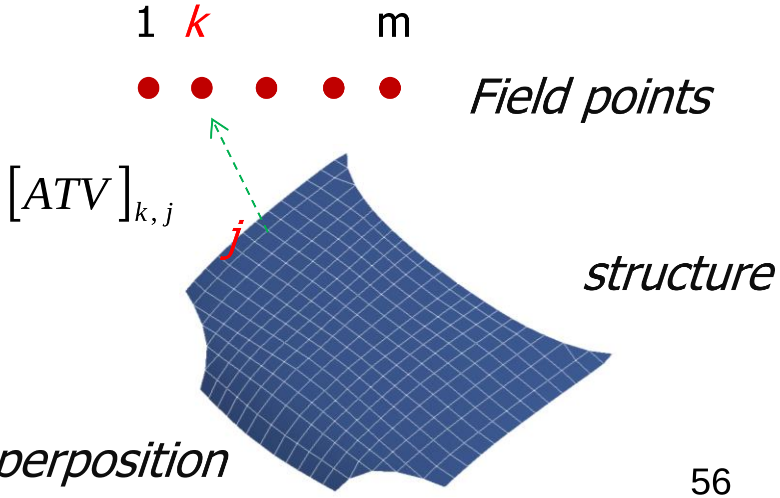
ATV and MATV

FREQUENCY_DOMAIN_ACOUSTIC_BEM_*ATV
FREQUENCY_DOMAIN_ACOUSTIC_BEM_*MATV

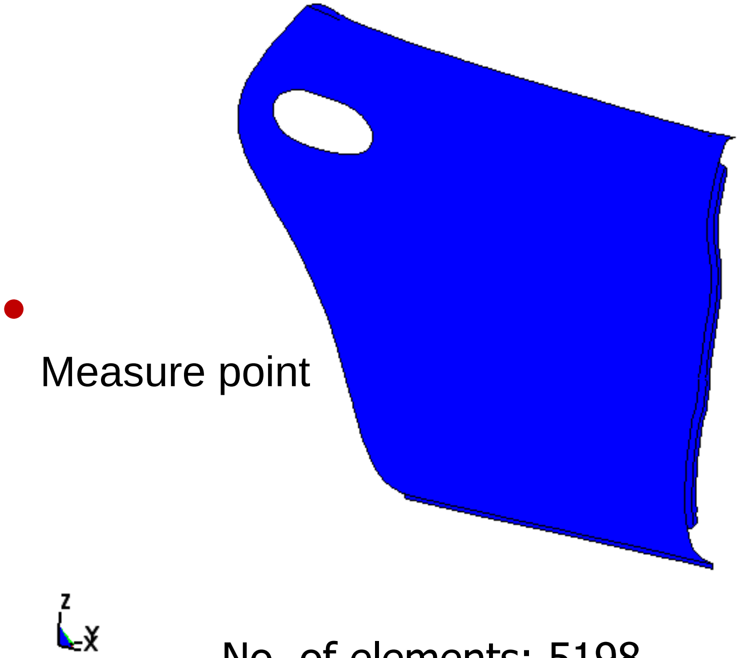
- **ATV calculates acoustic pressure at field points due to unit normal velocity of each surface node.**
- **MATV calculates acoustic pressure at field points due to vibration in eigen-modes.**
- **ATV / MATV is dependent on structure model, properties of acoustic fluid as well as location of field points.**
- **ATV / MATV is useful if the same structure needs to be studied under multiple load cases.**

$$
\begin{aligned}
\{P\} &= [ATV]\{V\} \\
&= [ATV](-i\omega[\Phi]\{q\}) \\
&= -i\omega[ATV][\Phi]\{q\} \\
&= [MATV]\{q\}
\end{aligned}
$$

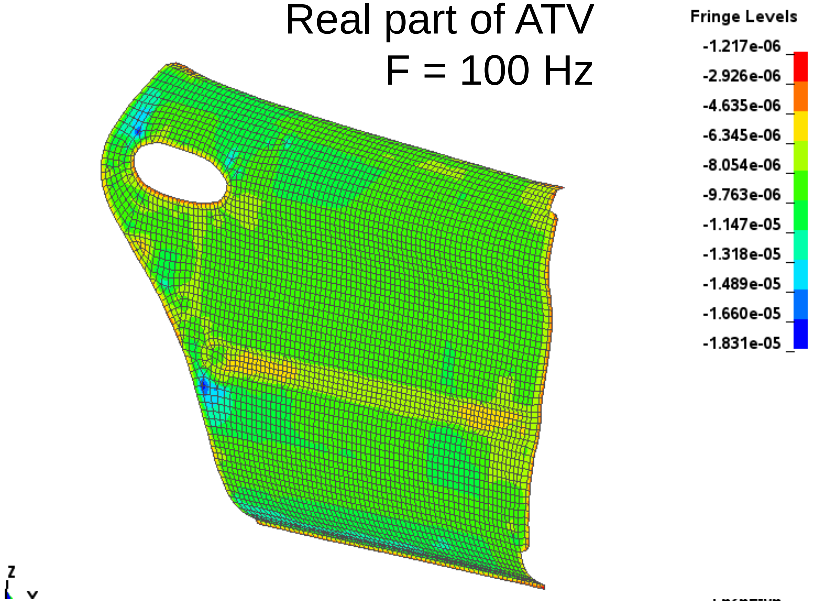
Velocity vector is obtained by modal superposition

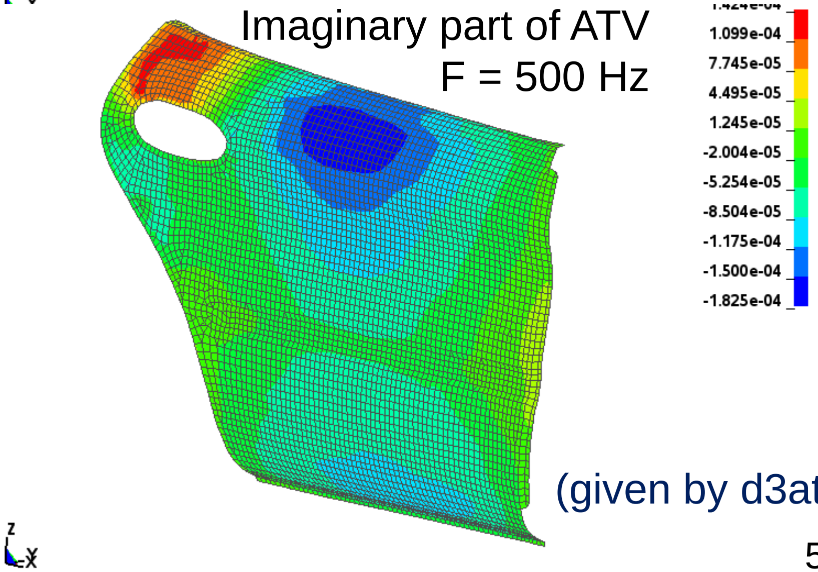


Example: ATV of car door model



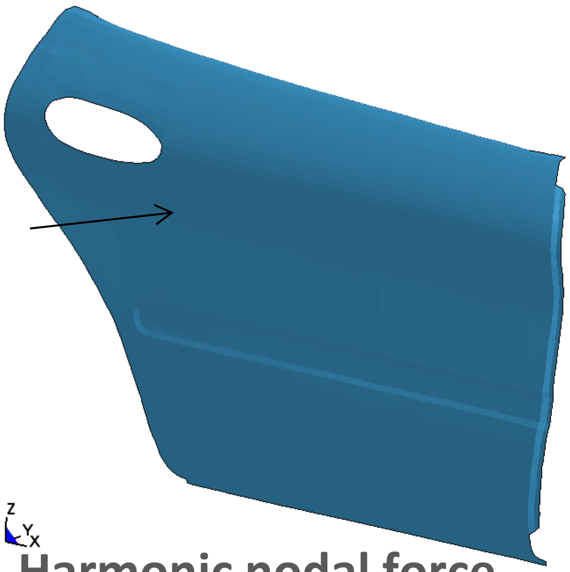
No. of elements: 5198
No. of frequencies: 101


Real part of ATV
F = 100 Hz

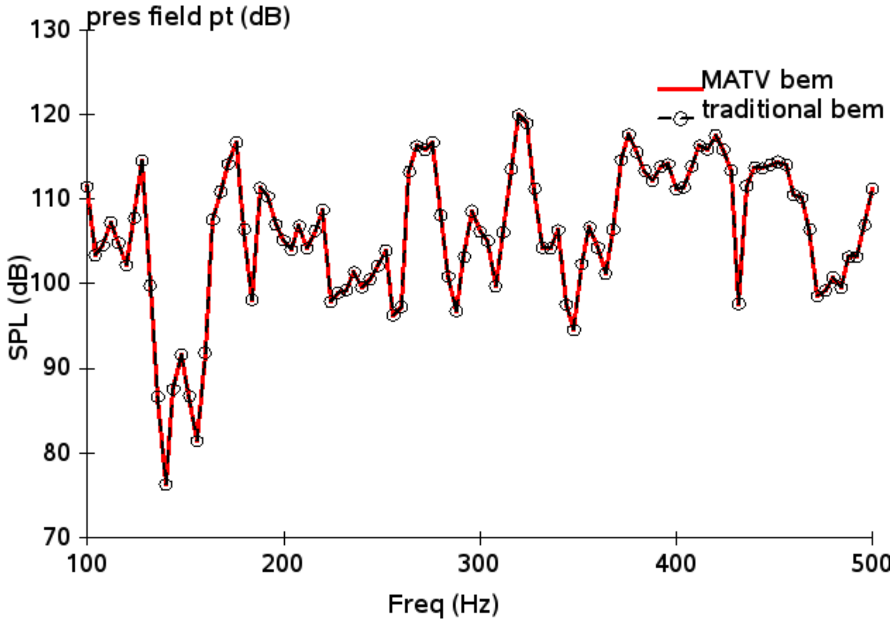

Imaginary part of ATV
F = 500 Hz

(given by d3atv)

MATV BEM is efficient



Harmonic nodal force
100-500 Hzc

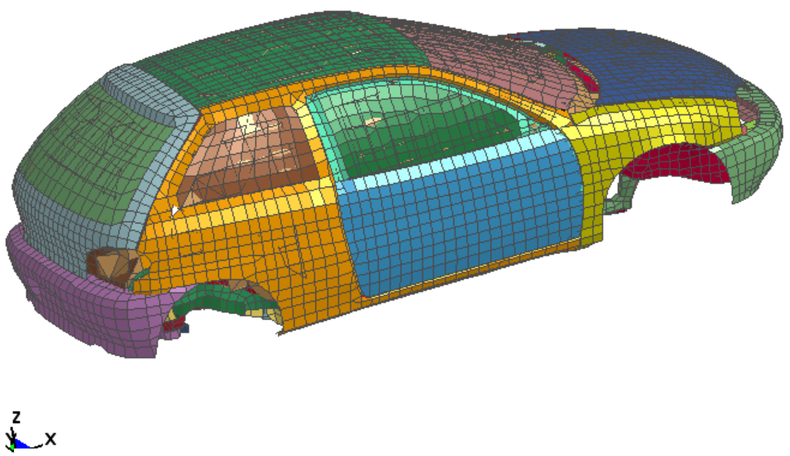


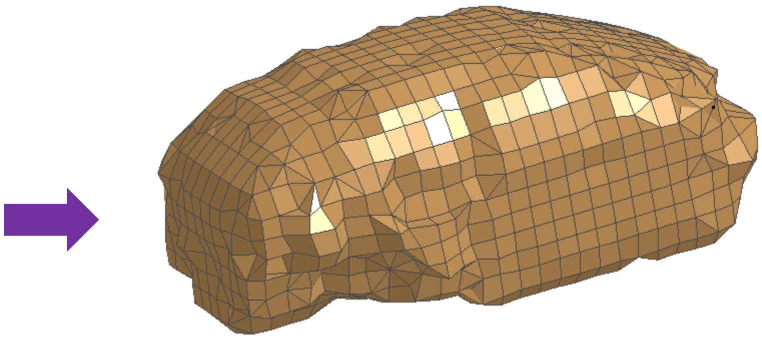
CPU time (1 core) Intel Xeron CPU E5504 @2.00 GHz

Cases	traditional BEM	MATV BEM
1 load case	2 h 39 m 50 s	4 h 40 m 56 s
10 load cases	**26 h 38 m 18 s**	**4 h 41 m 10 s**

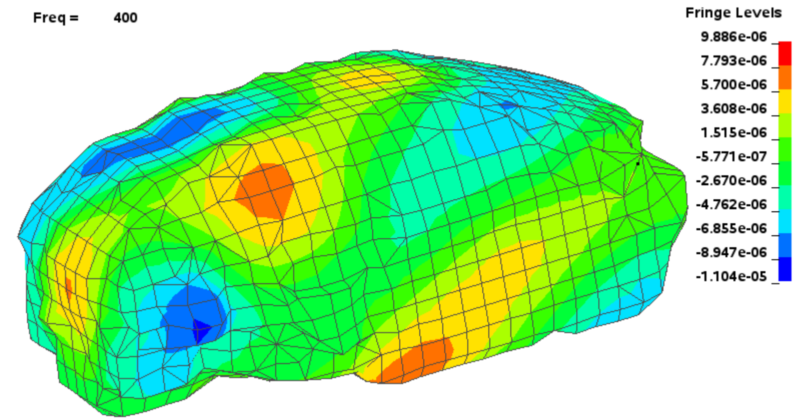
D3ACS for BEM acoustics

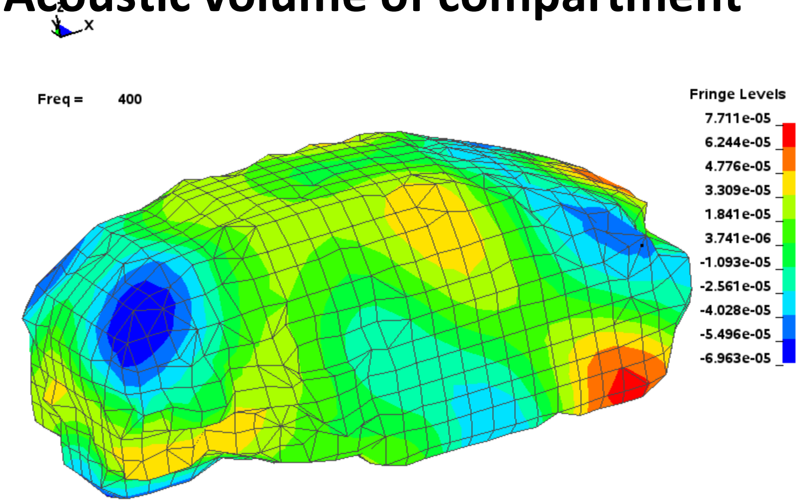
***DATABASE_FREQUENCY_BINARY_D3ACS**





Acoustic volume of compartment



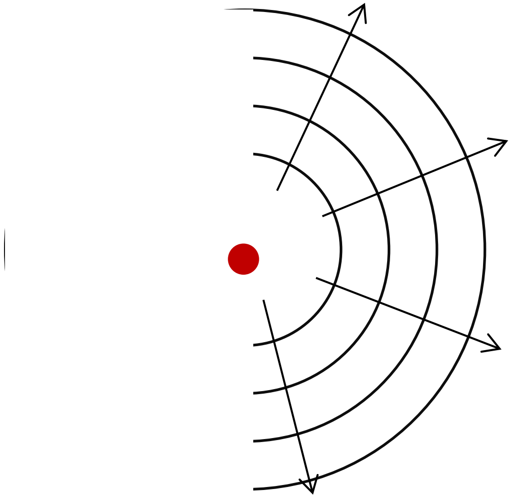


Real part of surface pressure

Imaginary part of surface pressure

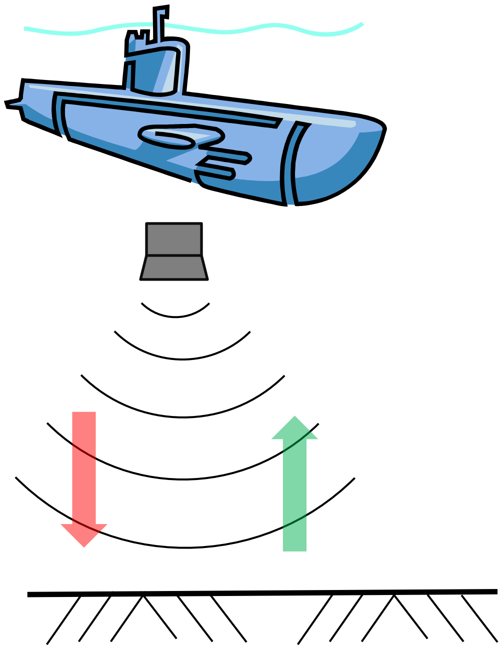
Incident waves for acoustic analysis

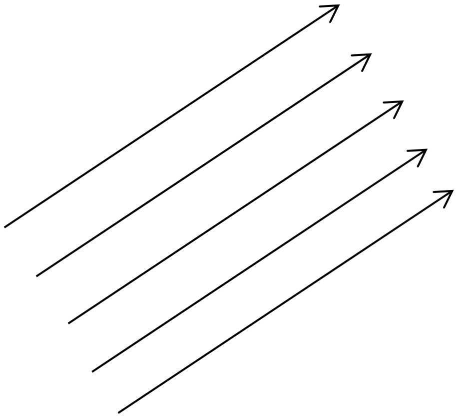
***FREQUENCY_DOMAIN_ACOUSTIC_INCIDENT_WAVE**



Spherical wave

$$p^i = A\frac{e^{-ikR}}{R}$$

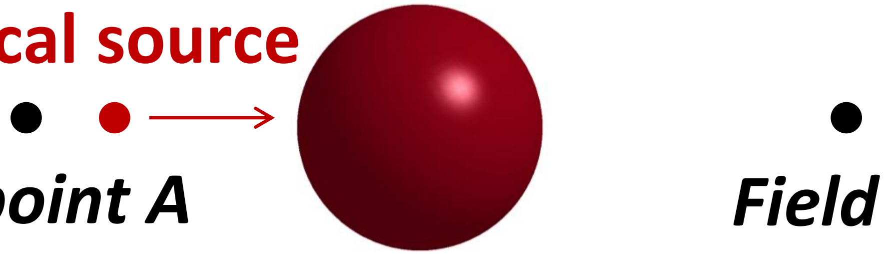




Plane wave

$$p^i = Ae^{-ik(\alpha x+\beta y+\gamma z)}$$

Example: sound scattering on rigid sphere

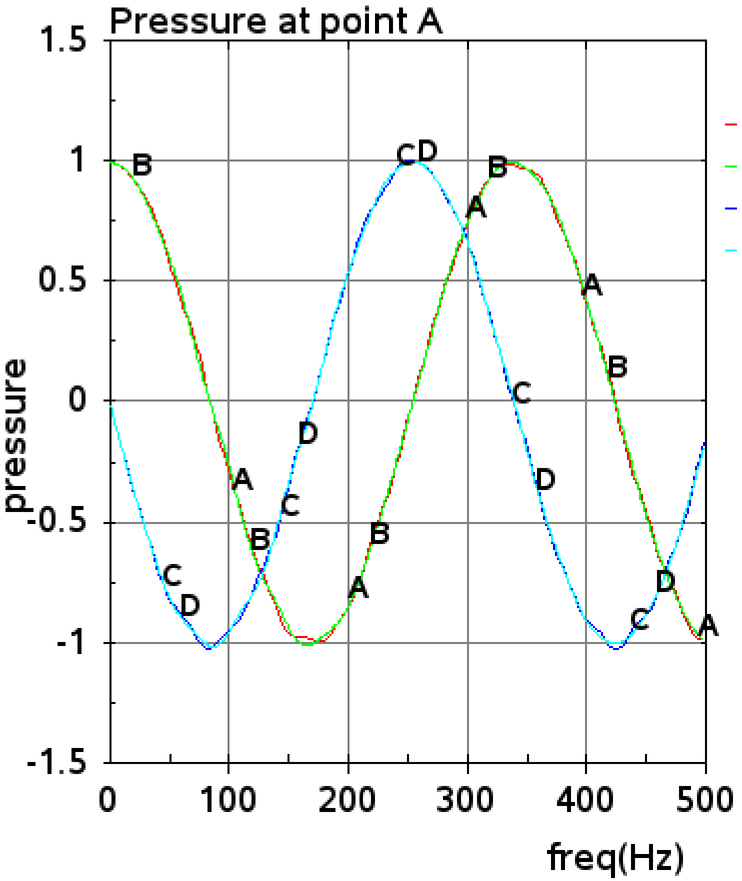


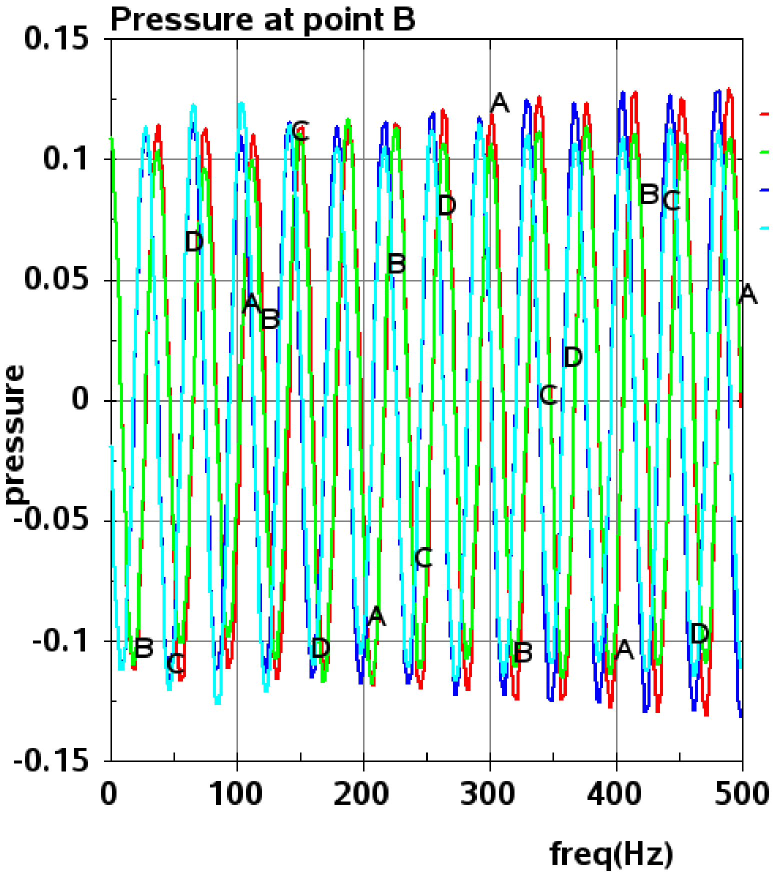
Spherical source

Field point A

Field point B

- A Real part (LS-DYNA)
- B Real part (Analytical)
- C Imaginary part (LS-DYNA)
- D Imaginary part (Analytical)



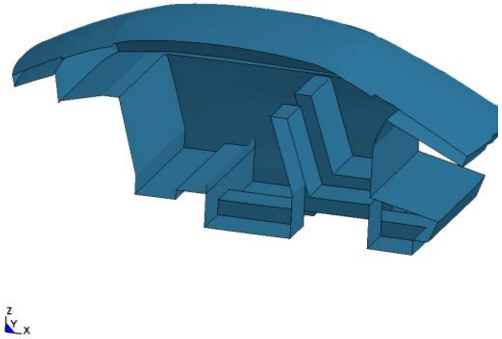


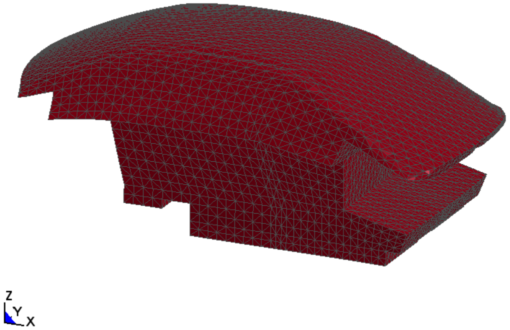
FEM Acoustics

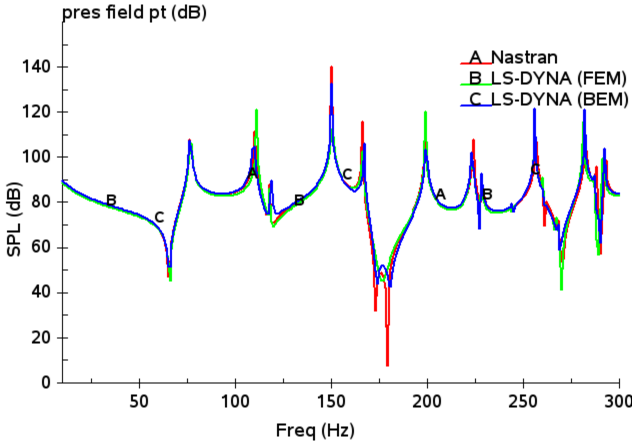
*FREQUENCY_DOMAIN_ACOUSTIC_FEM

- An alternative method for acoustics. It helps predict and improve sound and noise performance of various systems. The FEM simulates the entire propagation volume -- being air or water
- Compute acoustic pressure and SPL (sound pressure level)
- Output frequency range dependent on mesh size
- Available elements: hexahedron, tetrahedron, and pentahedron

Output: D3ACS; PRESS_PA; PRESS_DB





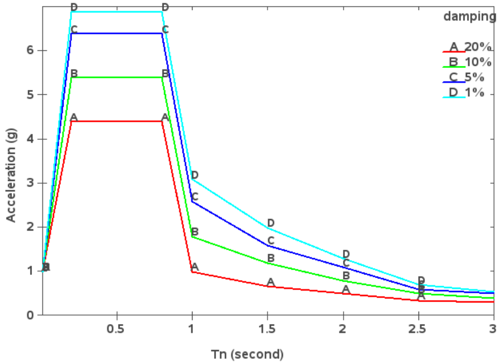


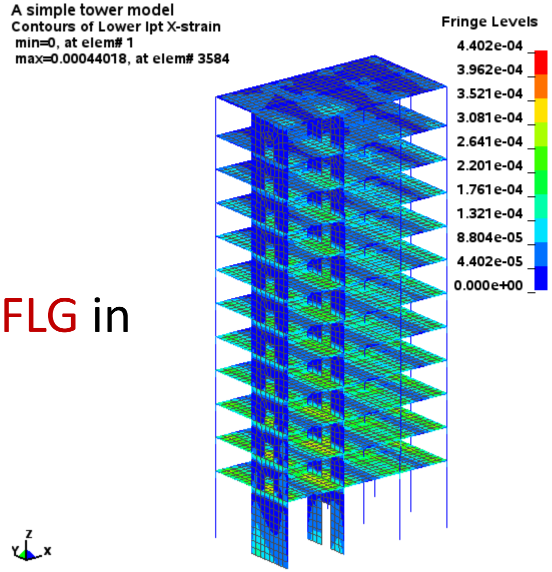
Response spectrum analysis

*FREQUENCY_DOMAIN_RESPONSE_SPECTRUM

- Various mode combination methods
  - SRSS method
  - NRC Grouping method
  - CQC method
  - Double Sum methods
  - NRL SUM method
- Evaluate peak response of structure
- The input spectrum is dependent on damping (using ***DEFINE_TABLE** to define the series of excitation spectrum corresponding to each damping ratio).
- Application: *earthquake engineering,* *nuclear power plants design* etc.
- Strain results are obtained by turning on STRFLG in *DATABASE_EXTENT_BINARY.

Output: D3SPCM



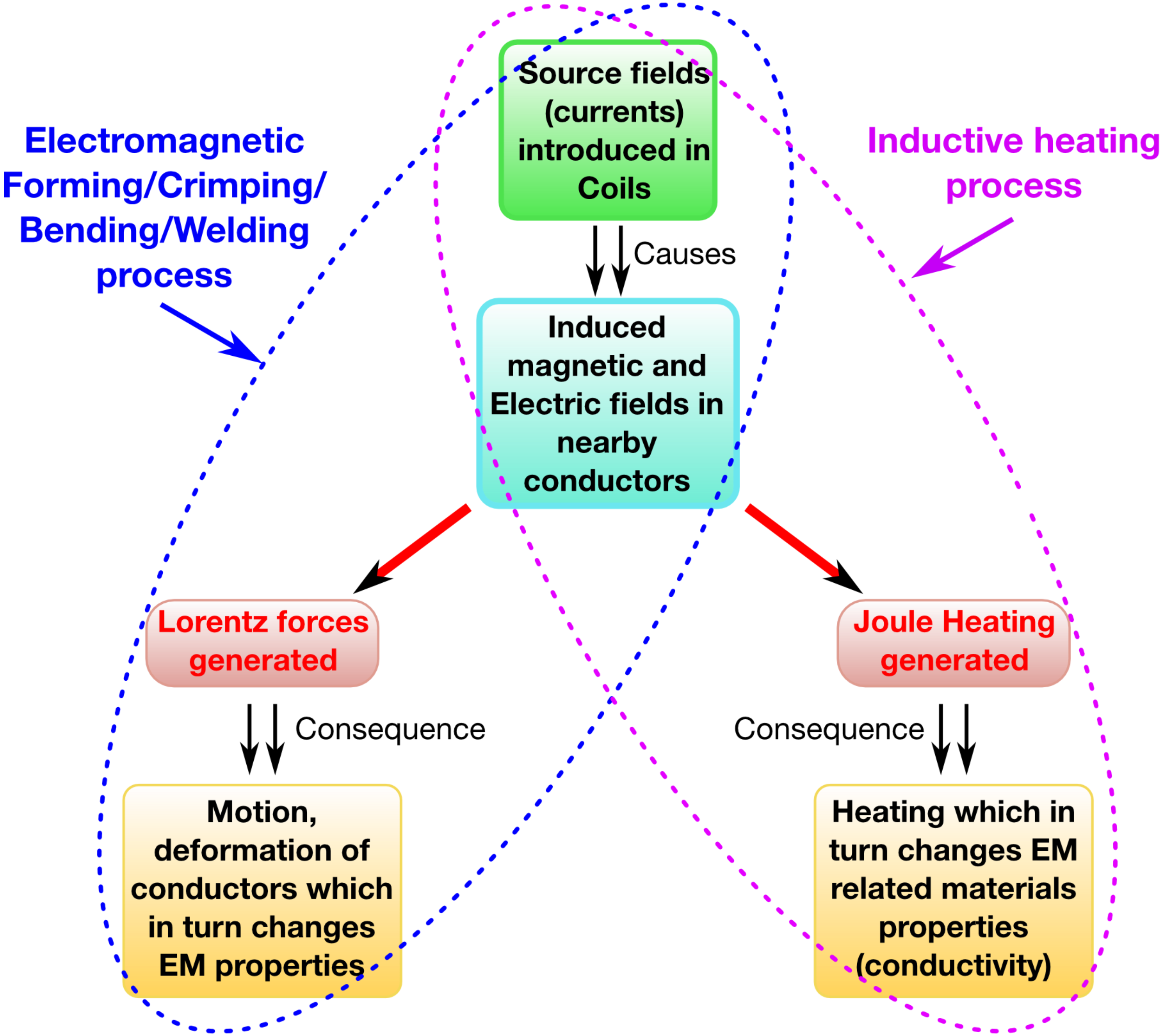


Thank You!

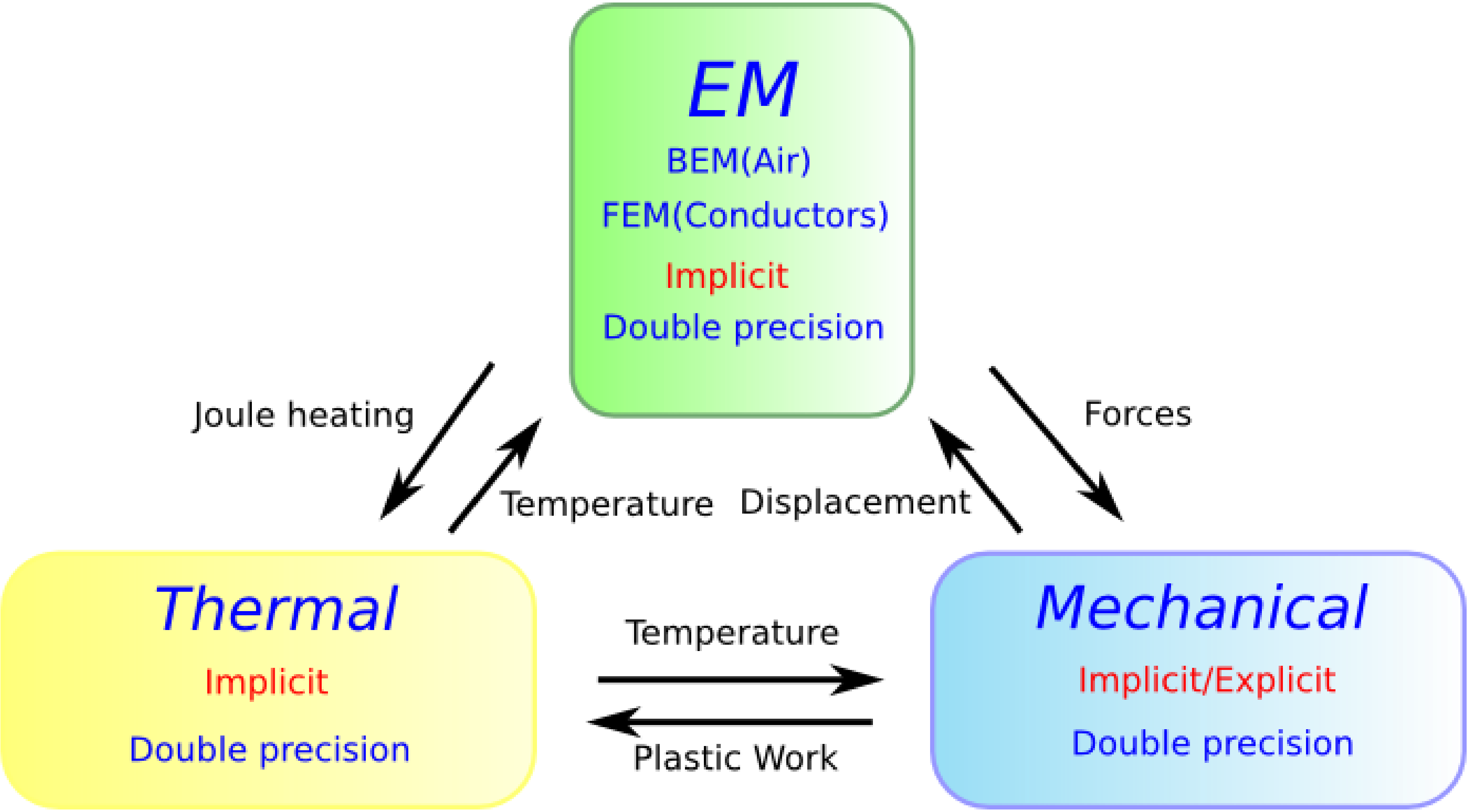
EM, ICFD & LS-PrePost

Iñaki Çaldichoury

EM Solver Introduction



EM Solver Introduction



EM

BEM(Air)

FEM(Conductors)

Implicit

Double precision

Joule heating

Temperature

Displacement

Forces

Thermal

Implicit

Double precision

Temperature

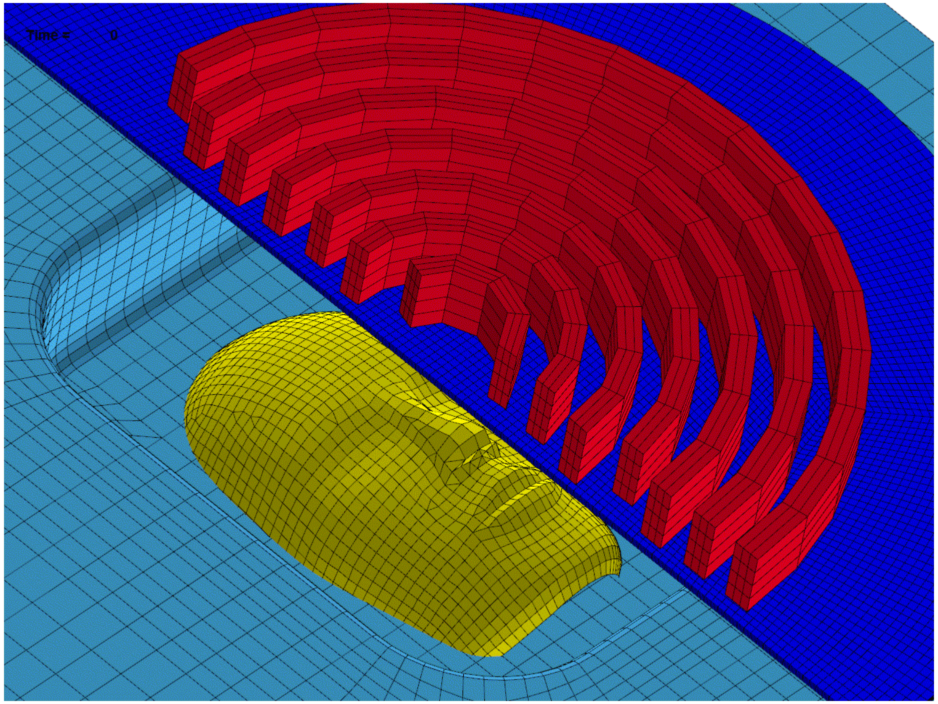
Plastic Work

Mechanical

Implicit/Explicit

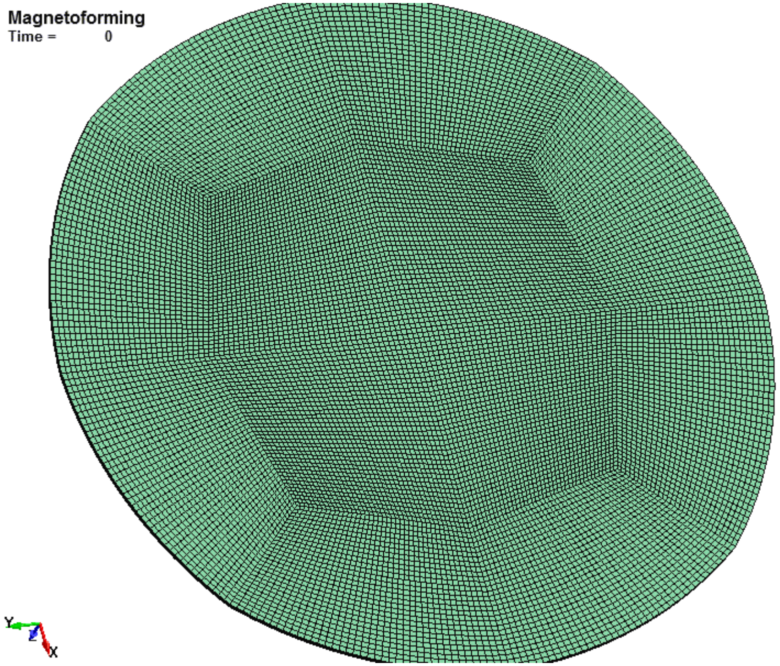
Double precision

EM Solver Introduction



MMF: High velocity forming process

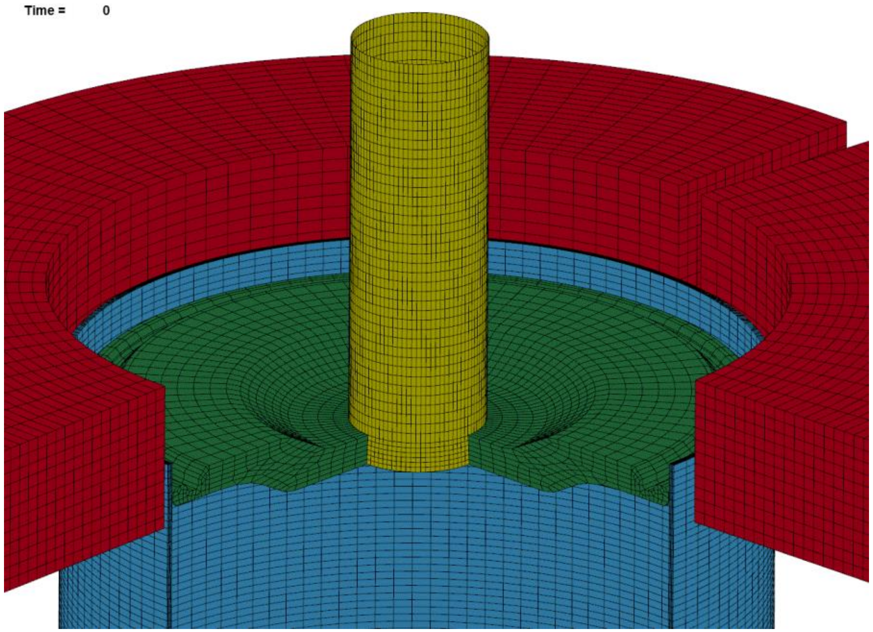
- Forming limits increased
- Springback reduced
- Wrinkling reduced
- High reproducibility



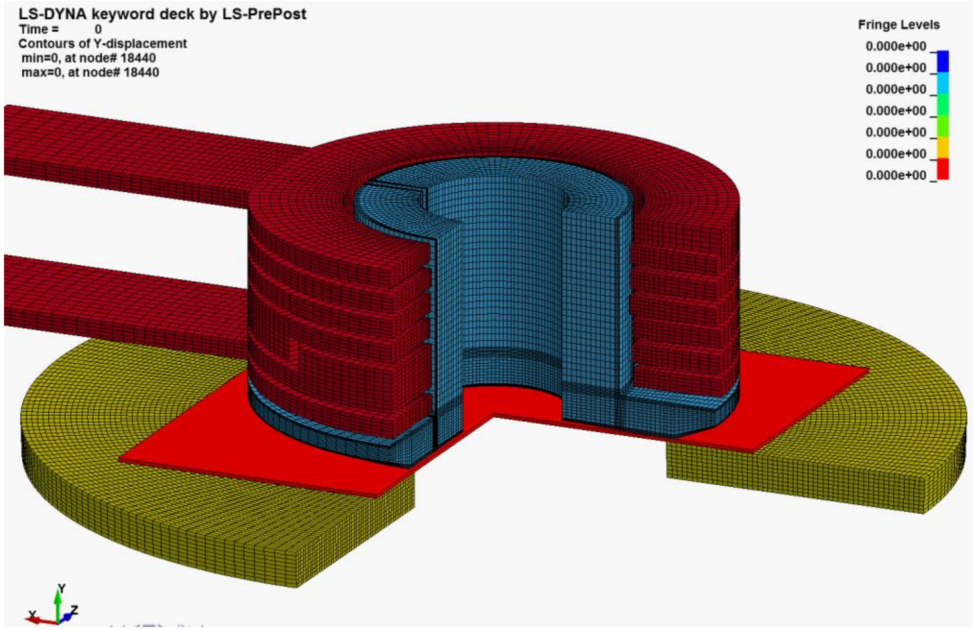
In collaboration with:
G. Mazars & G. Avrillaud:
Bmax, Toulouse, France

Bmax
shaping your future

EM Solver Introduction



Magneto crimping



Magneto forming

Fraunhofer
IWU

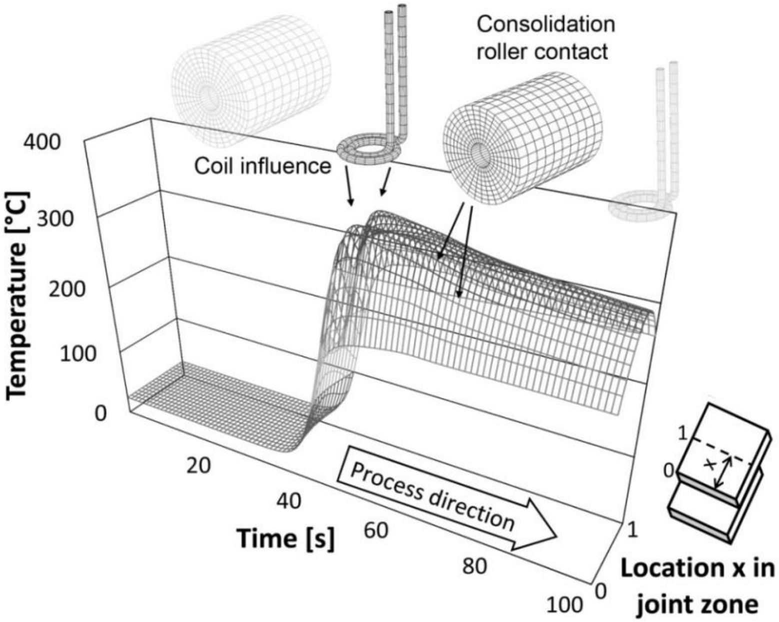
In collaboration with:
G. Mazars and G. Avrillaud:
Bmax, Toulouse, France

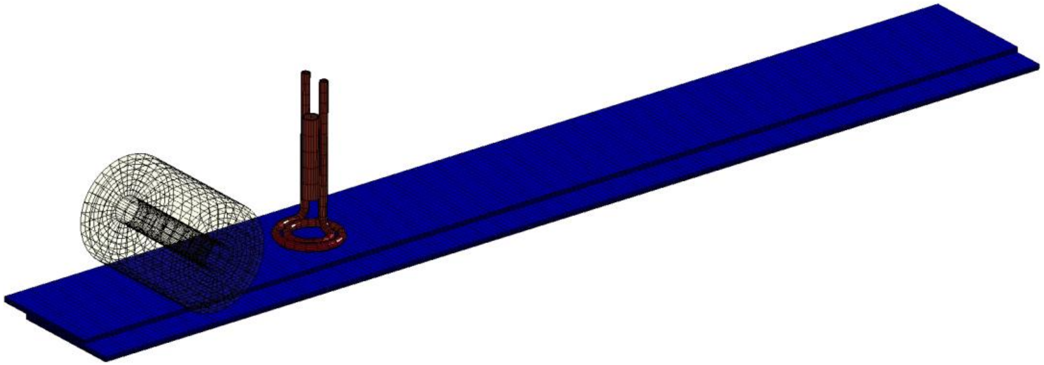
Bmax
shaping your future

In collaboration with:
Christian Scheffler, Chemnitz, Germany

EM Solver Introduction

Heating by Joule losses pressure applied for consolidation and maintained during cooling.



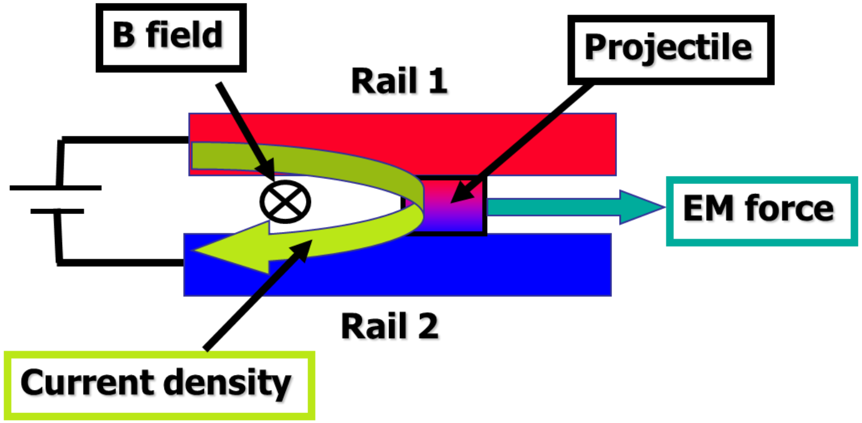


Institut für Verbundwerkstoffe

In collaboration with:
M. Duhovic, Institut für Verbundwerkstoffe, Kaiserslautern, Germany

EM Solver Introduction

Rail gun simulation


B field

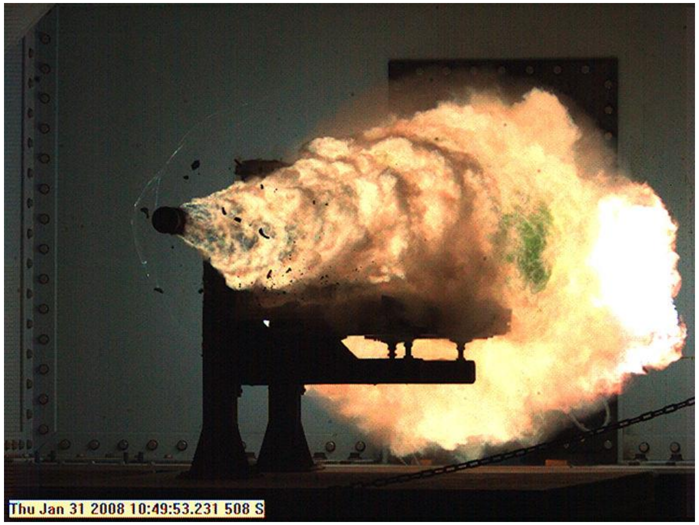
Rail 1

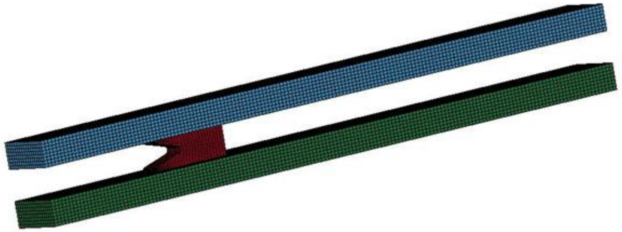
Projectile

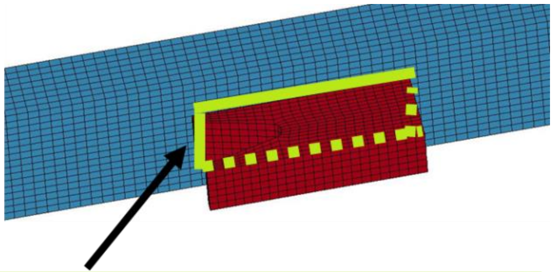
EM force

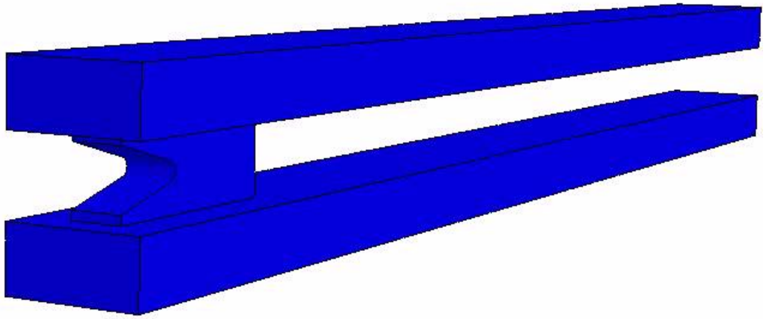
Rail 2

Current density



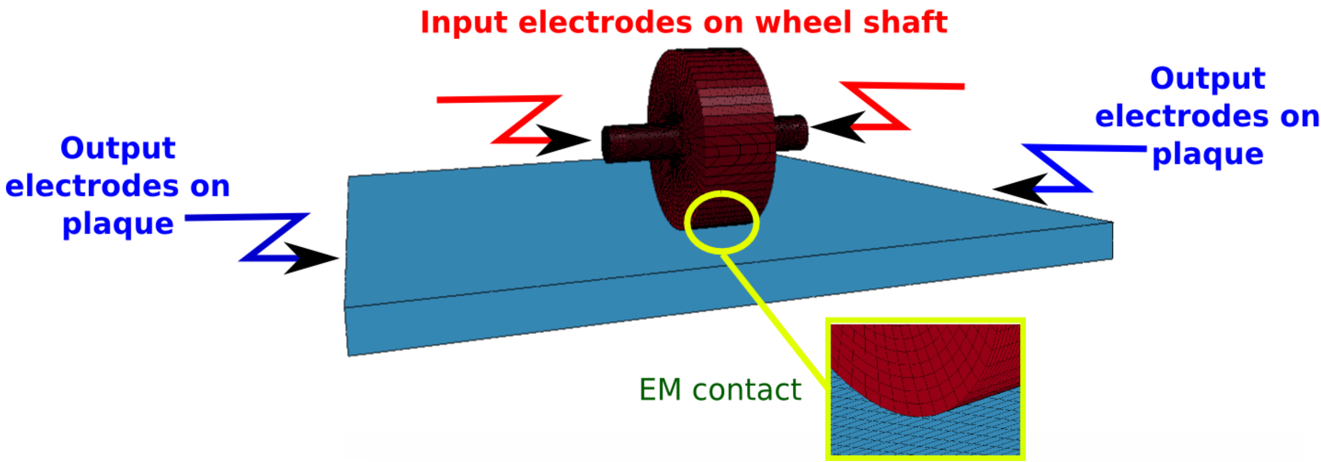


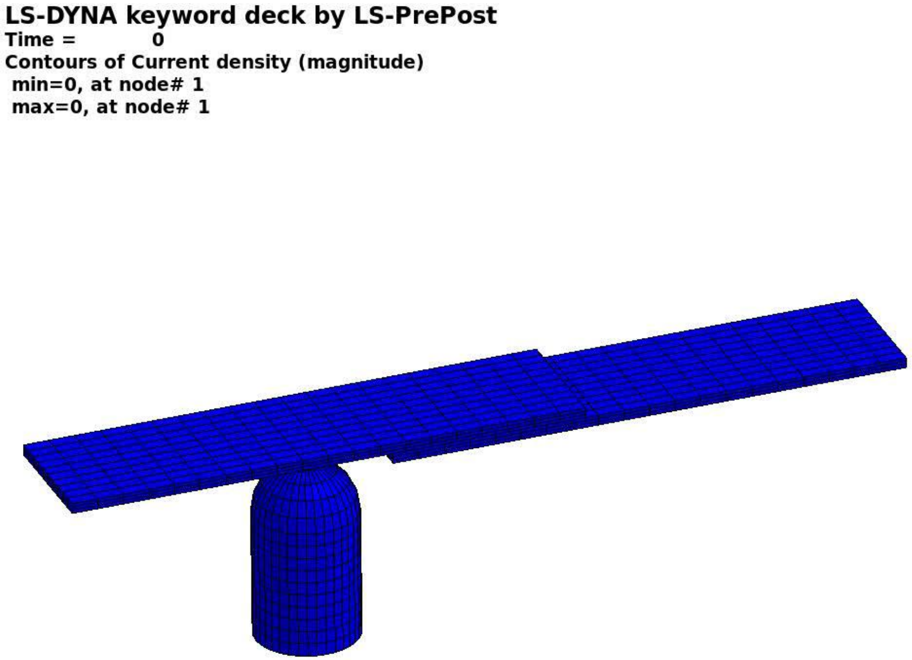

Sliding contact between the rails and the projectile



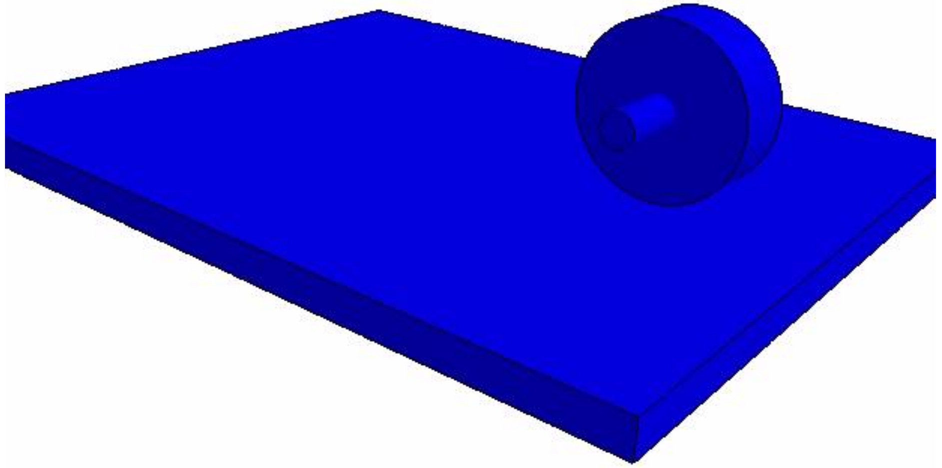
EM Solver Introduction

- **Joule heating study**
- **Study of shortcuts**

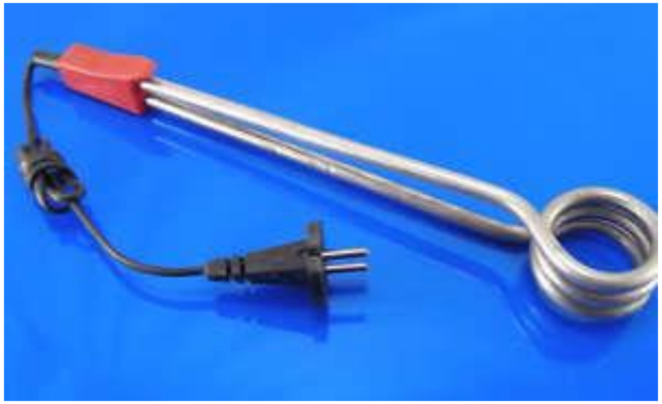




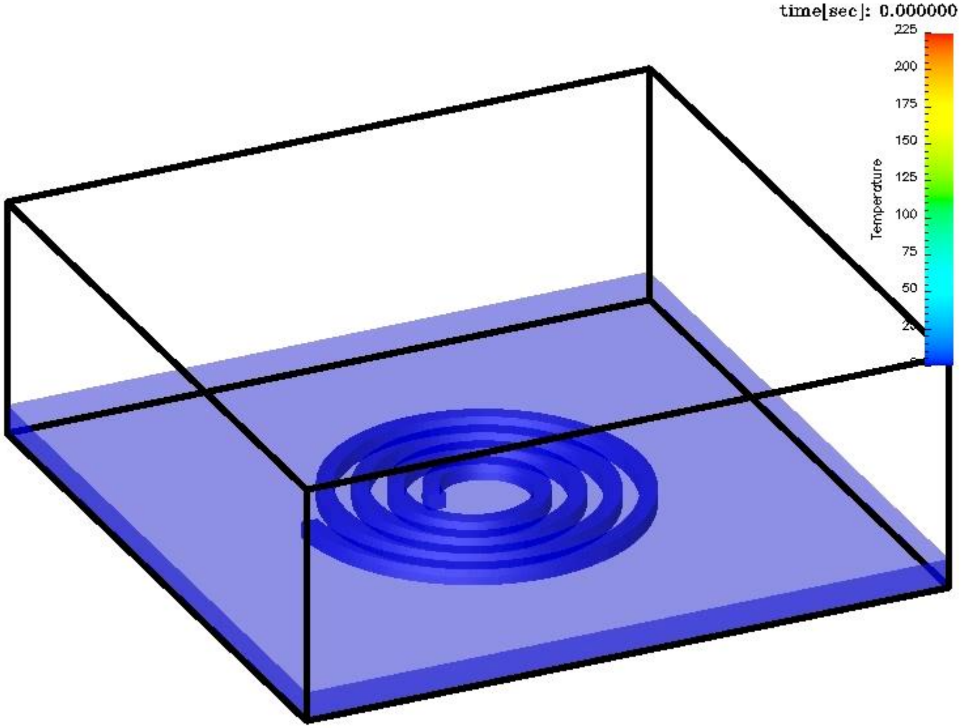
```
LS-DYNA keyword deck by LS-PrePost
Time =          0
Contours of Current density (magnitude)
 min=0, at node# 1
 max=0, at node# 1
```



EM Solver Introduction



- **EM heats up a coil plunged in a kettle**
- **ICFD with conjugate heat transfer heats up the water**



Water stream lines colored by the temperature level.

EM Solver
New Developments

EM Solver Introduction

Application: Conducting shells

Development : Allowing user to define conductive shells in 3D problems.

Advancement stage : Conducting shells can be used in the development version with the Eddy Current, Inductive heating and Resistive heating solvers. EM contact also available. SMP and MPP versions fully functional.

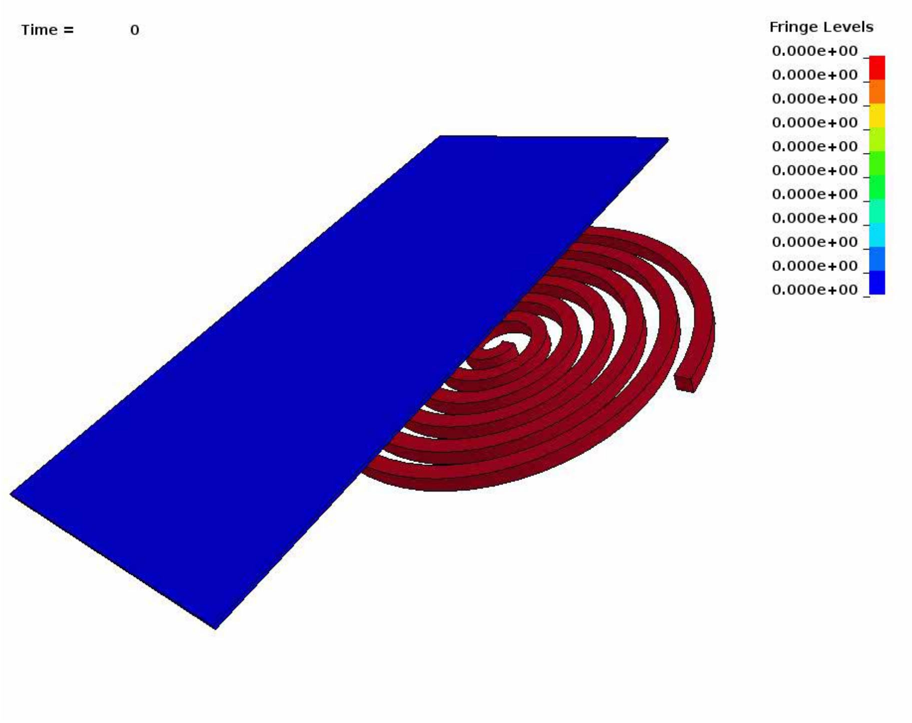
- **Eddy currents i.e the combination of inductive diffusive effects is essentially a 3D effect which means that elements with thickness are usually required for correct solve. Consequently, up to now, only solid conductive elements were allowed, shells could be insulators only.**

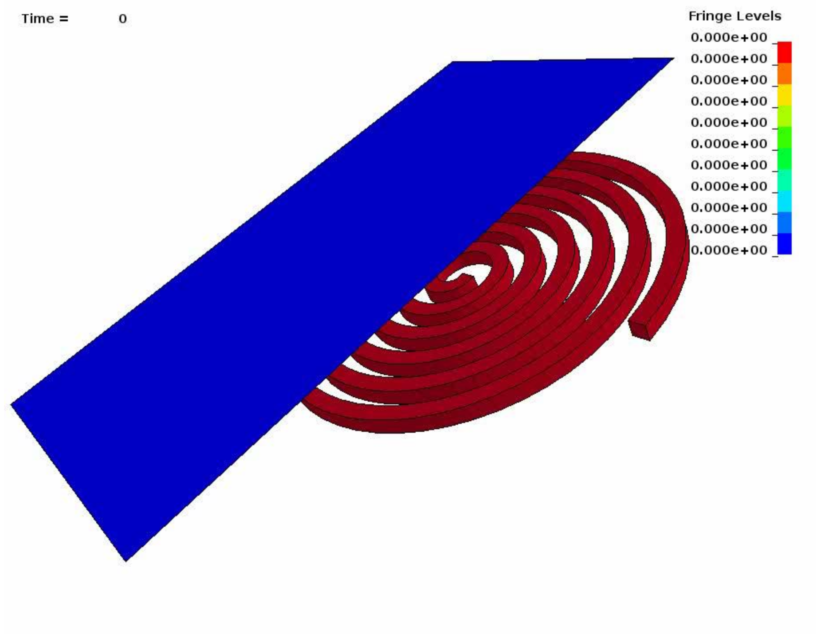
- **However, some users have expressed the wish to be able to use conductive shells in 3D problems in order to maintain their associated mechanical and thermal properties.**

- **From the EM perspective, those shells are treated like “invisible” solids i.e the EM solver will build an underlying equivalent solid mesh to solve for the EM quantities.**

EM Solver Introduction

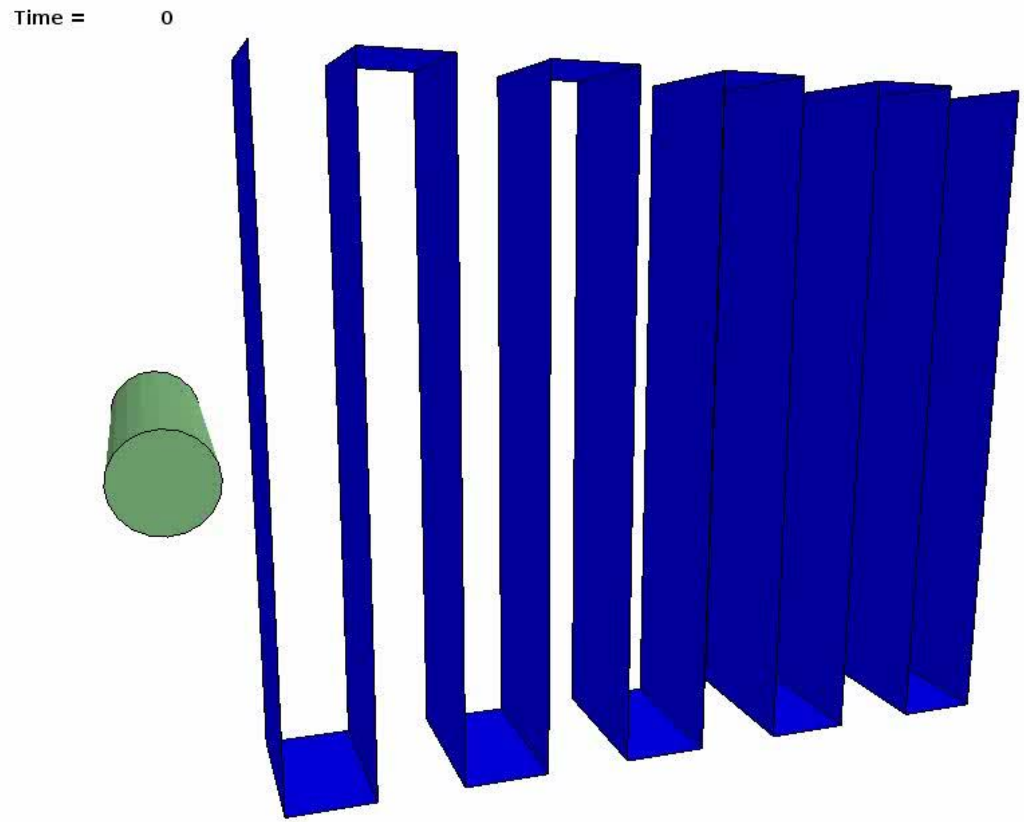
EM forming using conductive shells vs solids





EM Solver Introduction

Impact against battery : shortcut study



EM Solver Introduction

Application: Resistive spot welding and others

Development: Calculation of a additional resistance term and local Joule heating due to contact occurring between two conductors.

- **Resistive spot welding is a process in which contacting metal surfaces are joined by the heat obtained from resistance to electric current.**
- **Frequently encountered in the automobile manufacturing industry where it is used almost universally to weld metal sheets together (often automated process).**
- **Several formulas exist, often variation of Holm's law:**

$$\boxed{R_{contact} = \frac{\rho}{2a}}$$

With ρ the material's resistivity and a the radius of the equivalent contact circle area.

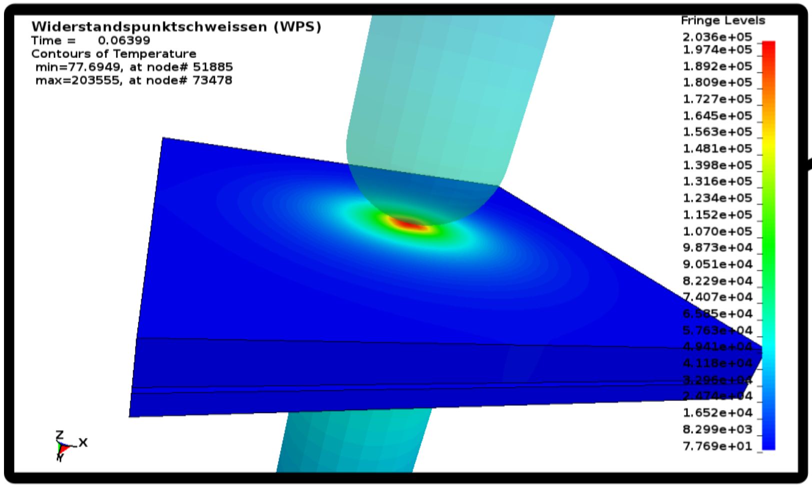
EM Solver Introduction

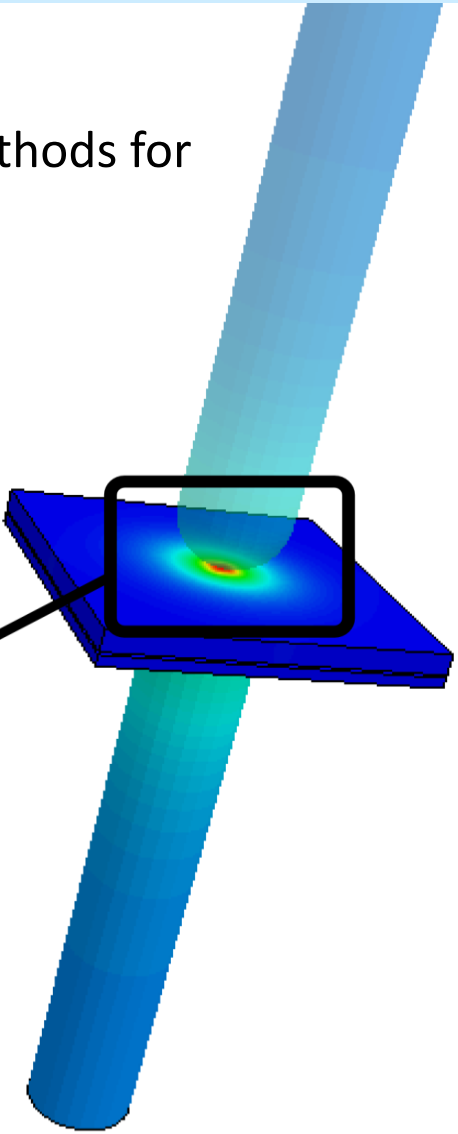
Application: Resistive spot welding and others

Status: Available in the development version. Several methods for calculating contact Resistance available. See *EM_CONTACT_RESISTANCE card.

Local heating spot between electrodes and work piece due to Contact Resistance

$$R_{contact} = \frac{\rho}{2a}$$





EM Solver Introduction

Application: magnetic field lines

Development: Have the solver compute some magnetic field lines in the air based on user defined parameters for analysis and post treating purposes.

- **The use of the BEM method i.e no air mesh does not allow the visualization of the EM fields in the air hence the interest of this new feature.**

- **From a starting point given by the user the field line is computed step by step using a explicit numerical integration scheme (RK4, DOP853).**

- **Approximation methods are available in order to speed up the computation of the second member of the magnetic field line equations :**
  - **multipole method**
  - **"multicenter" method: this method has been developed at LSTC**

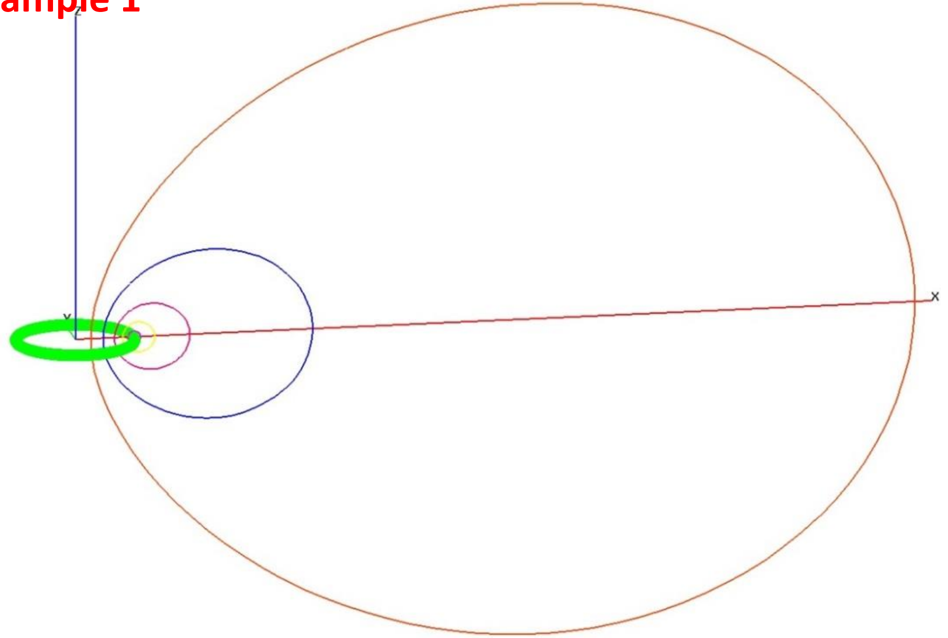
- **This feature along with the multicenter/multipole methods are treated as a research project that could be used at a latter stage to compute and store efficiently the BEM matrix and thus speed up the computational time of the EM solver.**

EM Solver Introduction

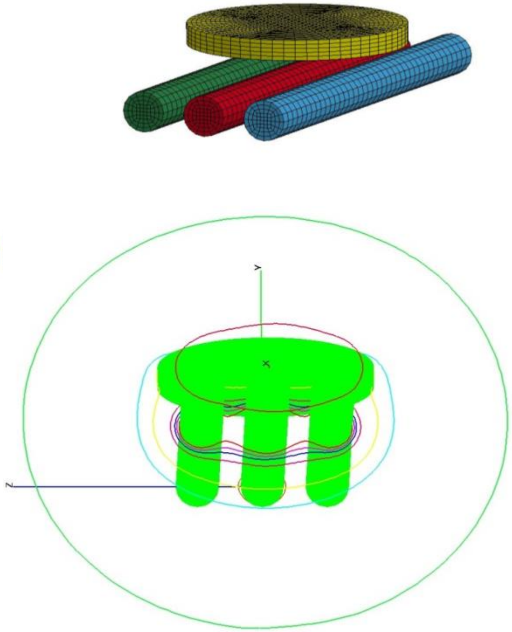
Application: Magnetic field lines

Current status: The magnetic field lines are now exported as individual lsprepost readable files at each output state, and will soon be integrated in the d3plot files. In a future development cycle, they will be automatically generated in lsprepost at any time without the user having to specify the output times before the run.

Example 1



Example 2



EM Solver Introduction

Future Developments

- Symmetry conditions
- Piezo electric materials
- Magnetic materials

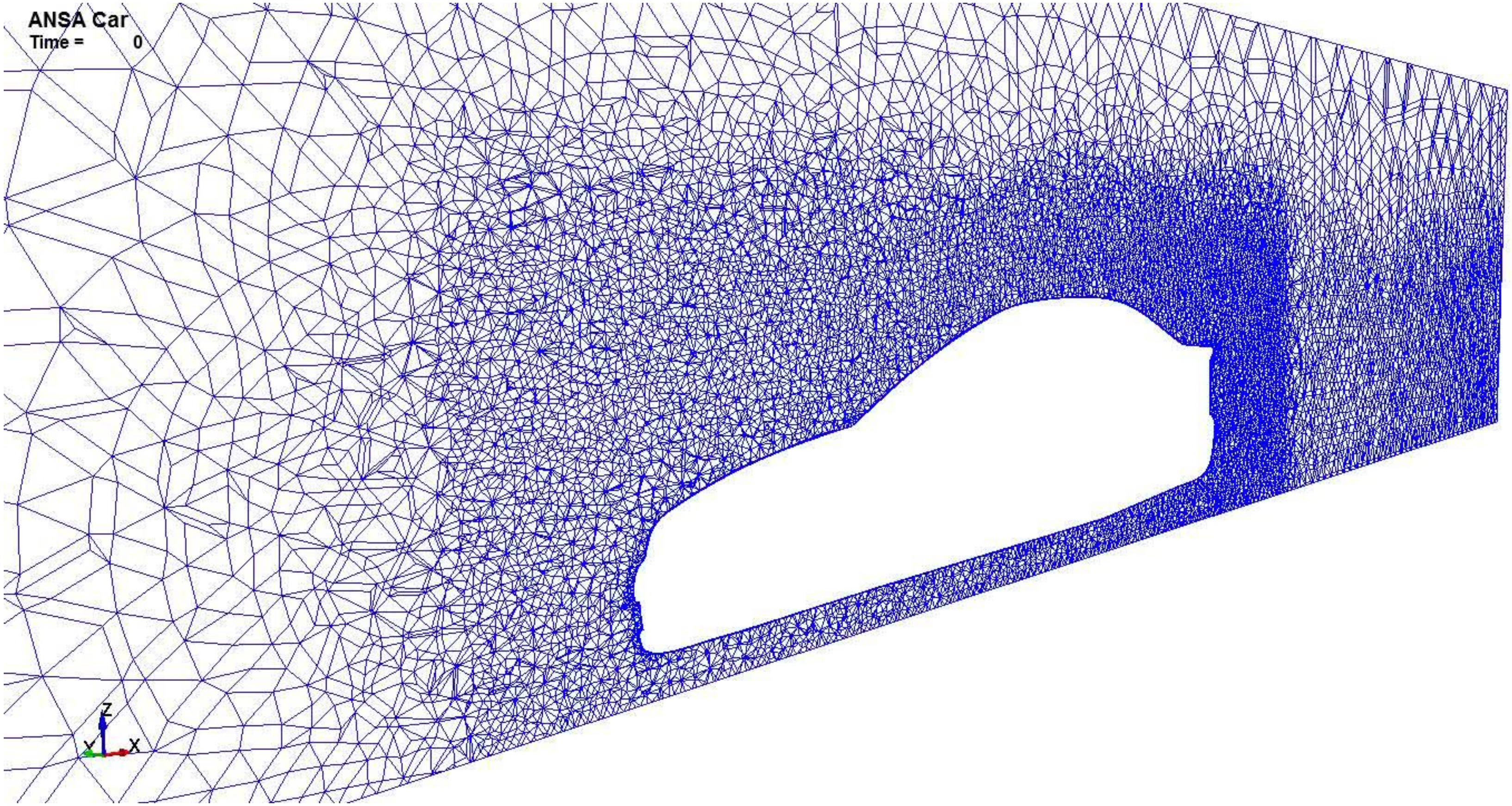
ICFD Solver

ICFD Solver Introduction

- **A CFD solver for incompressible flows (ICFD solver).**
- **Fully implicit.**
- **Double precision.**
- **SMP and MPP versions available. Highly scalable in MPP.**
- **New set of keywords starting with *ICFD/*MESH.**
- **Can run as a stand alone CFD solver.**

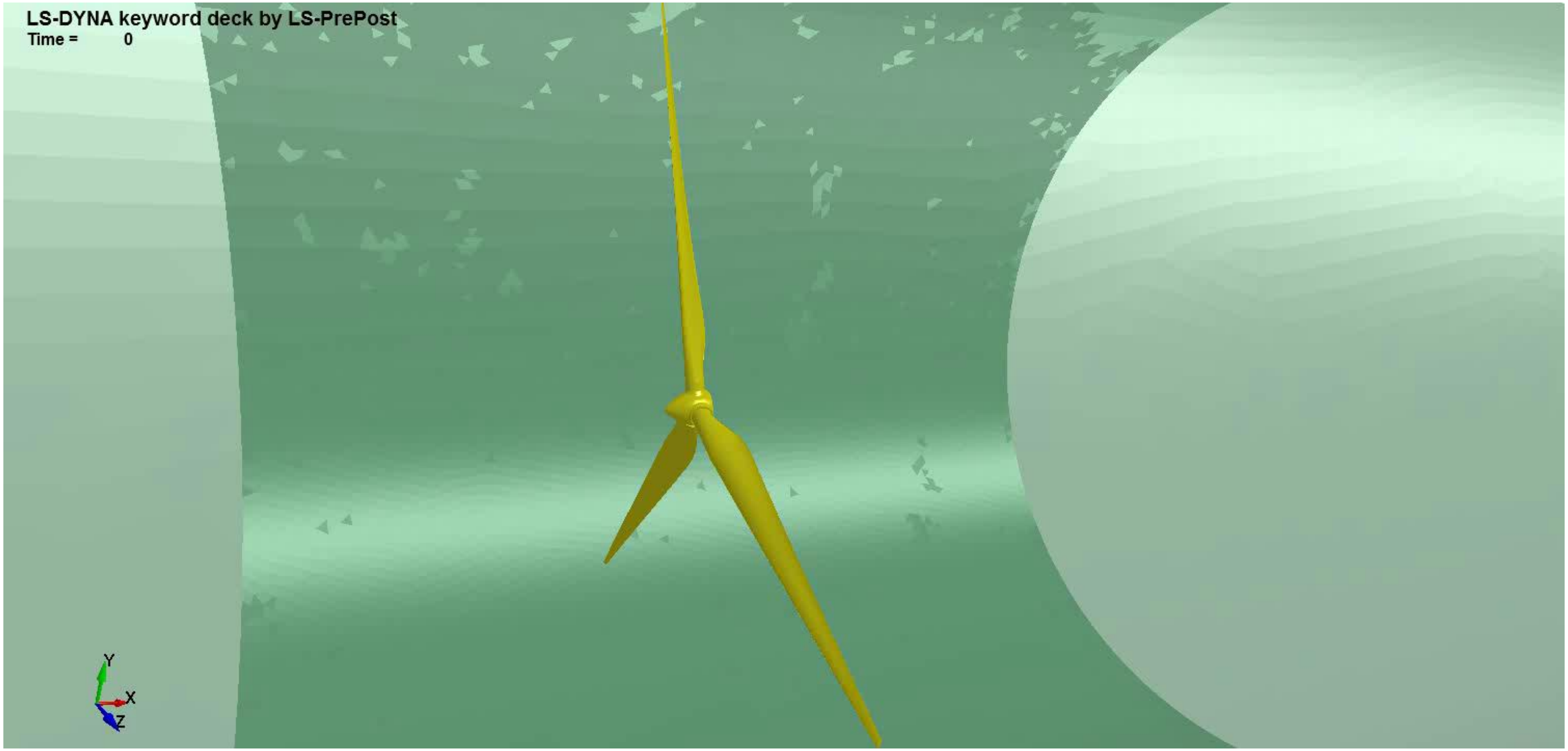
ICFD Solver Introduction

Aerodynamics: study of turbulent flow around a car



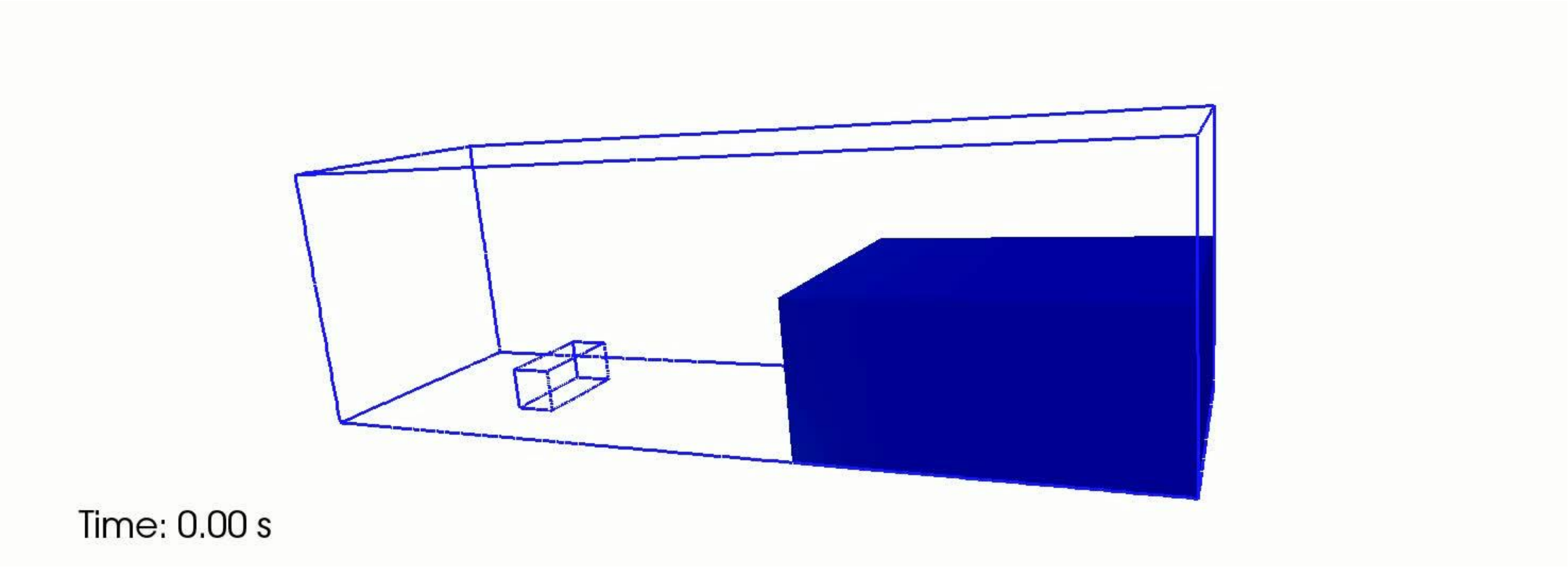
ICFD Solver Introduction

Aerodynamics: study of flow vortexes around a wind turbine



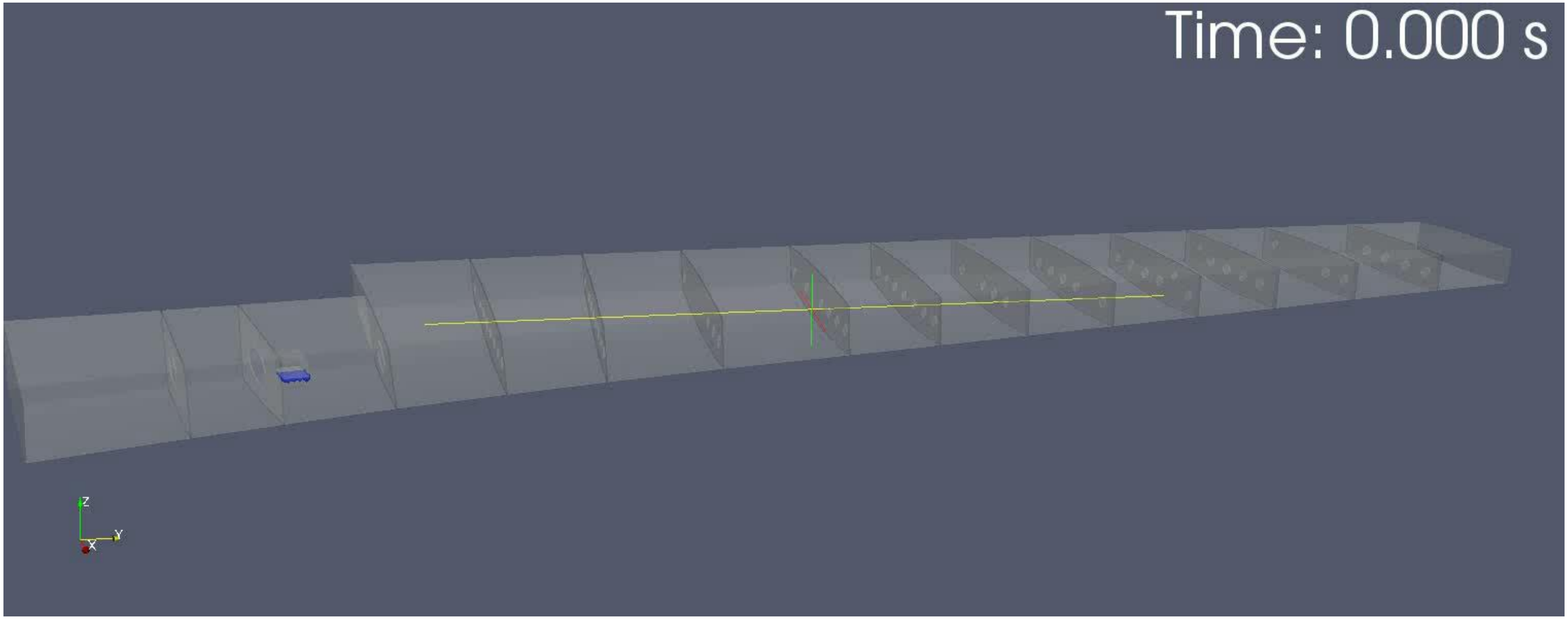
ICFD Solver Introduction

Wave impact: study of pressure forces on a body



ICFD Solver Introduction

Aircraft wing filling: study of fuel repartition and filling time



ICFD Solver Introduction

Thoughts about FSI

- A widely approach used in engineering is to assume that the FSI problem is linear and to use two different software products and licenses.

- The solid work group finds the need for FSI simulation.

- The geometry is sent to the fluid group which builds a mesh and runs the fluid problem with a CFD solver until it reaches steady state.

- The fluid stresses together with the mesh is brought back to the solid group which handles data with scripts to convert it into the input data for the solid solver.

- The solid solver performs a modal analysis.

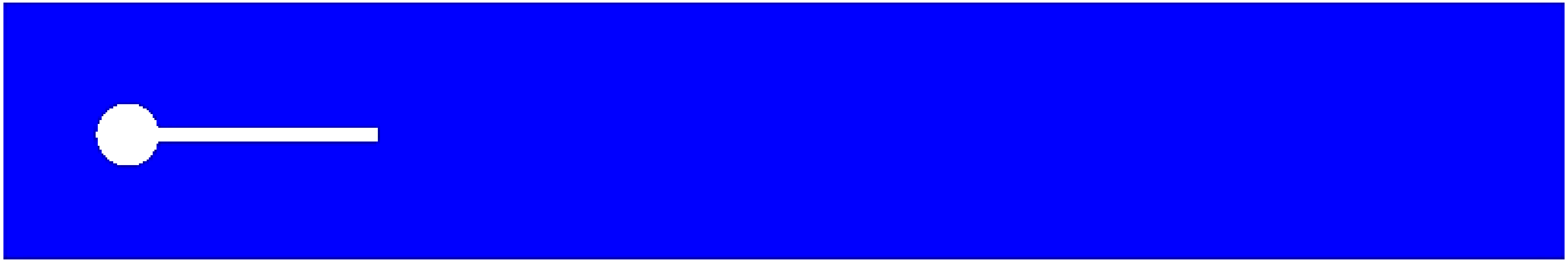
ICFD Solver Introduction

The LS-DYNA approach

- LS-DYNA has immense solid mechanics capabilities as well as a huge material library. It can run both in explicit and implicit and already has a thermal solver for solids.
- LS-DYNA offers the perfect environment in order to develop a CFD solver allowing complex fluid structure interactions as well as the solving of conjugate heat transfer problems.
- The set up of the coupled problem is greatly simplified with only a few additional keywords necessary.
- On top of the classic “loose” or “explicit” FSI coupling, the ICFD solver offers a state of the art strong coupling method which opens up new applications.
- “All in one code” strategy.

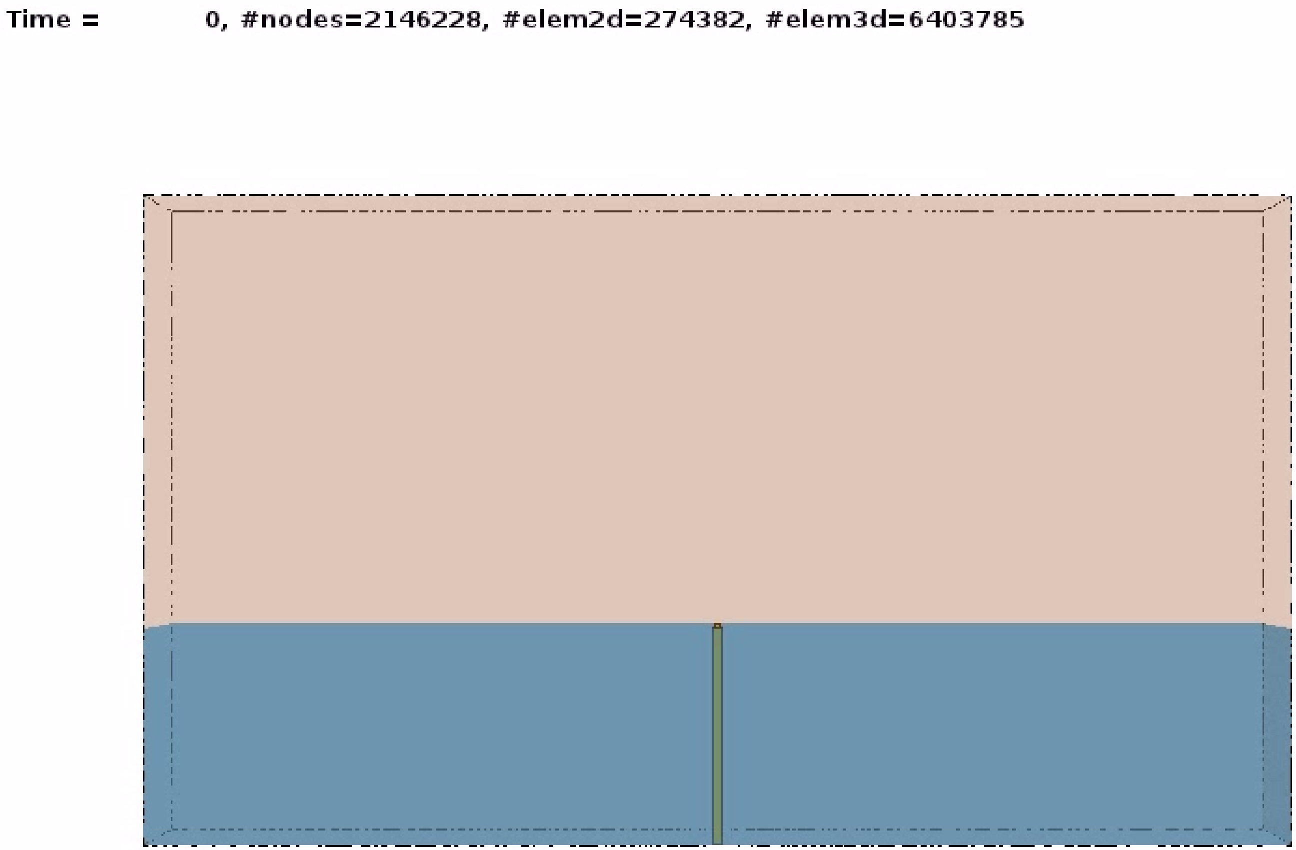
ICFD Solver Introduction

Aerodynamics: highly non linear FSI problem with flap oscillating in the wind *(Turek benchmark problem)*



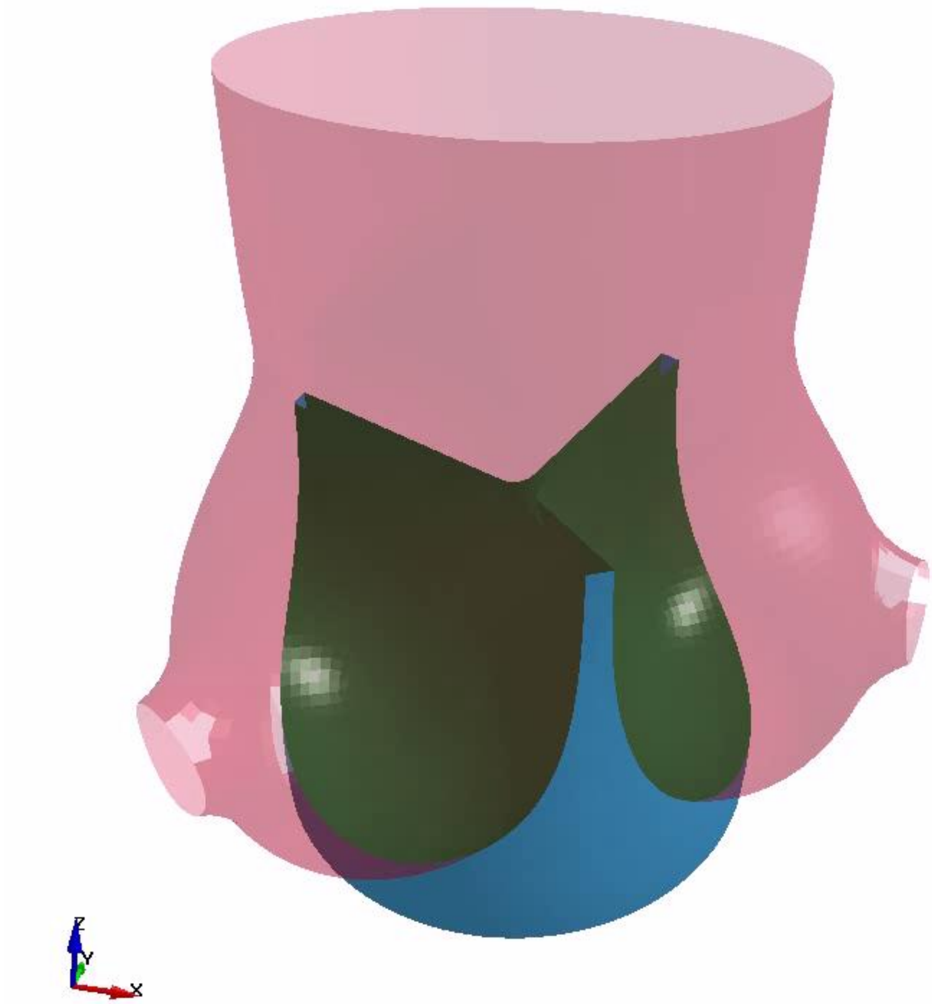
ICFD Solver Introduction

Sloshing: Flap oscillating in partially filled tank



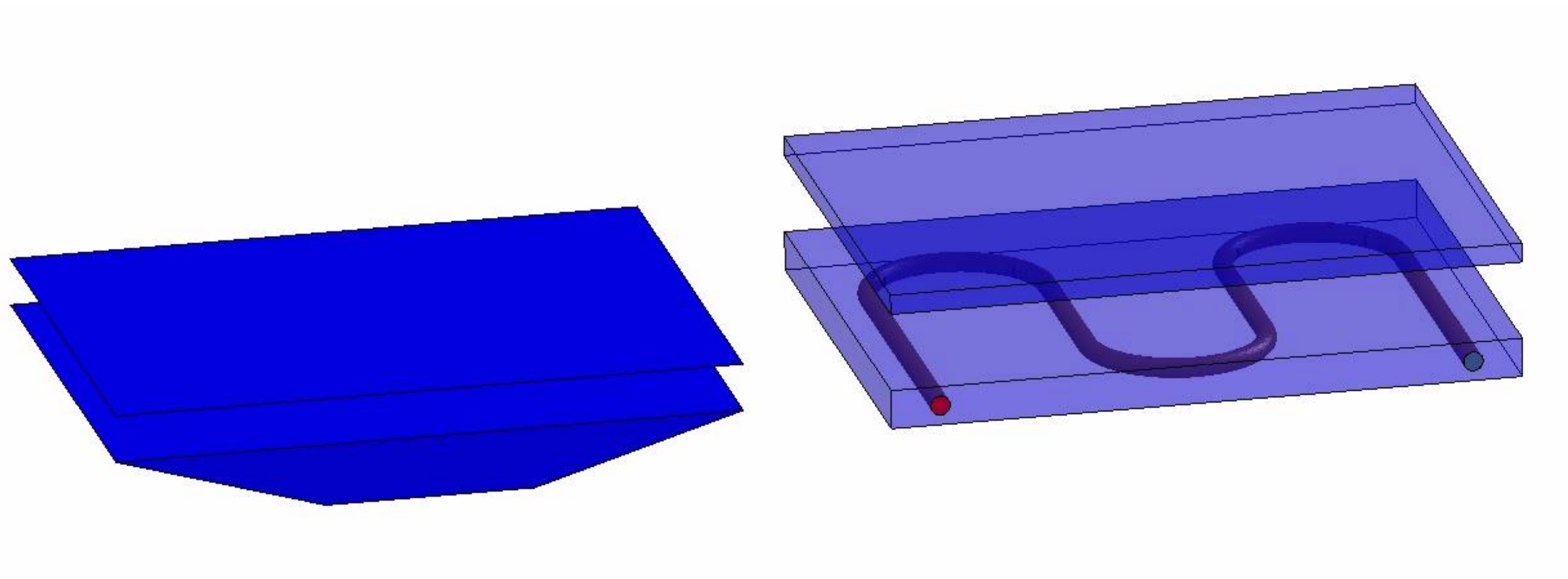
ICFD Solver Introduction

Artificial Heart valve: Strong pressure gradients forces leaflets open



ICFD Solver Introduction

Stamping and conjugate heat transfer:
flow in serpentine causes dye to cool off



ICFD Solver
New Developments

ICFD Solver Introduction

Application: Multiphase problems

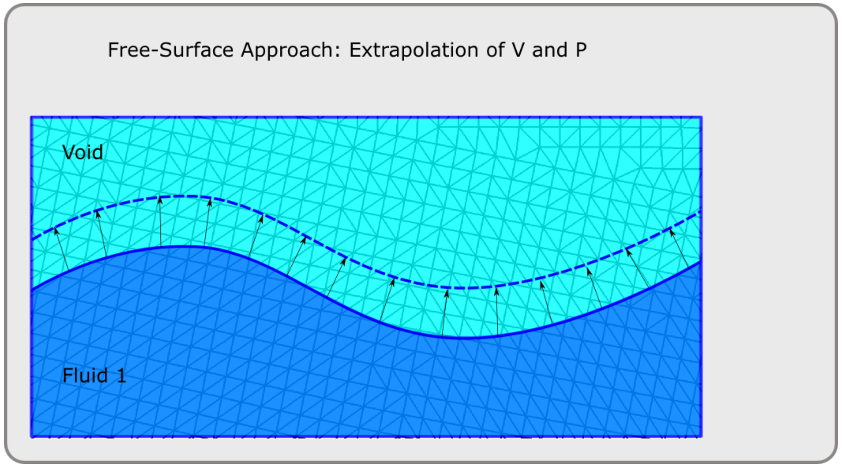
Development: Being able to solve problems with two fluids of different densities (water+air).

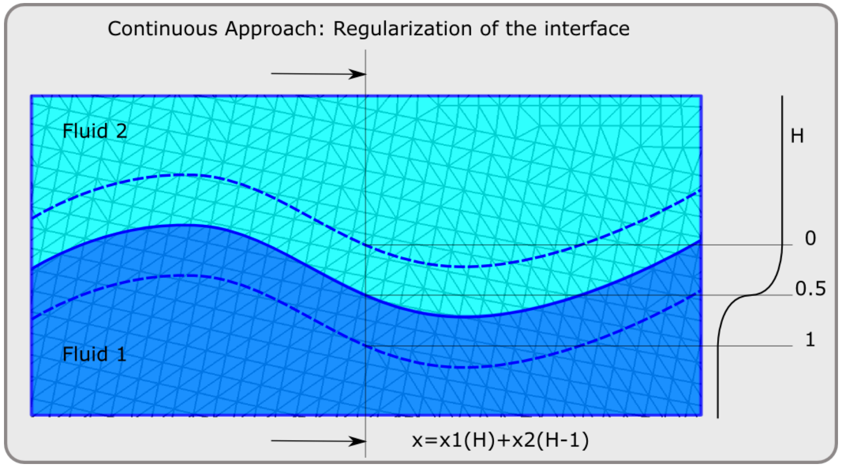
Current Status: Implementation stage.

- **Numerous applications such as lubrication, droplets, sloshing in closed tanks, etc.**
- **Two immiscible phases.**
- **Level Set approach for interface tracking.**
- **Continuous and discontinuous approach to model pressure jumps (surface tension).**
- **When the inertial effects of the second fluid can be simplified, the Free surface approach can be used.**

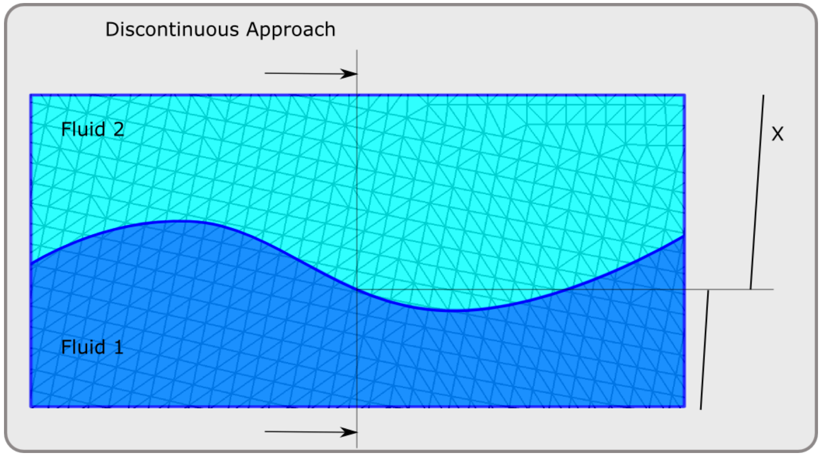
ICFD Solver Introduction

Application: Multiphase problems





- **Free surface approach: suitable for problems where the inertial effects of the lighter fluid may be neglected.**
- **Continuous approach: works in most multiphase problems.**
- **Discontinuous approach: used in problems where surface tension effects are important.**



ICFD Solver Introduction

Application: Porous media

Development: Implementation of a generalization of the Navier Stokes equations that will allow the definition of sub-domains with different permeability/porosity.

Current Status: Validation stage. Available in the development version. See 4th card of ICFD_MAT.

Being ε, κ the porosity and the permeability of the medium respectively :

$$u_i = \varepsilon u_{if} \qquad \varepsilon = \frac{void\ volume}{total\ volume}$$

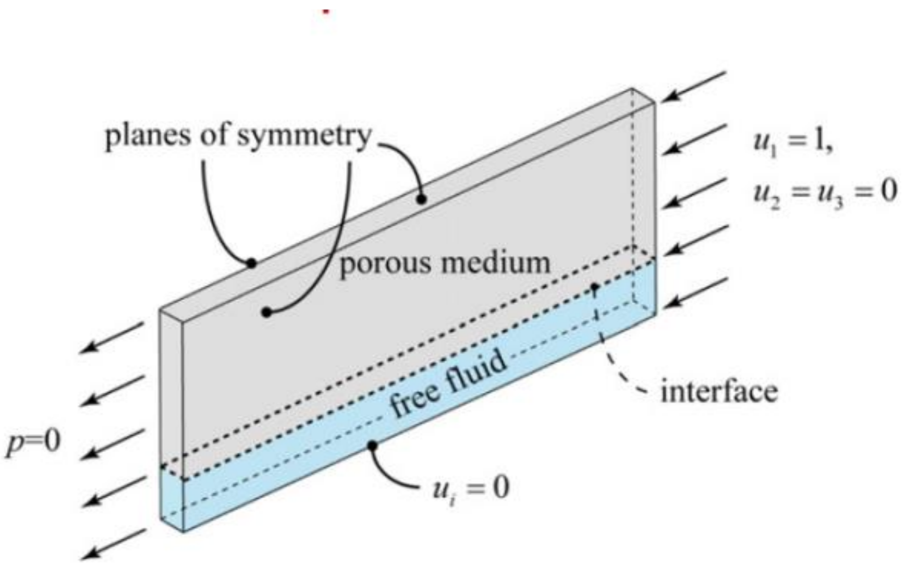
$$\frac{\rho}{\varepsilon}\left[\frac{\partial u_i}{\partial t} + \frac{\partial\left(\frac{u_i u_j}{\varepsilon}\right)}{\partial x_j}\right] = -\frac{1}{\varepsilon}\frac{\partial (P\ \varepsilon)}{\partial x_i} + \frac{\mu}{\varepsilon}\frac{\partial^2 u_i}{\partial {x_i}^2} + \rho g_i - D_i$$

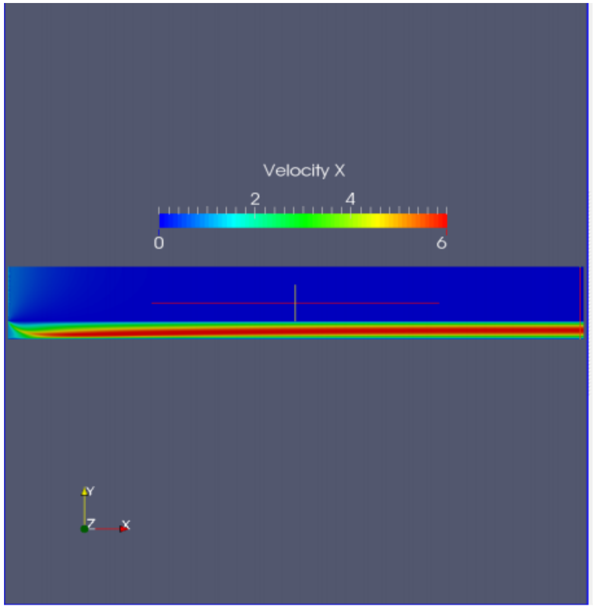
Ergun correlation : $$D_i = -\frac{\mu U_i}{\kappa} + \frac{1{,}75\rho|u|}{\sqrt{150}\sqrt{\kappa}\varepsilon^{1.5}} U_i \qquad \kappa = \tilde{\kappa} \text{ or } \kappa = \kappa_{ij}$$

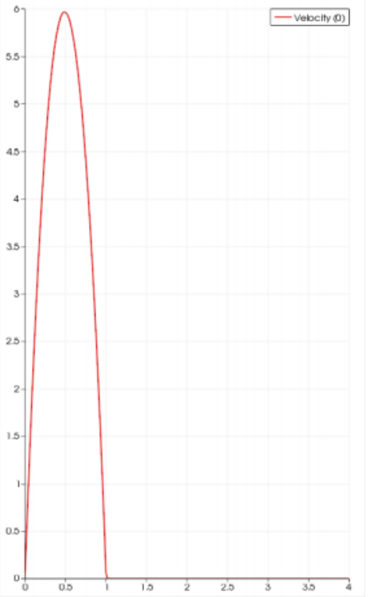
ICFD Solver Introduction

Application: Porous media

Validation stage: analysis of references cases involving porous and fluid domains. Study of FEM solution and analytical/reference solutions







ICFD Solver Introduction

Application: Thermal problems

Development: Calculation of the convection coefficient “h” based on a rigorous approach for the estimation of the bulk temperature Tm.

Current Status: Available in the Development version.

$$\boxed{h = \frac{q}{T_s - T_m}}$$

With q the heat flux, Ts the temperature at the surface and Tm the “bulk temperature”

- Frequently used by engineers in cooling applications in order to approximate the effect of the fluid cooling on the structure (See *BOUNDARY_CONVECTION).
- The h can be found in empirical tables based on the fluid properties and the geometry of the pipe.
- However, for complex cases and geometries, it may be useful to run the CFD problem in order to check the value of the h along the pipe and to look for potential zones or pipe bents where the cooling becomes less or too efficient.

ICFD Solver Introduction

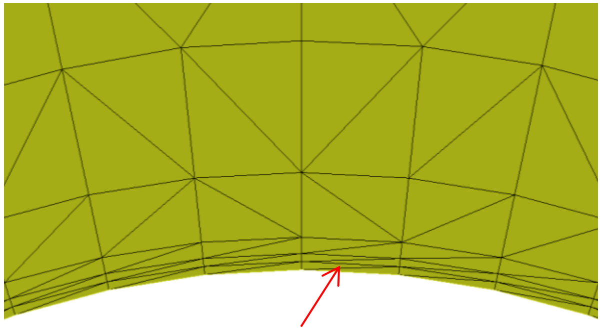
Application: External and internal aerodynamics with turbulence

Development: Adding more HRN and LRN laws of the wall for the turbulence models. Providing more tools for the boundary layer mesh generation.

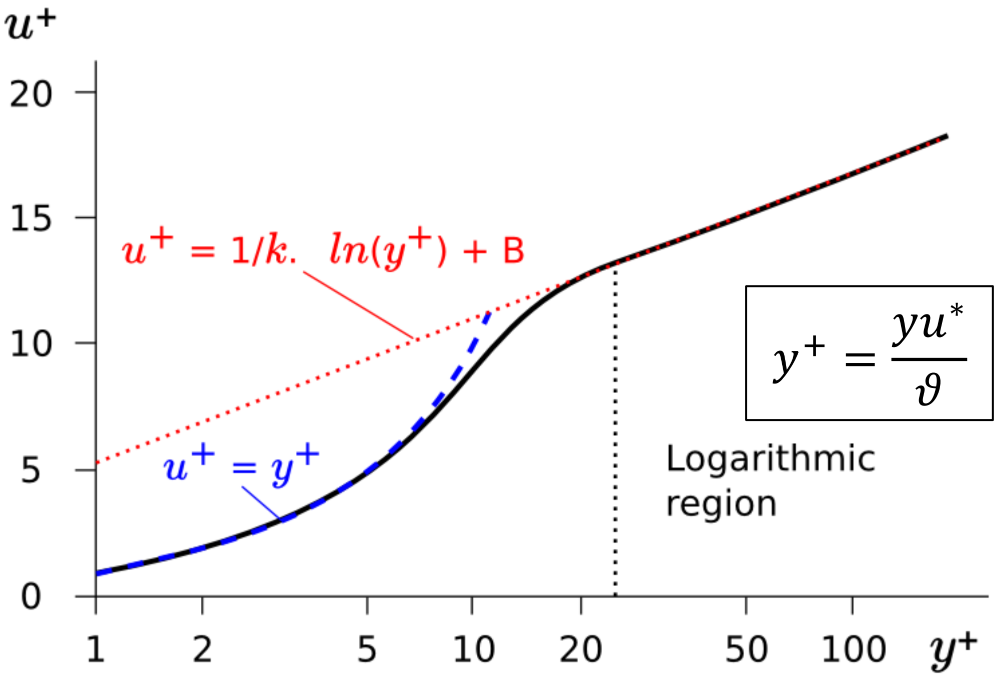
Current status: Implementation stage

$$\boxed{u^* = \sqrt{\frac{\tau_w}{\rho}}}$$

$$\boxed{u^+ = \frac{U}{u^*}}$$



BL mesh



$$\boxed{y^+ = \frac{yu^*}{\vartheta}}$$

ICFD Solver Introduction

Future Developments

- A wave generator for free surface problems.
- Porous media with FSI problems.
- Specific porous media models for parachutes.
- Adaptive surface remeshing.
- Embedded approach for FSI problems.

LS-PrePost
ICFD Post Treatments

ICFD Post Processor Introduction

- Since the official release of the ICFD solver in the R7.0 version, developments have been continuous and the number of users has been steadily growing.

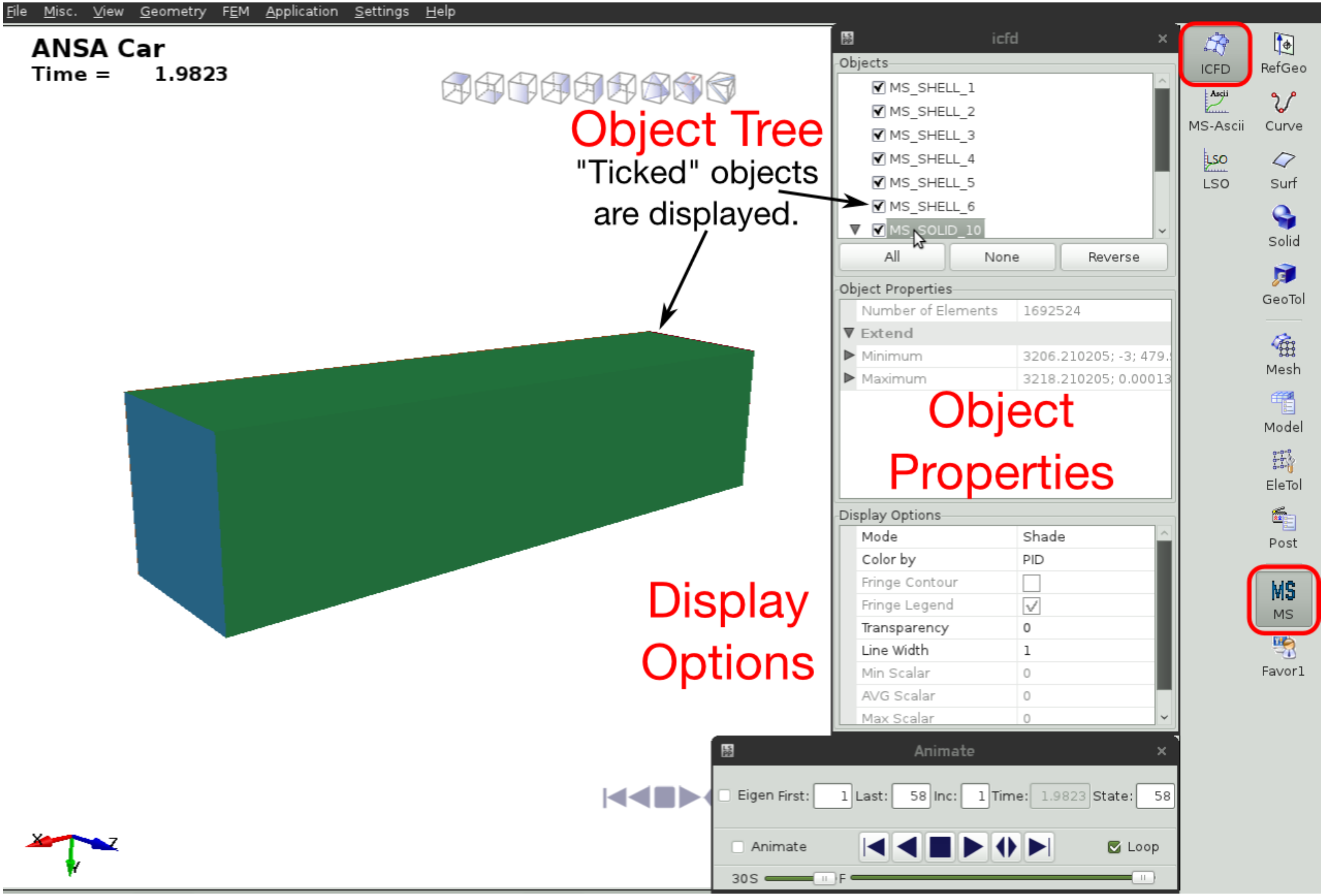
- Currently, LS-PrePost offers some tools in order to post treat the results from the ICFD solver based on its solid mechanic counterpart.

- However, the requirements for CFD post treatment are often quite different and challenging. This meant that a radically new approach was needed for LS-PrePost to meet those specific requirements.

- LS-PrePost 4.2 will be the first version to incorporate post treatments specific to the ICFD Solver and to CFD solvers in general.

ICFD Post Processor Introduction

Object oriented structure:



ANSA Car
Time = 1.9823

Object Tree
"Ticked" objects are displayed.

Object Properties

Display Options

Objects
- ☑ MS_SHELL_1
- ☑ MS_SHELL_2
- ☑ MS_SHELL_3
- ☑ MS_SHELL_4
- ☑ MS_SHELL_5
- ☑ MS_SHELL_6
- ▼ ☑ MS_SOLID_10

Object Properties

Number of Elements	1692524
▼ Extend	
▶ Minimum	3206.210205; -3; 479.
▶ Maximum	3218.210205; 0.00013

Display Options

Mode	Shade
Color by	PID
Fringe Contour	☐
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	0
AVG Scalar	0
Max Scalar	0

Animate

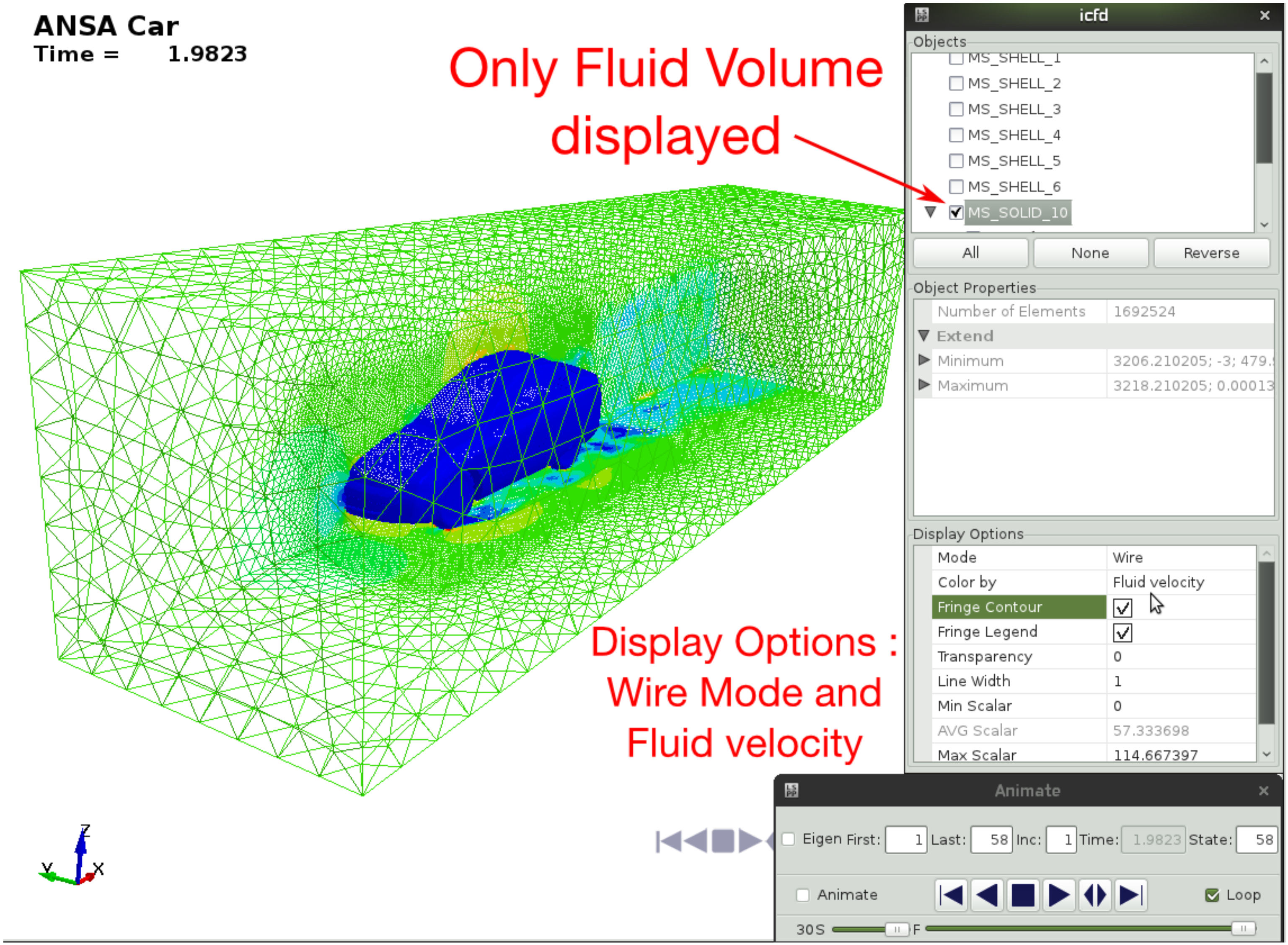
☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

30S F

ICFD Post Processor Introduction

Object oriented structure:



ANSA Car
Time = 1.9823

Only Fluid Volume displayed

Display Options : Wire Mode and Fluid velocity

icfd

Objects

- ☐ MS_SHELL_1
- ☐ MS_SHELL_2
- ☐ MS_SHELL_3
- ☐ MS_SHELL_4
- ☐ MS_SHELL_5
- ☐ MS_SHELL_6
- ☑ MS_SOLID_10

Object Properties

Number of Elements	1692524
Extend	
Minimum	3206.210205; -3; 479.
Maximum	3218.210205; 0.00013

Display Options

Mode	Wire
Color by	Fluid velocity
Fringe Contour	☑
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	0
AVG Scalar	57.333698
Max Scalar	114.667397

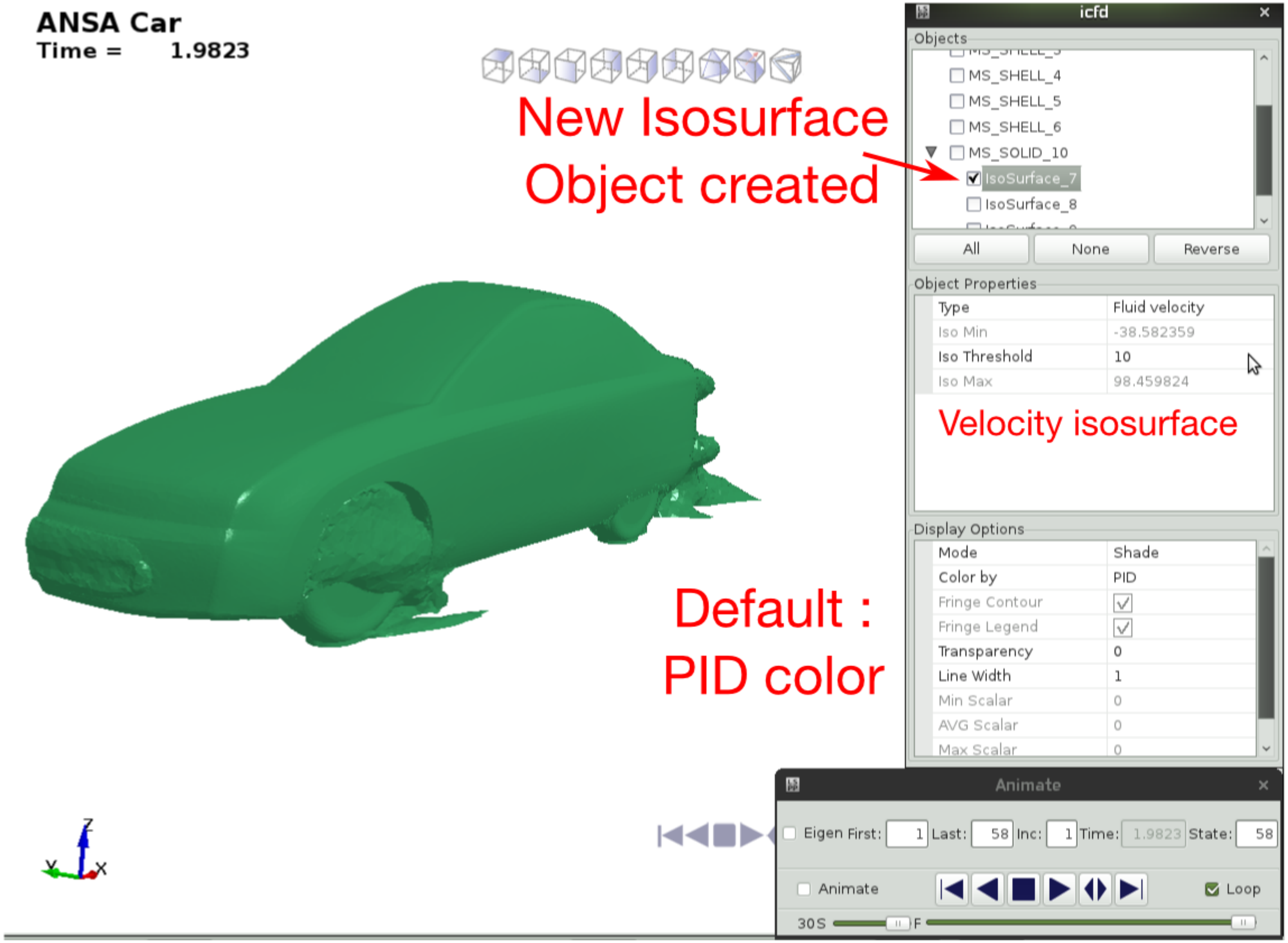
Animate

☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

ICFD Post Processor Introduction

A right click on the initial Object (here the fluid volume) pops up a Menu which allows the user to create new objects



New Isosurface Object created

Velocity isosurface

Default : PID color

☐ MS_SHELL_4
☐ MS_SHELL_5
☐ MS_SHELL_6
▼ ☐ MS_SOLID_10
☑ IsoSurface_7
☐ IsoSurface_8

Object Properties

Type	Fluid velocity
Iso Min	-38.582359
Iso Threshold	10
Iso Max	98.459824

Display Options

Mode	Shade
Color by	PID
Fringe Contour	☑
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	0
AVG Scalar	0
Max Scalar	0

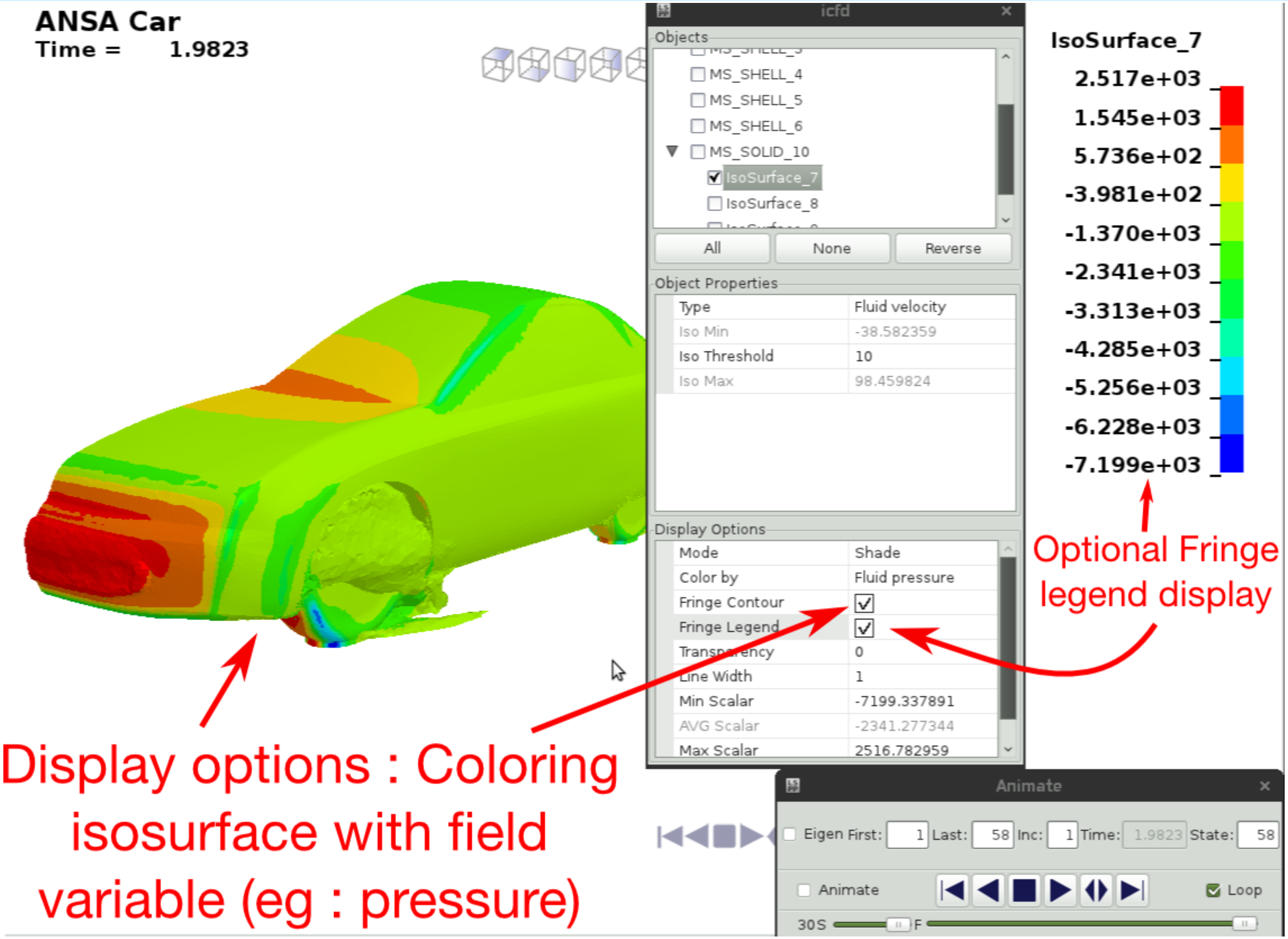
☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

30S F

ICFD Post Processor Introduction

Modifying newly created object:



ANSA Car

Time = 1.9823

icfd

Objects

- ☐ MS_SHELL_4
- ☐ MS_SHELL_5
- ☐ MS_SHELL_6
- ▼ ☐ MS_SOLID_10
  - ☑ IsoSurface_7
  - ☐ IsoSurface_8

Object Properties

Type	Fluid velocity
Iso Min	-38.582359
Iso Threshold	10
Iso Max	98.459824

Display Options

Mode	Shade
Color by	Fluid pressure
Fringe Contour	☑
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	-7199.337891
AVG Scalar	-2341.277344
Max Scalar	2516.782959

Animate

☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

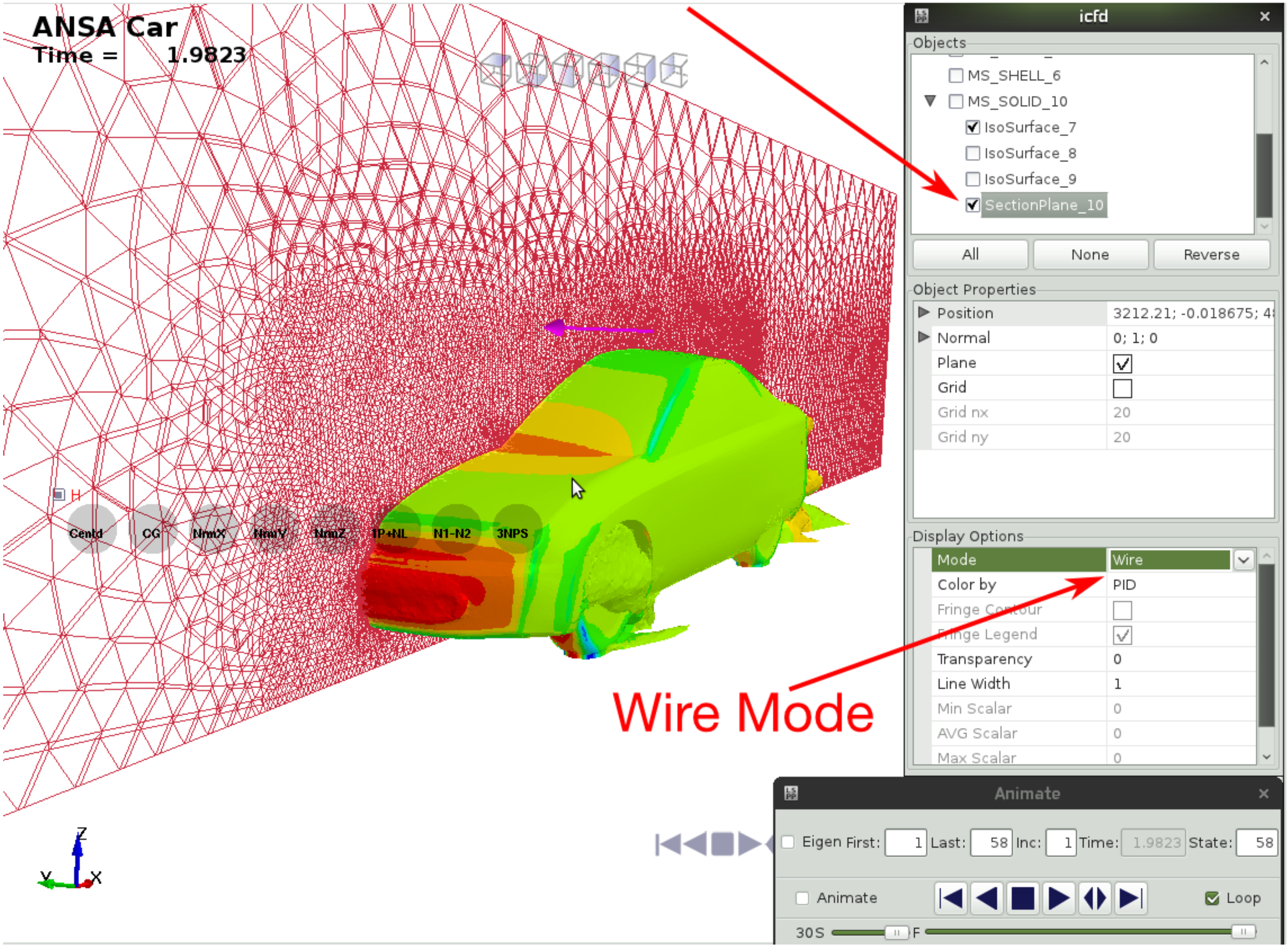
☐ Animate ☑ Loop

Optional Fringe legend display

Display options : Coloring isosurface with field variable (eg : pressure)

ICFD Post Processor Introduction

New object created : Splane



Wire Mode

ANSA Car
Time = 1.9823

icfd

Objects
- ☐ MS_SHELL_6
- ☐ MS_SOLID_10
  - ☑ IsoSurface_7
  - ☐ IsoSurface_8
  - ☐ IsoSurface_9
  - ☑ SectionPlane_10

Object Properties

Position	3212.21; -0.018675; 4
Normal	0; 1; 0
Plane	☑
Grid	☐
Grid nx	20
Grid ny	20

Display Options

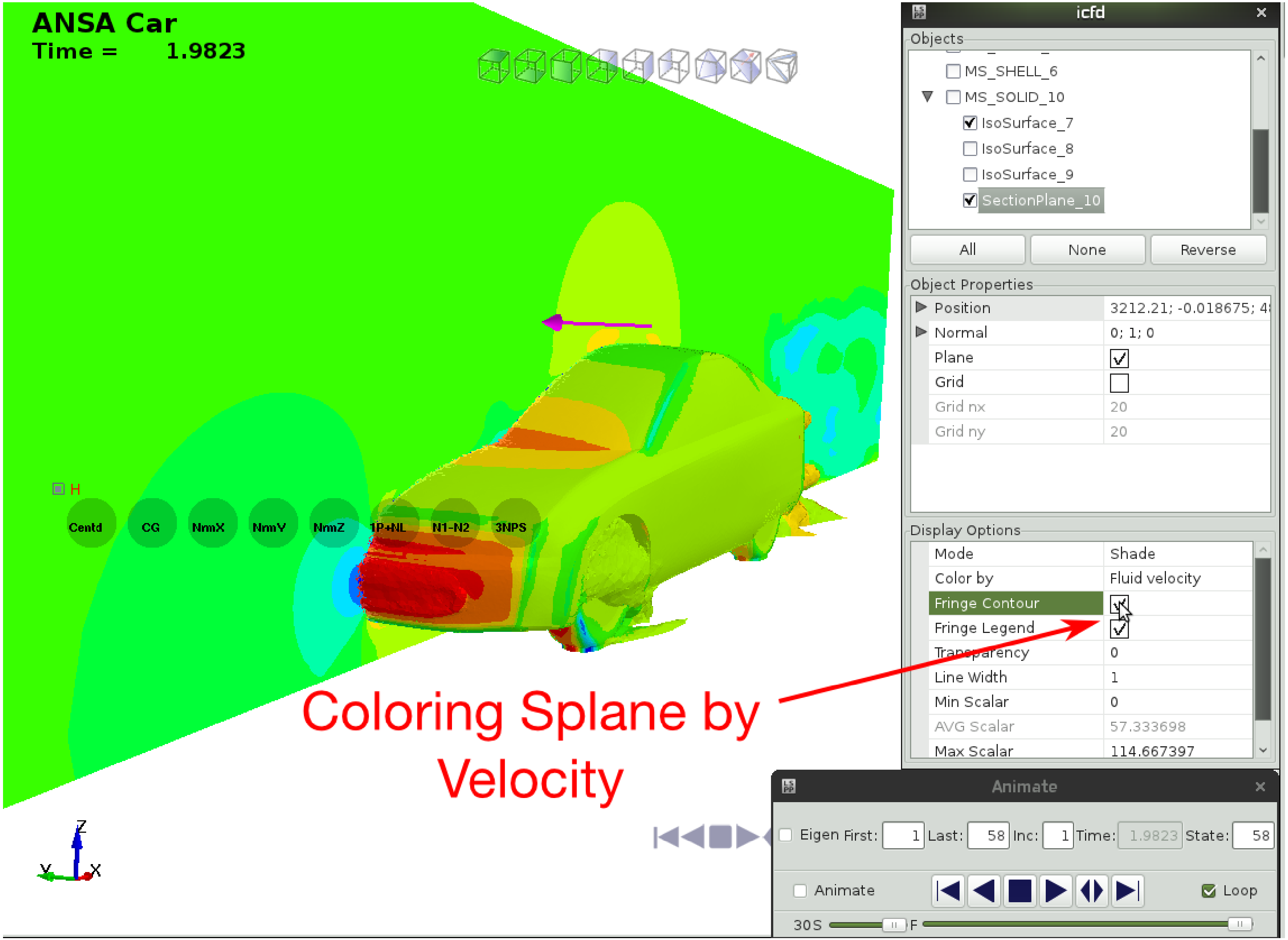
Mode	Wire
Color by	PID
Fringe Contour	☐
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	0
AVG Scalar	0
Max Scalar	0

Animate

☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

ICFD Post Processor Introduction



icfd

Objects

☐ MS_SHELL_6
▼ ☐ MS_SOLID_10
☑ IsoSurface_7
☐ IsoSurface_8
☐ IsoSurface_9
☑ SectionPlane_10

Object Properties

▶ Position	3212.21; -0.018675; 4
▶ Normal	0; 1; 0
Plane	☑
Grid	☐
Grid nx	20
Grid ny	20

Display Options

Mode	Shade
Color by	Fluid velocity
Fringe Contour	☑
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	0
AVG Scalar	57.333698
Max Scalar	114.667397

Animate

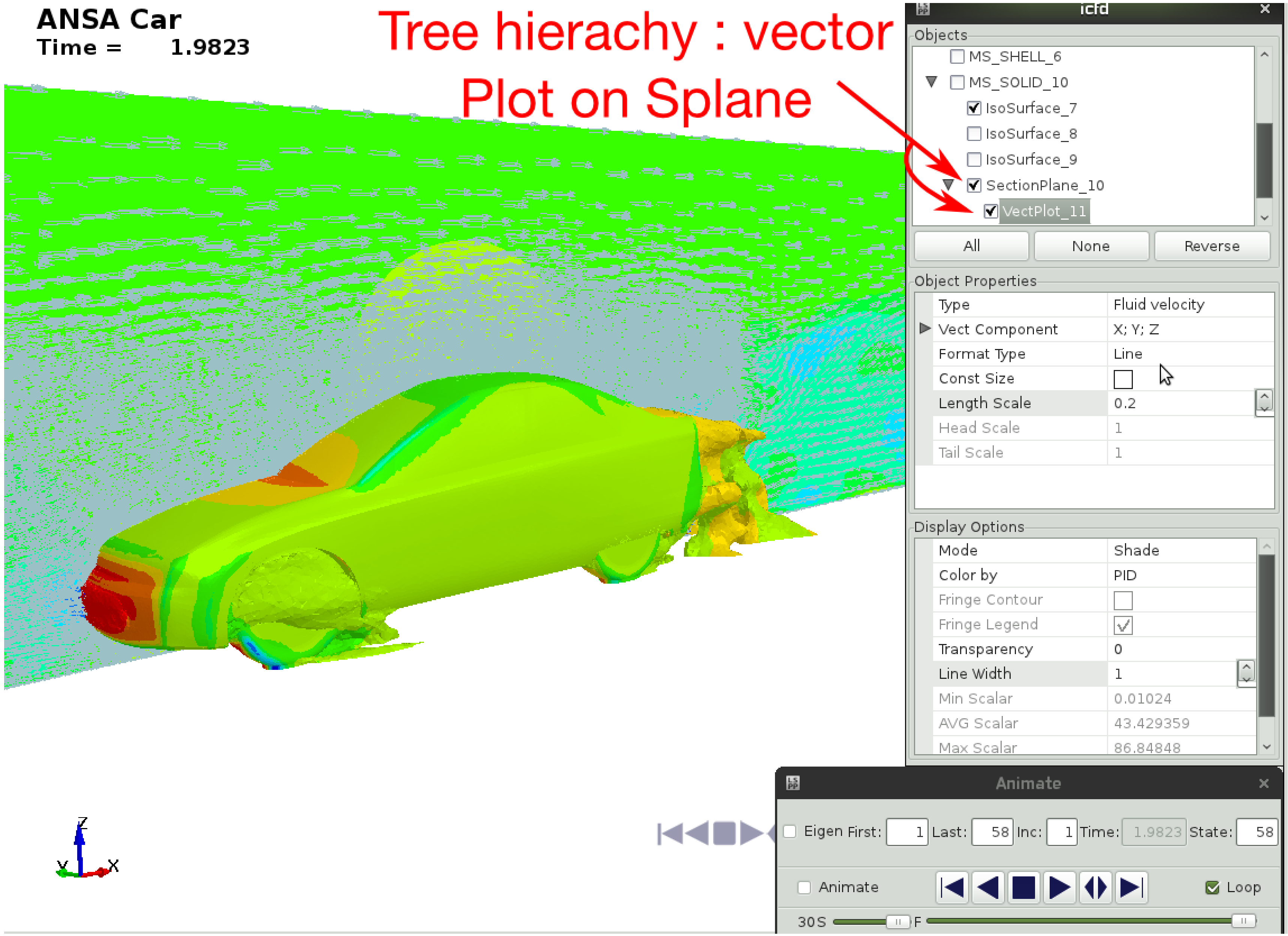
☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

30S F

Coloring Splane by Velocity

ICFD Post Processor Introduction



Tree hierachy : vector Plot on Splane

ANSA Car
Time = 1.9823

☐ MS_SHELL_6
☐ MS_SOLID_10
  ☑ IsoSurface_7
  ☐ IsoSurface_8
  ☐ IsoSurface_9
  ☑ SectionPlane_10
    ☑ VectPlot_11

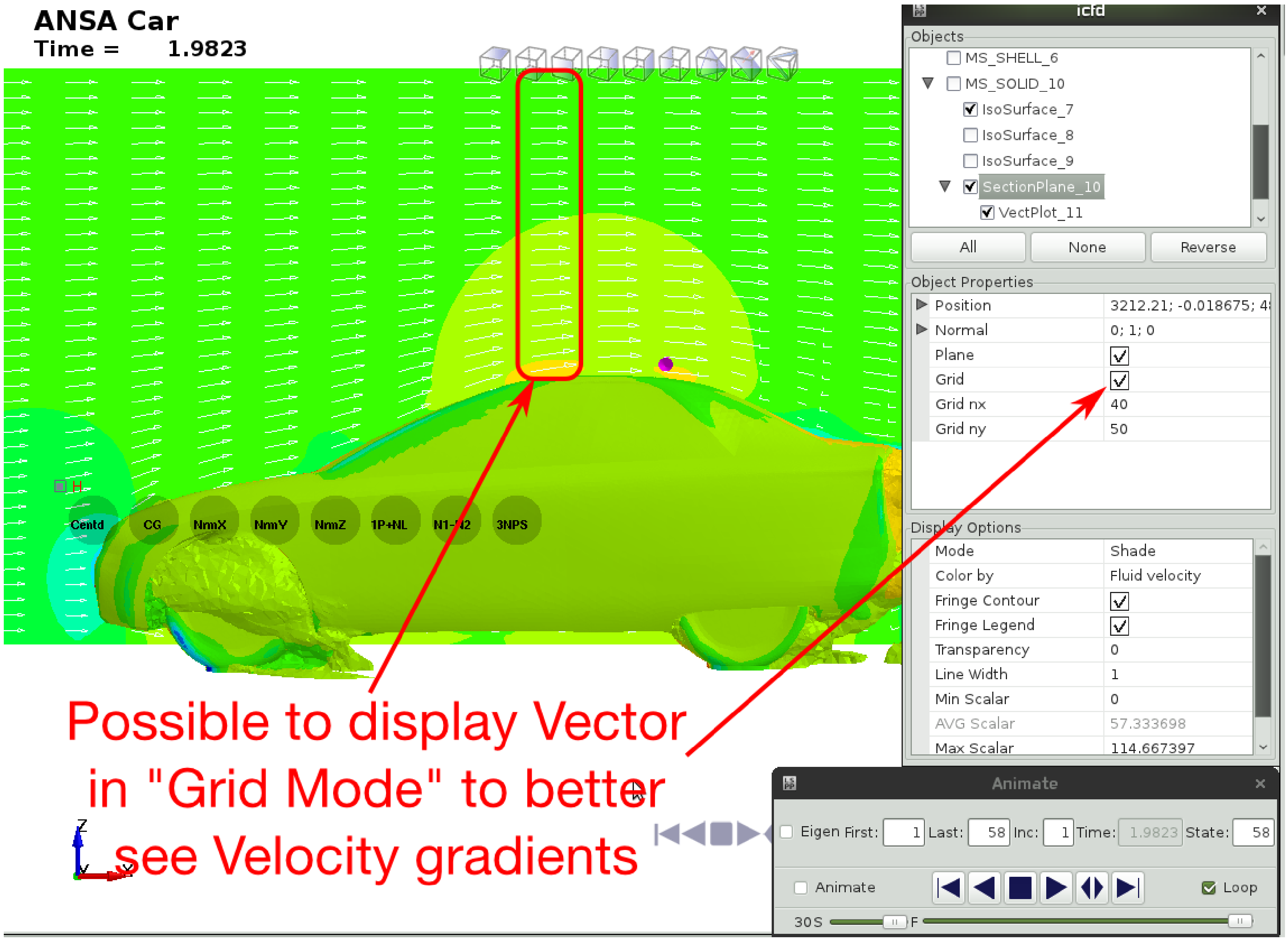
Type	Fluid velocity
Vect Component	X; Y; Z
Format Type	Line
Const Size	☐
Length Scale	0.2
Head Scale	1
Tail Scale	1

Mode	Shade
Color by	PID
Fringe Contour	☐
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	0.01024
AVG Scalar	43.429359
Max Scalar	86.84848

☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

ICFD Post Processor Introduction



ANSA Car
Time = 1.9823

icfd

Objects
- ☐ MS_SHELL_6
- ▼ ☐ MS_SOLID_10
  - ☑ IsoSurface_7
  - ☐ IsoSurface_8
  - ☐ IsoSurface_9
  - ▼ ☑ SectionPlane_10
    - ☑ VectPlot_11

Object Properties

▶ Position	3212.21; -0.018675; 4
▶ Normal	0; 1; 0
Plane	☑
Grid	☑
Grid nx	40
Grid ny	50

Display Options

Mode	Shade
Color by	Fluid velocity
Fringe Contour	☑
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	0
AVG Scalar	57.333698
Max Scalar	114.667397

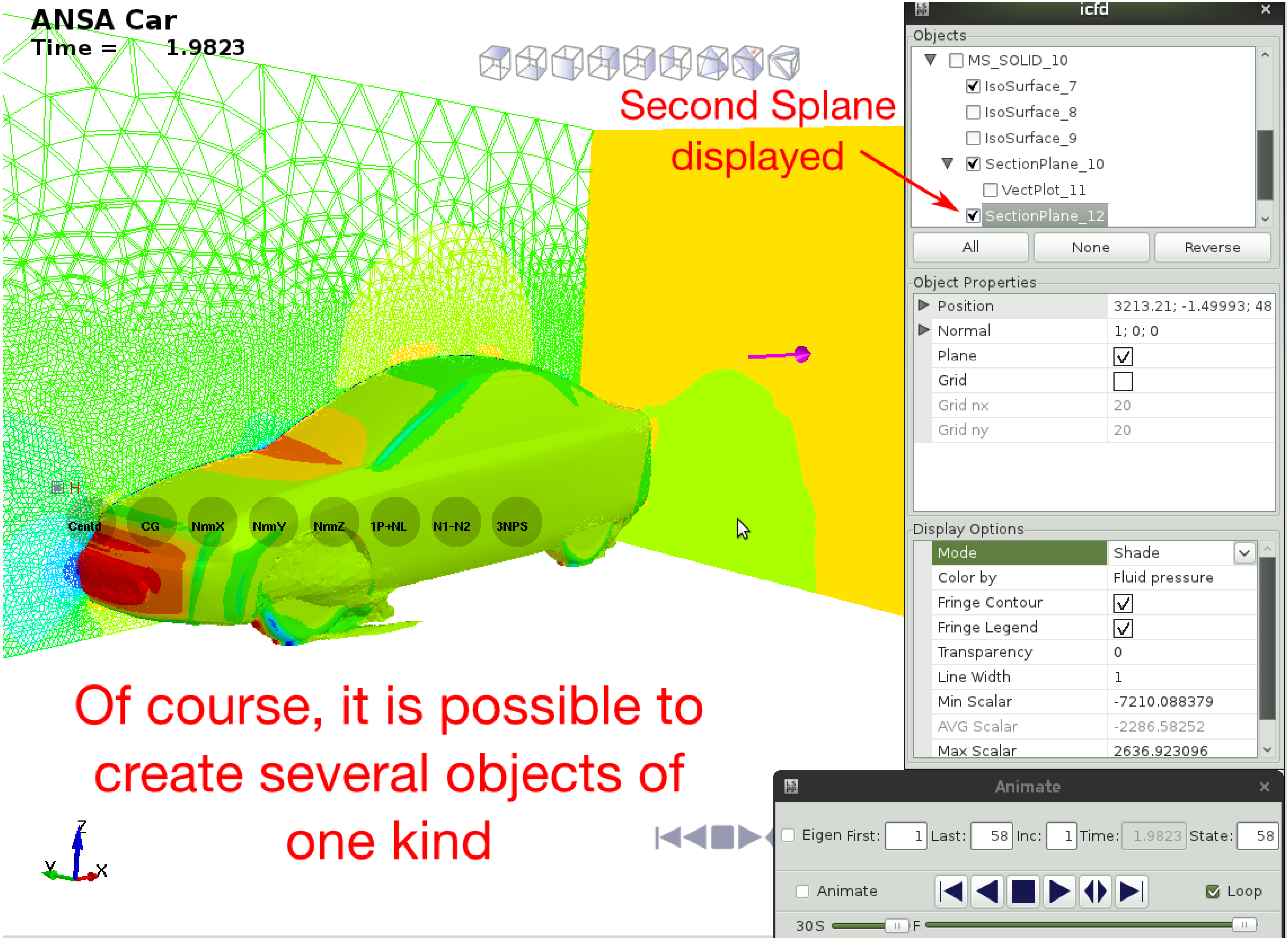
Animate

☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

Possible to display Vector in "Grid Mode" to better see Velocity gradients

ICFD Post Processor Introduction



Second Splane displayed

Objects

- ☐ MS_SOLID_10
  - ☑ IsoSurface_7
  - ☐ IsoSurface_8
  - ☐ IsoSurface_9
  - ☑ SectionPlane_10
    - ☐ VectPlot_11
  - ☑ SectionPlane_12

Object Properties

Position	3213.21; -1.49993; 48
Normal	1; 0; 0
Plane	☑
Grid	☐
Grid nx	20
Grid ny	20

Display Options

Mode	Shade
Color by	Fluid pressure
Fringe Contour	☑
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	-7210.088379
AVG Scalar	-2286.58252
Max Scalar	2636.923096

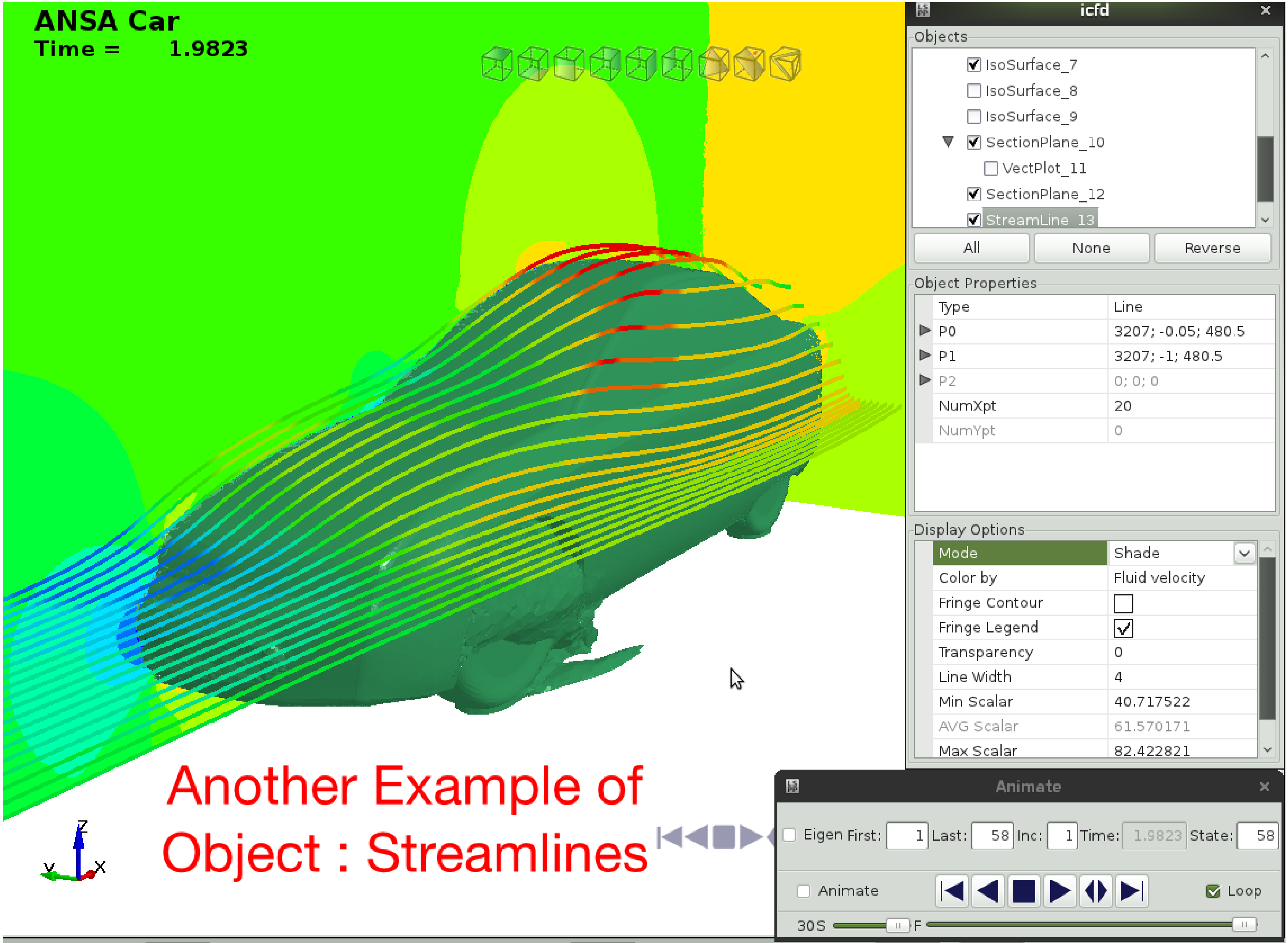
☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

30S F

Of course, it is possible to create several objects of one kind

ICFD Post Processor Introduction



icfd

Objects

- ☑ IsoSurface_7
- ☐ IsoSurface_8
- ☐ IsoSurface_9
- ☑ SectionPlane_10
  - ☐ VectPlot_11
- ☑ SectionPlane_12
- ☑ StreamLine_13

Object Properties

Type	Line
P0	3207; -0.05; 480.5
P1	3207; -1; 480.5
P2	0; 0; 0
NumXpt	20
NumYpt	0

Display Options

Mode	Shade
Color by	Fluid velocity
Fringe Contour	☐
Fringe Legend	☑
Transparency	0
Line Width	4
Min Scalar	40.717522
AVG Scalar	61.570171
Max Scalar	82.422821

Animate

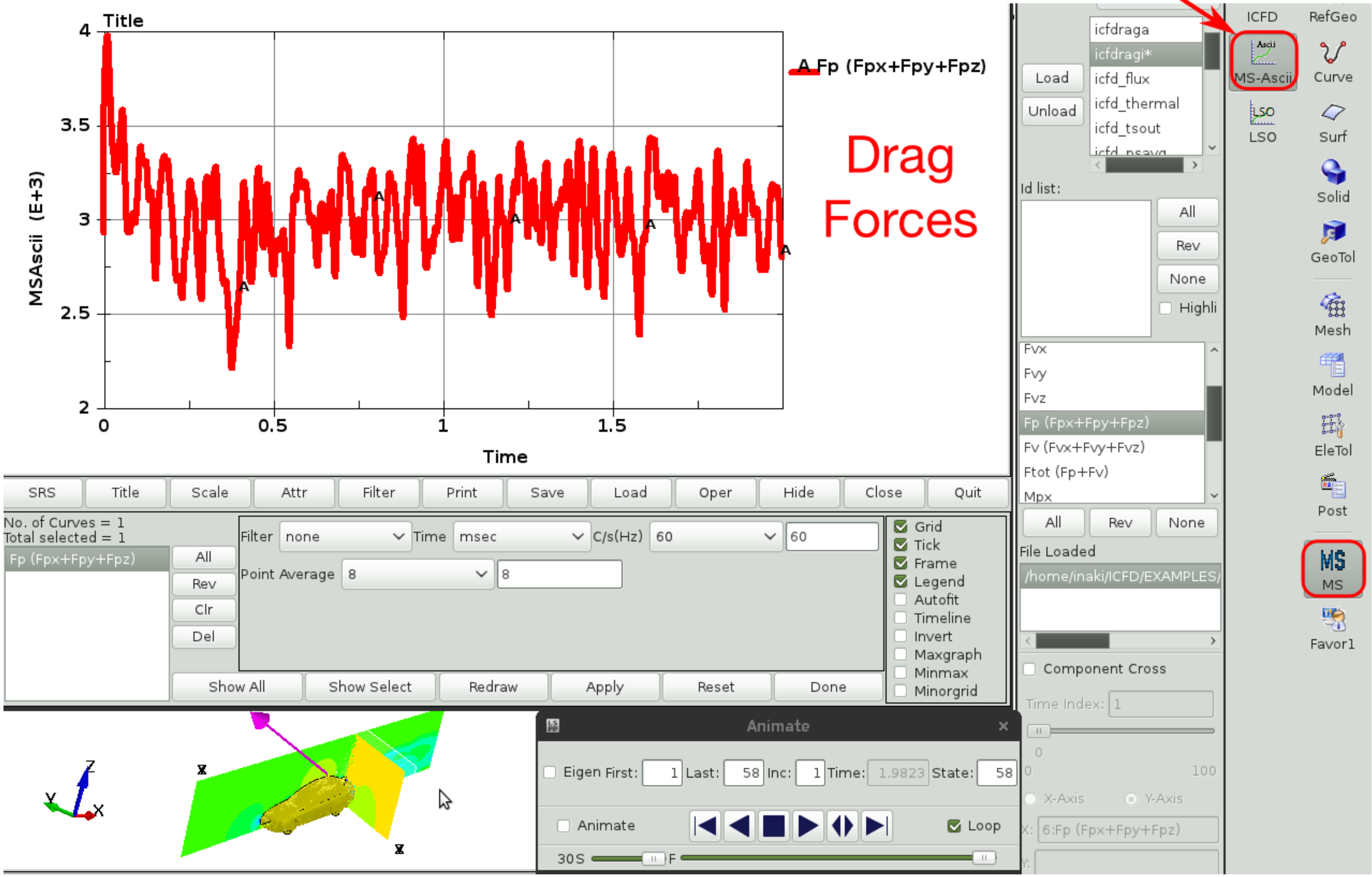
☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

Another Example of Object : Streamlines

ICFD Post Processor Introduction

It is also possible to post treat all ASCII files (See *ICFD_DATABASE family) dumped by the ICFD solver (forces, flux, point data etc)



No. of Curves = 1
Total selected = 1
Fp (Fpx+Fpy+Fpz)

Filter: none
Time: msec
C/s(Hz): 60 60
Point Average: 8 8

☑ Grid
☑ Tick
☑ Frame
☑ Legend
☐ Autofit
☐ Timeline
☐ Invert
☐ Maxgraph
☐ Minmax
☐ Minorgrid

File Loaded
/home/inaki/ICFD/EXAMPLES/

☐ Component Cross
Time Index: 1
X: 6:Fp (Fpx+Fpy+Fpz)
Y:

☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58
☐ Animate
☑ Loop

Drag Forces

ICFD Post Processor Introduction

- Currently, four type of objects can be created : Splane, isosurfaces, streamlines and vectors. Those are the most commonly used visualization tools in order to study flow patterns.

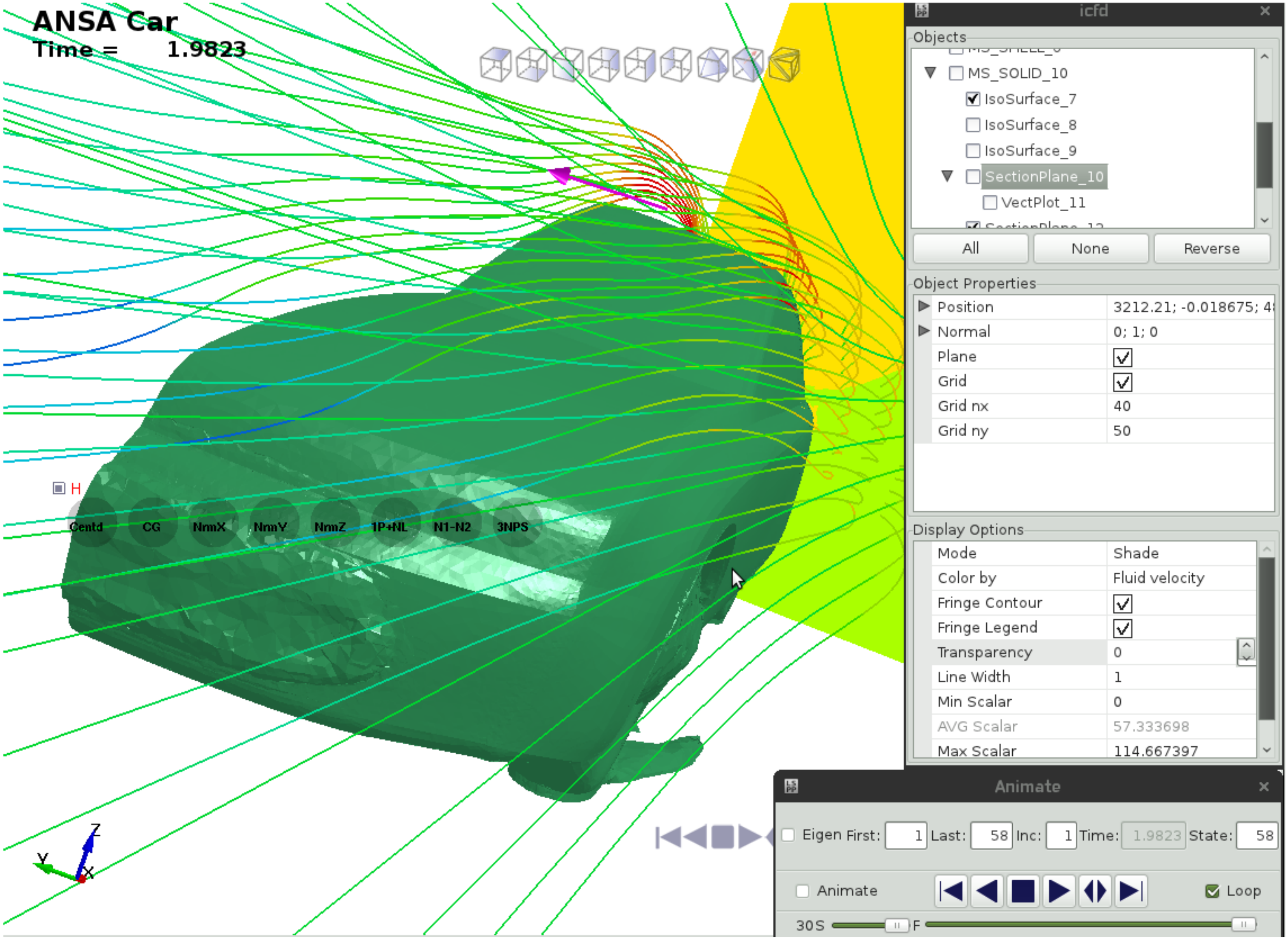
- More object types may be implemented in the future.

- The next step of development will include some new features for a more flexible and dynamic post treatment of results and data (easy extraction of values from the mesh, curve plotting options etc..)

- LS-PrePost 4.2 is currently in beta stage and is available to users eager to beta test its current functionalities.

ICFD Post Processor Introduction

Thank you for your attention !



Objects

- MS_SOLID_10
  - ☑ IsoSurface_7
  - ☐ IsoSurface_8
  - ☐ IsoSurface_9
  - ☐ SectionPlane_10
    - ☐ VectPlot_11

All | None | Reverse

Object Properties

Position	3212.21; -0.018675; 4
Normal	0; 1; 0
Plane	☑
Grid	☑
Grid nx	40
Grid ny	50

Display Options

Mode	Shade
Color by	Fluid velocity
Fringe Contour	☑
Fringe Legend	☑
Transparency	0
Line Width	1
Min Scalar	0
AVG Scalar	57.333698
Max Scalar	114.667397

☐ Eigen First: 1 Last: 58 Inc: 1 Time: 1.9823 State: 58

☐ Animate ☑ Loop

30S F

Thank You!

Particle Methods

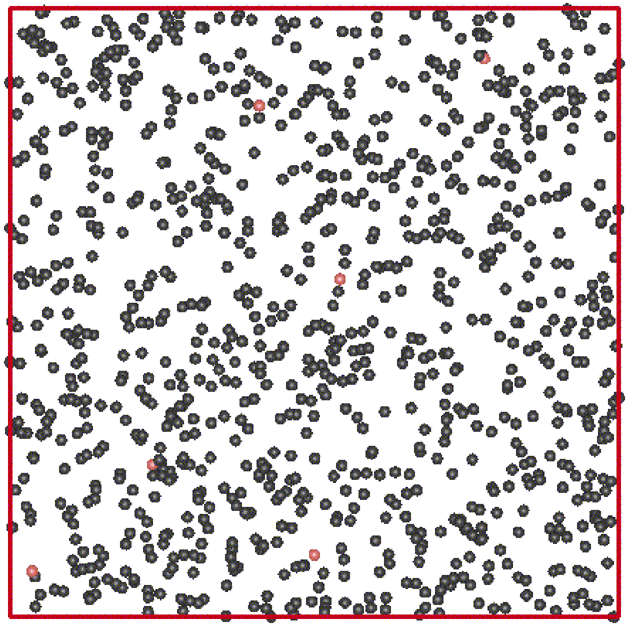
Jason Wang

Meshless Particle Solvers

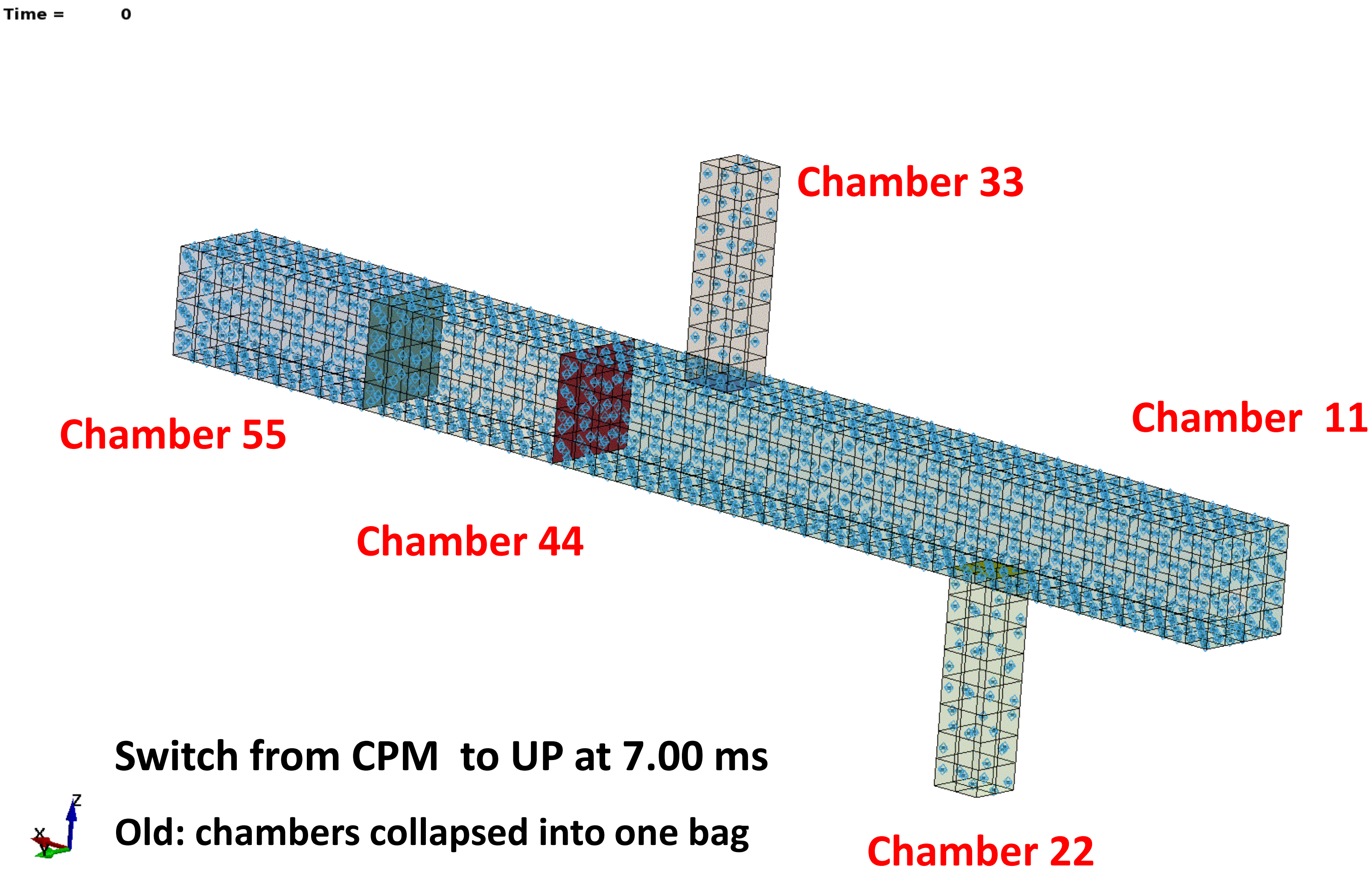
1) **Particle Gas**
   - **CPM - Ideal Gas Law**
   - **Particle Blast - Real Gas Law**
2) **SPH**
3) **Discrete Element Method (DEM)**
   - **Discrete Element Sphere**
   - **Discrete Element Method with Bond**
4) **Coupled Multi-Physics Solvers**

Particle Gas, CPM-Ideal Gas Law

- **Modeled by ideal gas law: pV=*n*RT**
- **The volume of molecules is neglected**
- **maintain the same Maxwell-Boltzmann velocity distribution at thermal equilibrium**
- **Work for low pressure and moderate temperature**



CPM/UP switch with Chambers



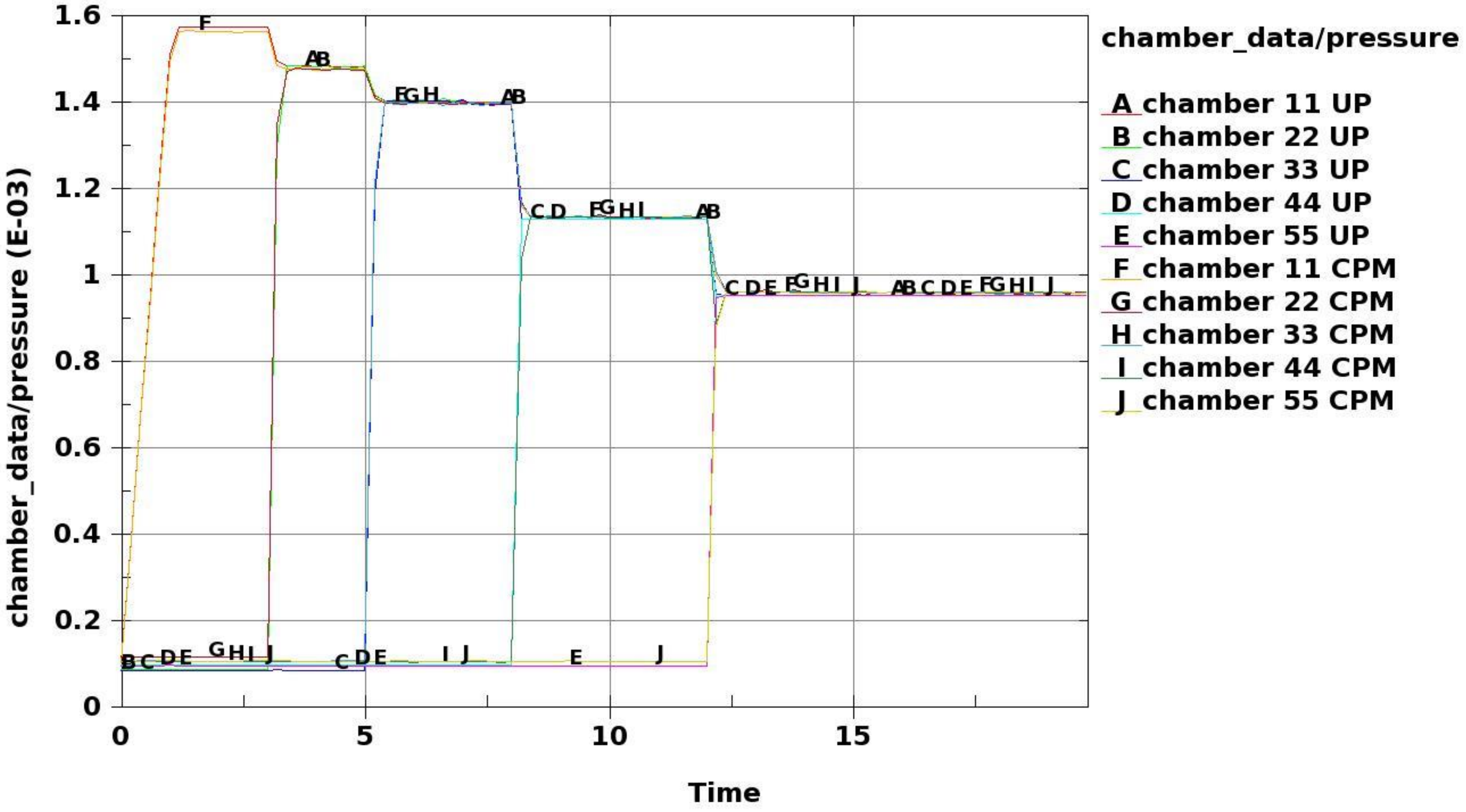
Switch from CPM to UP at 7.00 ms

Old: chambers collapsed into one bag

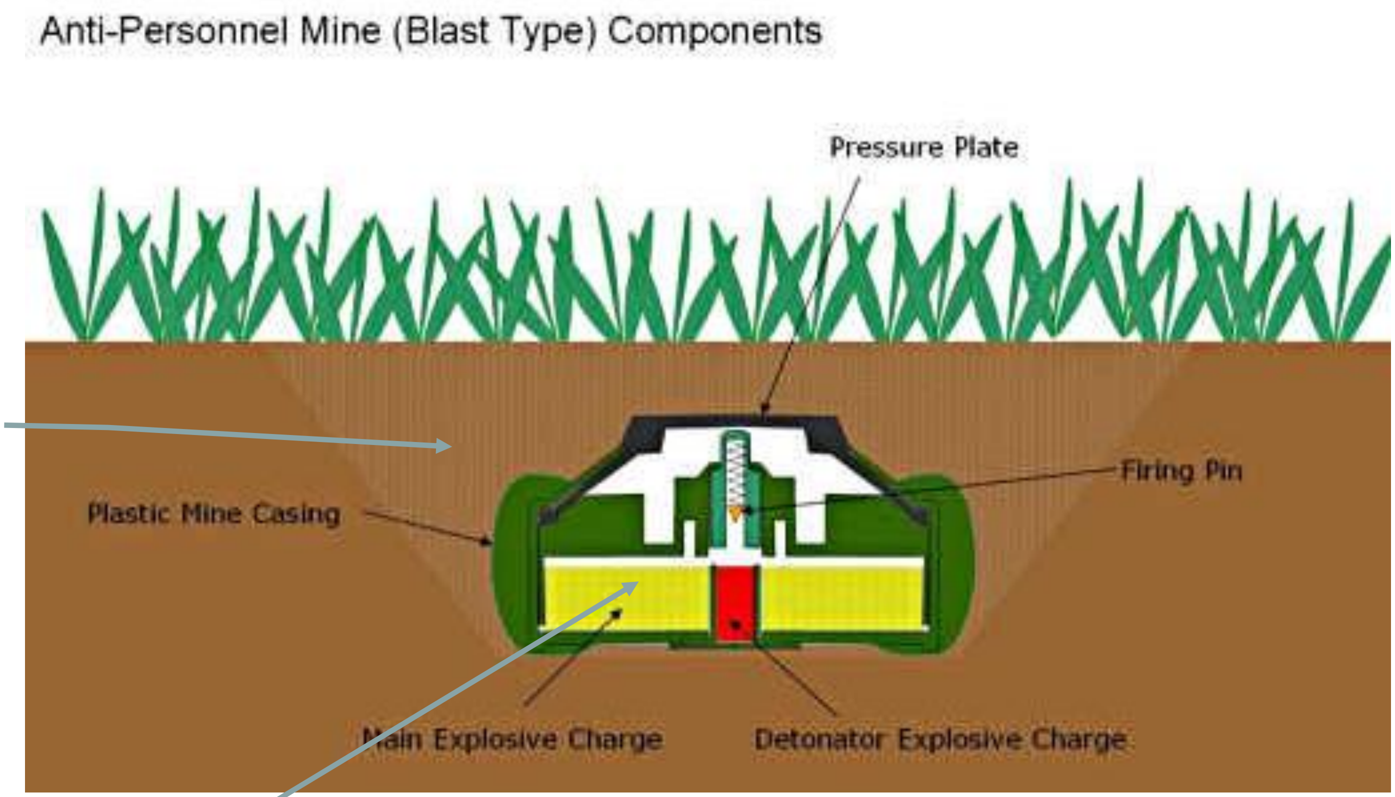
New: chambers becomes separated UP domain

CPM/UP switch with Chambers

- **CPM/UP Switch at 7 ms, curves A, B, C, D, E**
- **CPM all the way, curves F, G, H, I, J**



Particle Gas, Particle Blast-Real Gas Law



Air particle
(Ideal Gas Law)

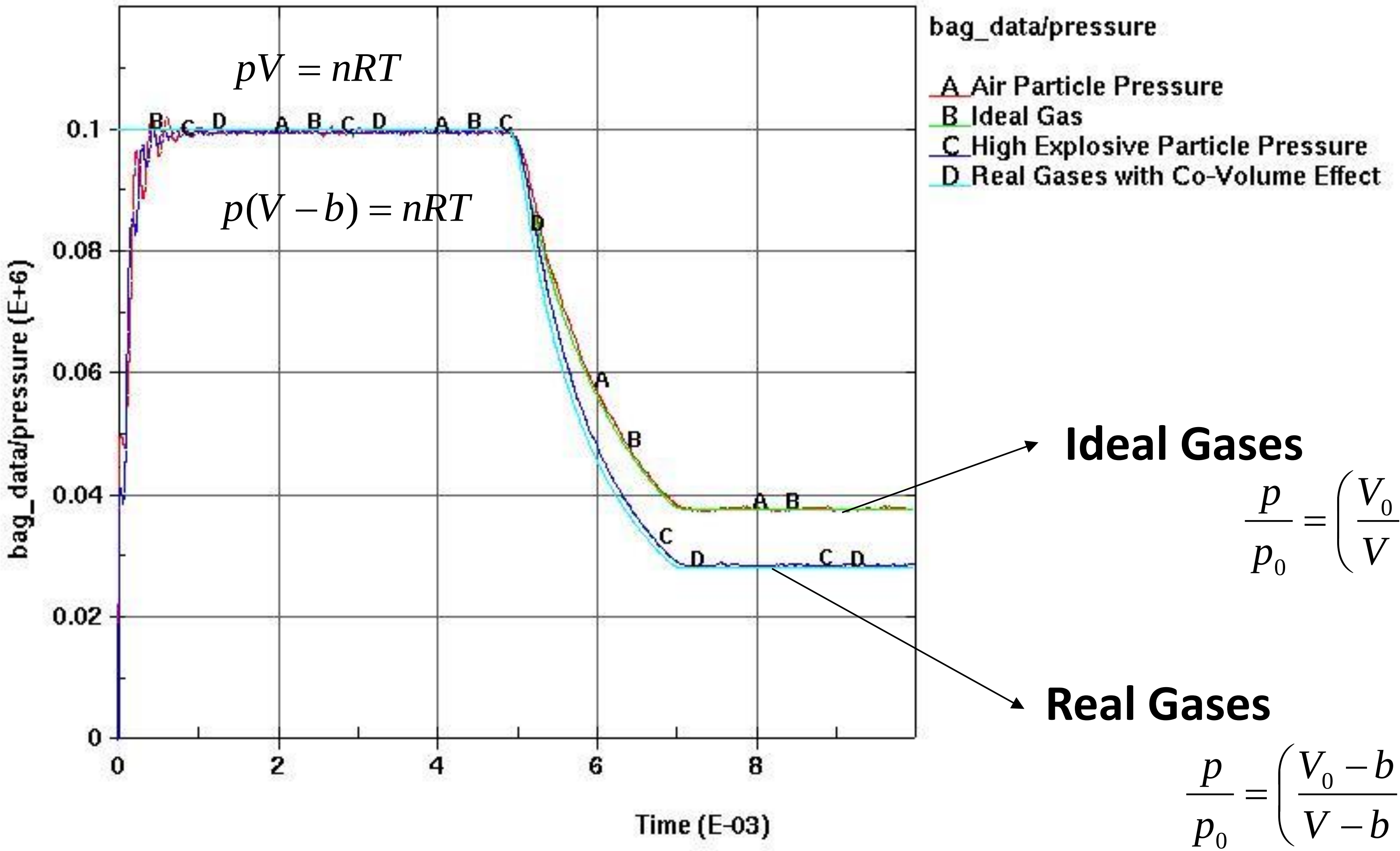
Discrete Element
Method (DEM)

HE Particle
(Ideal Gas Law)

Particle Gas, Particle Blast–Real Gas Law

- **Air Particle**
  - Modeled by ideal gas law (CPM): $pV=nRT$
- **High Explosive (HE) Particles**
  - Modeled by real gases: $p(V-b)=nRT$
  - The co-volume effect is included
  - Work for high pressure and high temperature
  - Pressure drop sharply during adiabatic expansion

Particle Gas, Ideal and Real Gas Law



$$pV = nRT$$

$$p(V-b) = nRT$$

Ideal Gases

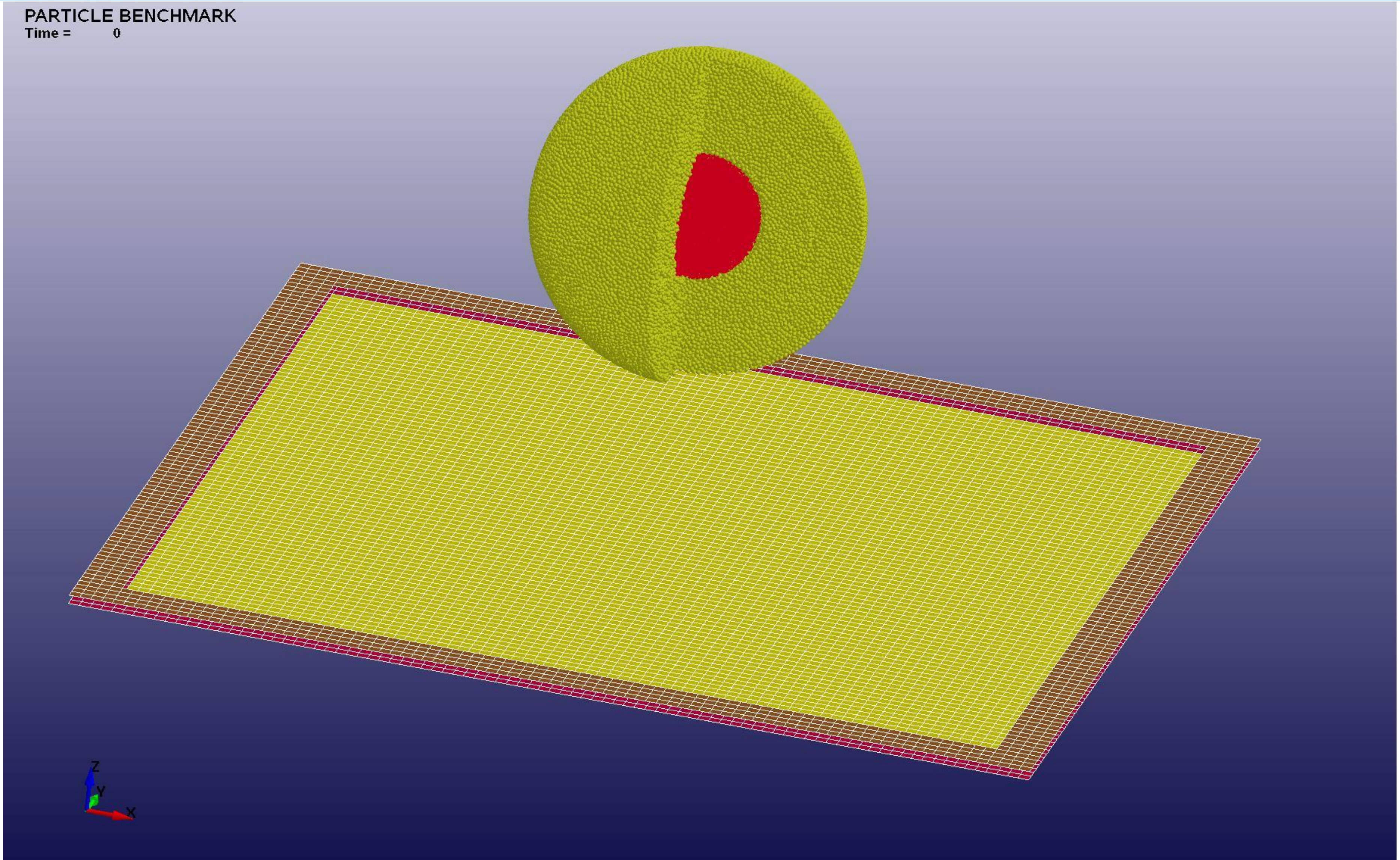
$$\frac{p}{p_0} = \left(\frac{V_0}{V}\right)^{\gamma}$$

Real Gases

$$\frac{p}{p_0} = \left(\frac{V_0-b}{V-b}\right)^{\gamma}$$

Particle Blast, Real Gas Law

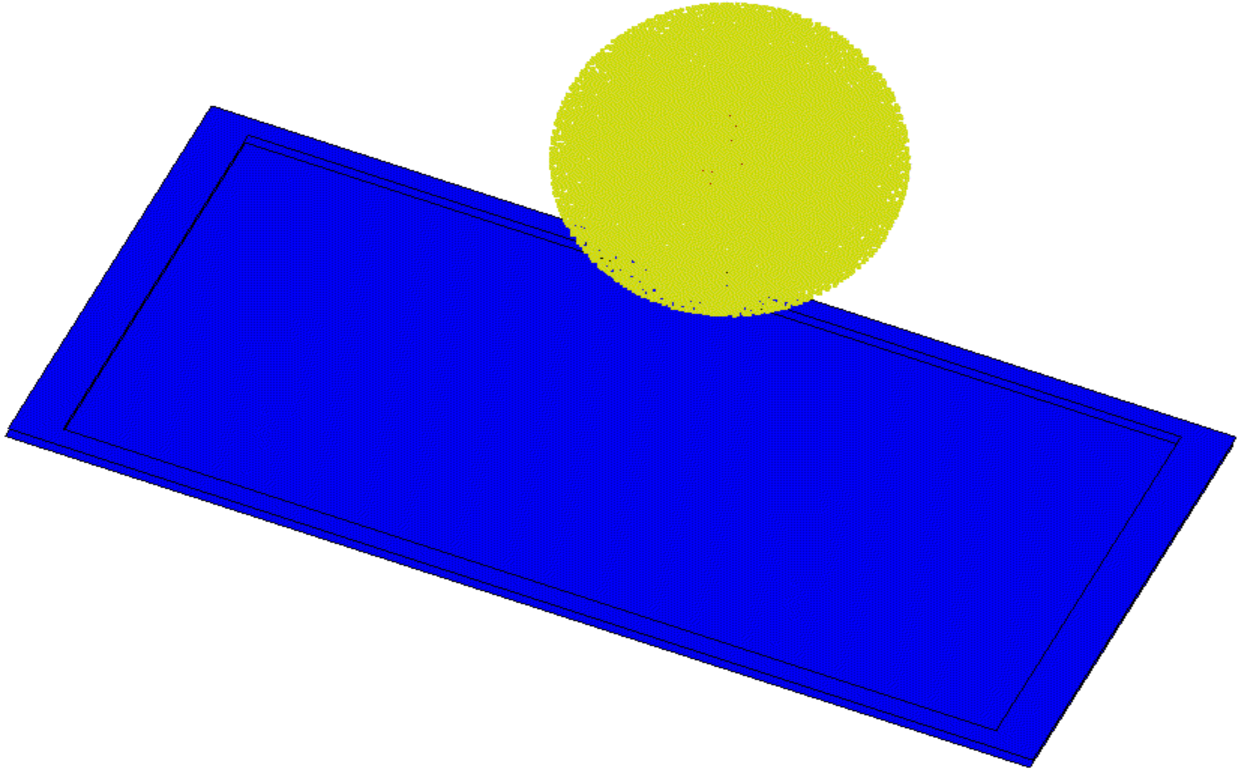
Numerical Example



Particle Blast, Real Gas Law

Numerical Example

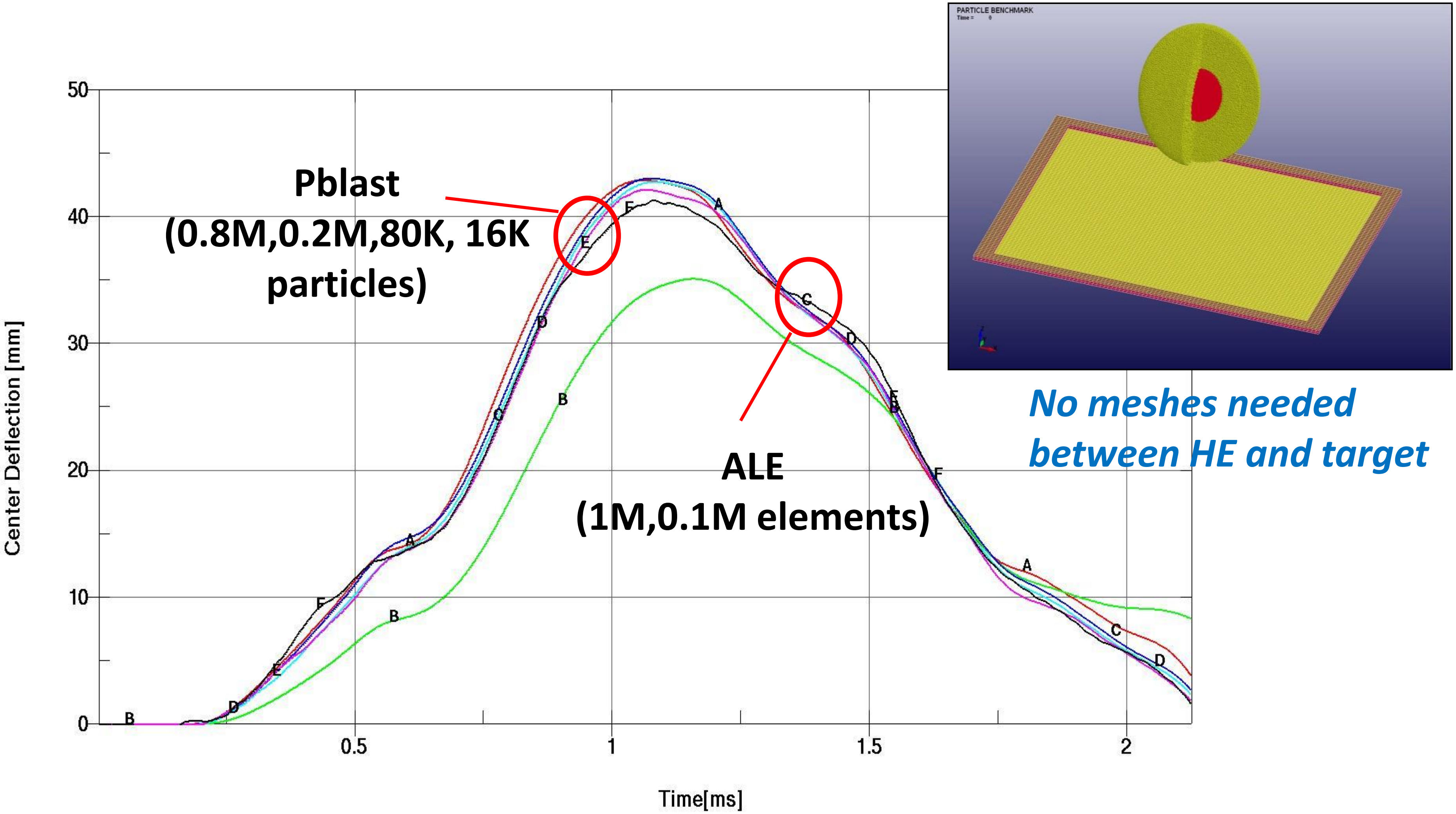
Blast simulation with sand



Particle Blast, Real Gas Law

Numerical Example

Simulation Results for 700mm Model



Pblast
(0.8M,0.2M,80K, 16K particles)

ALE
(1M,0.1M elements)

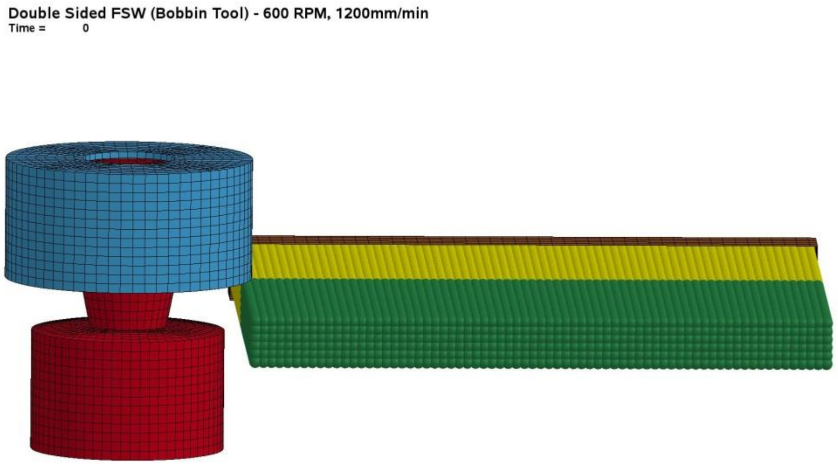
No meshes needed between HE and target

Enhancement of SPH

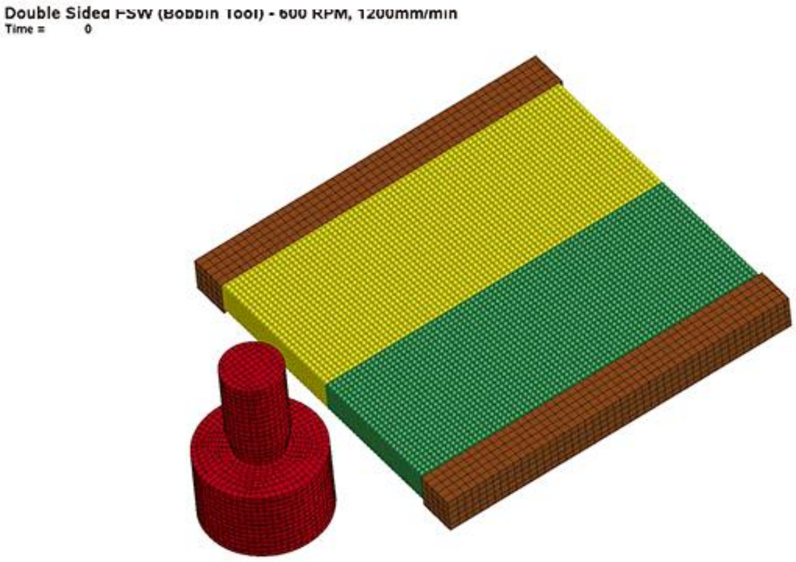
1. **Friction Stir Welding**
2. **SPH to SPH contact**
3. **SPH active region and new bucket sort**

Friction Stir Welding

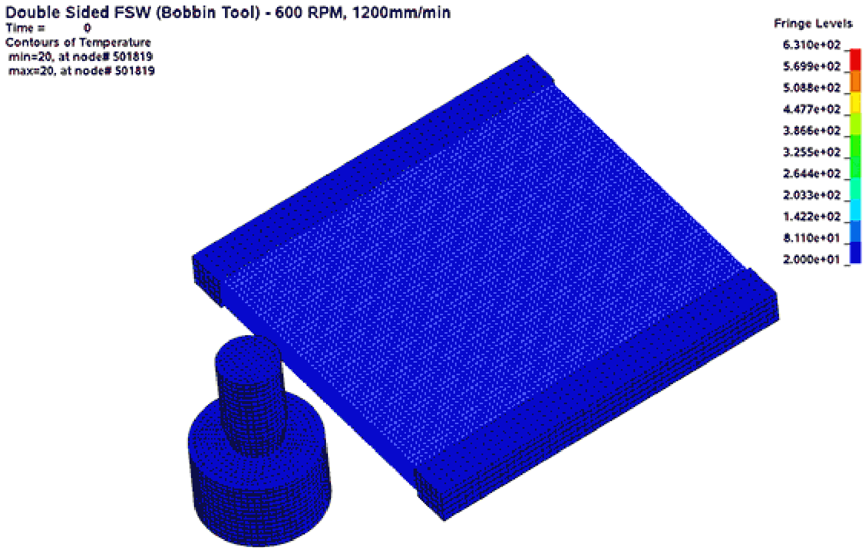
Extended SPH thermal solver for SPH form 7 and 8



Double sided FSW 600 RPM, 1200 mm/min(plastic work and friction energy to heat)
Courtesy of Kirk A. Fraser @ PredictiveEngineering



Material Mixing



Temperature Contours

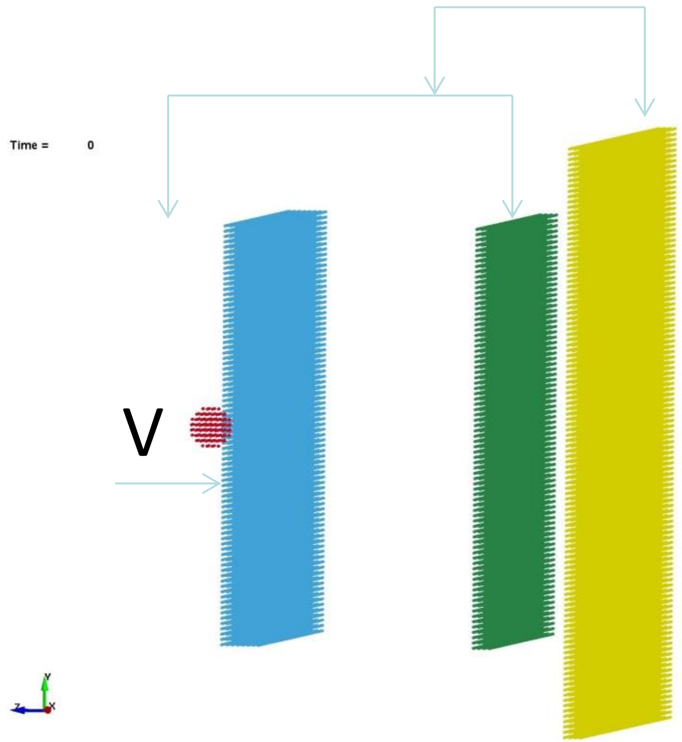
SPH Interaction

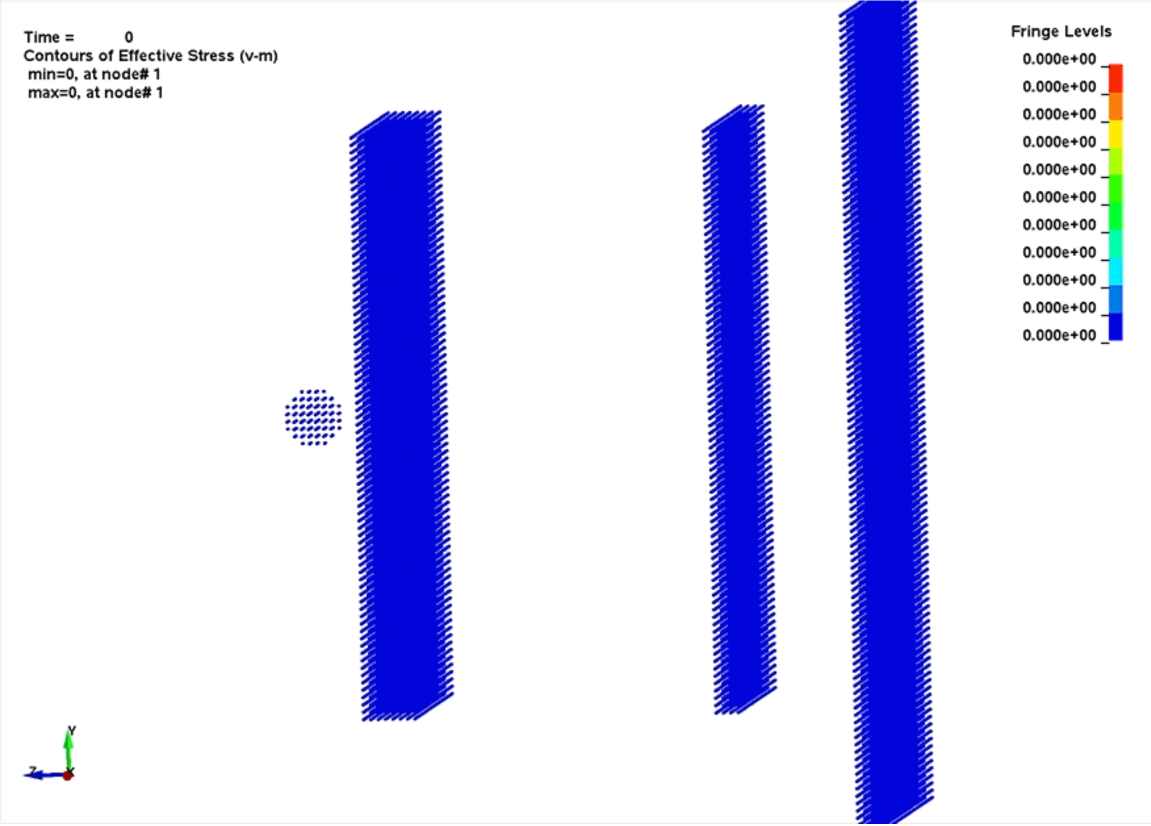
Multiple impacts with Keyword: SECTION_SPH_INTERACTION

Define the different type of interactions between SPH parts.

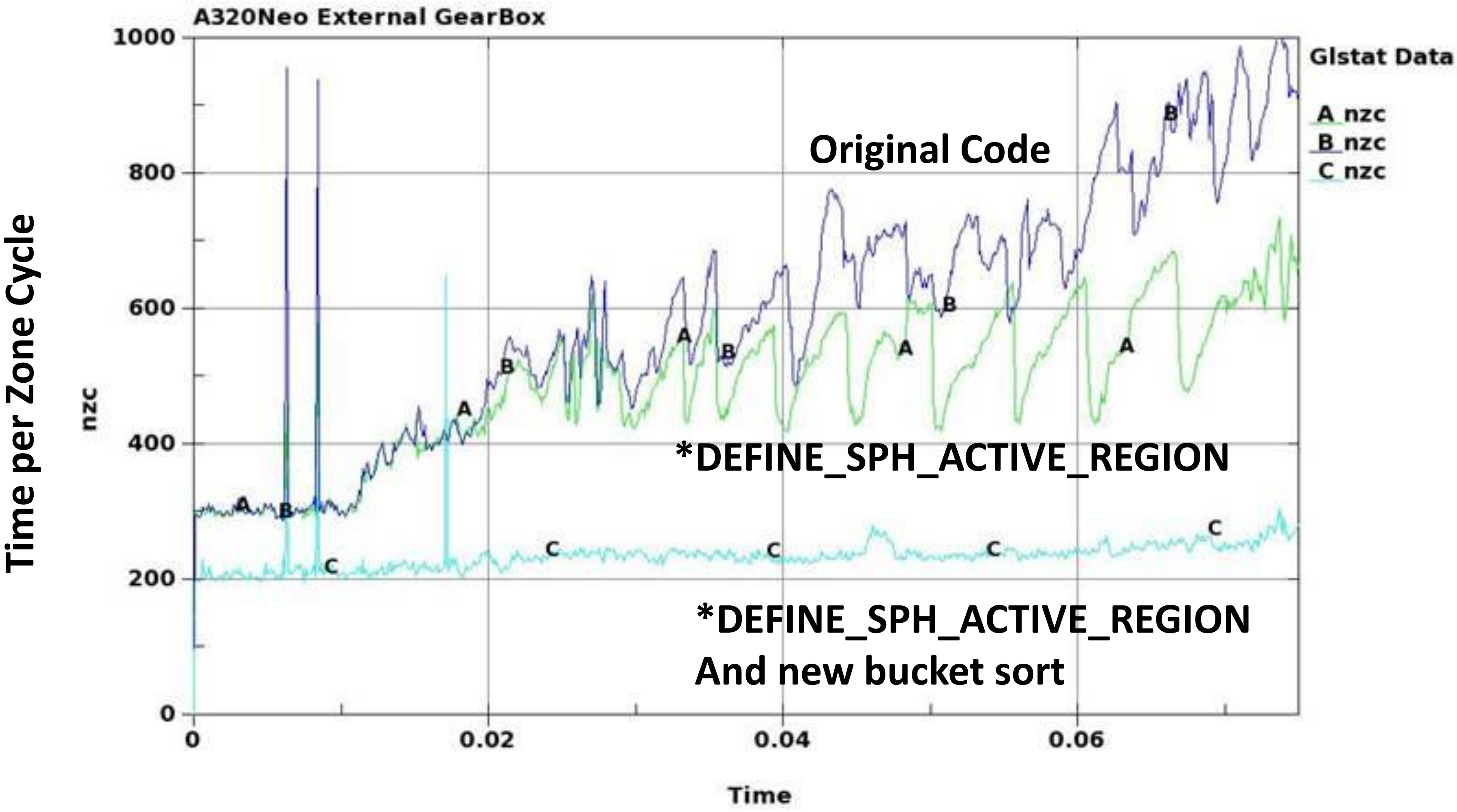
Metals with Standard interactions

Ceramic with node to node contacts





SPH Active Region & Better Bucket Sort



Original Code

*DEFINE_SPH_ACTIVE_REGION

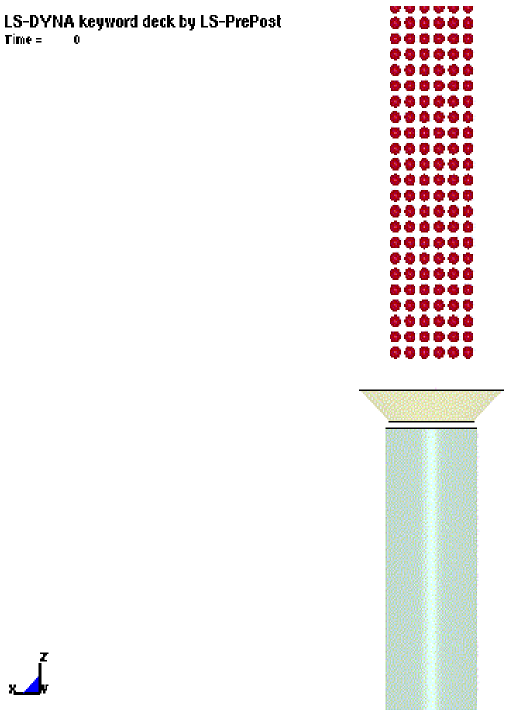
*DEFINE_SPH_ACTIVE_REGION
And new bucket sort

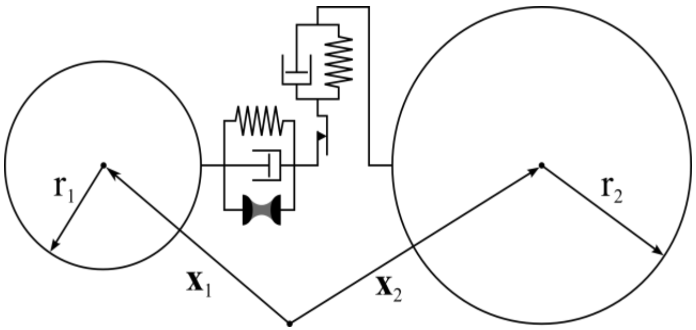
Discrete Element Method (DEM)

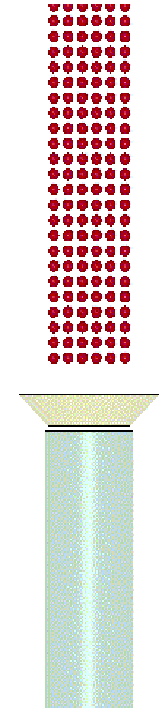
1. **Discrete Element Sphere**
2. **Discrete Element Method with Bond**

Discrete Element Method (DEM)

Discrete Element Sphere







Dry Particle

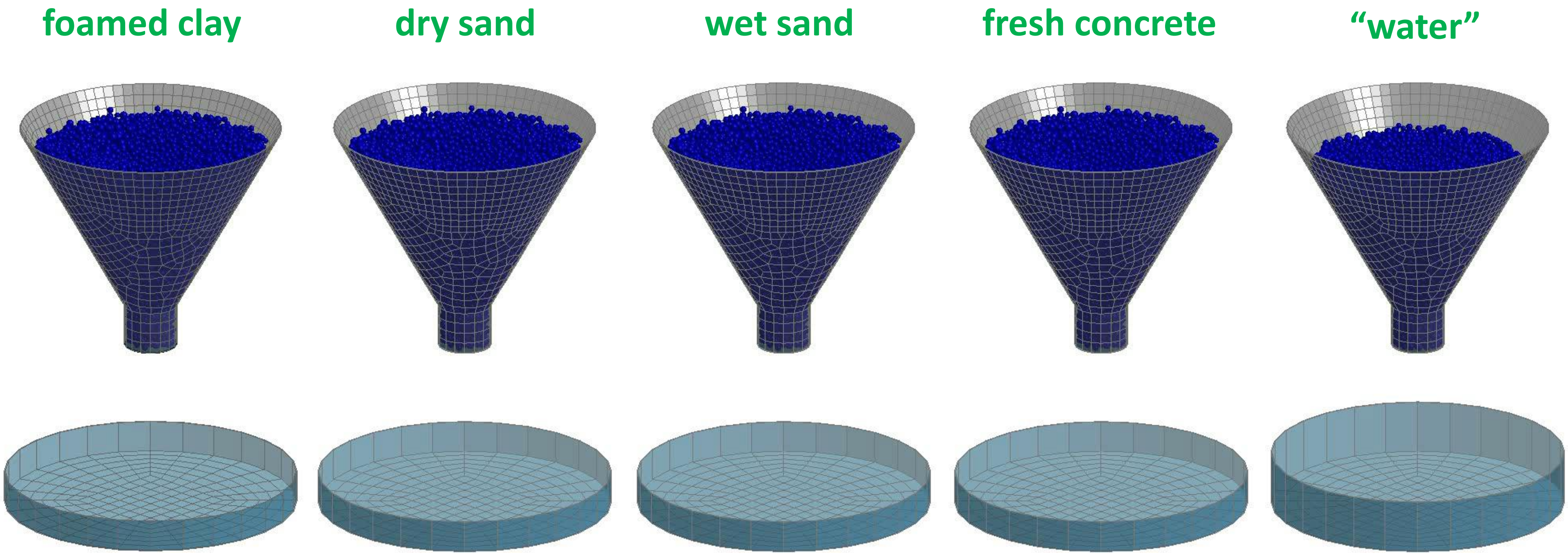
Wet Particle

Discrete Element Method (DEM)

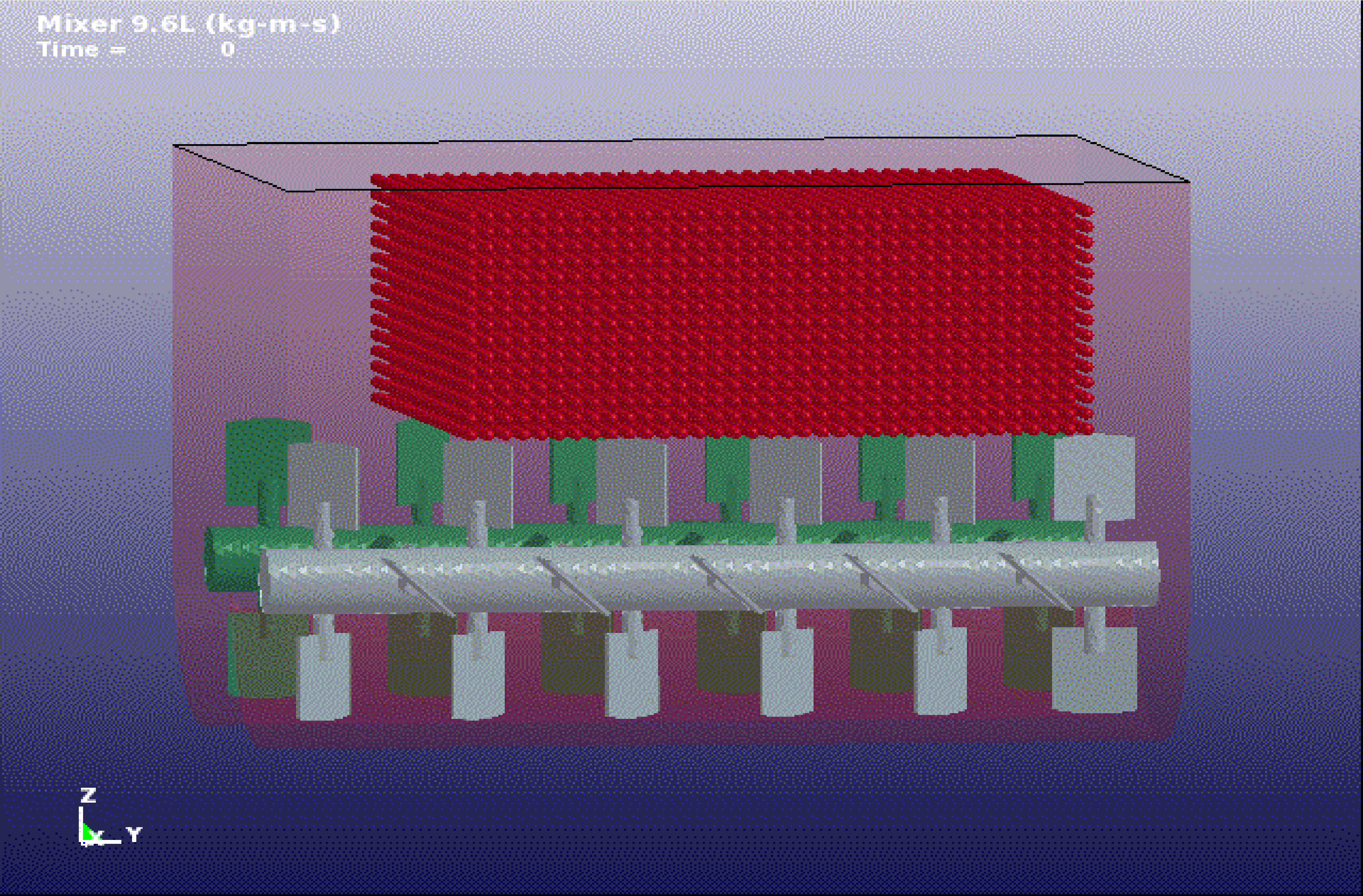
Discrete Element Sphere

Variation of the parameters Courtesy of Dr.-Ing. Nils Karajan, Dynamore GmbH

```
$-------+-------1--------+--------2---------+--------3---------+--------4---------+--------5
RHO       0.80E-6           2.63E-6            2.63E-6            2.63E-6             1.0E-6
P-P Fric     0.57              0.57               0.57               0.10               0.00
P-P FricR    0.10              0.10               0.01               0.01               0.00
P-W FricS    0.27              0.30               0.30               0.10               0.01
P-W FricD    0.01              0.01               0.01               0.01               0.00
CAP             0                 0                  1                  1                  1
Gamma        0.00              0.00            7.20E-8            2.00E-6             7.2E-8
$-------+-------1--------+--------2---------+--------3---------+--------4---------+--------5
```

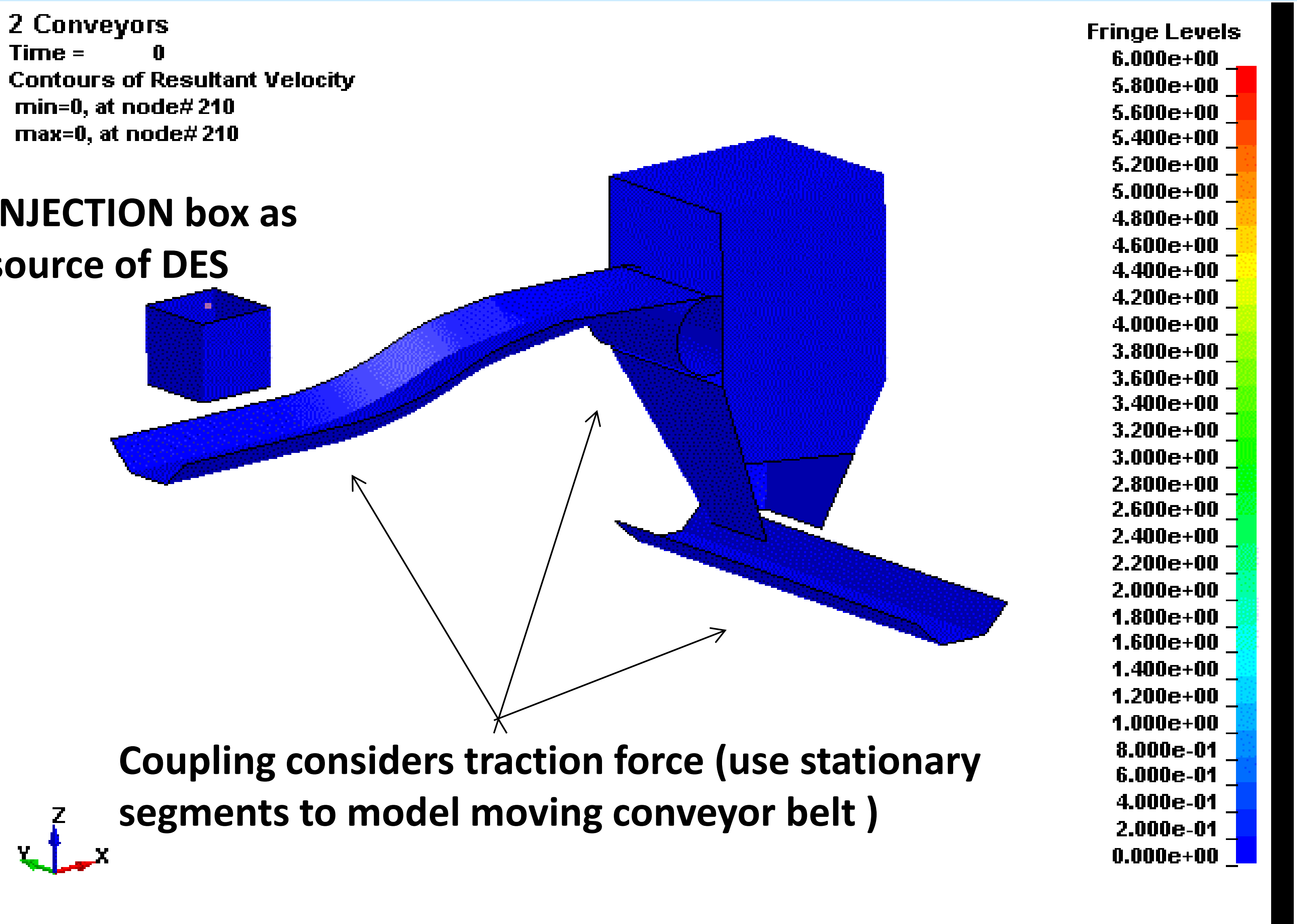


DEM Mixer



Node to Surface Coupling

*DEFINE_DE_TO_SURFACE_COUPLING

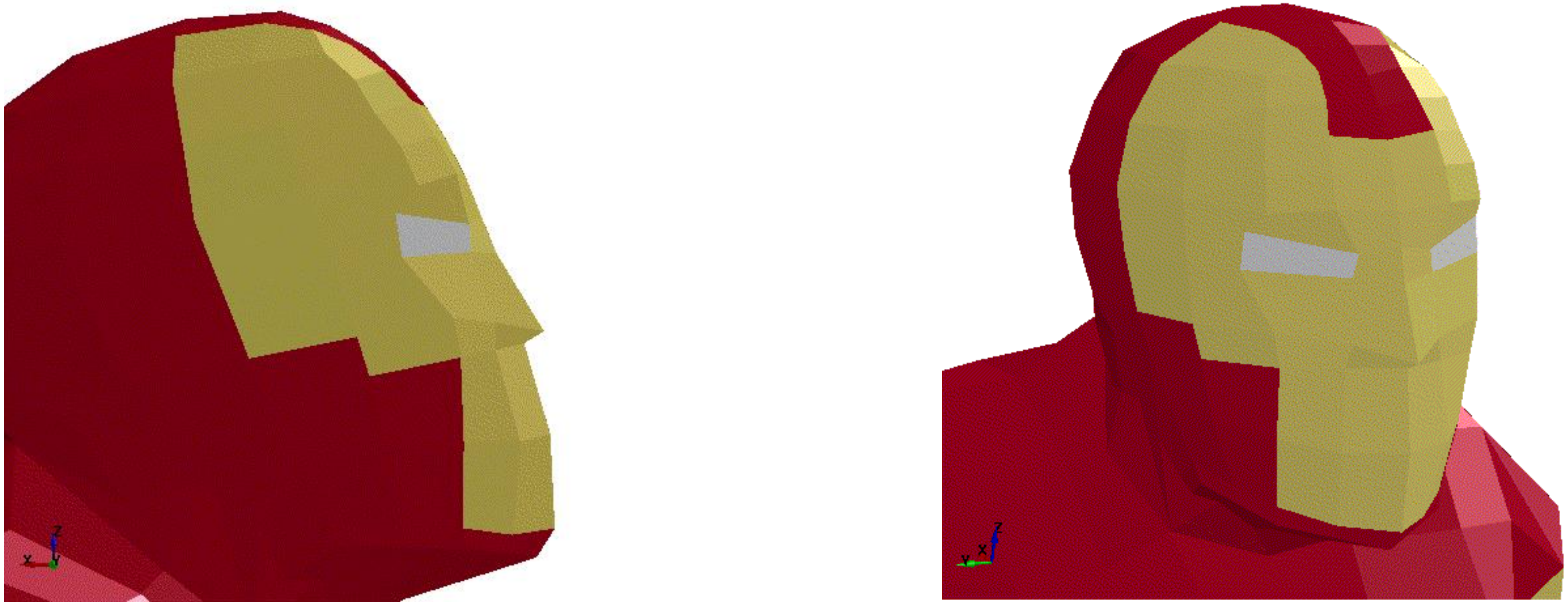


INJECTION box as source of DES

Coupling considers traction force (use stationary segments to model moving conveyor belt)

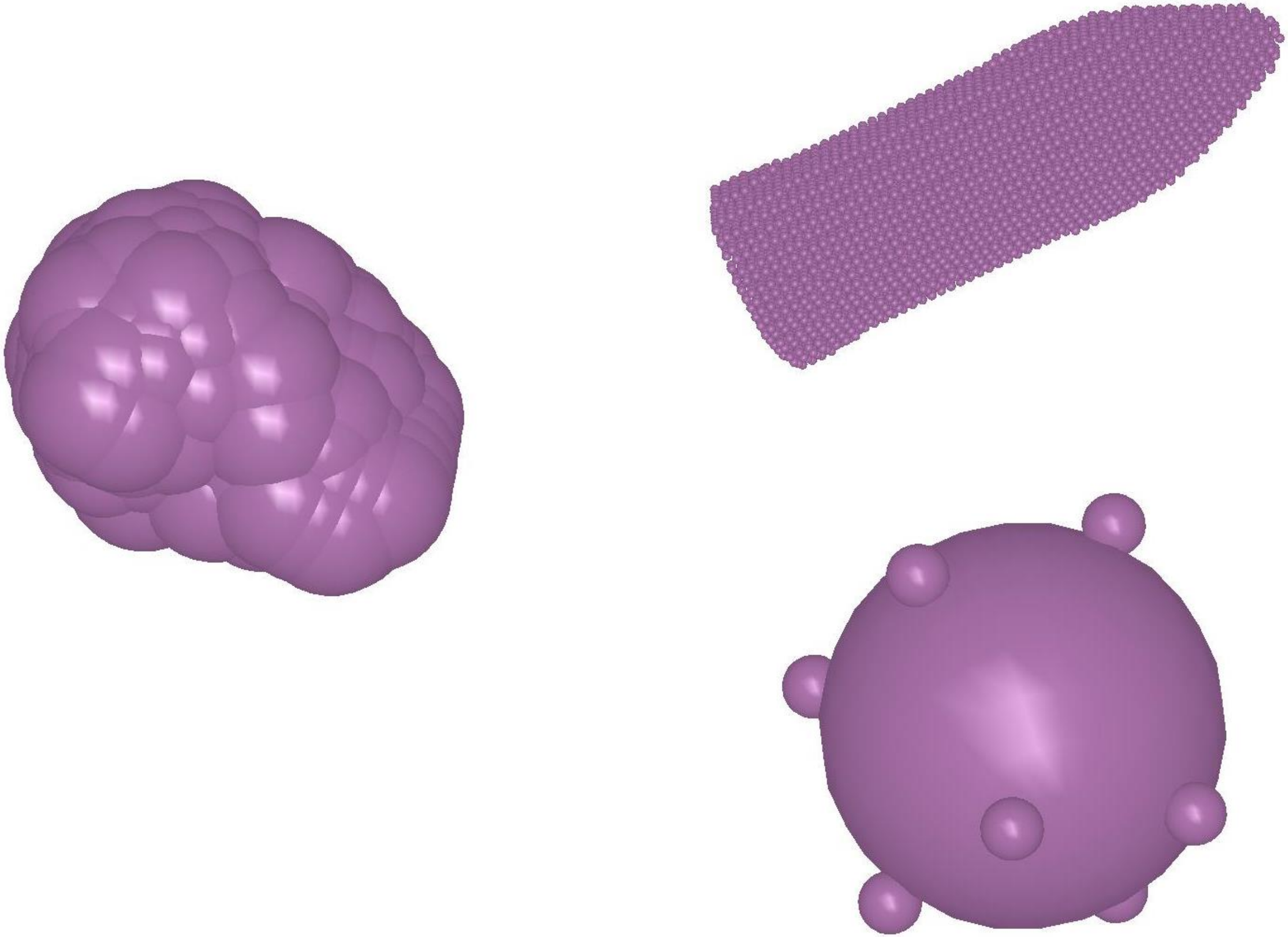
Tied Node to Surface Coupling

***DEFINE_DE_TO_SURFACE_TIED**



Throwing a pie in the face, Courtesy of Kazuya, Lancemore

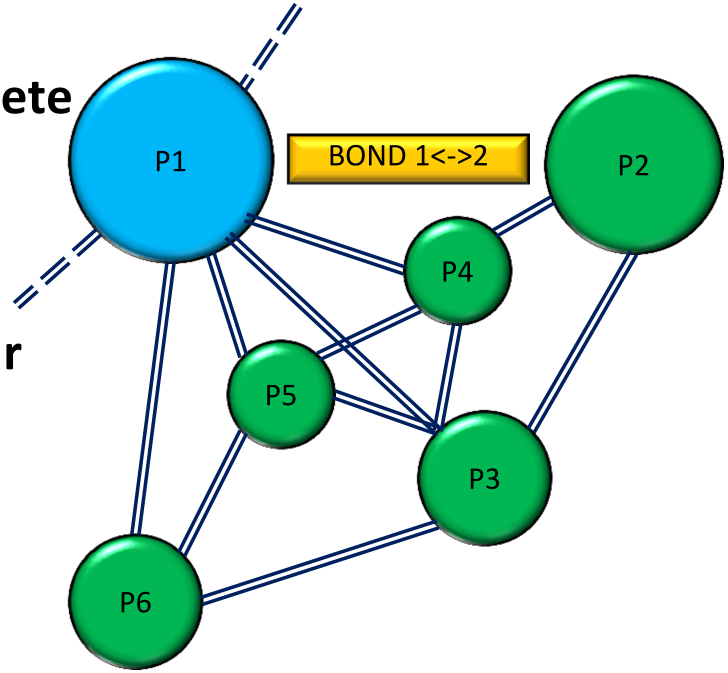
How to Form Other Shapes



Discrete Element Sphere with Bond Model

Emerge into Continuum Mechanics

- All particles are linked to their neighboring particles through Bonds.
- The properties of the bonds represent the complete mechanical behavior of Solid Mechanics.
- The bonds are independent from the DES model.
- They are calculated from Bulk Modulus and Shear Modulus of materials.
- Contact is disabled between bonded pair



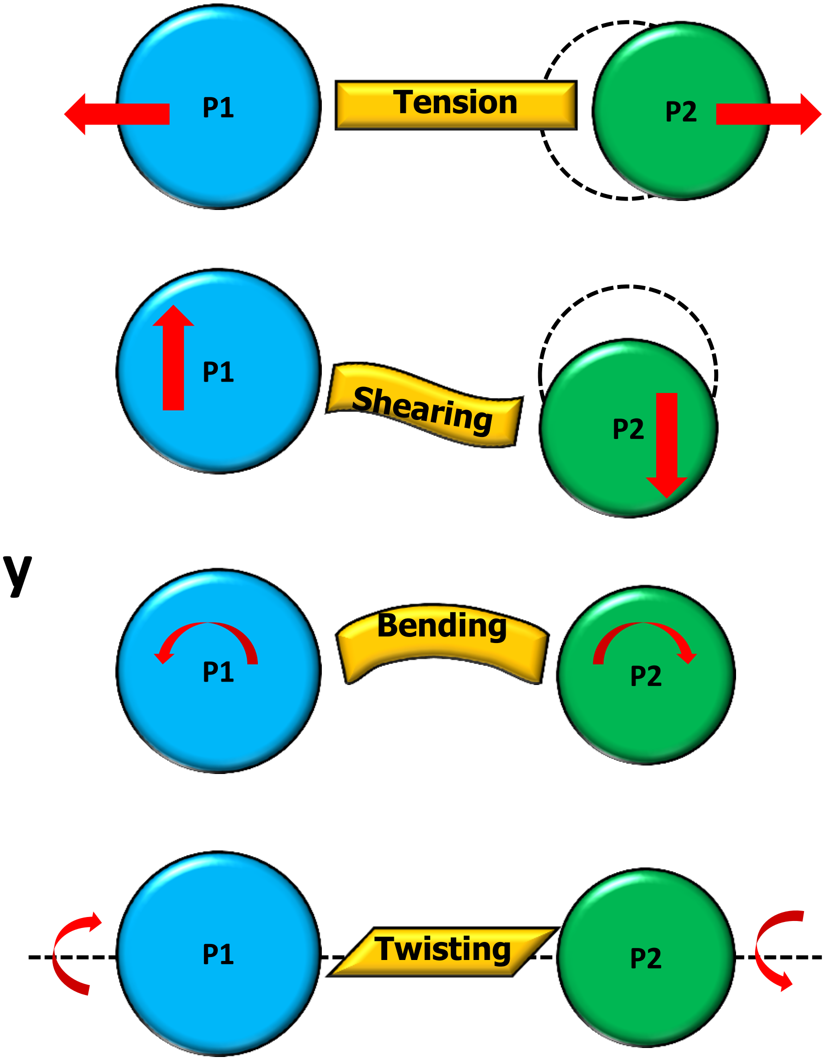
DEM Bonds

- **DE Bond Type I**
    - Simple links, truss or beam, etc…
    - Extended Peridynamics

- **DE Bond Type II**
    - Heterogeneous links to model continuum mechanics
    - Extended features and will use regular *MAT properties

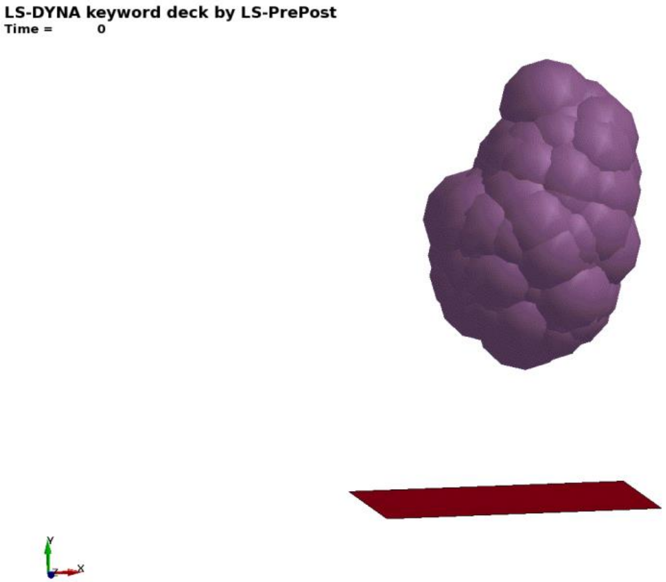
DE Bond TYPE I

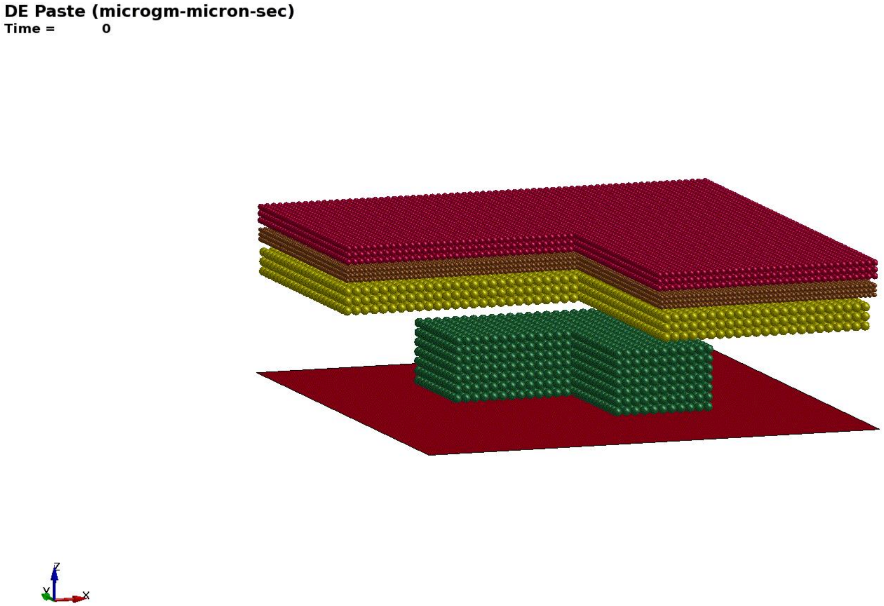
- **Every bond is subjected to:**
    - *Stretching*
    - *Shearing*
    - *Bending*
    - *Twisting*
- **The breakage of a bond results to Micro-Damage which is controlled by the critical fracture energy value J_{IC}.**



DEM Bond TYPE I

Form different shapes of particles using DEM

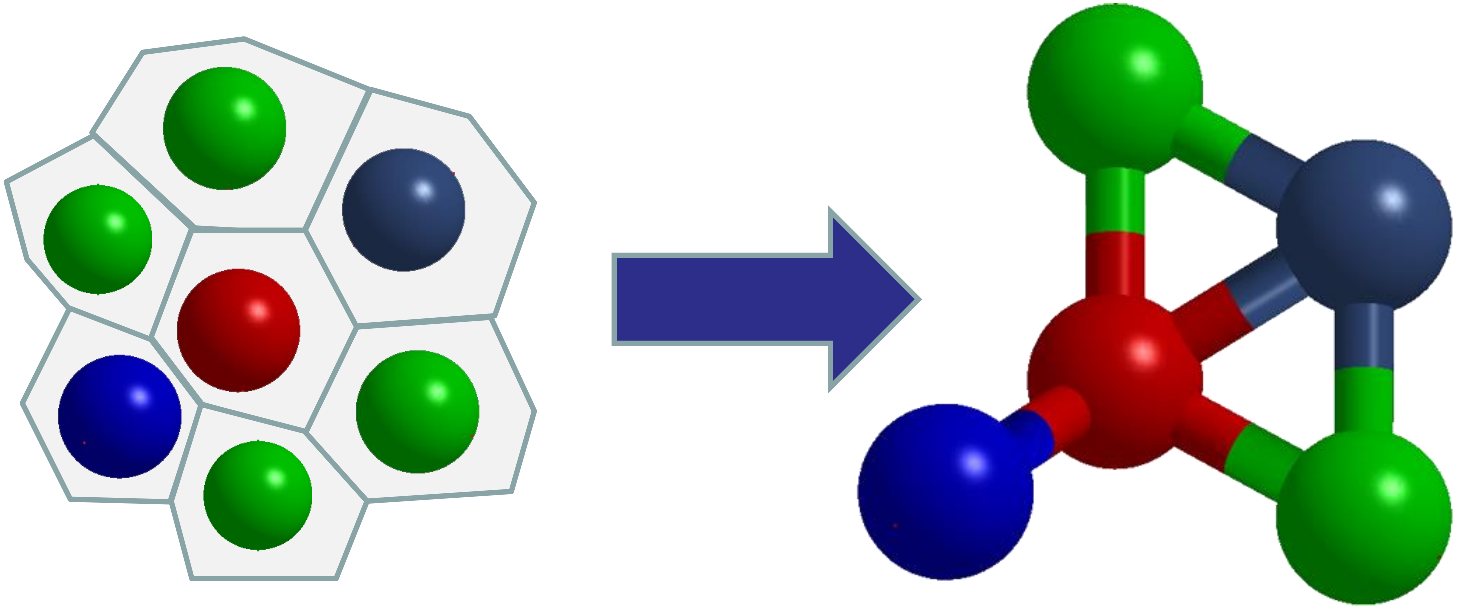




DEM TYPE II

Heterogeneous BOND (HBOND)

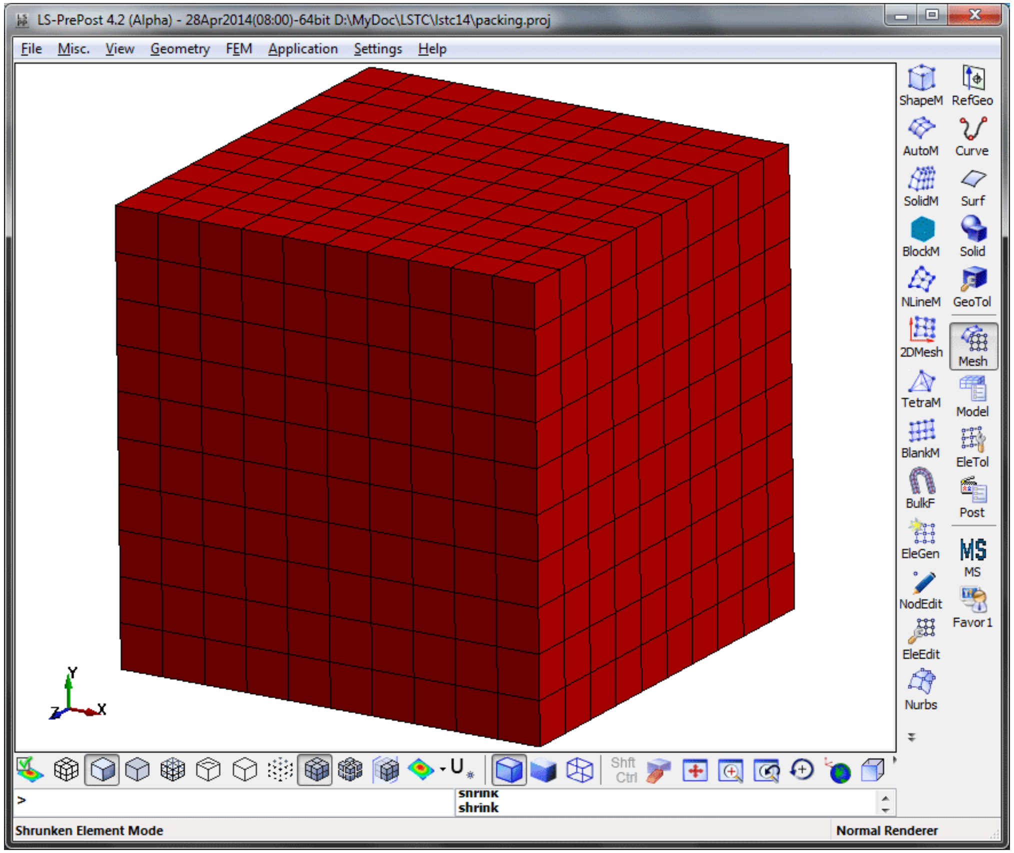
Continuum Particle Method



- ***DEFINE_DE_HBOND** connects two spheres with a heterogeneous bond.
- ***MAT** properties are used to determine the stiffness of the bonds automatically.
- Strains, stresses, and history variables are computed for each particle independently.
- ***INTERFACE_DE_HBOND** specifies various damage/failure models.
- Self contact will be activated for broken bonds.

One Particle Method

From “Continuum” to “Discrete”



LS-PrePost 4.2 (Alpha) - 28Apr2014(08:00)-64bit D:\MyDoc\LSTC\lstc14\packing.proj

```
>
shrink
shrink
```

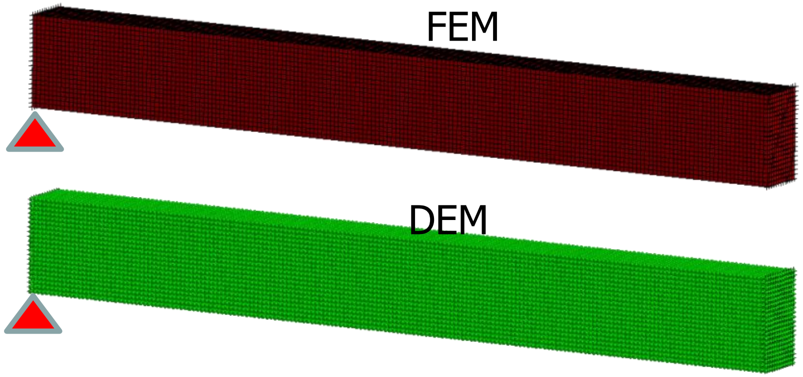
Shrunken Element Mode

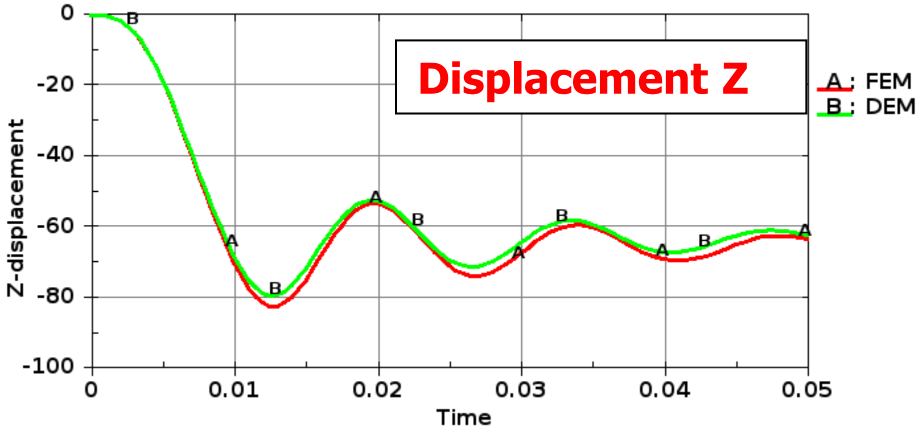
Normal Renderer

- One model setup
- One solver
- Same material models
- Multi-physics:
  Continuum Mechanics
  Damage Mechanics
  Fracture Mechanics
  Discrete Mechanics
- Built-in self contact
- Coupling with other FEM and particle methods

HBOND Verification

A simply-supported beam under a body force



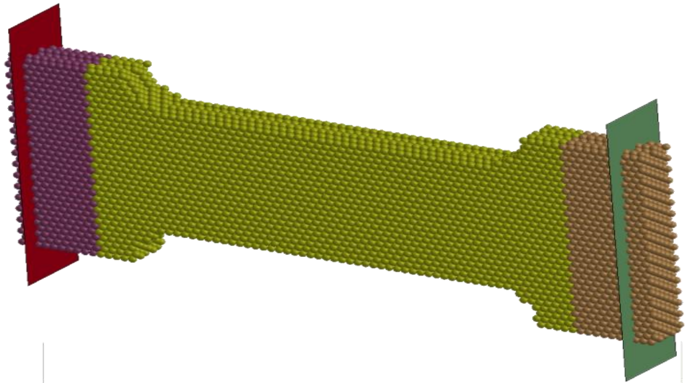


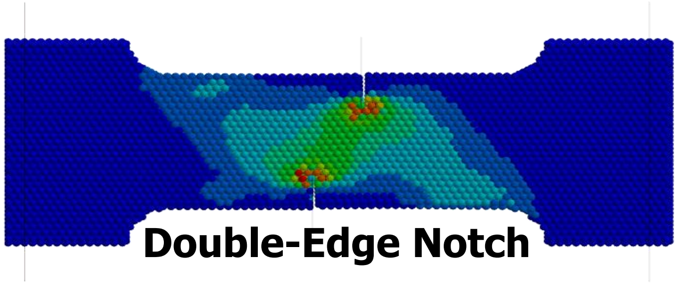
- **FEM & DEM models are created for one half beam with the symmetric boundary conditions in the middle.**
- **The displacements & stresses obtained by the DEM are very close to those by the FEM.**
- **No boundary effects.**

Tension Stress X

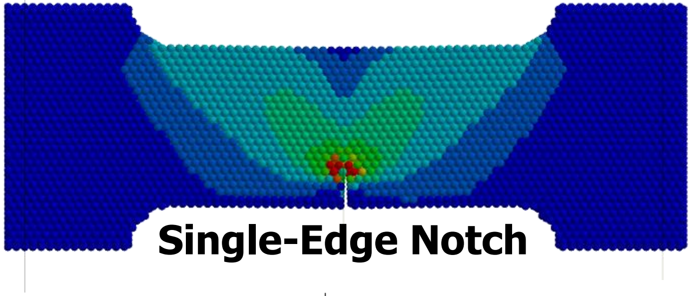
HBOND Verification

Specimen under Tension

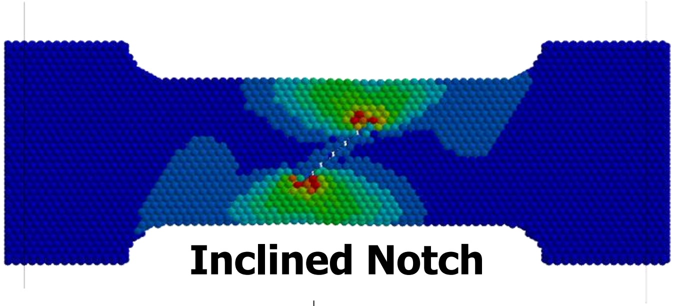




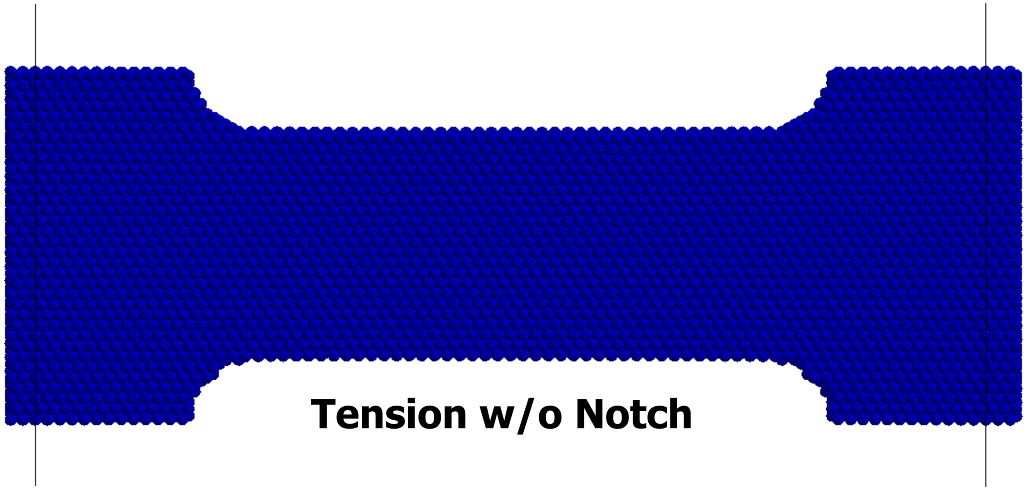
Double-Edge Notch



Single-Edge Notch



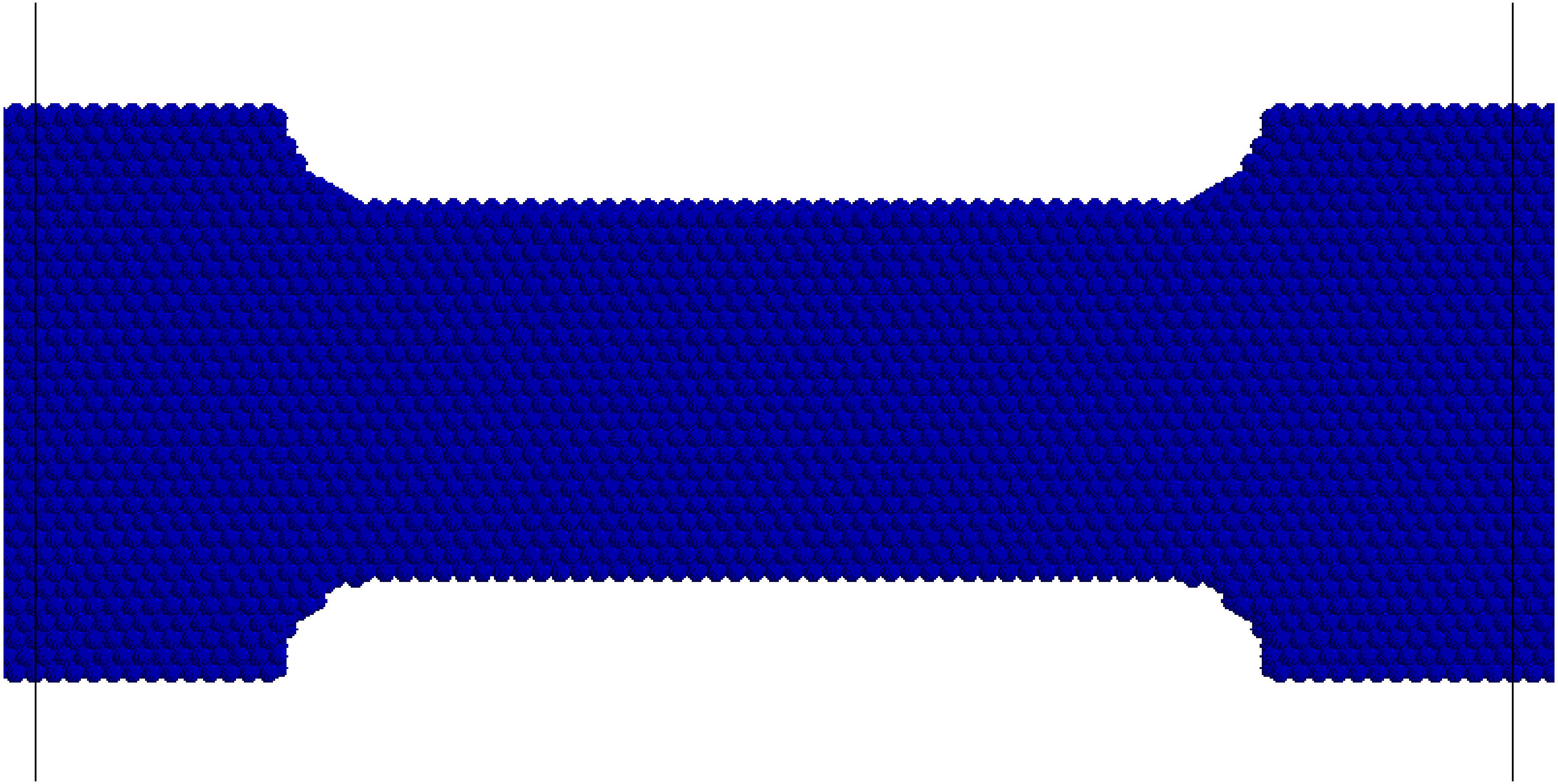
Inclined Notch



Tension w/o Notch

HBOND Verification

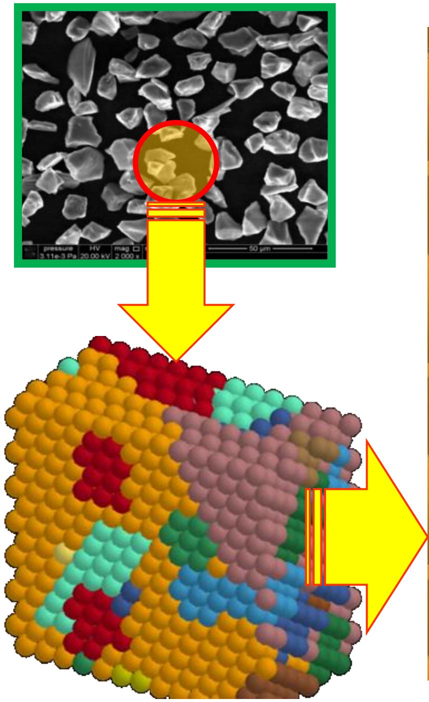
Specimen under Compression without Pre-Notch

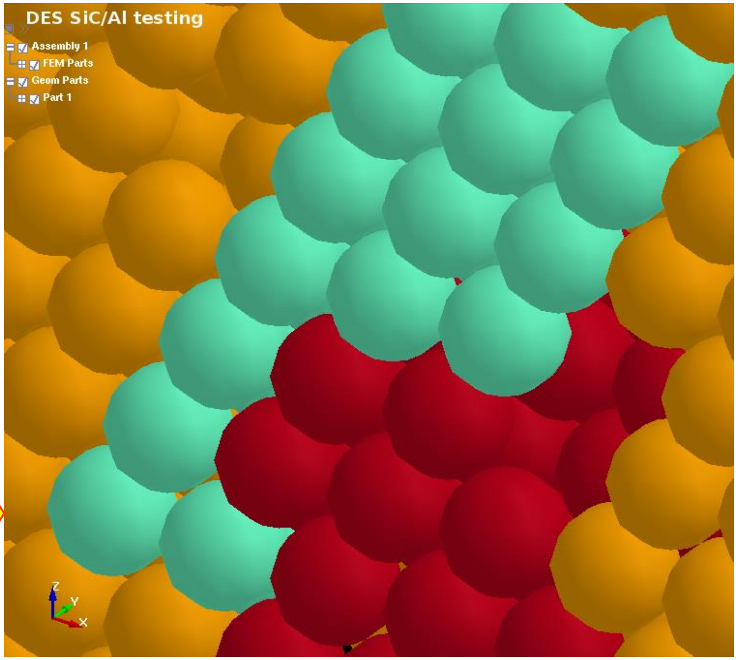


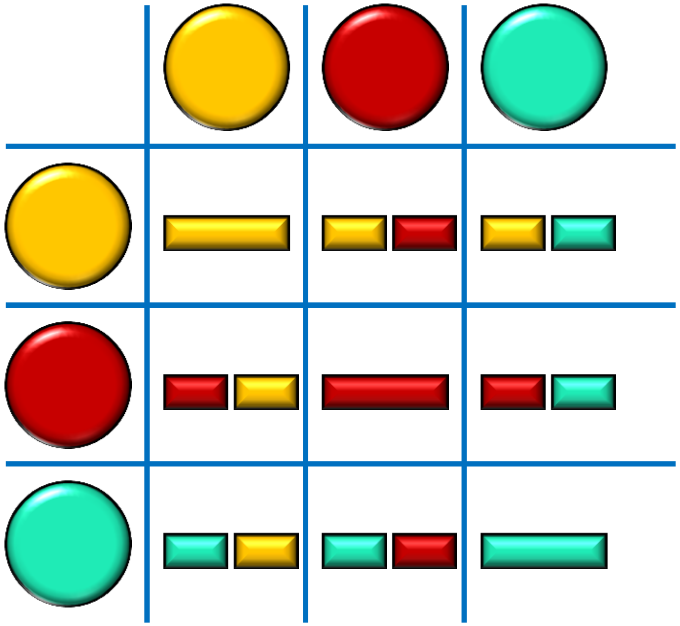
HBOND Micro-Mechanics

***DEFINE_DE_HBOND** creates a heterogeneous bond between different spheres.

***INTERFACE_DE_HBOND** defines various damage/failure models for the heterogeneous bonds based on the material properties of the connecting particles.



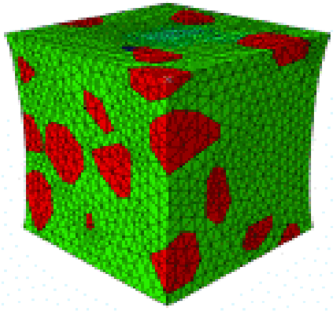




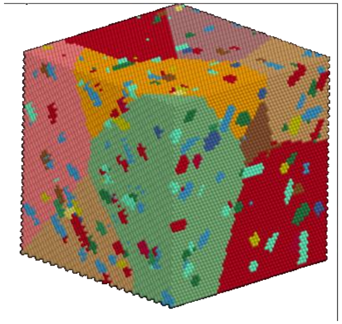
Various heterogeneous bonds

SiC/Al Metal Matrix Composite

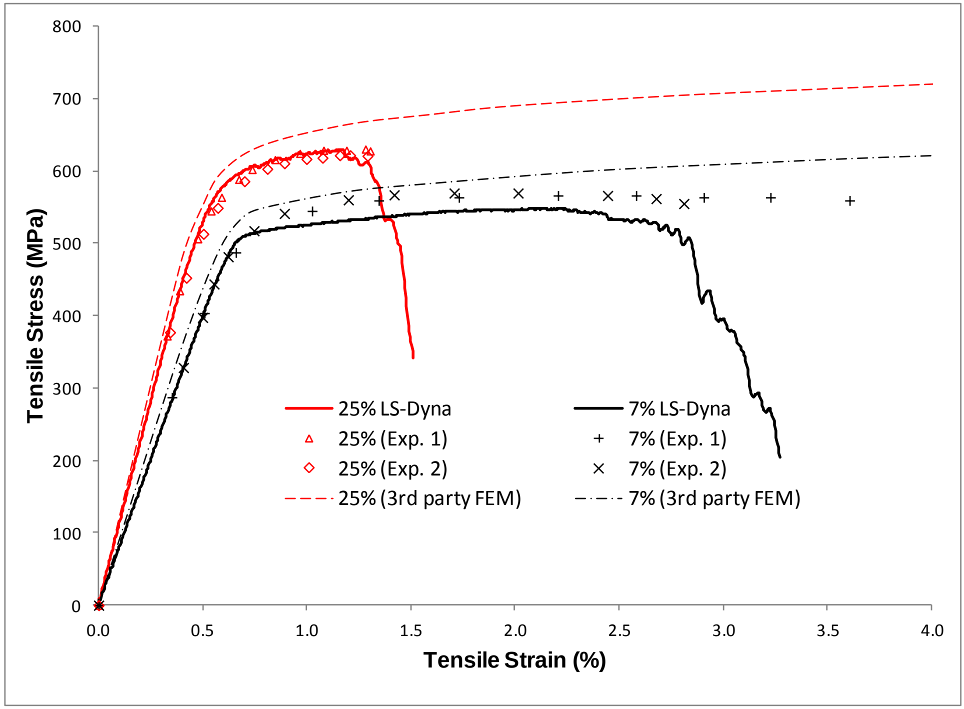
DEM for Material Design



3rd party FEM



LS-DYNA DEM

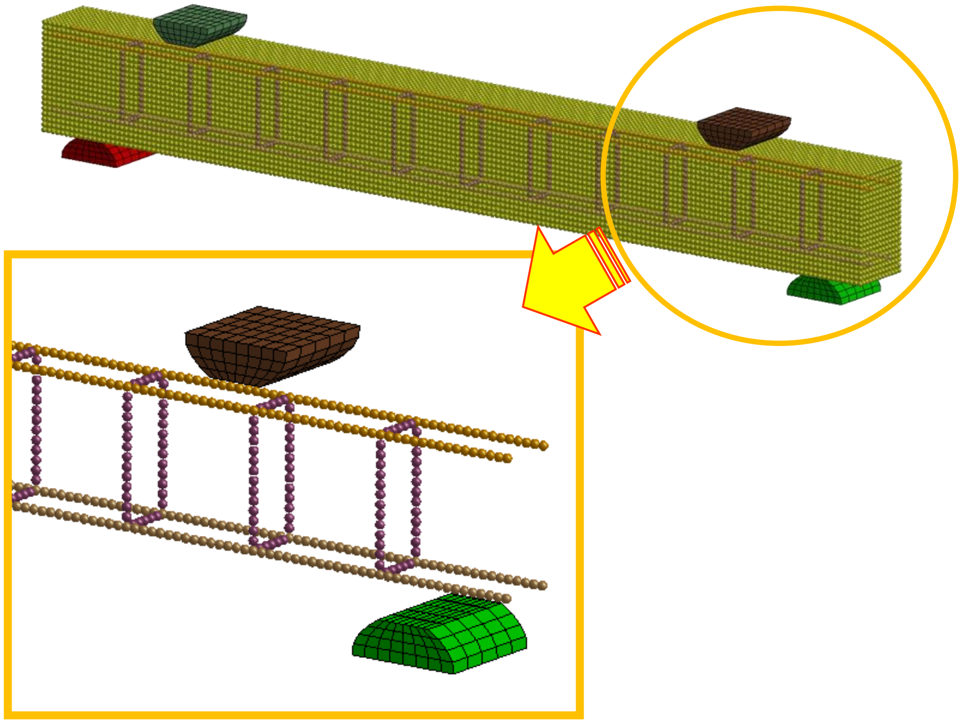


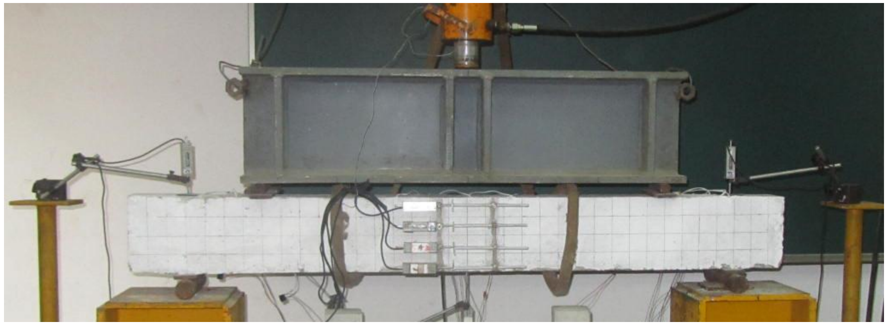
Material Properties	Al	SiC
Density: [kg/m^3]	2,700	3,100
Young's modulus [GPa]	71.7	427
Poisson's ratio:	0.33	0.17
Failure Energy Rate: [kN/m]	40	15
Average Particle size: [um]	-	13

LS-DYNA Results	7% vol		25% vol	
	Exp.	Num.	Exp.	Num.
Young's modulus [GPa]	**84.2**	**80.6**	**113.3**	**113.5**
Tensile strength [MPa]	**568.6**	**545.0**	**623.6**	**641.3**
Limit strain	**1.8%**	**1.9%**	**1.2%**	**1.4%**

HBOND

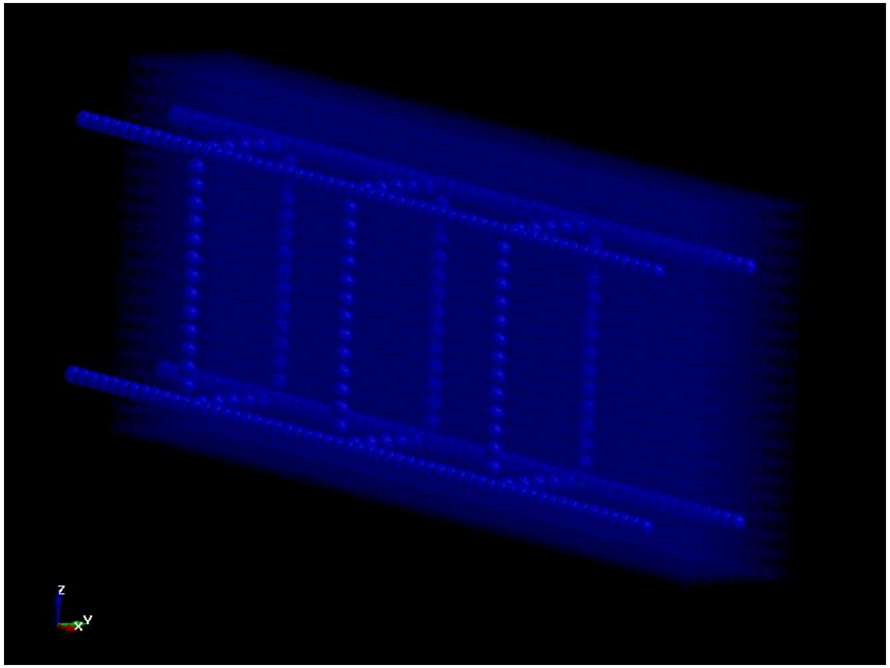
A Reinforced Bar under Four-Point Bending





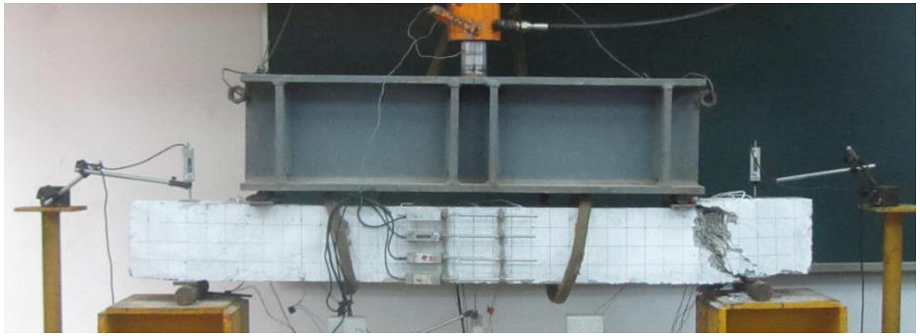
***DEFINE_DE_HBOND** bonds all parts.

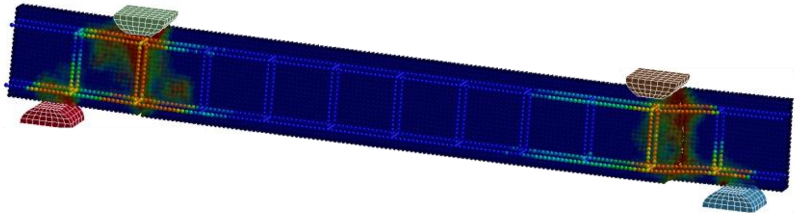
***INTERFACE_DE_HBOND** defines different de-bonding criteria between parts

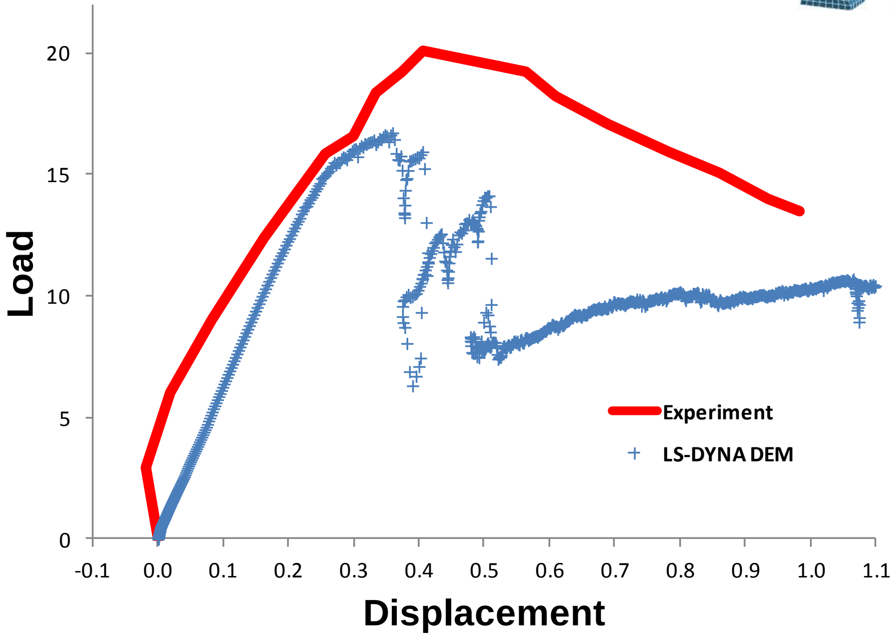


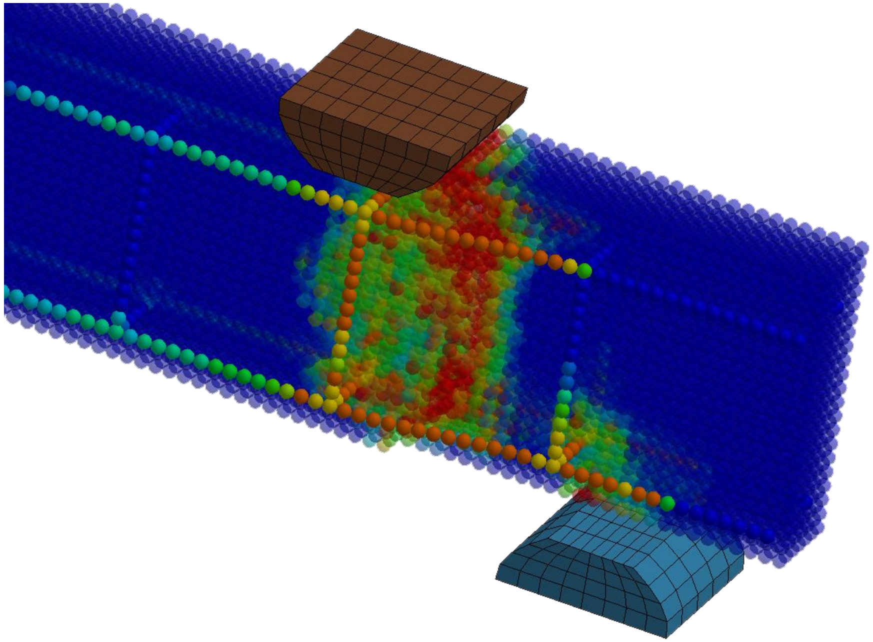
De-bonding process

Comparison between DEM & Experimental Results





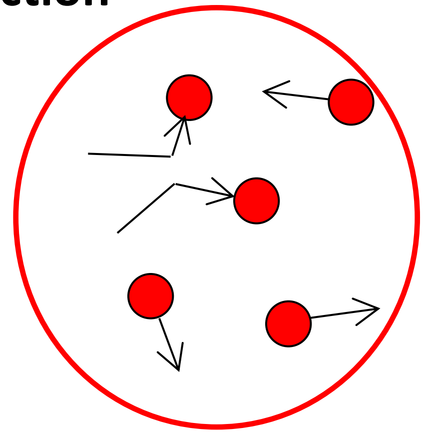


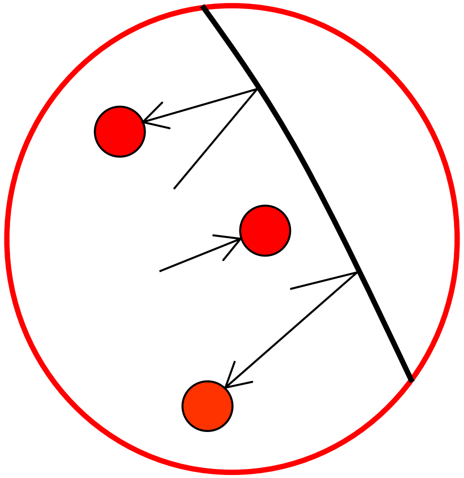


Coupled Multi-Physics Solvers

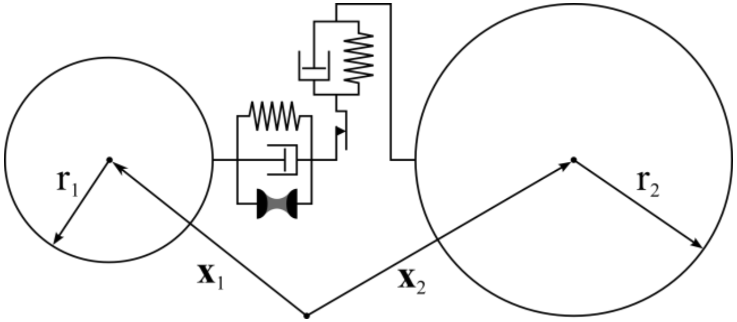
Particle-particle interaction

Particle Gas



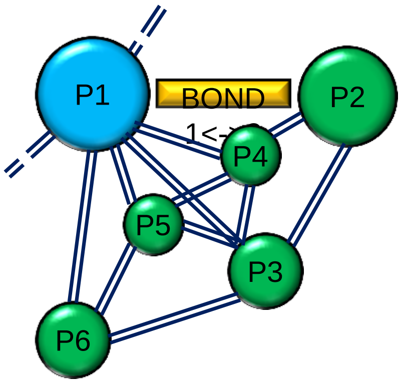


DEM



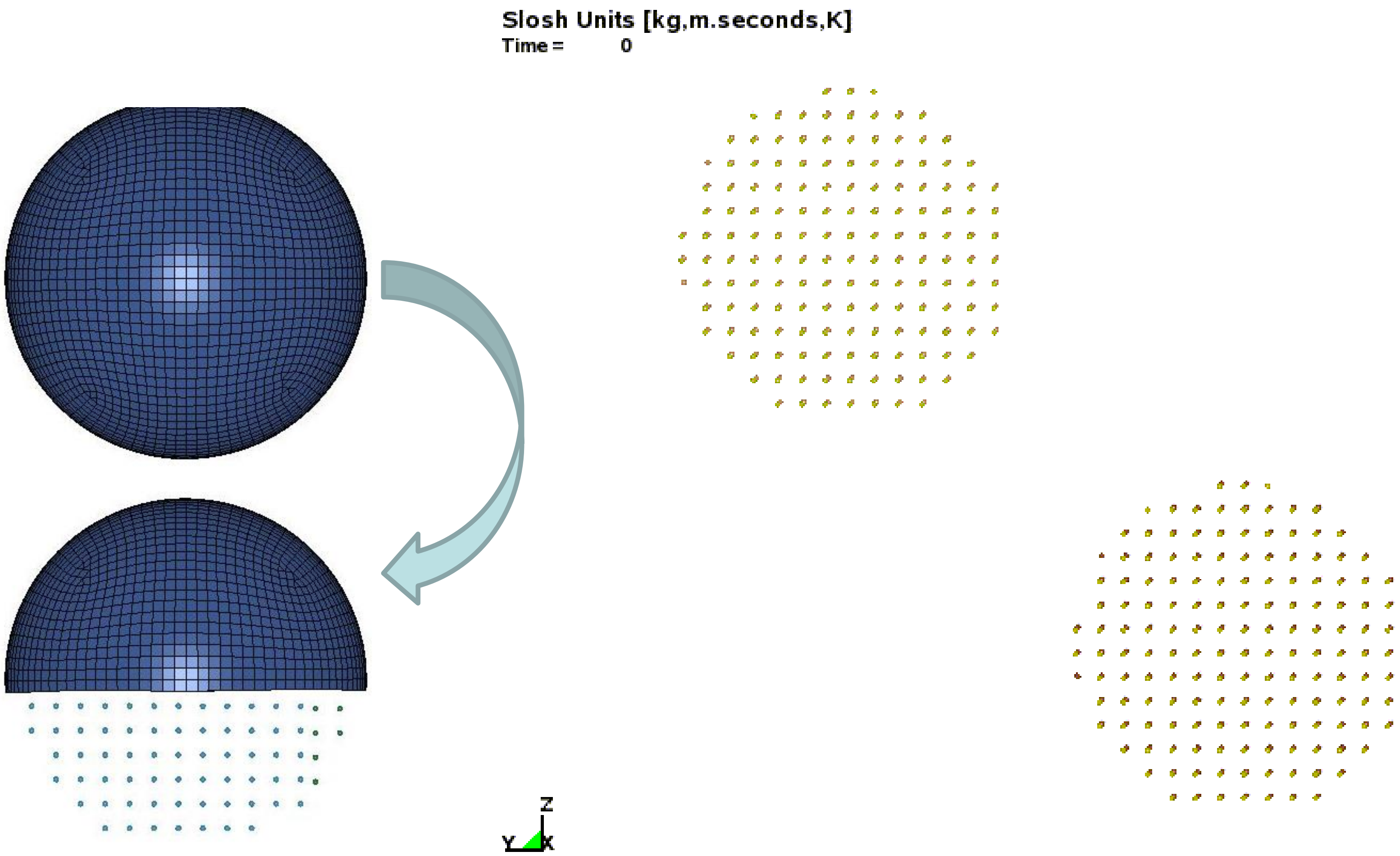
1) **Node to Node**
2) **Node to Beam**
3) **Node to Segment**
4) **Node to Volume**

DEM bond



Node to Node Coupling

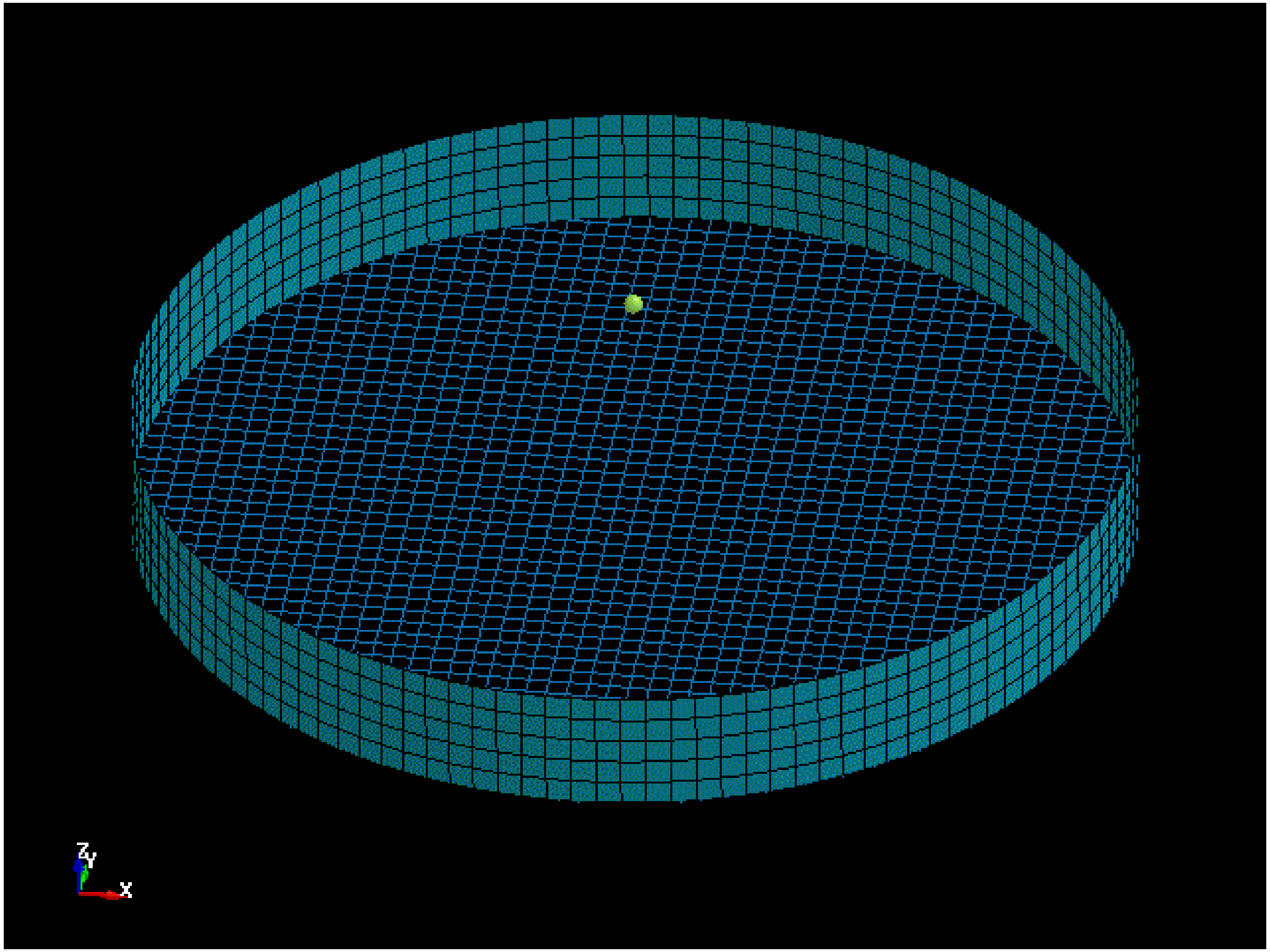
SPH to SPH Contact



Tank sloshing with fluid and vapor (node to node contact)
Density ratio ~ 1000

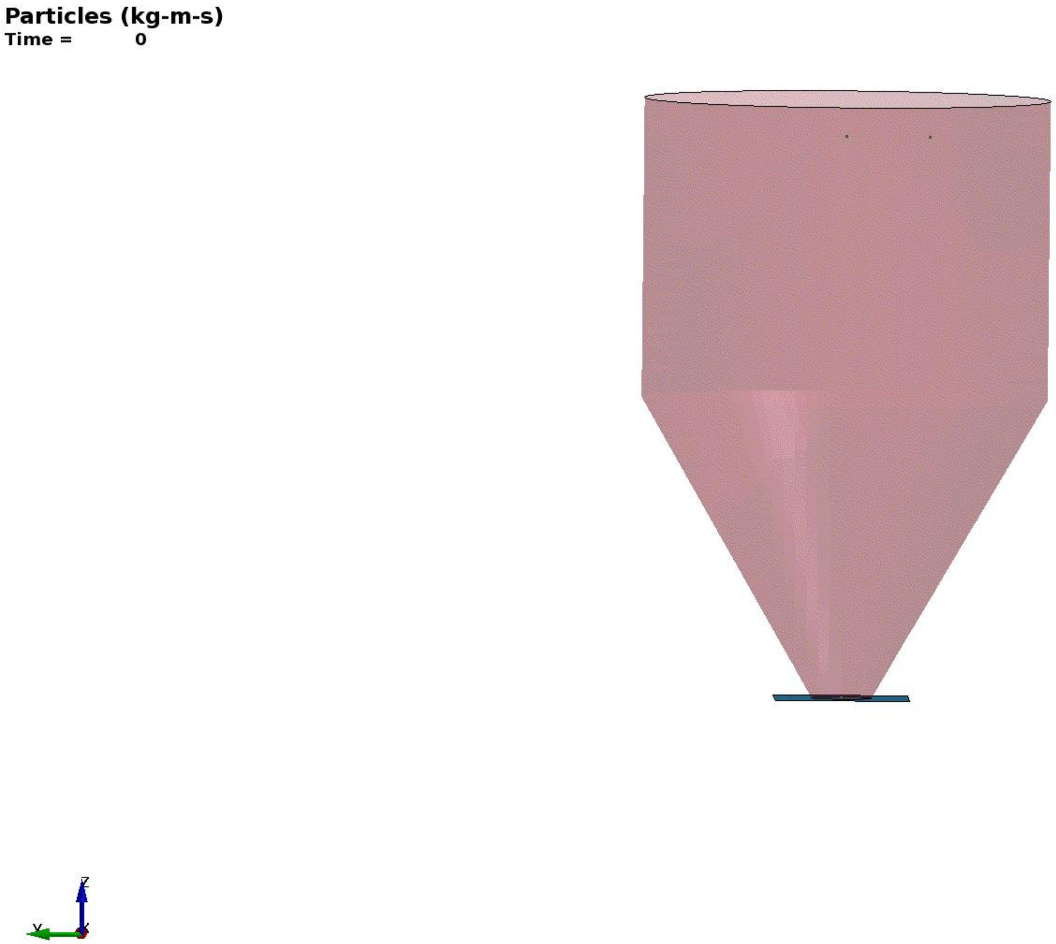
Node to Beam Coupling

DES to Beam Contact



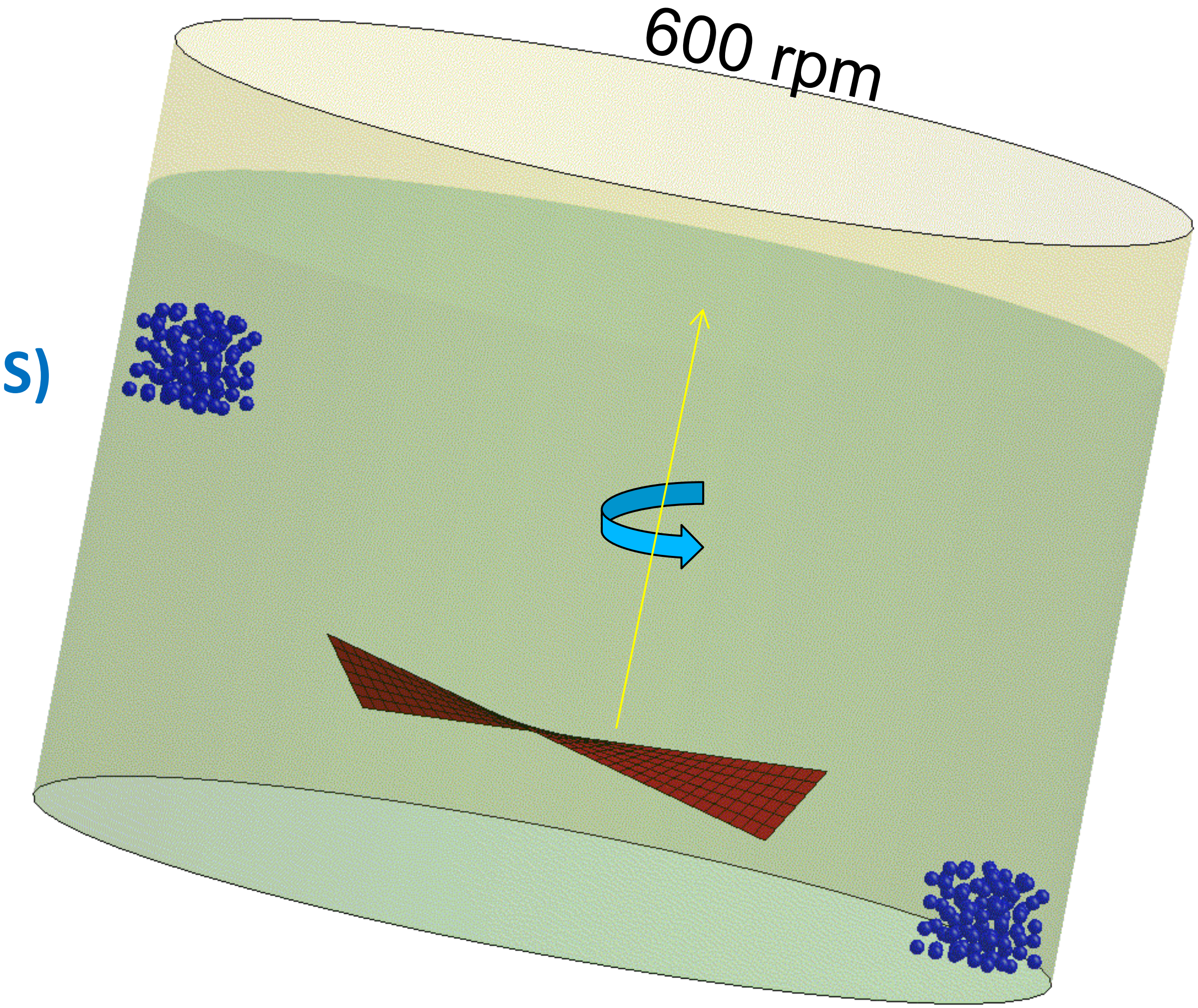
Node to Surface Coupling

DES to Segment Contact



Node to Volume Coupling

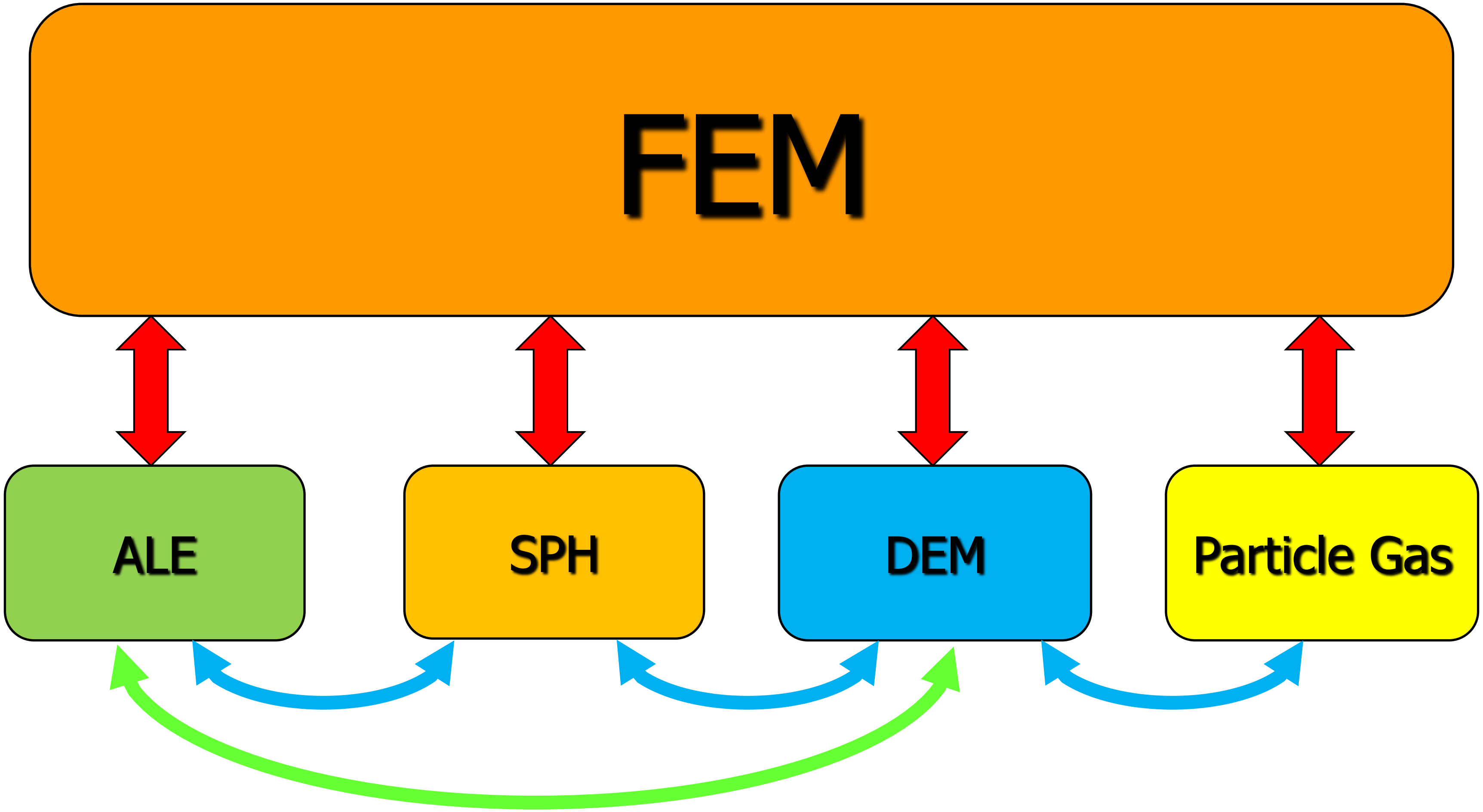
DES to ALE Contact



Powder(DES)

Water(ALE)

Coupled Multi-Physics Solvers



FEM

ALE

SPH

DEM

Particle Gas

Thank you!

Thank You!

Summary

- LSTC is working to be the leader in cost effective large scale numerical simulations
  - LSTC is providing dummy, barrier, and head form models to reduce customer costs.
  - LS-PrePost, LS-Opt, and LS-TaSC are continuously improving and gaining more usage within the LS-DYNA user community
  - LSTC is actively working on seamless multistage simulations in automotive crashworthiness, manufacturing, and aerospace
- The scalable implicit solver is quickly gaining market acceptance for linear/nonlinear implicit calculations and simulations
  - Robustness, speed, accuracy, and scalability have rapidly improved
  - Developments:
    - Acoustics
    - Rotational dynamics

Future

- LSTC is not content with what has been achieved
- New features and algorithms will be continuously implemented to handle new challenges and applications
  - Electromagnetics,
  - Acoustics,
  - Compressible and incompressible fluids
  - Isogeometric shell elements and NURB contact algorithms
  - Discrete element methodology for modeling granular materials, failure, etc.
  - Simulation based airbag folding and THUMS dummy positioning underway
- Multiscale capabilities are under development
  - Subcycling
- Hybrid MPI/OPENMP developments are showing significant advantages at high number of processors for explicit and implicit solutions

Thank You!

10th European LS-DYNA Users Conference

- 10th European LS-DYNA Users' Conference
- 16 – 17 June 2015 in Würzburg, Germany
- Maritim Hotel


LSTC
Livermore Software
Technology Corp.