# Recent Developments in LS-DYNA

LS-DYNA Forum
October 7, 2014
Bamberg

# Outline

- Introduction
- Developments
- Conclusions

# LS-DYNA Application Areas

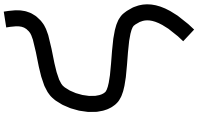
Development costs are spread across many industries

## Automotive

- Crash and safety
- NVH
- Durability
- FSI

## Aerospace

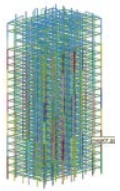
- Bird strike
- Containment
- Crash



## Manufacturing

- Stamping
- Forging

## Consumer Products

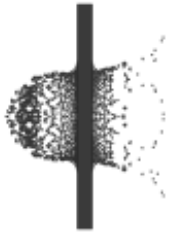


## Structural

- Earthquake safety
- Concrete structures
- Homeland security

## Electronics

- Drop analysis
- Package analysis
- Thermal

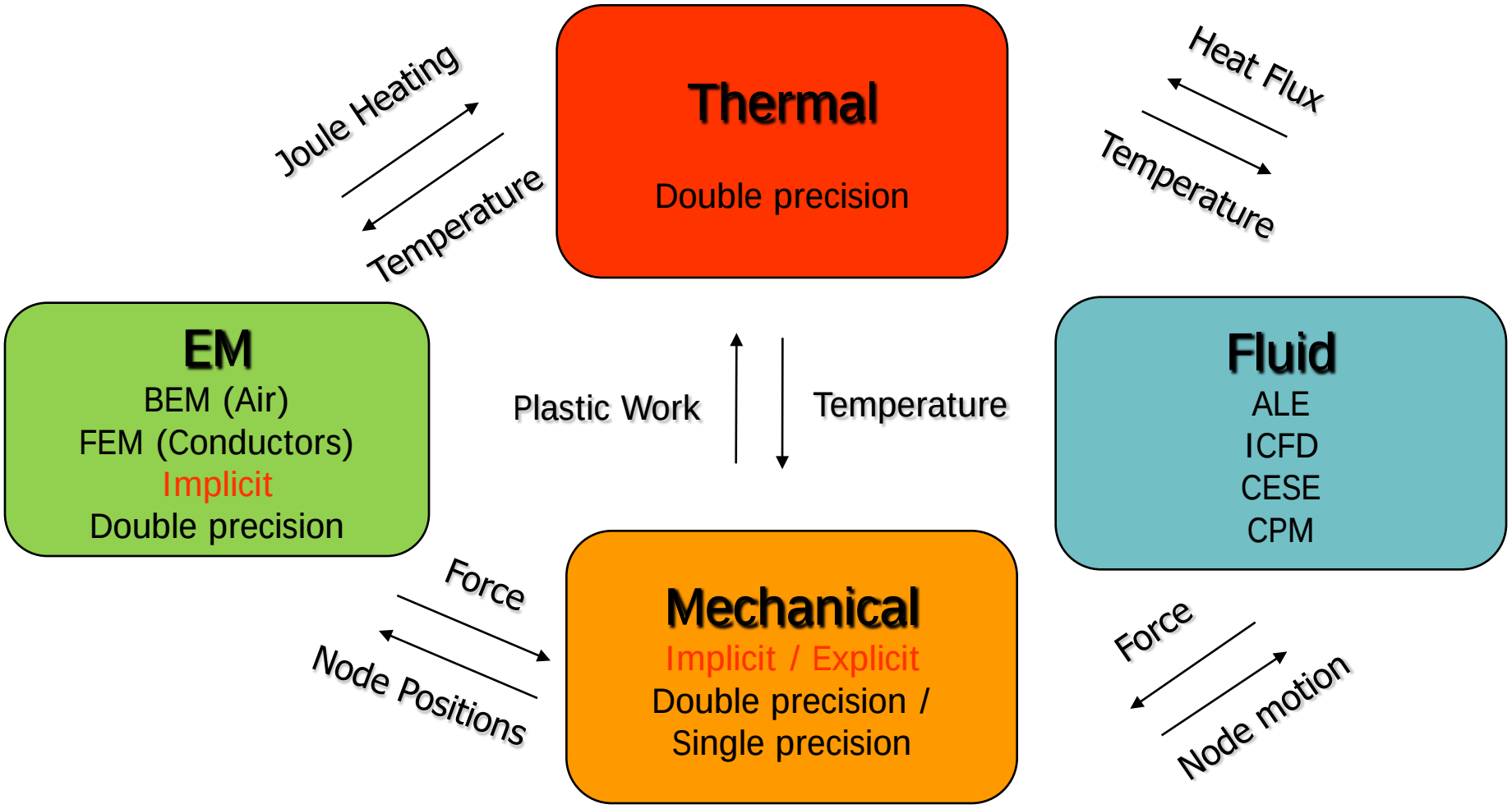


## Defense

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

# Accommodates Coupled Simulations

Multiple field equations are strongly coupled



# 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**

# One Model for All Applications

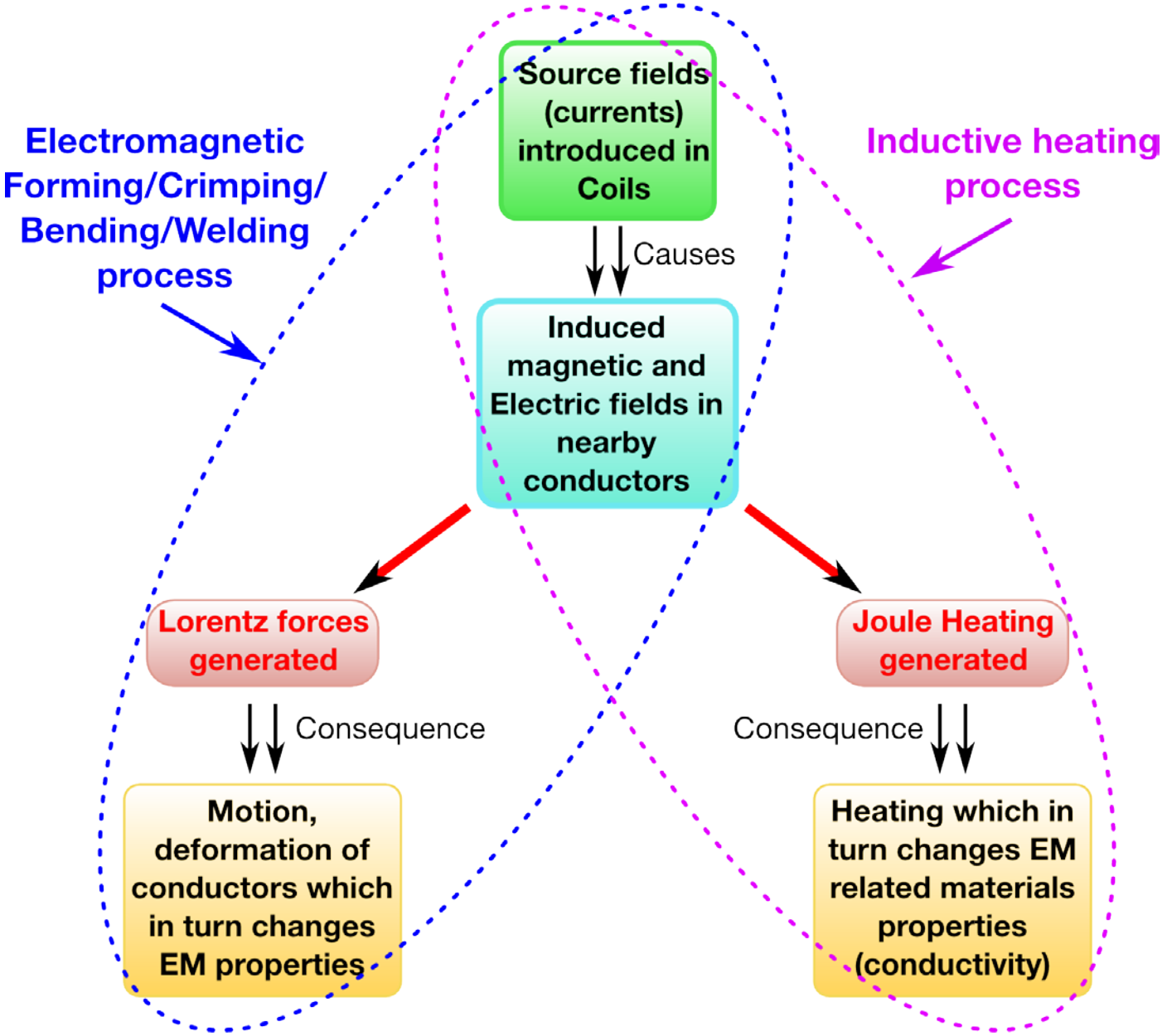
In crash, one structural model is used for frontal, side, offset, and rear impacts. The structural models used for durability, NVH, and FSI calculations are identical to the crash models with possible adjustments related to mesh density. The wide usage of the single model results in a model with fewer modeling errors:

- Only one model to revise for design changes.
- Only one model to check for errors.
- Multi-physics problems can be addressed.
- Easier database management

Initial stress, strain, and thickness distributions from manufacturing analysis are available for all simulations

# Electromagnetics

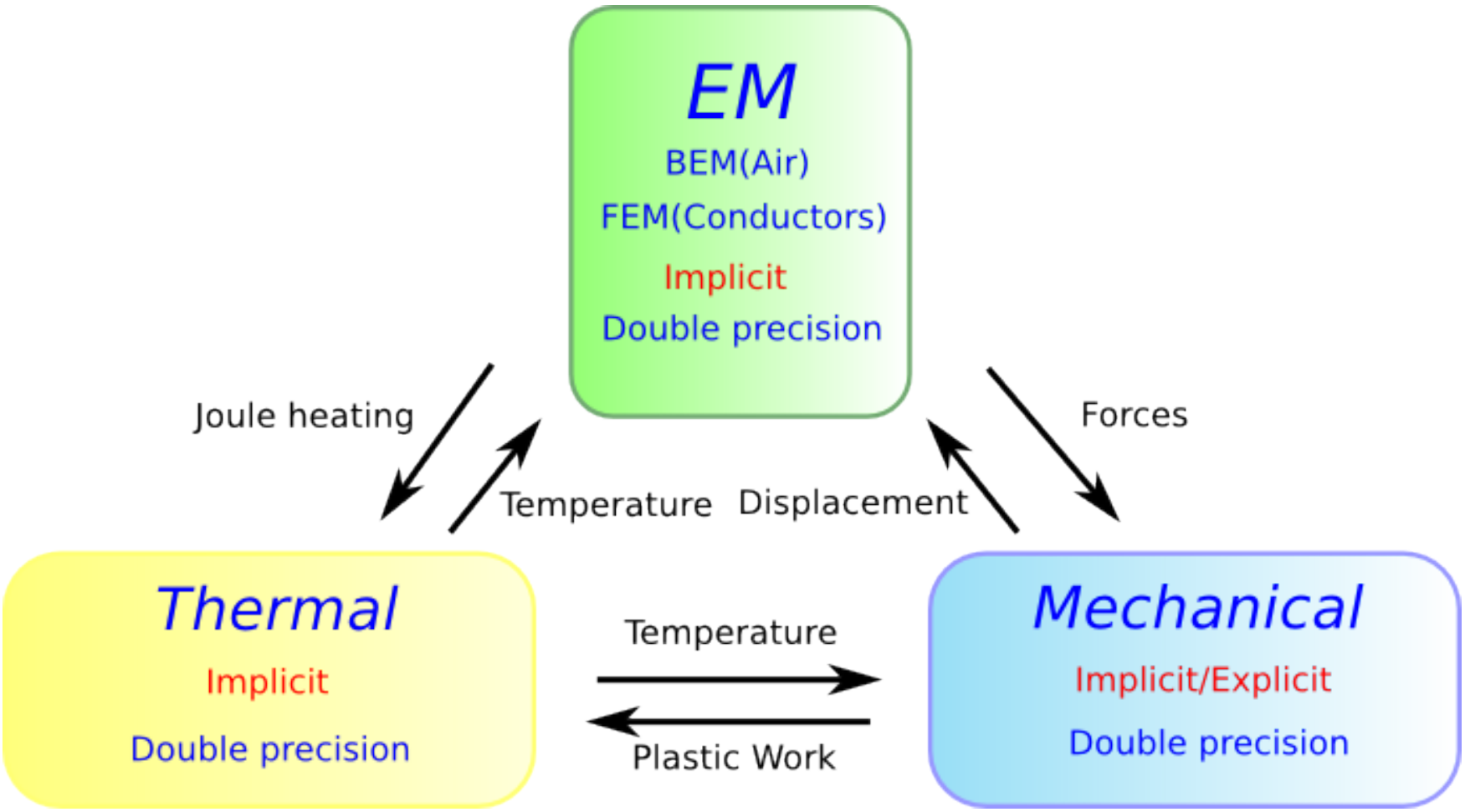
# EM Solver Introduction



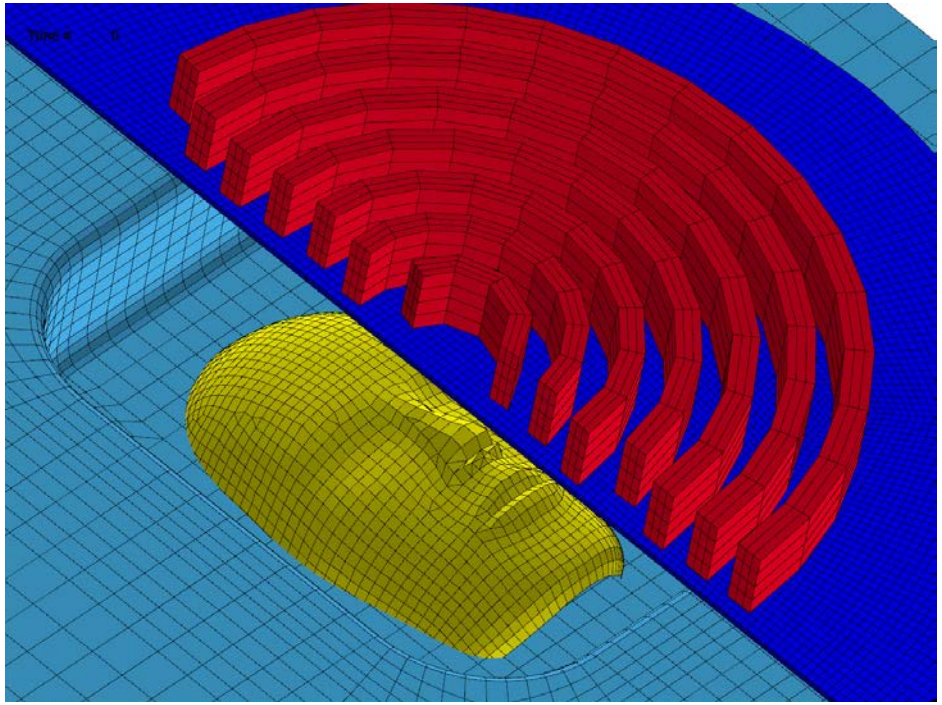
# EM Solver Introduction

- Fully implicit.
- Double precision.
- 2D axisymmetric solver /3D solver.
- SMP and MPP versions available.
- Automatically coupled with LS-DYNA solid and thermal solvers.
- FEM for conducting pieces, no air mesh needed (FEM-BEM method).
- New set of keywords starting with *EM.

# EM Solver Introduction

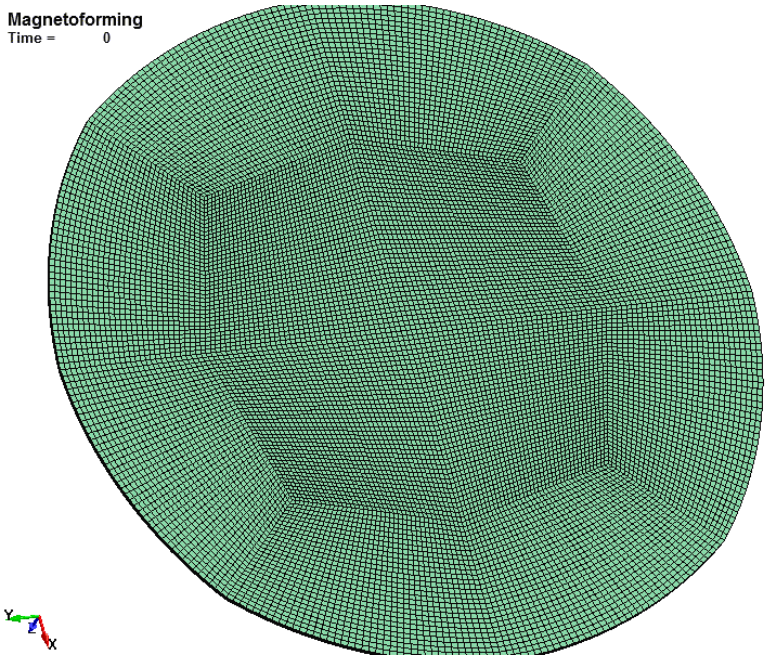


# EM Solver Introduction



**MMF: High velocity forming process**

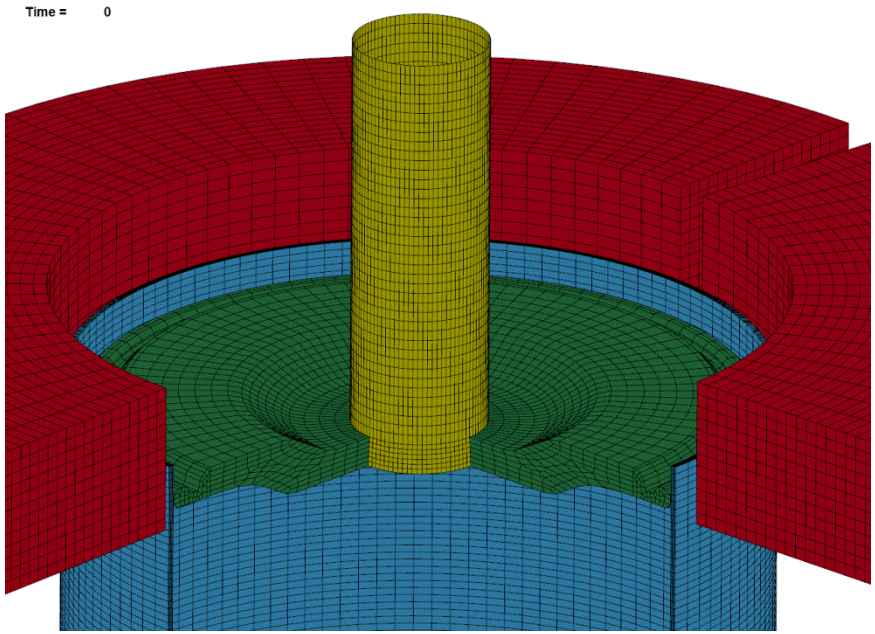
- Forming limits increased
- Springback reduced
- Wrinkling reduced
- High reproducibility




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

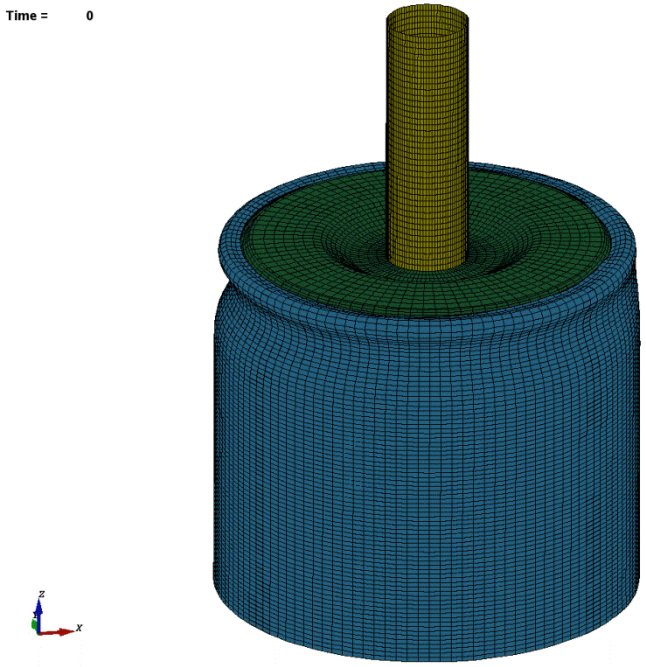
Bmax
shaping your future

# EM Solver Introduction




**Magneto crimping**

**Tensile test of the crimped parts (no EM)**

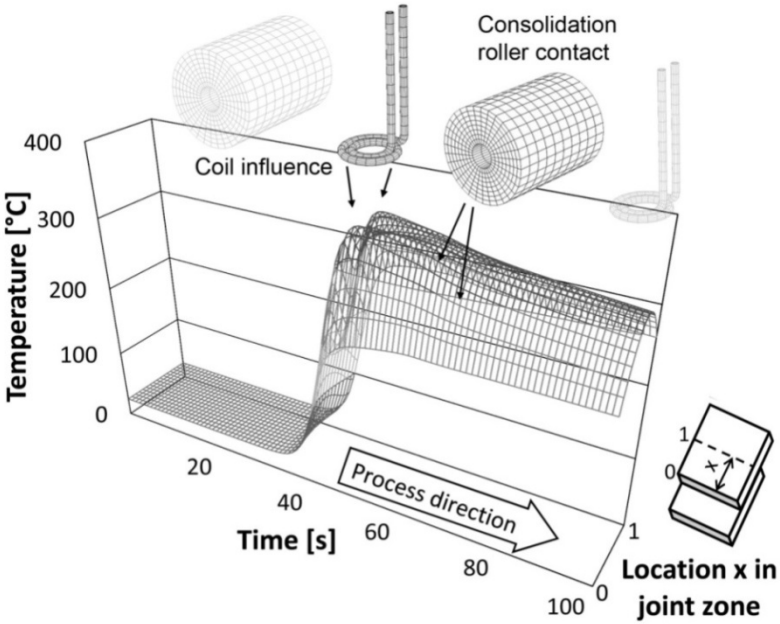


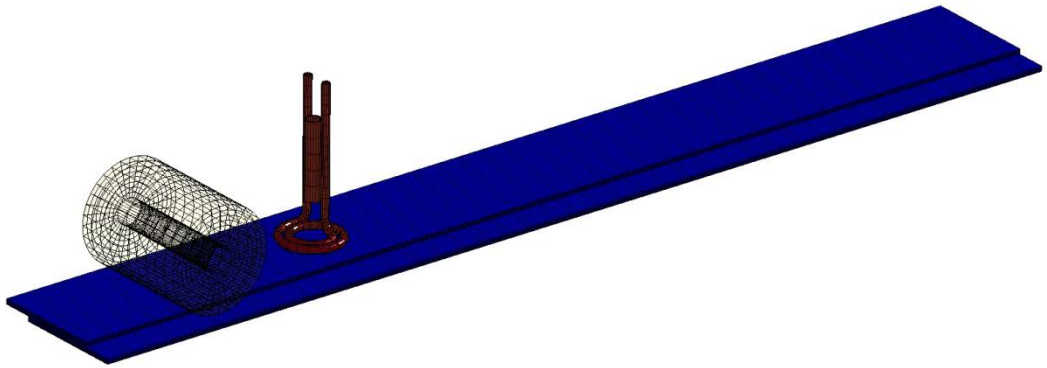

In collaboration with:
G. Mazars and G. Avrillaud:
Bmax, Toulouse, France

Bmax
shaping your future

# EM Solver Introduction

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



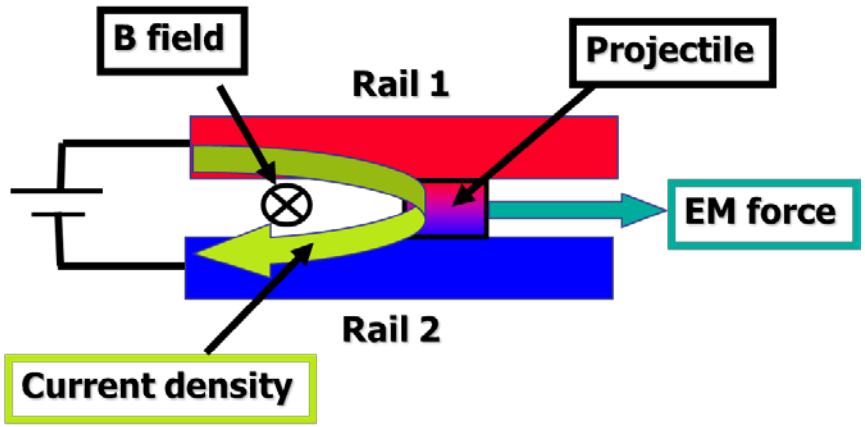


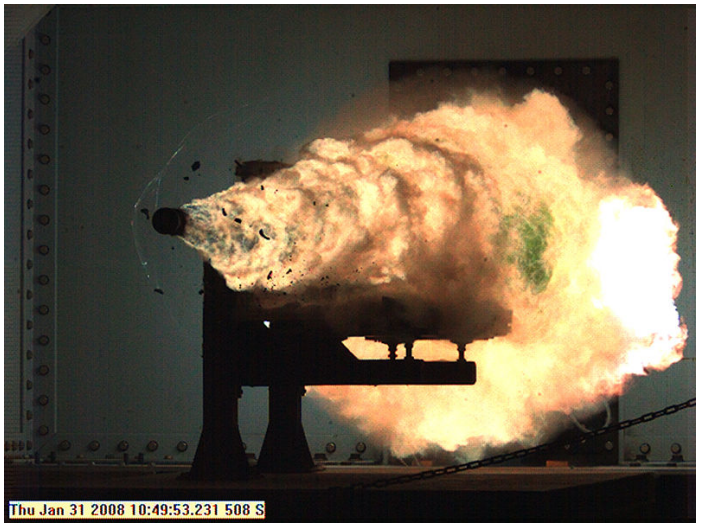
Institut für Verbundwerkstoffe

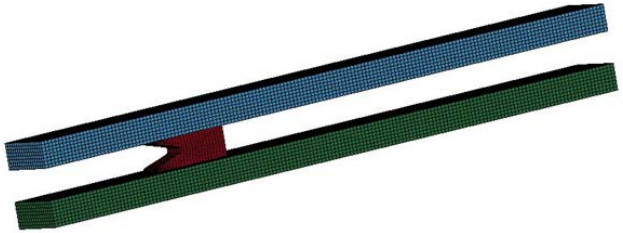
In collaboration with:
M. Duhovic, Institut für Verbundwerkstoffe, Kaiserslautern, Germany

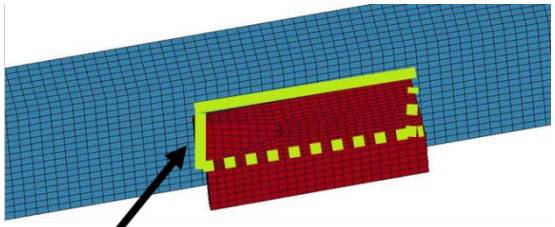
# EM Solver Introduction

## Rail gun simulation

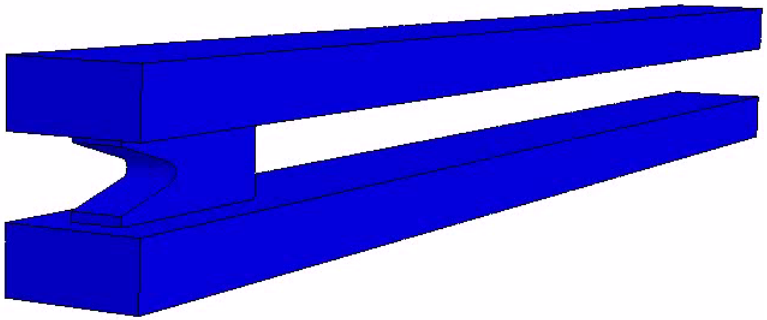


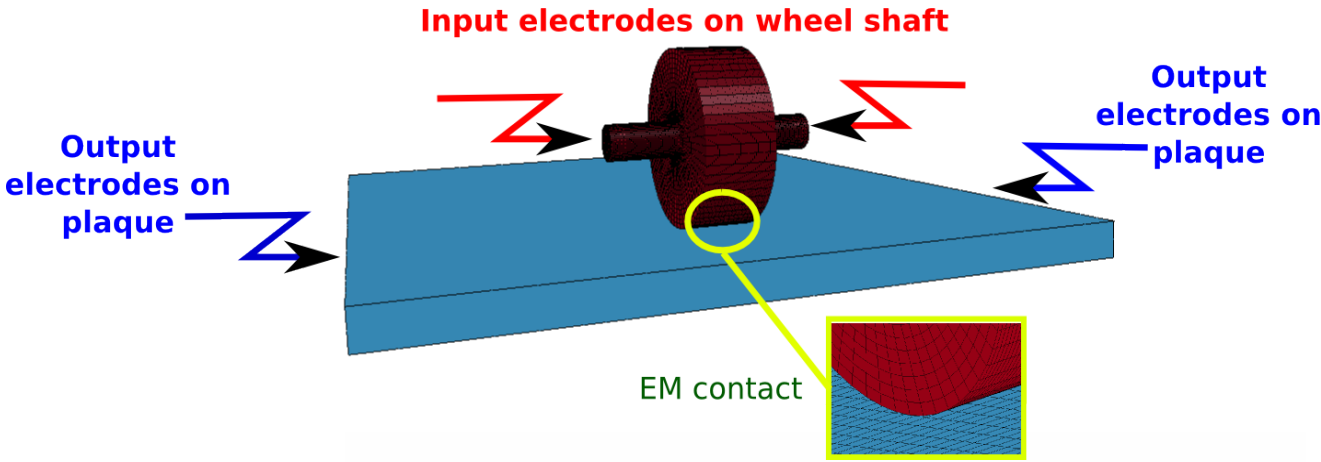


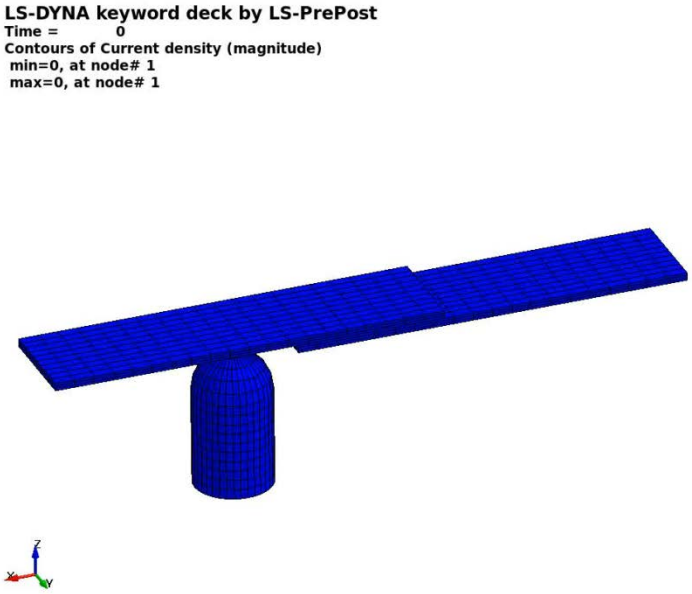
Sliding contact between the rails and the projectile

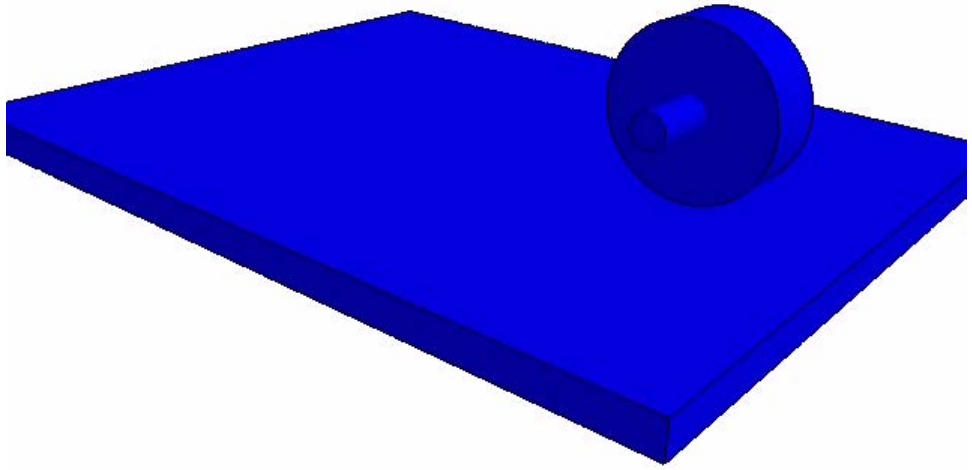


# EM Solver Introduction

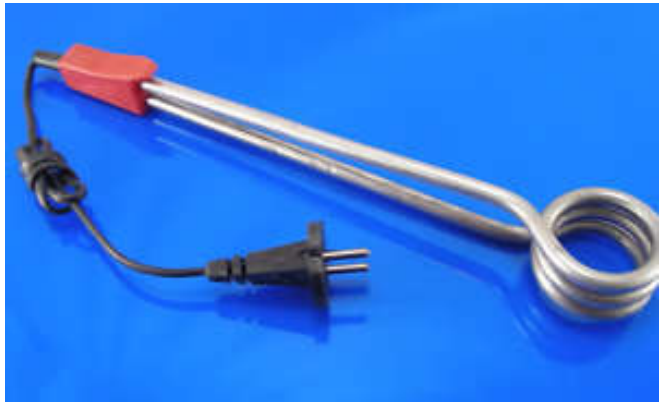
- **Joule heating study**
- **Study of shortcuts**



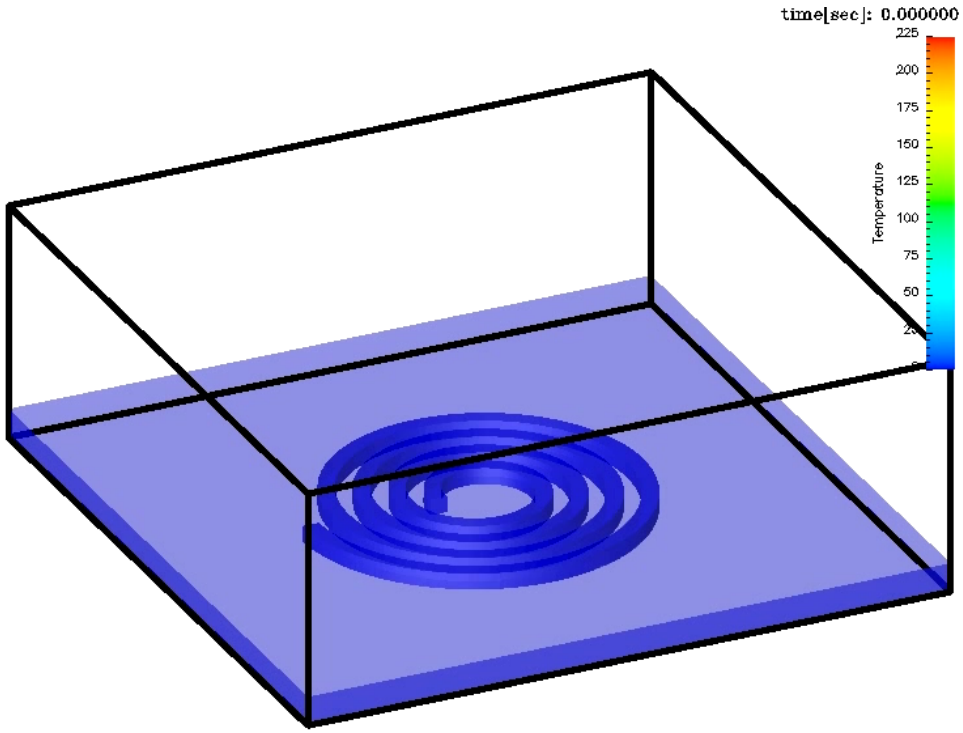



# EM Solver Introduction



- **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 Developments

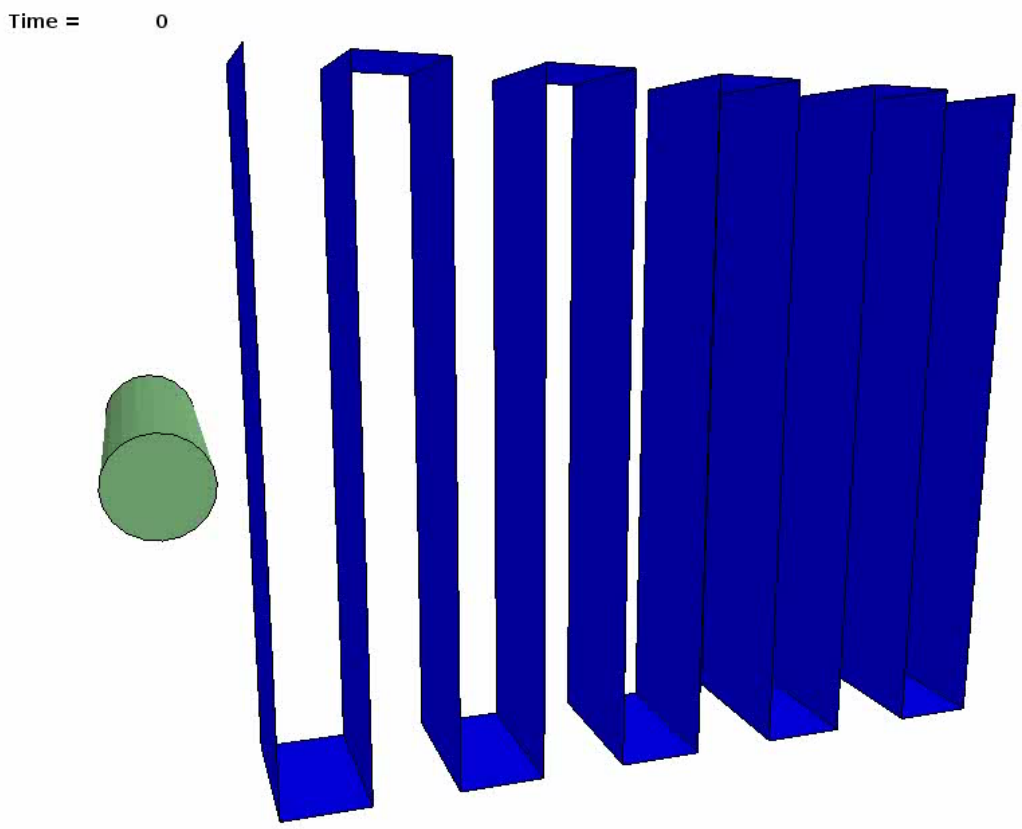
## Application: Conducting shells

**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 (the combination of inductive diffusive effects) is essentially a 3D phenomenon which means that elements with thickness are usually required for correct solution. Consequently, up to now, only solid conductive elements were allowed and 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 Developments



# EM Solver Developments

## 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:

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

With $\rho$ the material's resistivity and $a$ the radius of the equivalent contact circle area.

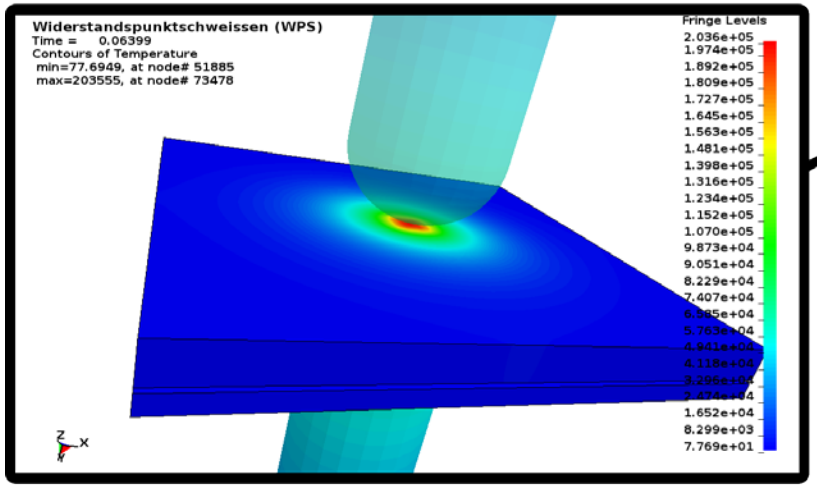
# EM Solver Developments

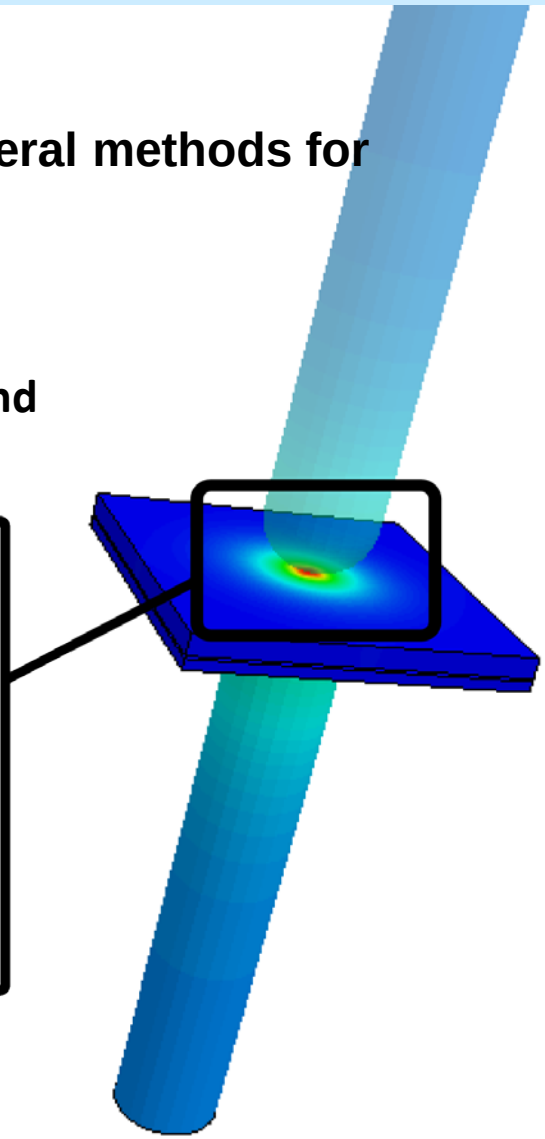
## Application: Resistive spot welding and others

**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 Developments

## 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 processing.**

- 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

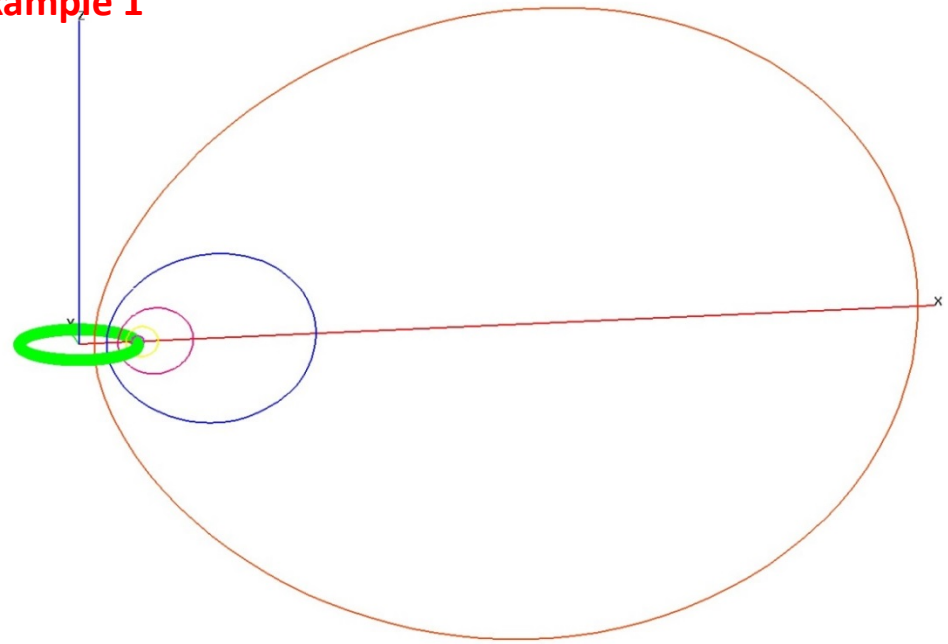
- This feature along with the multicenter/multipole methods are treated as a research project that could be used at a latter stage to compute and store efficiently the BEM matrix and thus speed up the computational time of the EM solver.

# EM Solver Developments

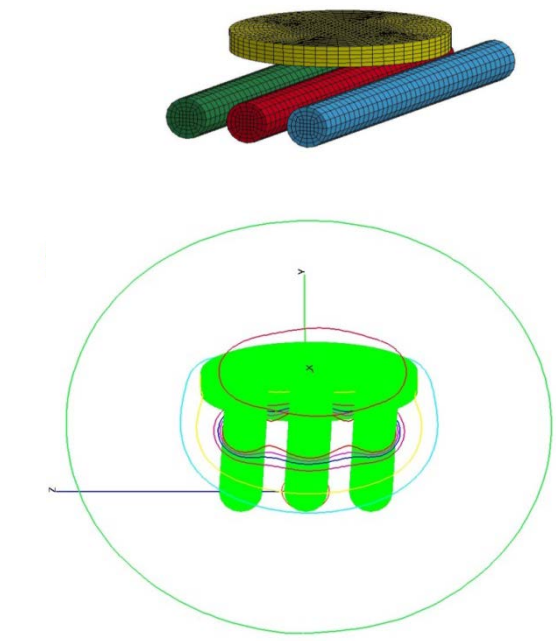
## Application: Magnetic field lines

**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 Developments

## Future Developments

- Symmetry conditions
- Piezoelectric materials
- Magnetic materials

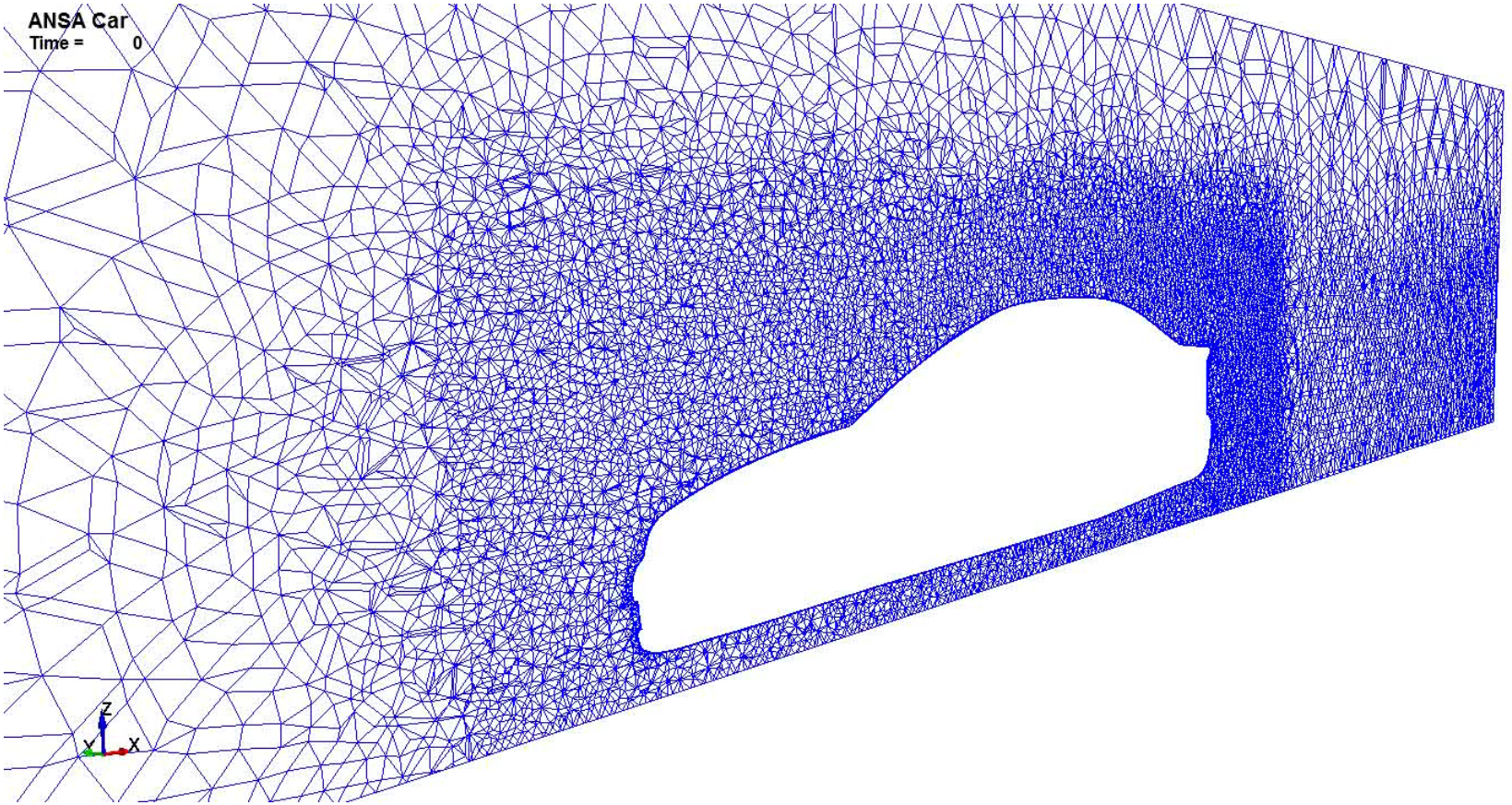
# Incompressible Fluid Dynamics

# 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.

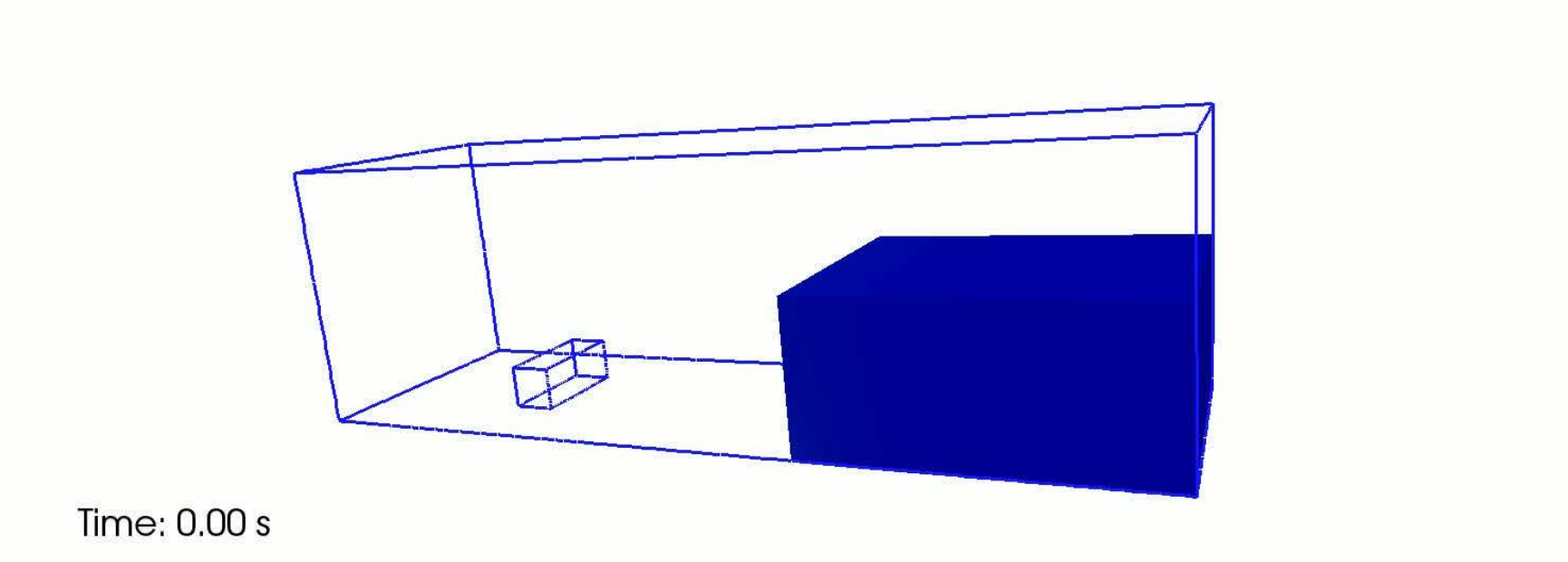
# ICFD Solver Introduction

**Aerodynamics: study of turbulent flow around a car**



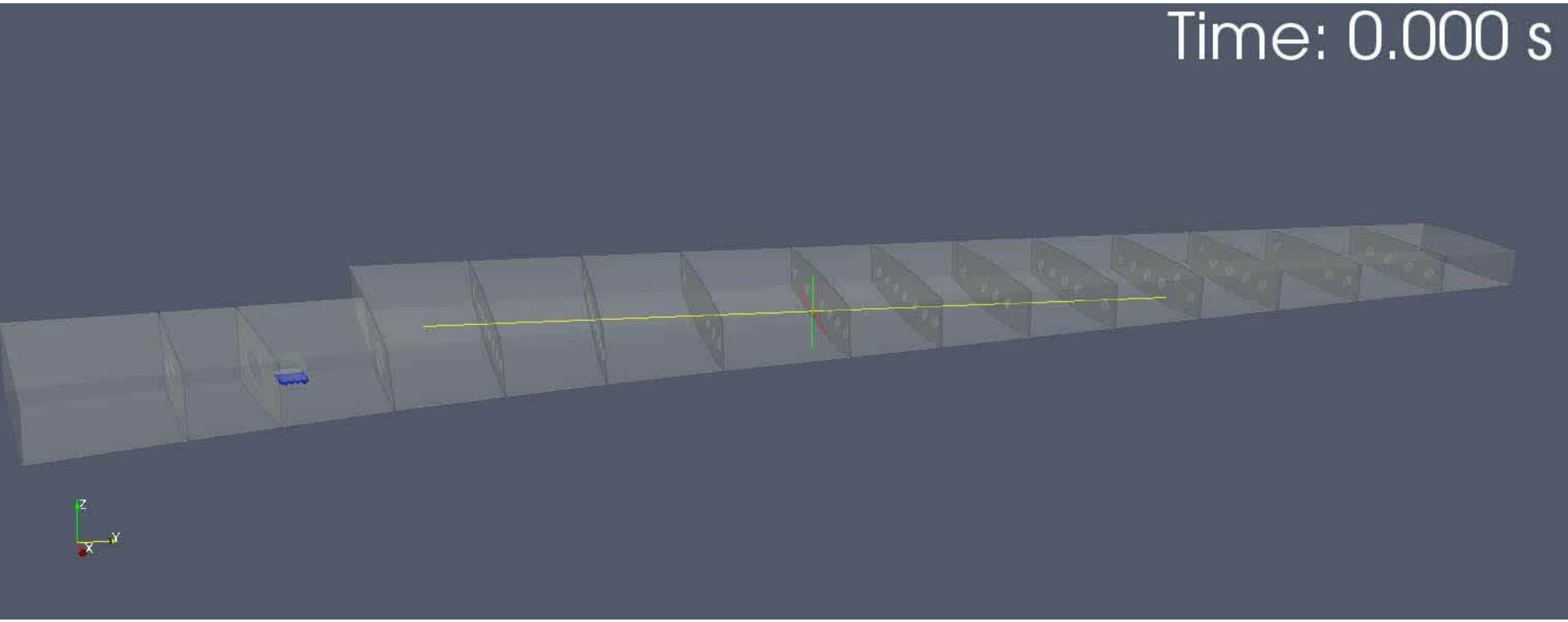
# ICFD Solver Introduction

**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

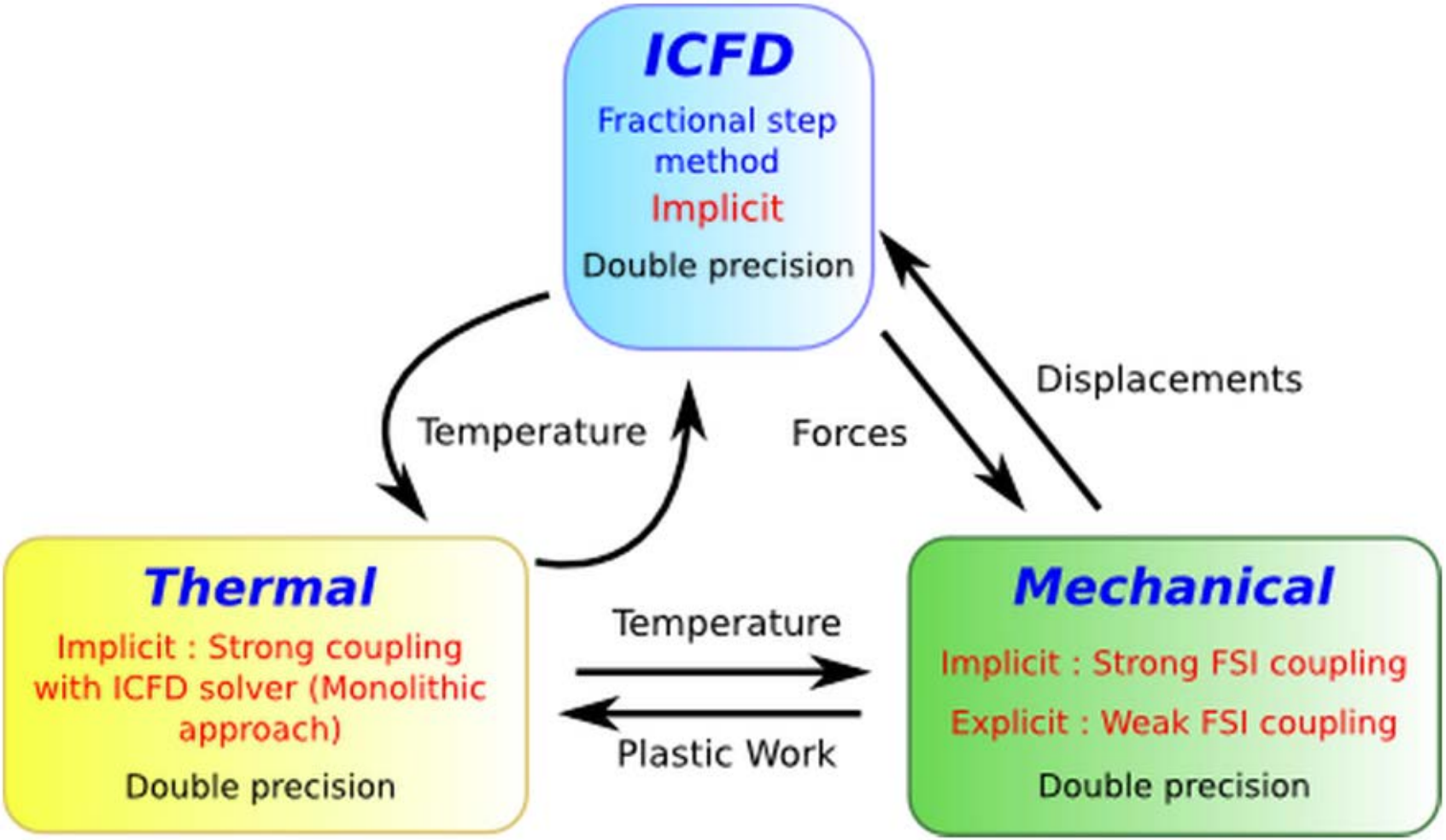
- A widely used approach in engineering is to assume that the FSI problem is linear and to use two different software products and licenses.
- The solid work group finds the need for FSI simulation.
- The geometry is sent to the fluid group which builds a mesh and runs the fluid problem with a CFD solver 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” approach.

# ICFD Solver Introduction



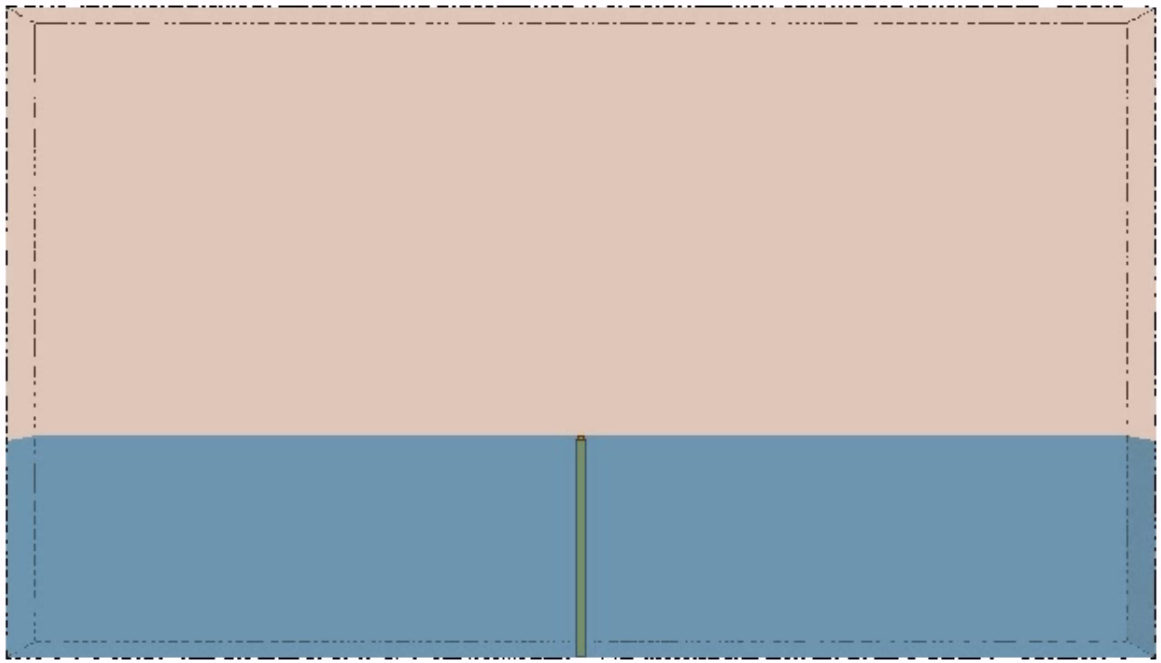
# ICFD Solver Introduction

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

# ICFD Solver Introduction

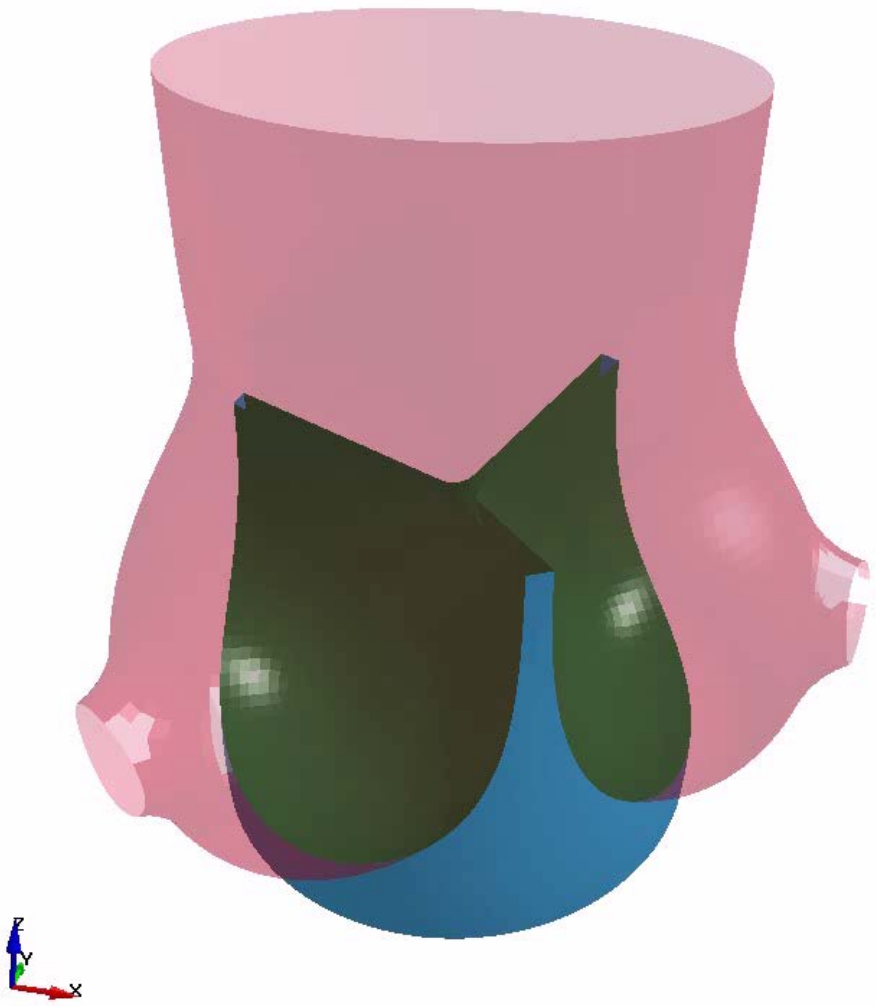
## Sloshing: Flap oscillating in partially filled tank

Time = 0, #nodes=2146228, #elem2d=274382, #elem3d=6403785



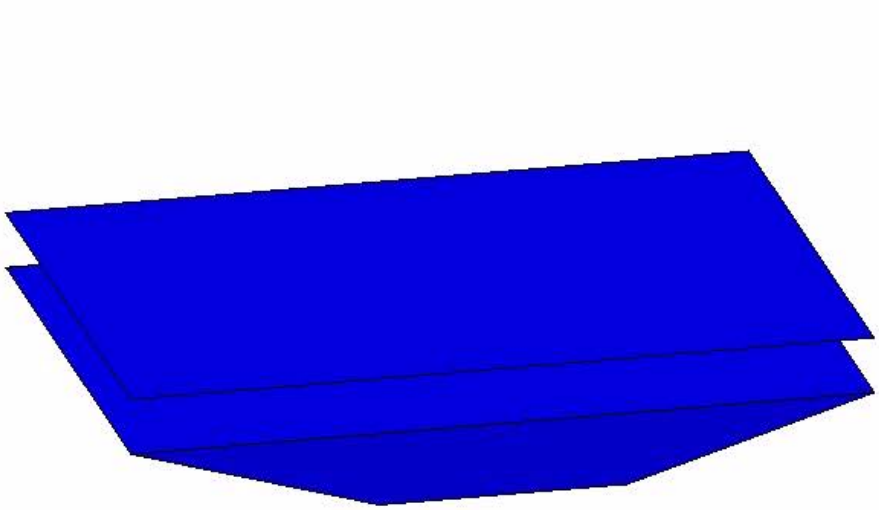
# ICFD Solver Introduction

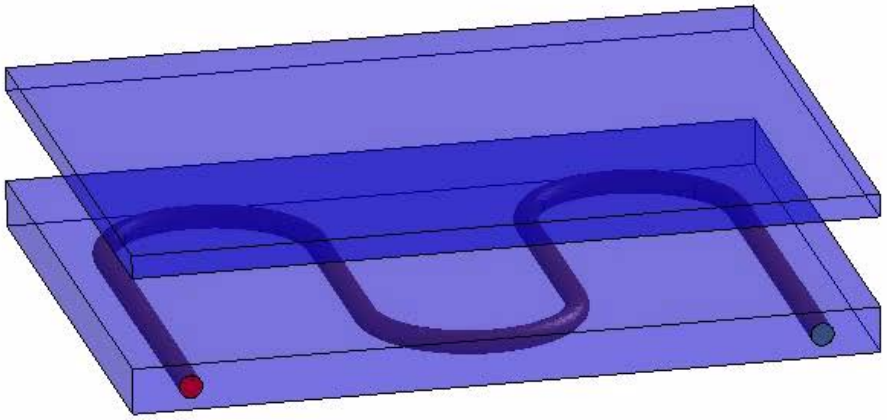
**Artificial Heart valve: Strong pressure gradients force leaflets open**



# ICFD Solver Introduction

**Stamping and conjugate heat transfer:**
**flow in serpentine pipe causes die to cool**





# ICFD Solver Developments

## Application: Multiphase problems

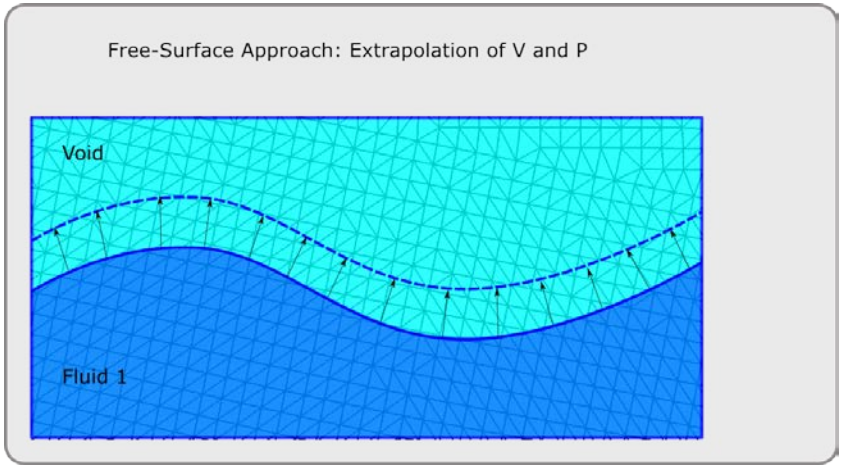
**Development : Being able to solve problems with two fluids of different densities (water+air).**

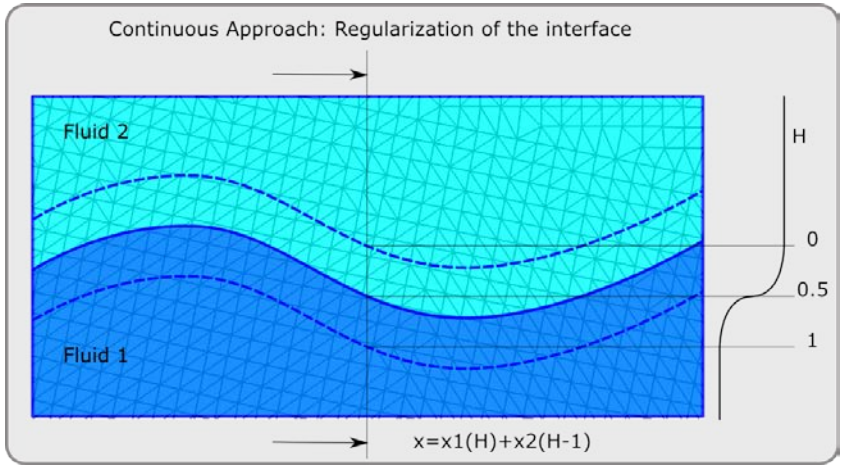
**Current Status : Implementation stage.**

- 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.

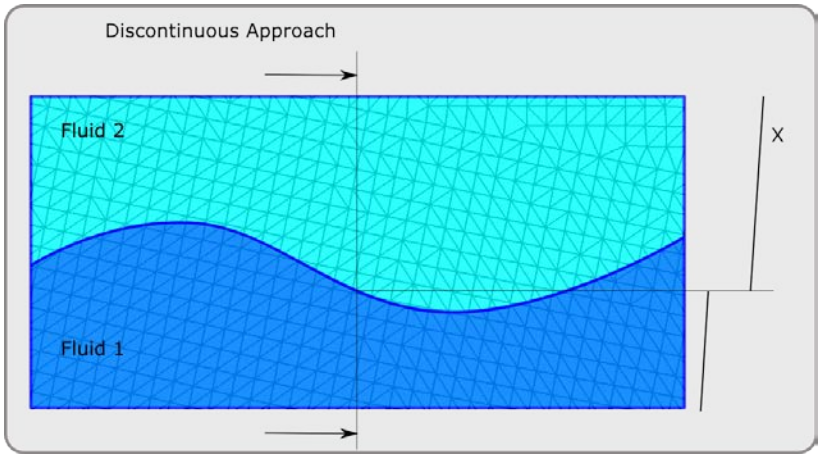
# ICFD Solver Developments

## 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 Developments

## Application: Porous Materials

**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 :

$$\varepsilon = \frac{void\ volume}{total\ volume}$$

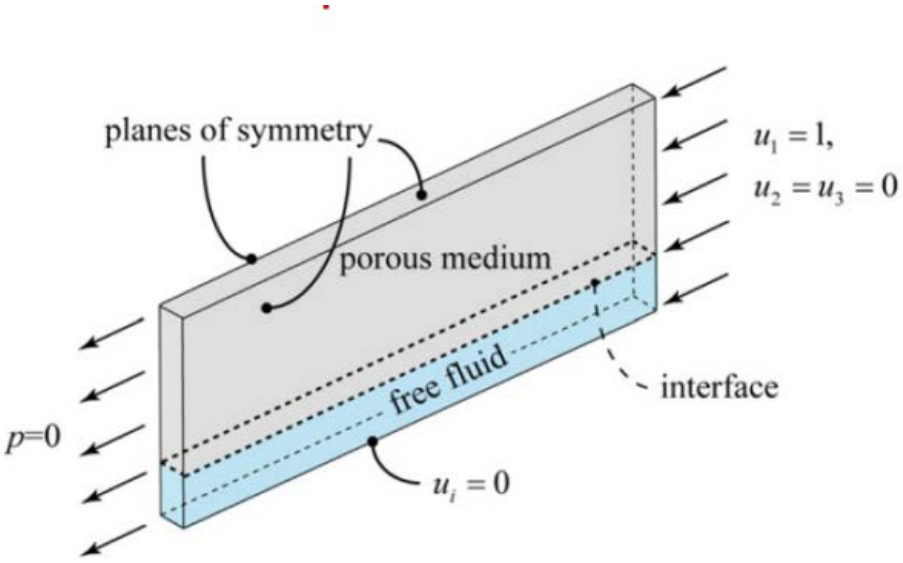
$$\frac{\rho}{\varepsilon}\left[\frac{\partial u_i}{\partial t} + \frac{\partial\left(\frac{u_i u_j}{\varepsilon}\right)}{\partial x_j}\right] = -\frac{1}{\varepsilon}\frac{\partial(P\,\varepsilon)}{\partial x_i} + \frac{\mu}{\varepsilon}\frac{\partial^2 u_i}{\partial {x_i}^2} + \rho g_i - D_i$$

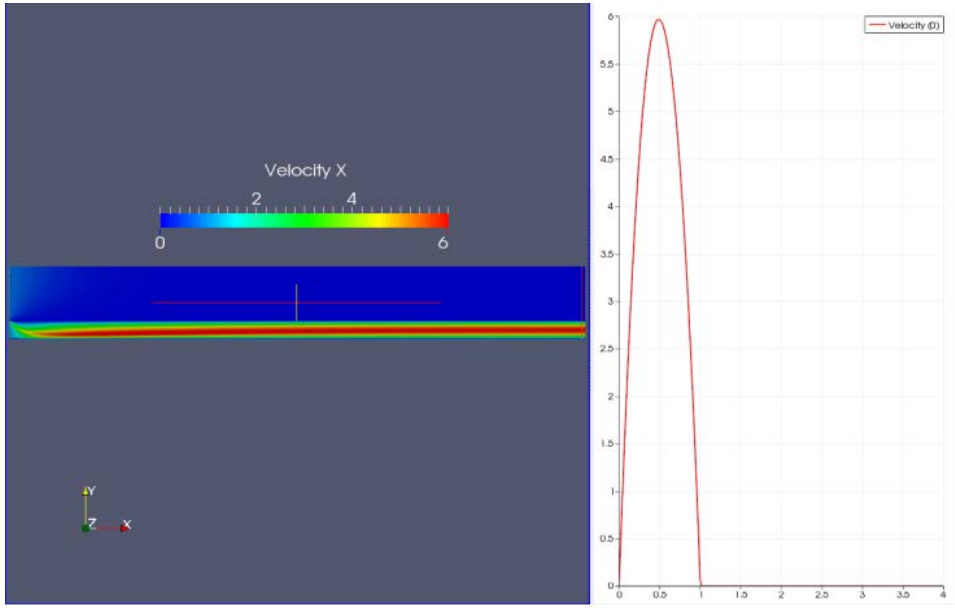
**Ergun correlation :** $$D_i = -\frac{\mu U_i}{\mathrm{K}} + \frac{1{,}75\rho|u|}{\sqrt{150}\sqrt{\kappa}\varepsilon^{1.5}}U_i$$

# ICFD Solver Developments

## Application: Porous Materials

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



# ICFD Solver Developments

## 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.**

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

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 Developments

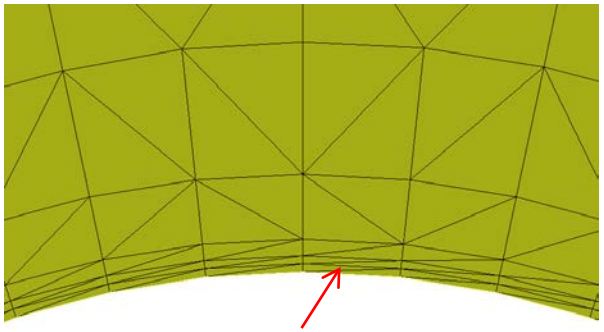
## Application: External and internal aerodynamics with turbulence

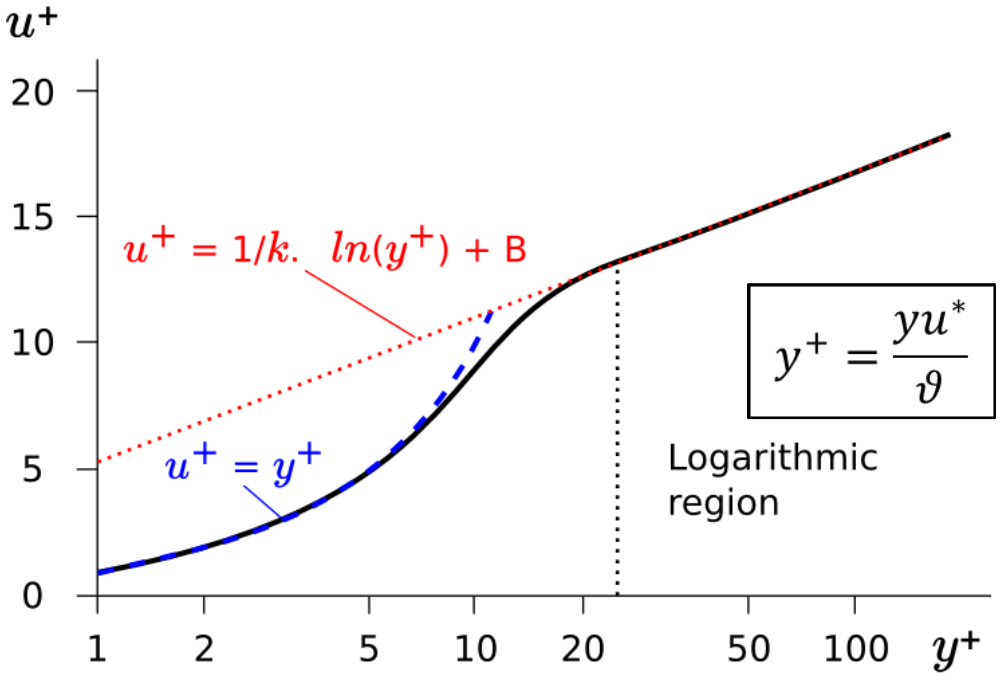
**Development: Adding more HRN and LRN laws of the wall for the turbulence models. Providing more tools for the boundary layer mesh generation.**

**Current status : Implementation stage**

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

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



# ICFD Solver Developments

## 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.

# Rotational Dynamics

# Rotational Dynamics for Implicit

**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 in machines with rapidly rotating parts.

The **resonant vibration** can lead to premature fatigue failure in those rotating components, as well as bearings and support structures.

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

# Implementation in LS-DYNA

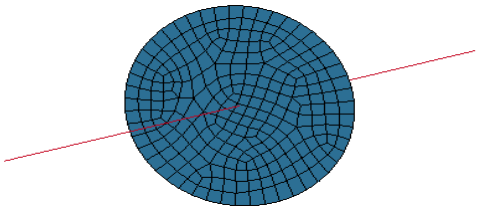
## *CONTROL_IMPLICIT_ROTATIONAL_DYNAMICS

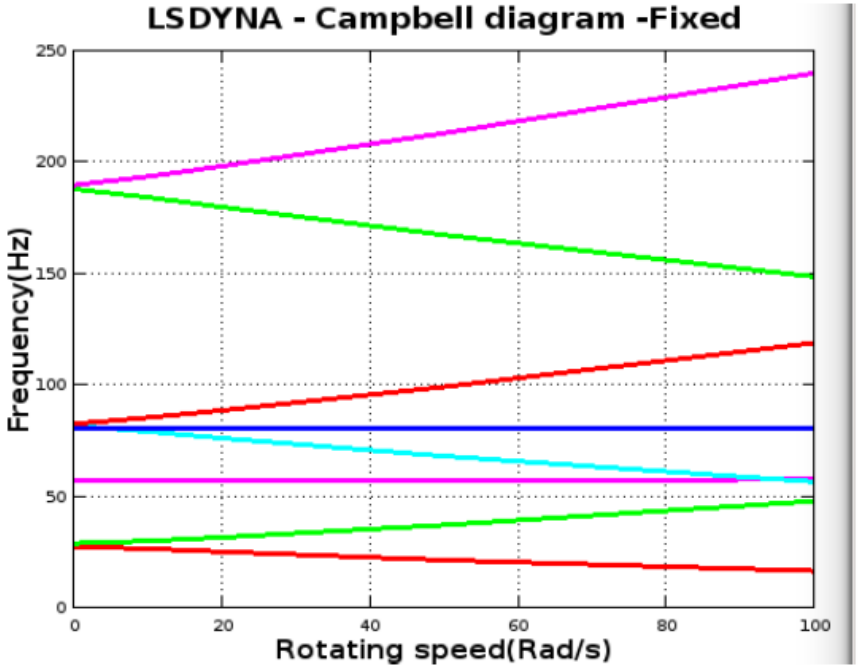
| | | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| Variable | sid | stype | omega | vid | nomega | iref | | |
| Type | I | I | F | I | I | I | | |
| Variable | omeg1 | omeg2 | omeg3 | omeg4 | omeg5 | omeg6 | omeg7 | omeg8 |
| Type | F | F | F | F | F | F | F | F |

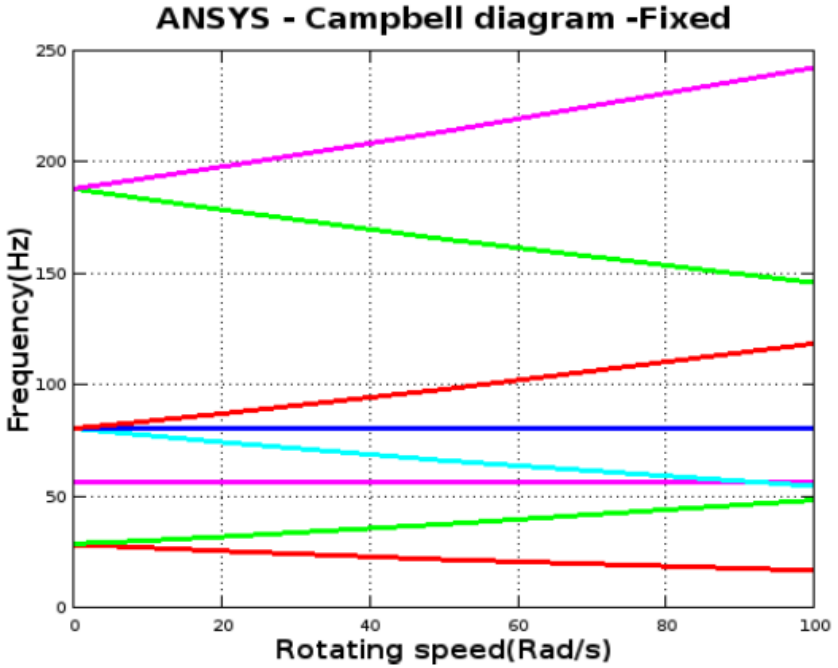
**sid**: Set ID of the rotational structure.
**stype**: Set type, 0: part;
1: part set.
**omega**: Rotating speed.
**vid**: Vector ID to define the rotating axis.
It is defined in *DEFINE_VECTOR.
**nomega**: No. of rotating speeds for Campbell diagram plotting (an example will be given on the next slide).
**iref**: Reference frame. 0: rotating coordinate system;
1: fixed coordinate system.
**omegi**: The ith rotating speed.
You can define as many cards as the number of rotating sets.

# Modal Analysis Example

**Campbell Diagram:** represents a system's eigen-frequencies as a function of rotating speeds.
**Study case** (on the right): a shaft and a disk, rotating along the shaft axis.
**The following two plots:** give the Campbell diagrams obtained from LS-DYNA and ANSYS.



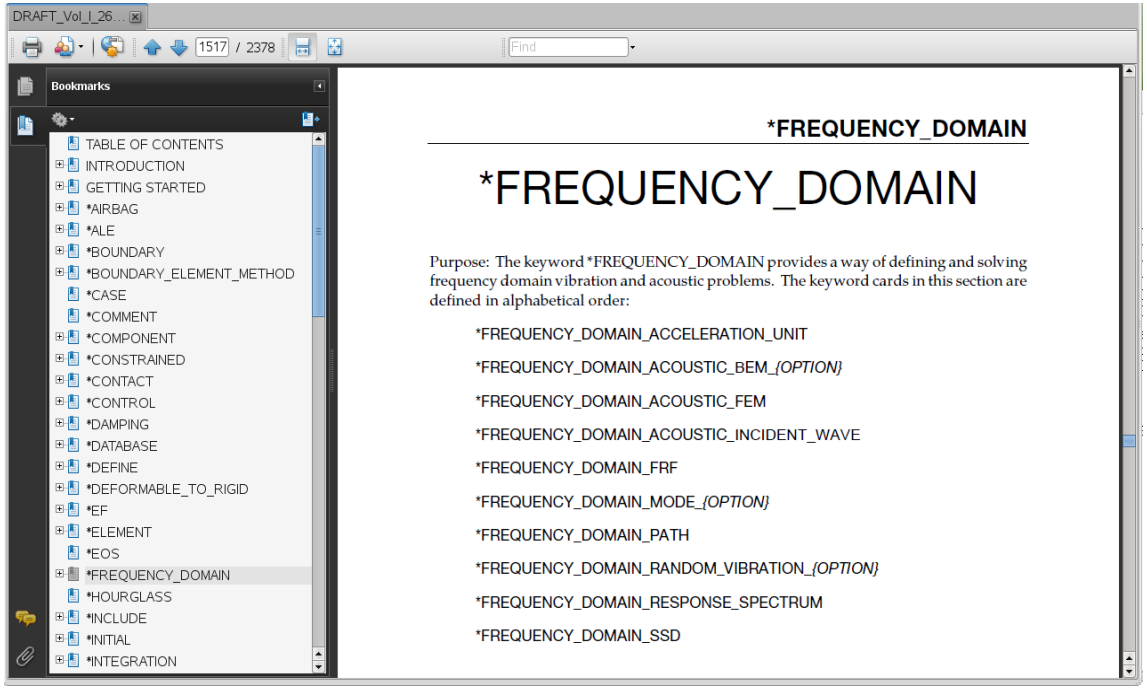


# Frequency Domain

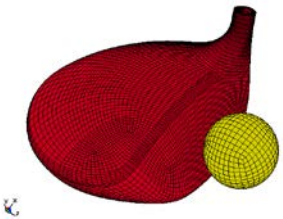
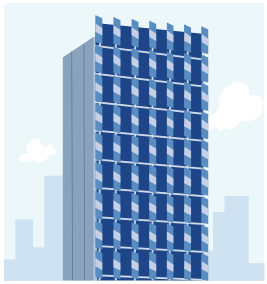
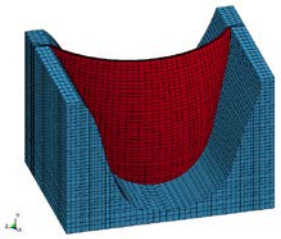
# Frequency domain features

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



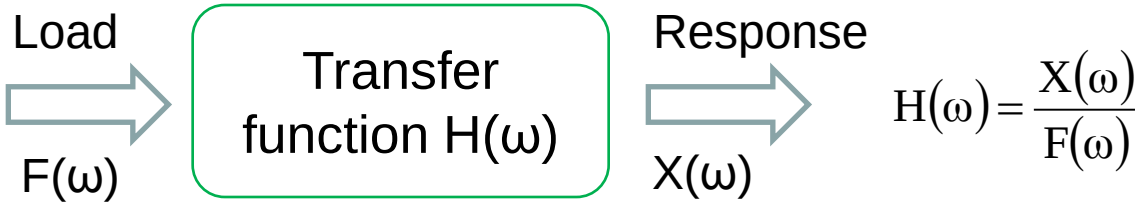
# Application

- NVH of automobiles and aircraft
- Acoustic design and analysis
- Defense industry
- Fatigue of machines and engines
- Civil and hydraulic engineering
- Earthquake engineering

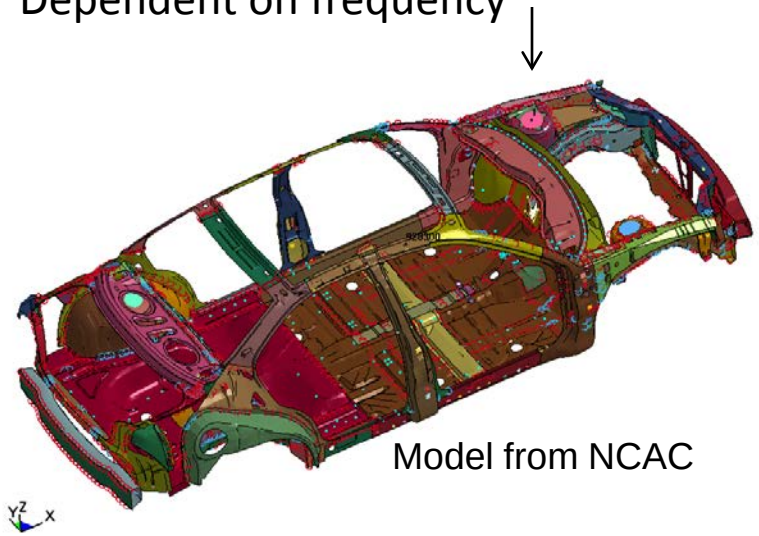




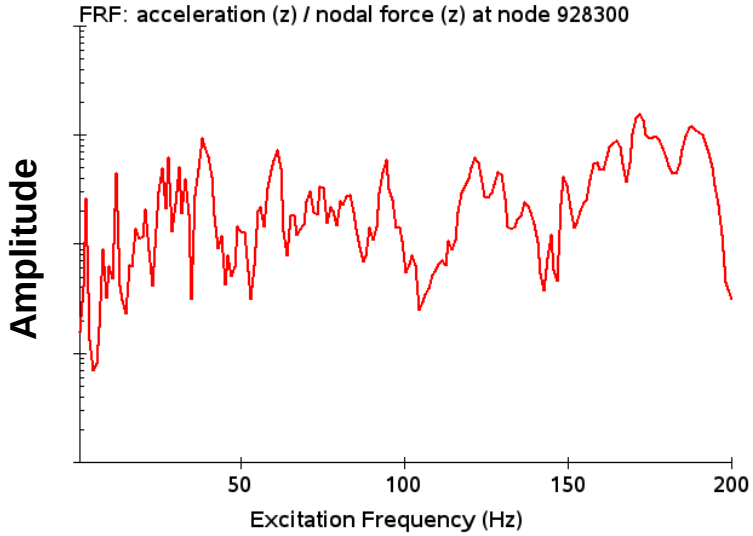
# FRF

## What is FRF?




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



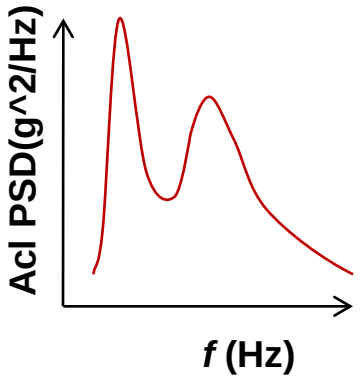
# Random vibration

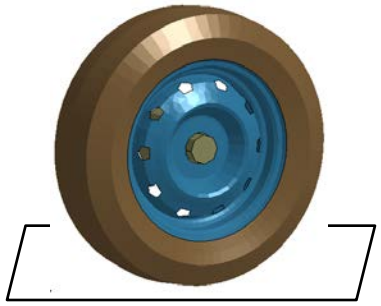
## Why do we need random vibration analysis?

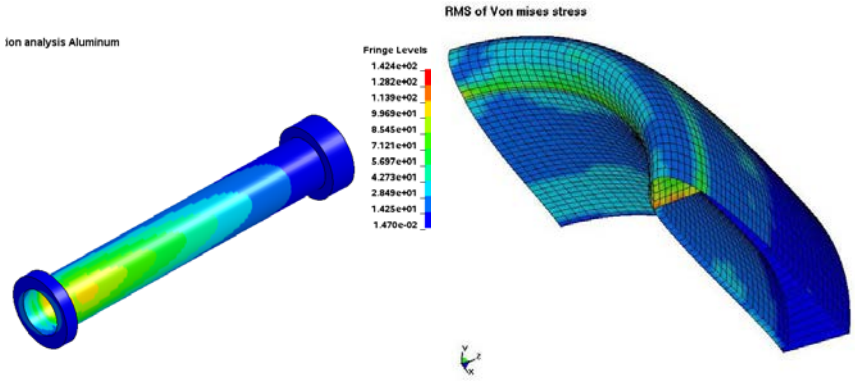
- The loading on a structure is not known in a definite sense
- Many vibration environments are not related to a specific driving frequency
- Examples:
  - Wind-turbine
  - Air flow over a wing or past a car body
  - Acoustic input from jet engine exhaust
  - Earthquake ground motion
  - Wheels running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

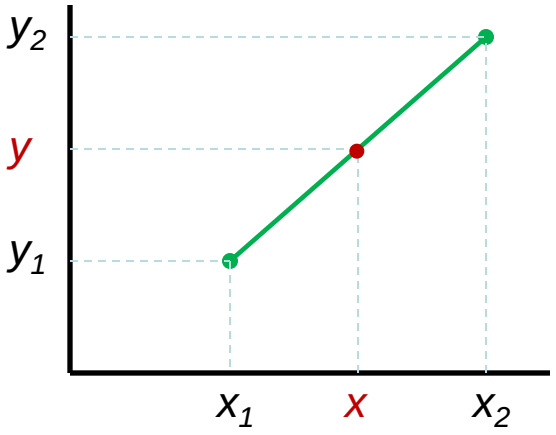


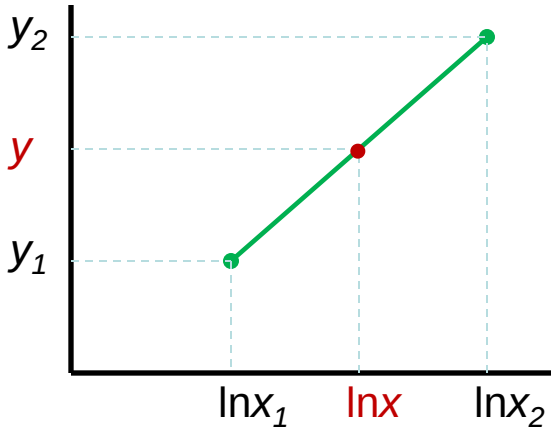


# New interpolation options on PSD curve

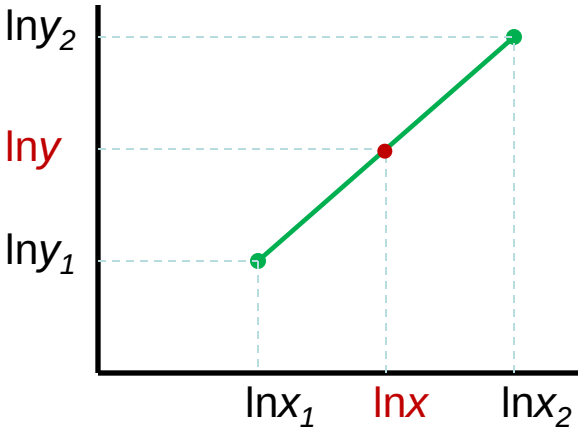
**Three types of interpolation for PSD curves**



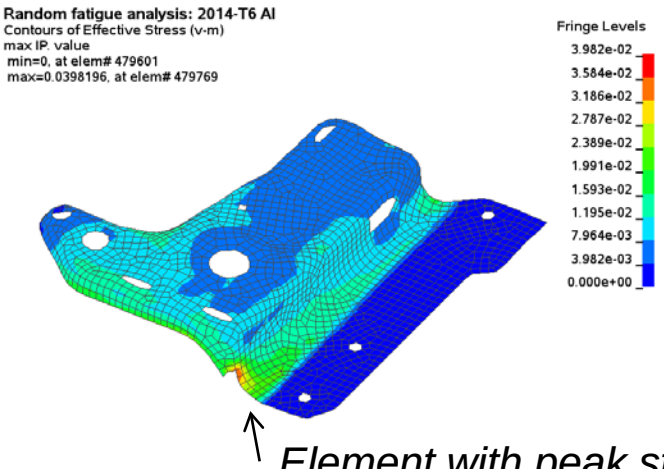

**linear-linear**



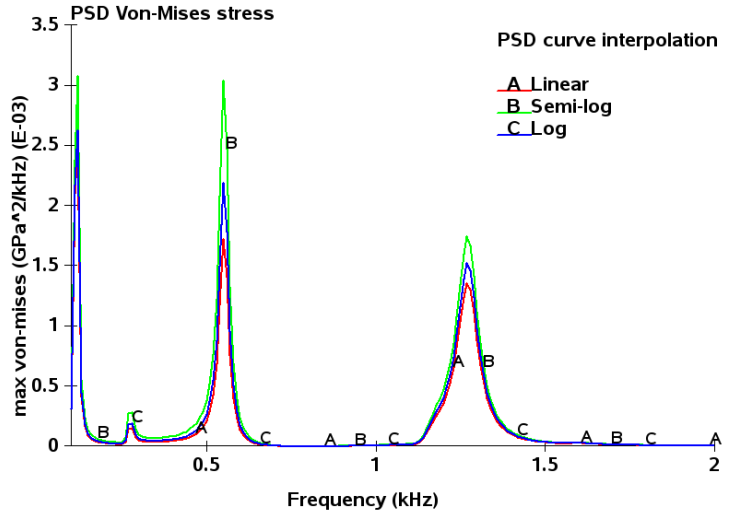

**linear-log** (semi-log)




**log-log**



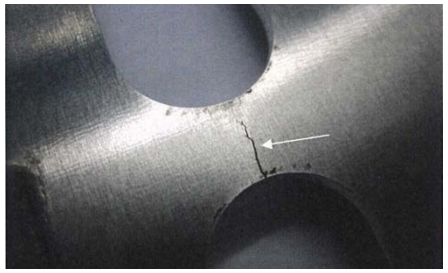

*Element with peak stress*



# 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



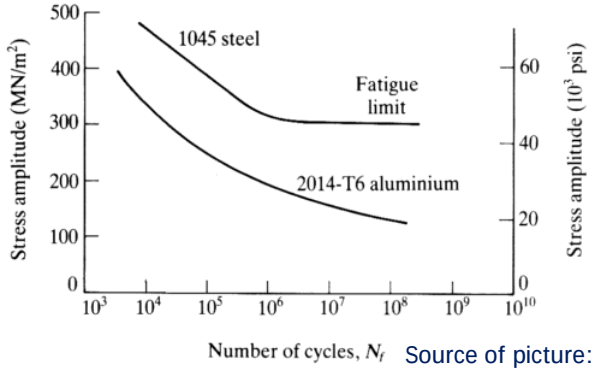
$$R = \sum_i \frac{n_i}{N_i}$$

## *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 | |

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

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




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

# SSD

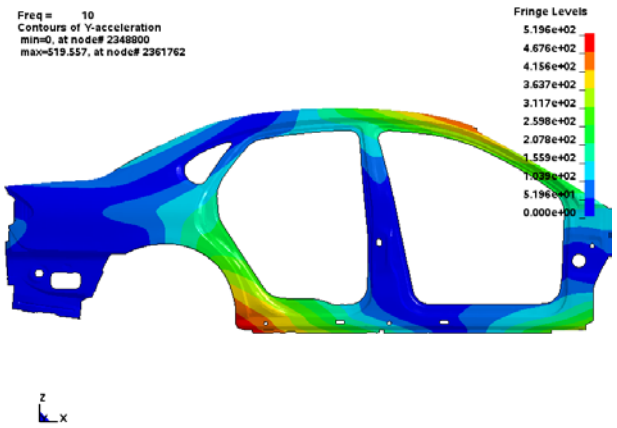
## Why do we need SSD (Steady State Dynamics)?

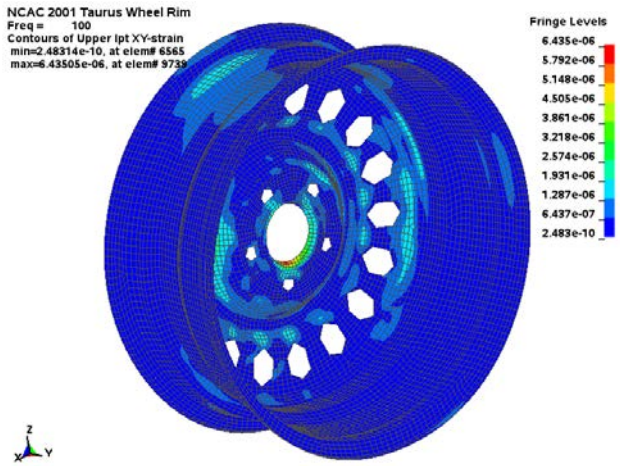
- Harmonic excitation is often encountered in engineering systems. E.g. it is commonly produced by the unbalance in rotating machinery.
- The load may also come from periodic load, e.g. in fatigue test.
- The excitation may also come from uneven base, e.g. the force on tires running on a zig-zag road

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

***Input***: frequency load spectrum (complex variable)

***Output***: D3SSD, NODOUT_SSD, ELOUT_SSD



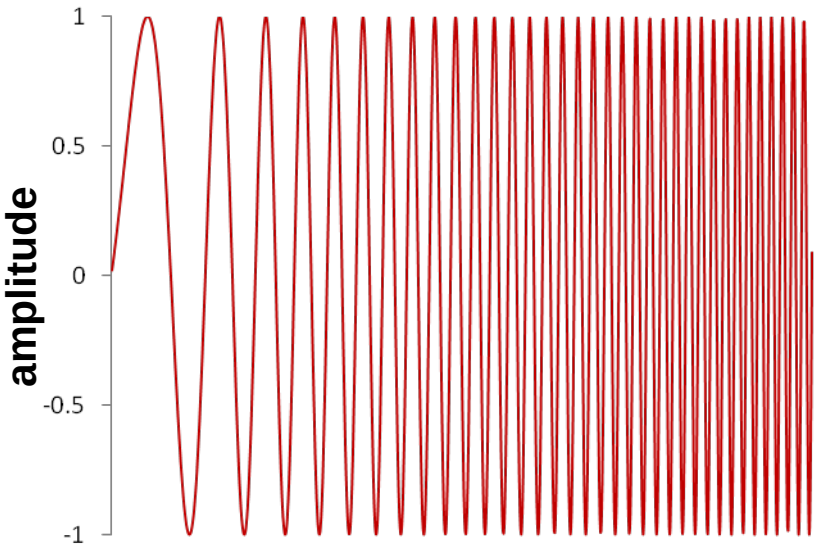



# SSD Fatigue (ongoing development)

***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
- Rainflow counting algorithm for each frequency for one period

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




Sine sweep

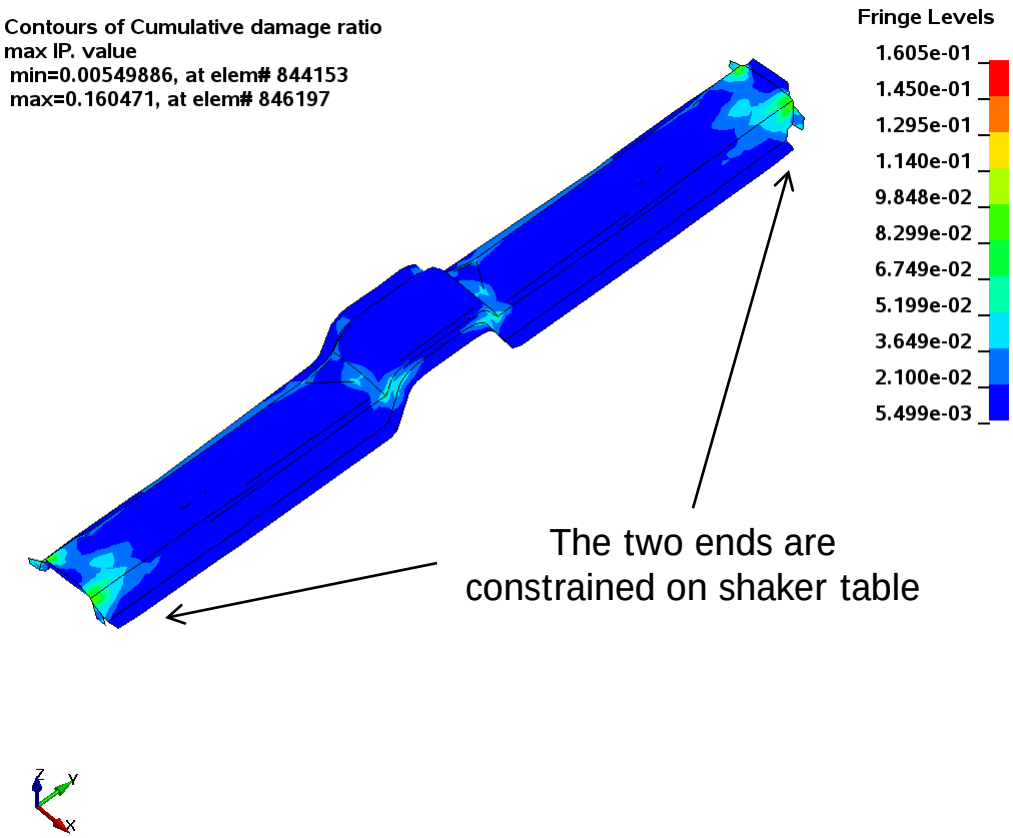
# Example of SSD Fatigue

## 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 |

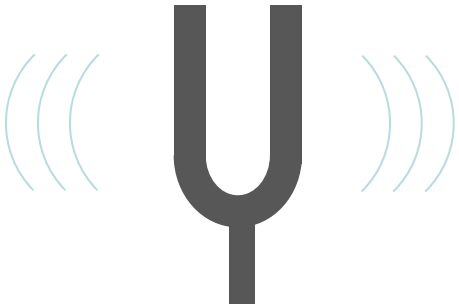
## S-N fatigue curve

| σ (MPa) | N |
|---|---|
| 100 | $8\times10^{4}$ |
| 10 | $8\times10^{5}$ |
| 1. | $8\times10^{6}$ |
| 0.1 | $8\times10^{7}$ |
| 0.01 | $8\times10^{8}$ |



# BEM Acoustics

## vibro-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*

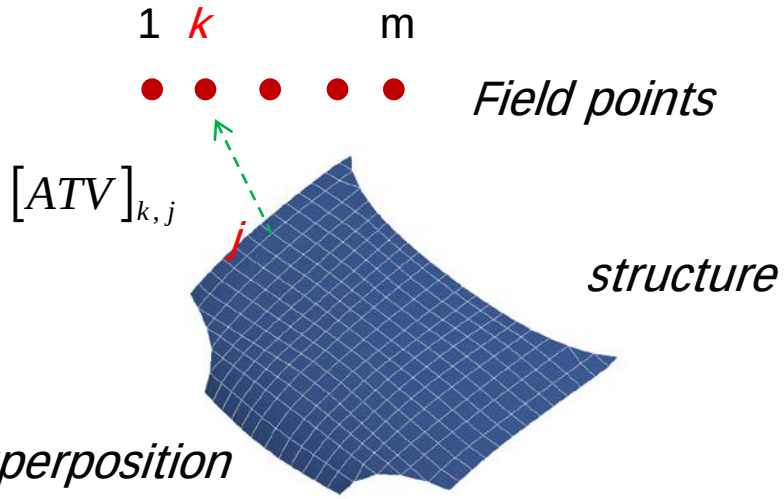
Acoustic pressure and SPL (dB) at field points

# 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.

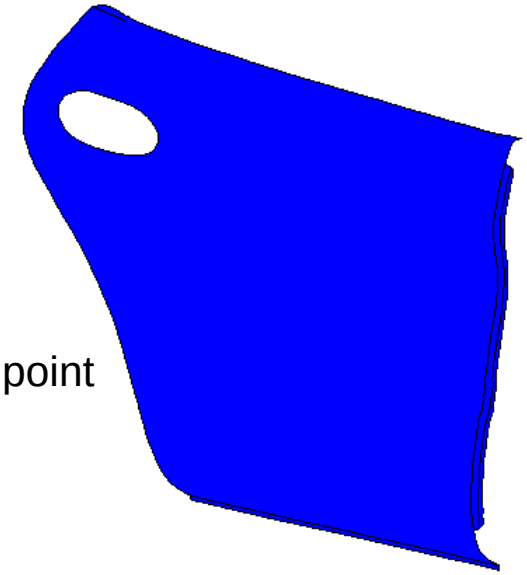
$$\{P\} = [ATV]\{V\}$$
$$= [ATV](-i\omega[\Phi]\{q\})$$
$$= -i\omega[ATV][\Phi]\{q\}$$
$$= [MATV]\{q\}$$




*Velocity vector is obtained by modal superposition*

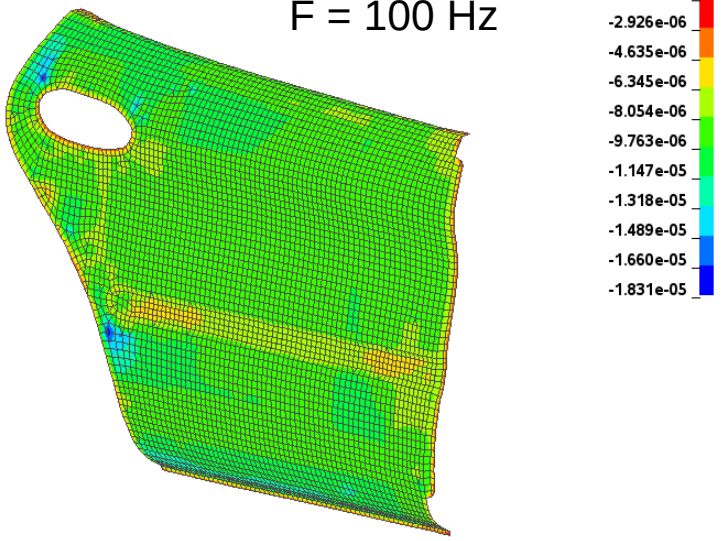
# Example: ATV of car door model

Measure point

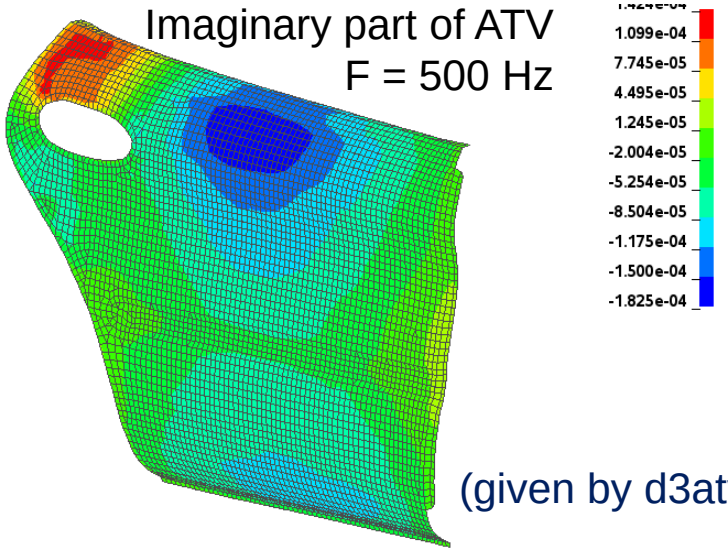


No. of elements: 5198
No. of frequencies: 101

Real part of ATV
F = 100 Hz

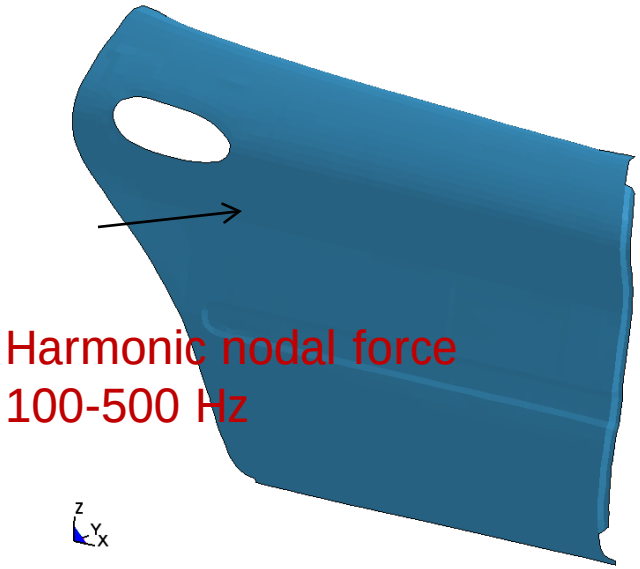


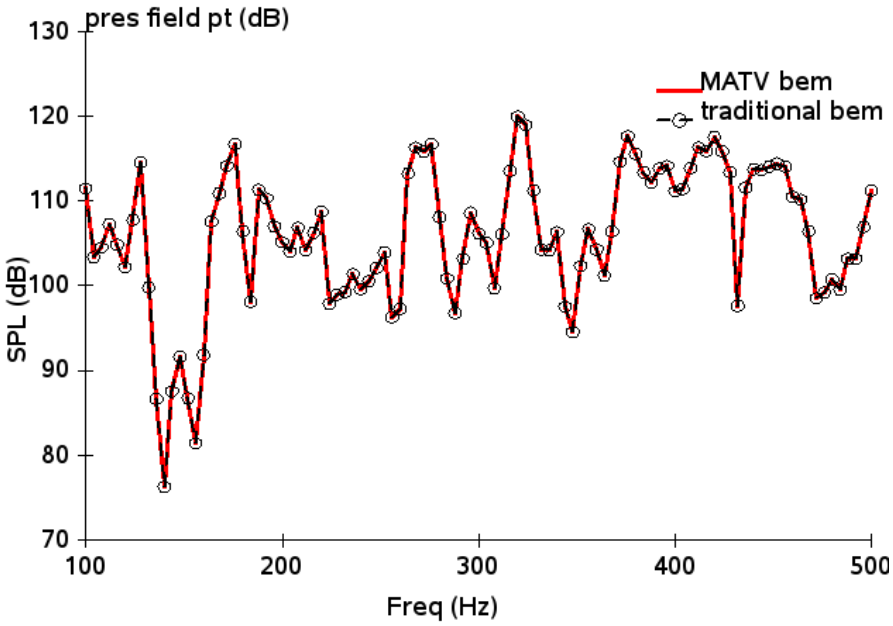

Imaginary part of ATV
F = 500 Hz




(given by d3atv)

# MATV BEM is efficient



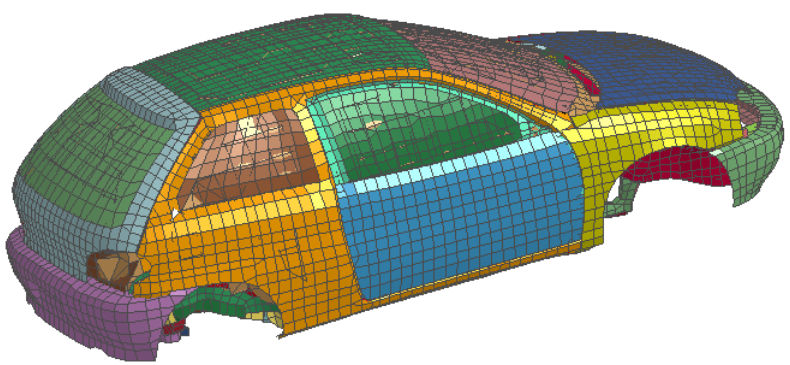



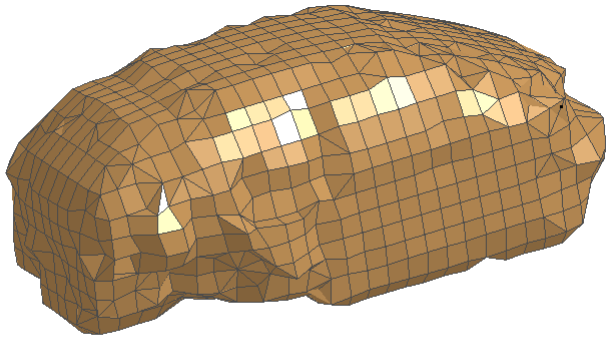

**CPU time (1 core)** Intel Xeron CPU E5504 @2.00 GHz

| 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** |

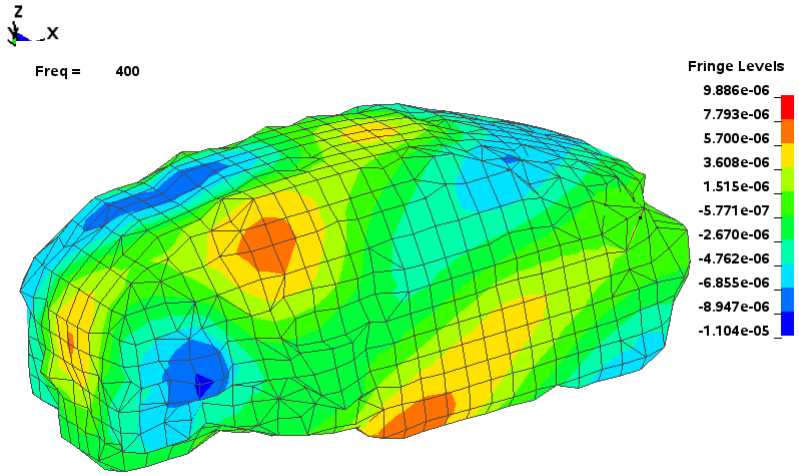
# D3ACS for BEM acoustics

***DATABASE_FREQUENCY_BINARY_D3ACS**

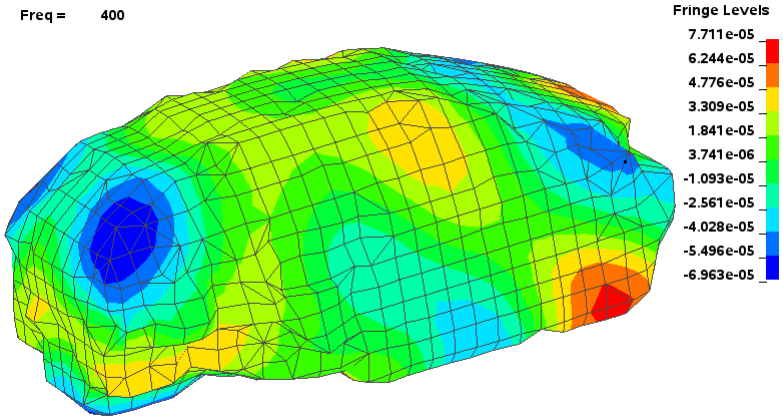




**Acoustic volume of compartment**



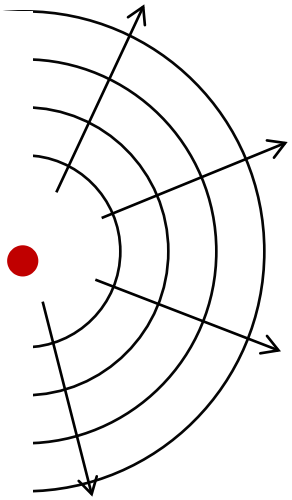

**Real part of surface pressure**




**Imaginary part of surface pressure**

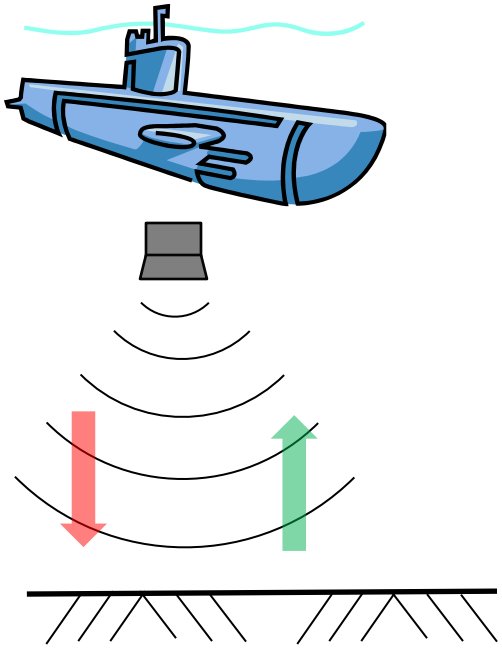
# Incident waves for acoustic analysis

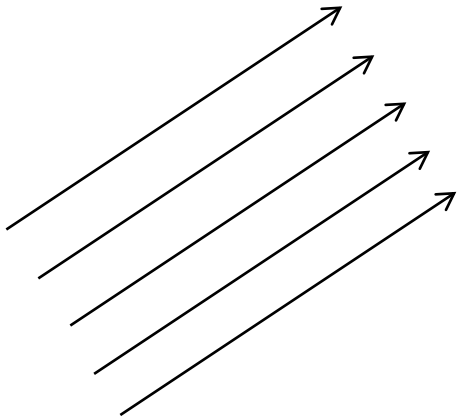
***FREQUENCY_DOMAIN_ACOUSTIC_INCIDENT_WAVE**



## Spherical wave

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

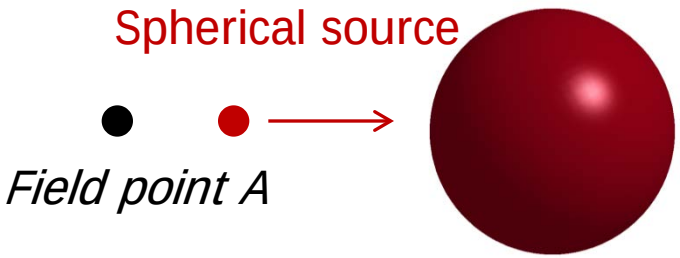


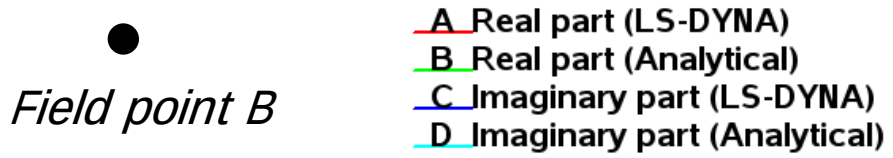


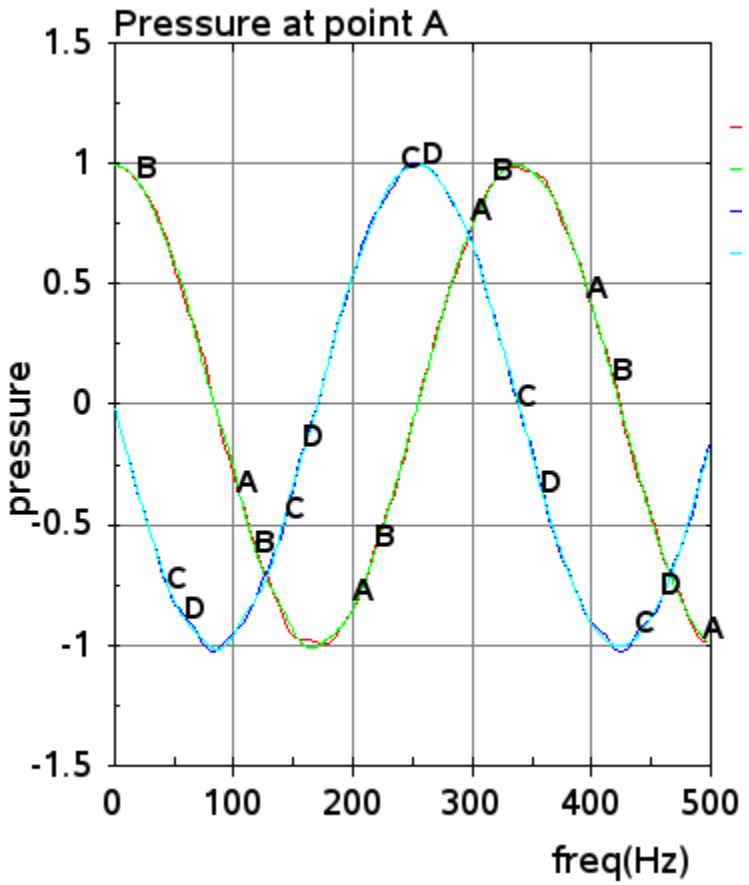
## Plane wave

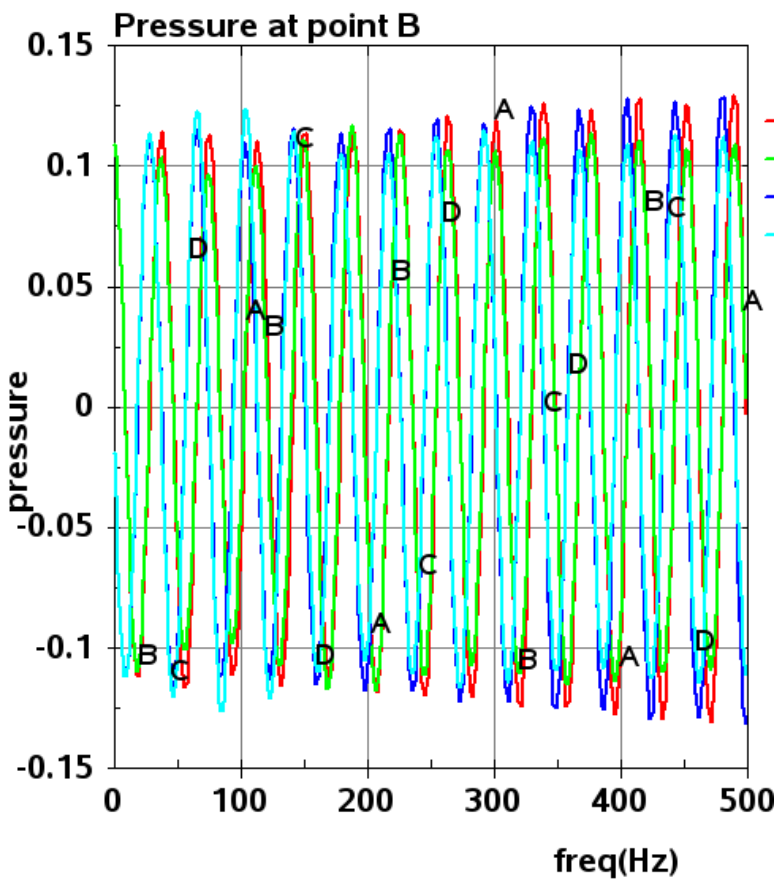
$$p^i = Ae^{-ik(\alpha x+\beta y+\gamma z)}$$

# Example: sound scattering on rigid sphere



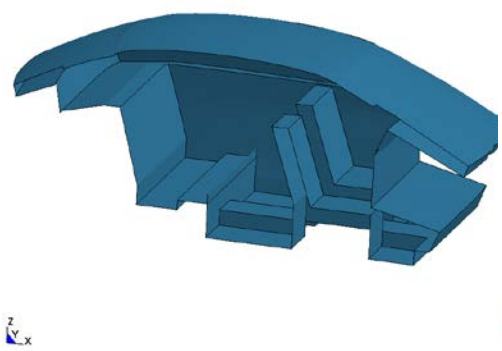



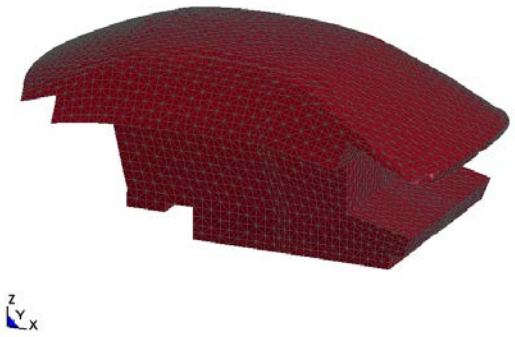
# FEM Acoustics

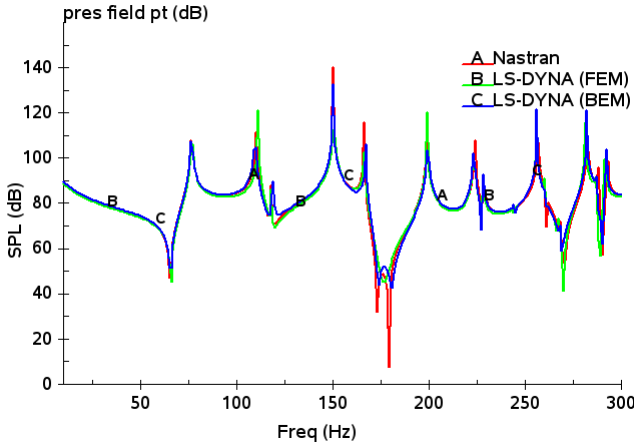
## *FREQUENCY_DOMAIN_ACOUSTIC_FEM

- 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





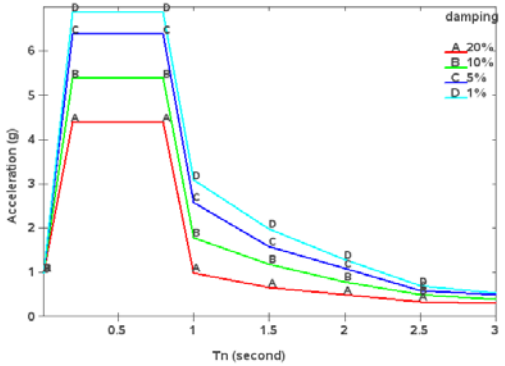


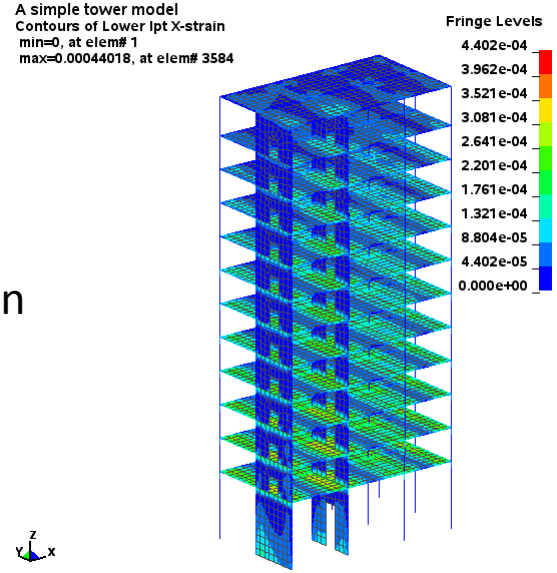
# Response spectrum analysis

## *FREQUENCY_DOMAIN_RESPONSE_SPECTRUM

- Various mode combination methods
  - o SRSS method
  - o NRC Grouping method
  - o CQC method
  - o Double Sum methods
  - o 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



# Recent Developments in LS-DYNA
(part 2)

LS-DYNA Forum
October 8, 2014
Bamberg

LSTC
Livermore Software
Technology Corp.

# Particle Methods

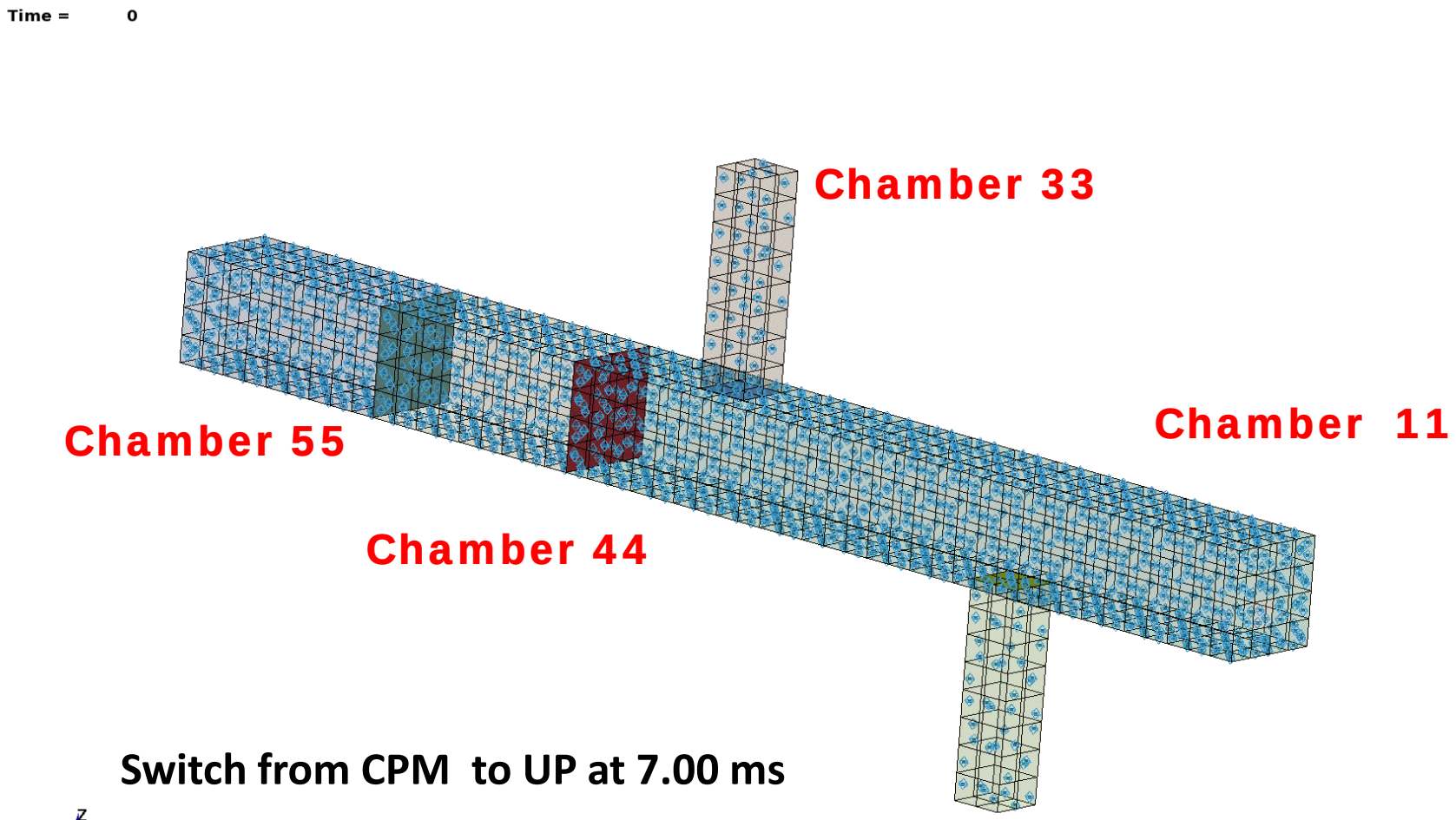
# Meshless Particle Solvers

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: $pV=nRT$
- 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



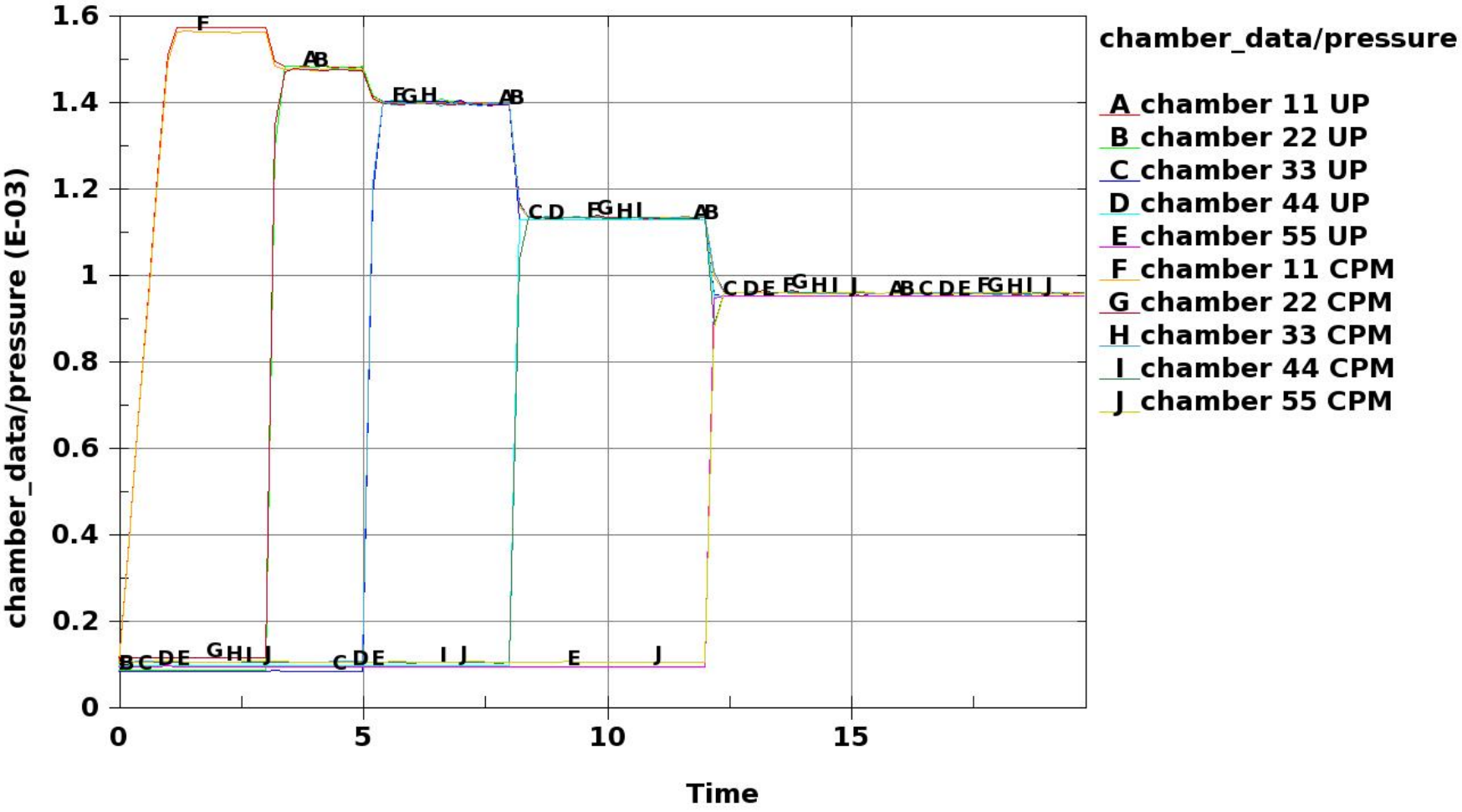

**Switch from CPM to UP at 7.00 ms**

**Old: chambers collapsed into one bag**

***New: chambers becomes separated UP domain***

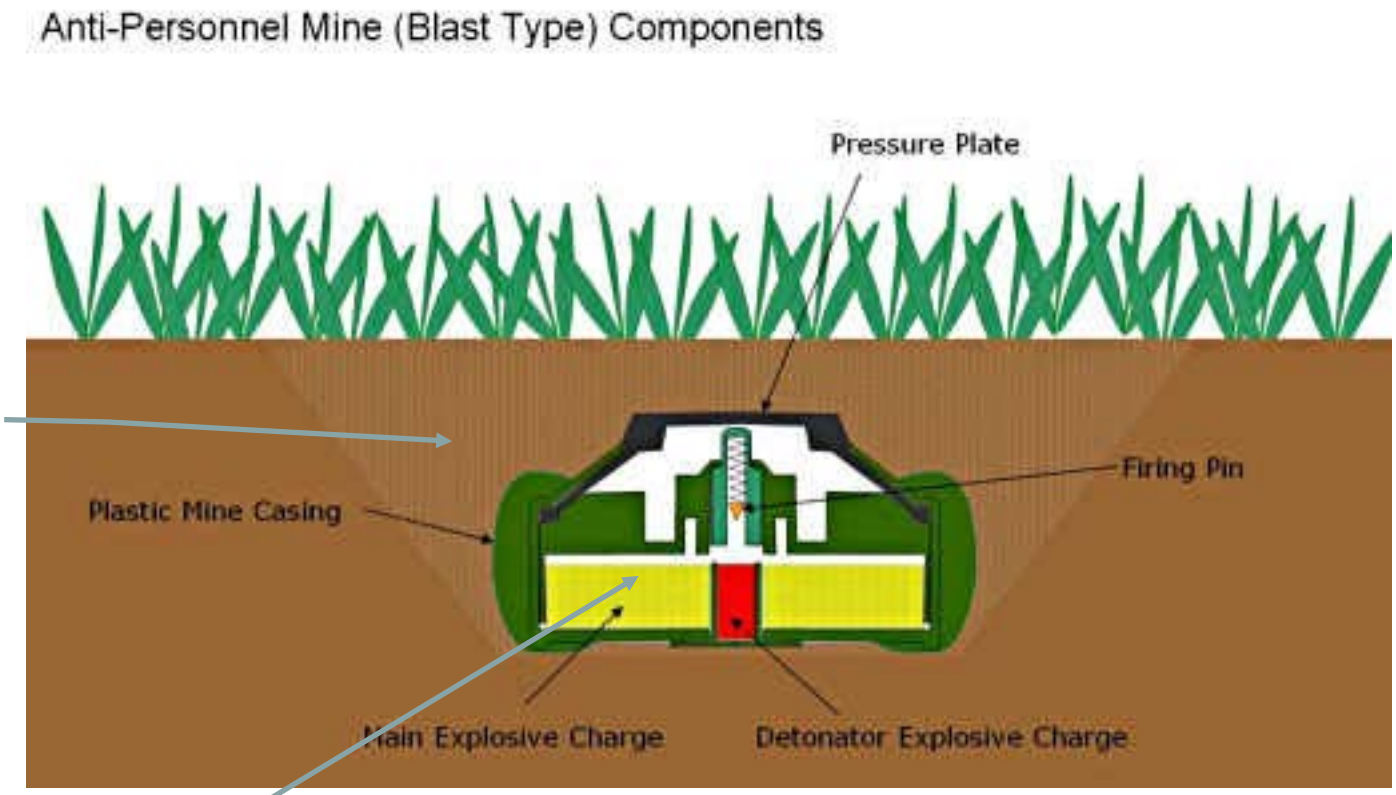
# CPM/UP switch with Chambers

- **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)**



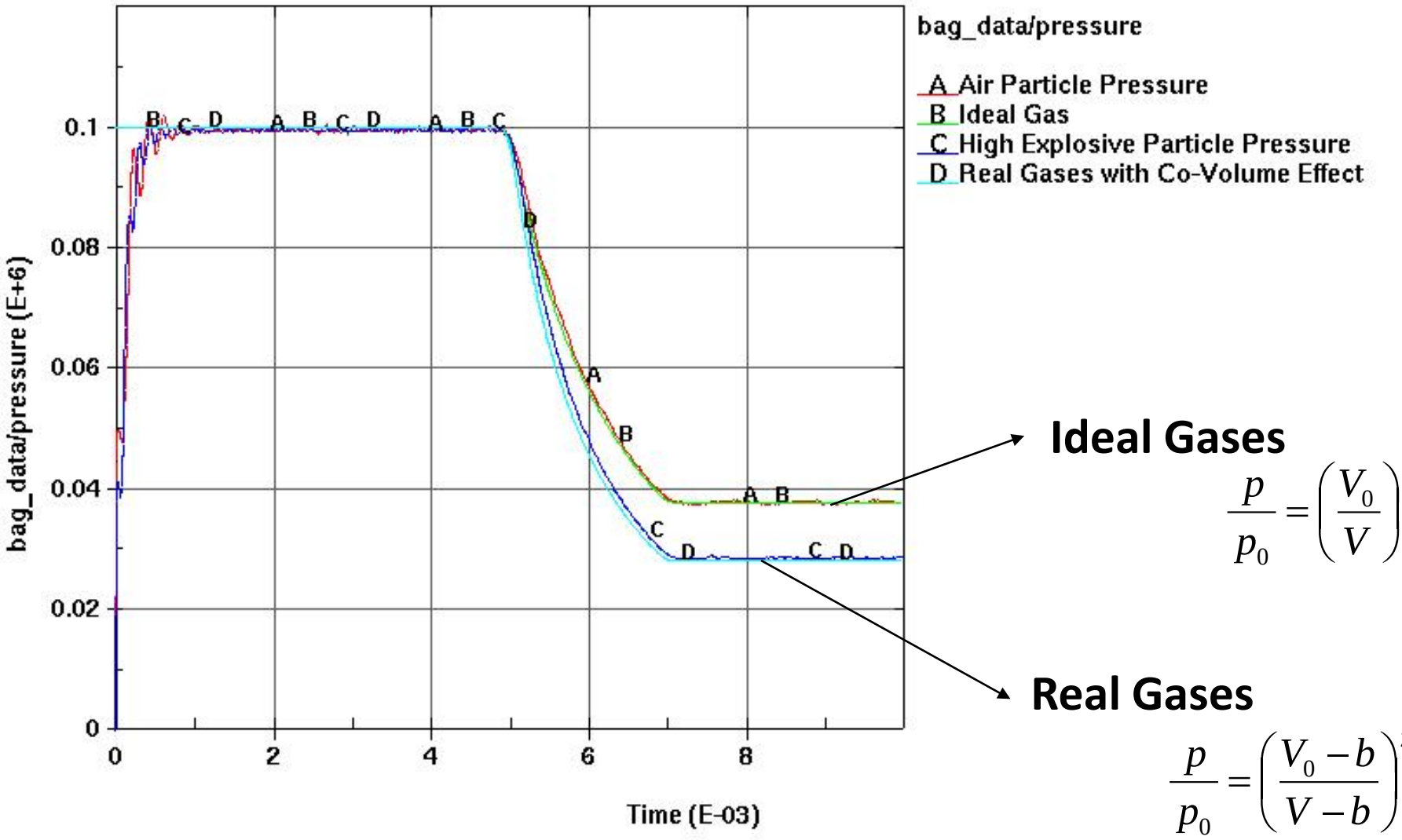

**Discrete Element Method (DEM)**

**HE Particle (Real Gas Law)**

# Particle Gas, Particle Blast–Real Gas Law

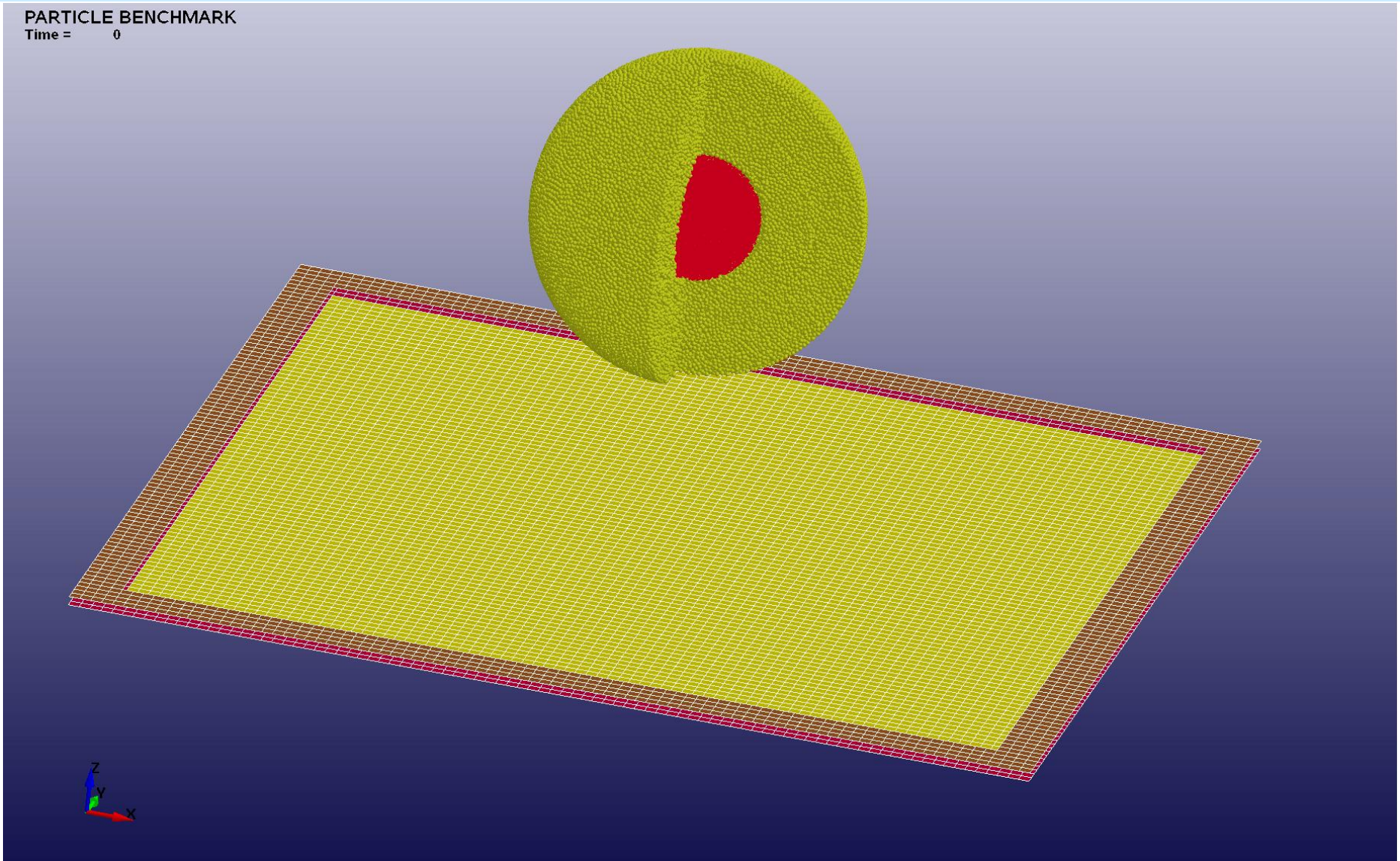
- Air Particle
  - Modeled by ideal gas law (CPM): $pV=nRT$
- High Explosive (HE) Particles
  - Modeled by real gases: $p(V-b)=nRT$
  - The co-volume effect is included
  - Work for high pressure and high temperature
  - Pressure drop sharply during adiabatic expansion

# Particle Gas, Ideal and Real Gas Law



# Particle Blast, Real Gas Law

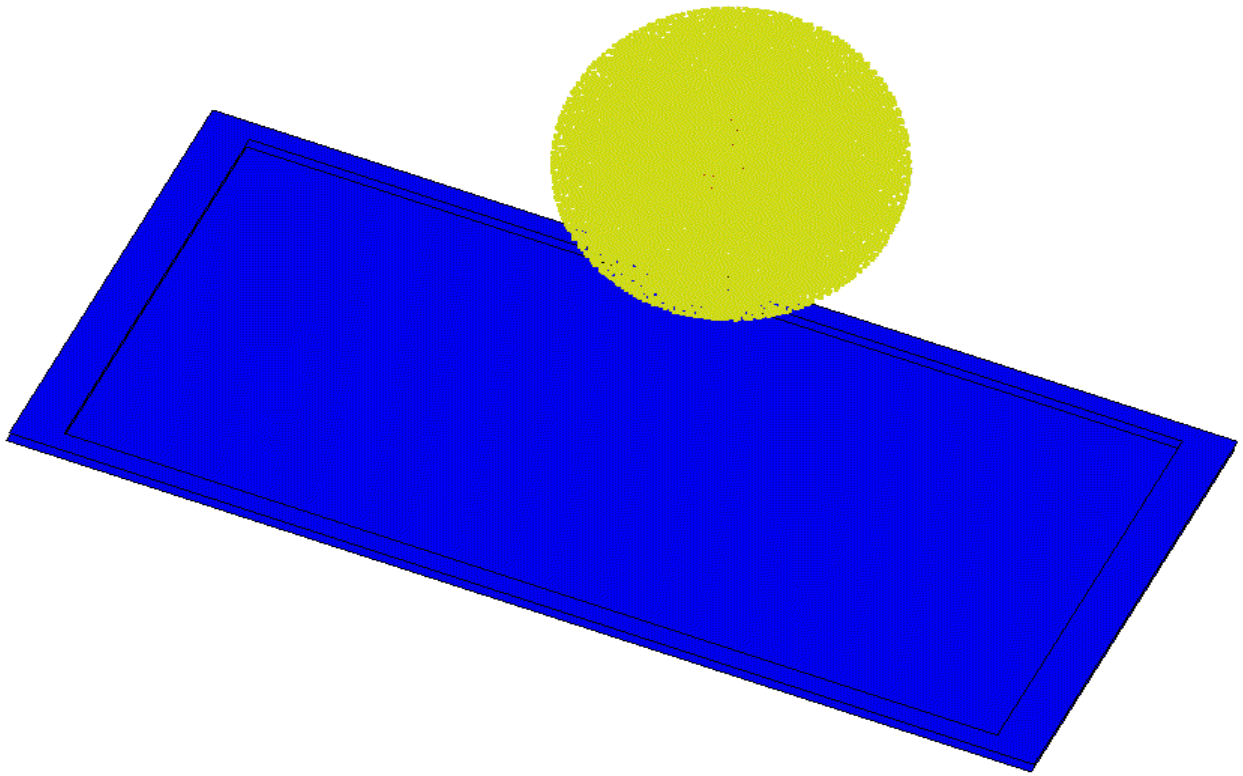
## Numerical Example



# Particle Blast, Real Gas Law

## Numerical Example

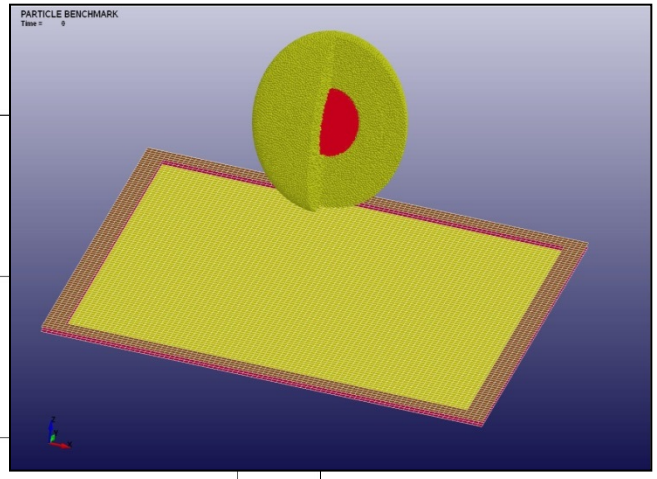
**Blast simulation with sand**

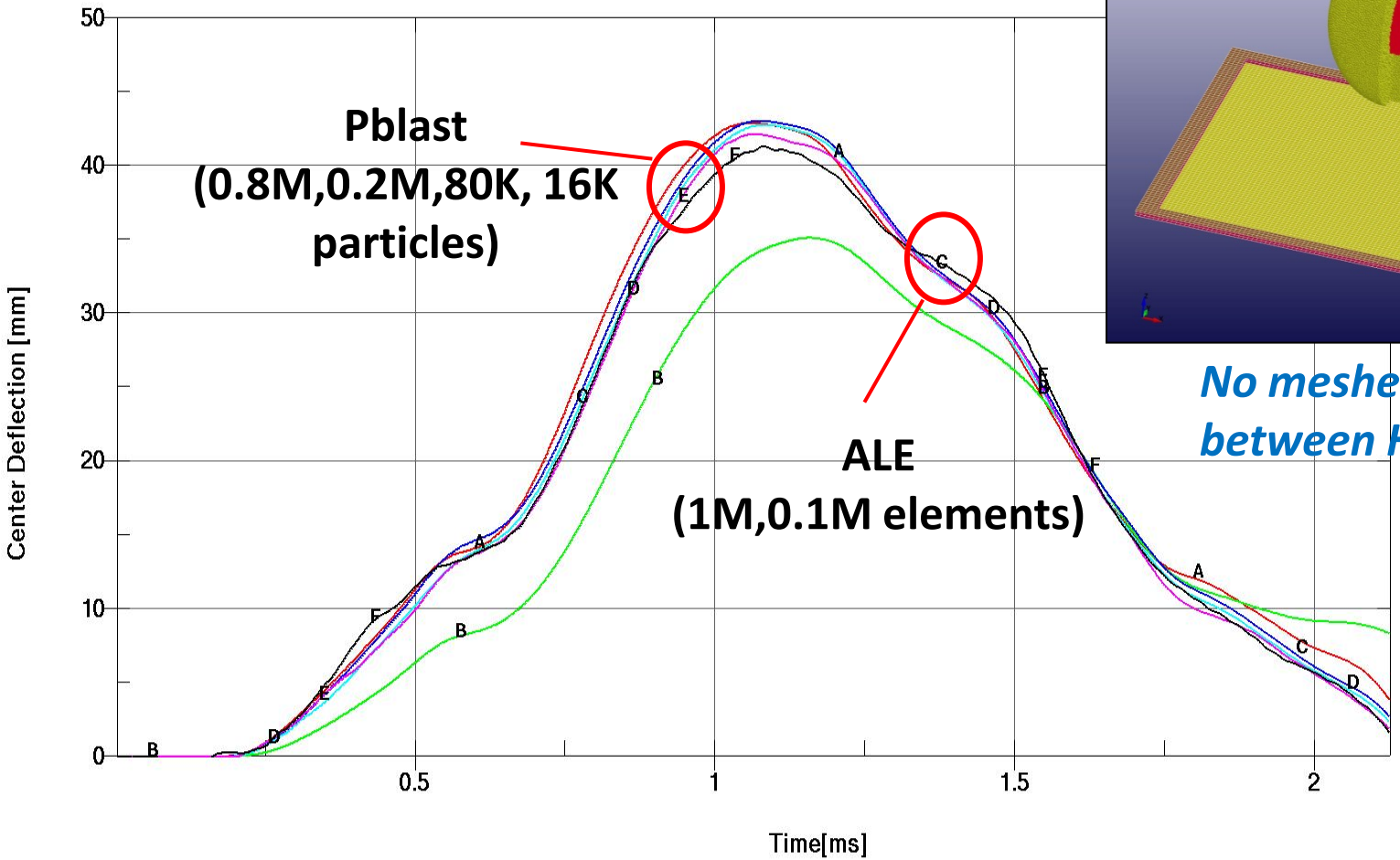


# Particle Blast, Real Gas Law

## Numerical Example

**Simulation Results for 700mm Model**



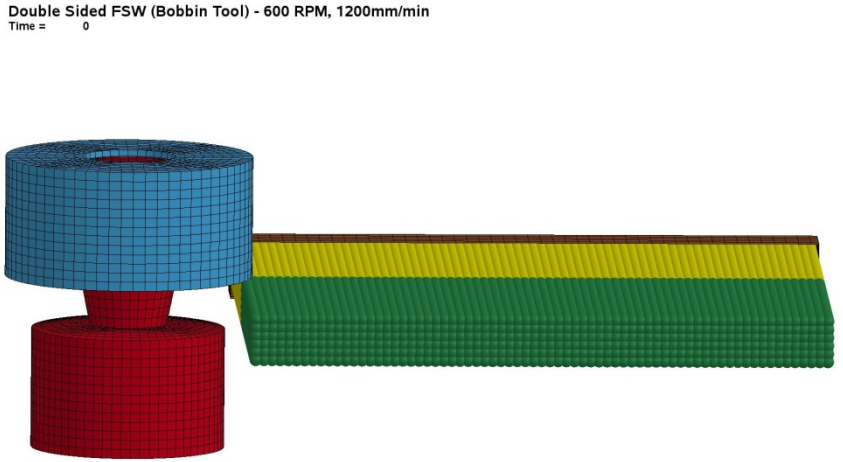




***No meshes needed between HE and target***

# Enhancement of SPH

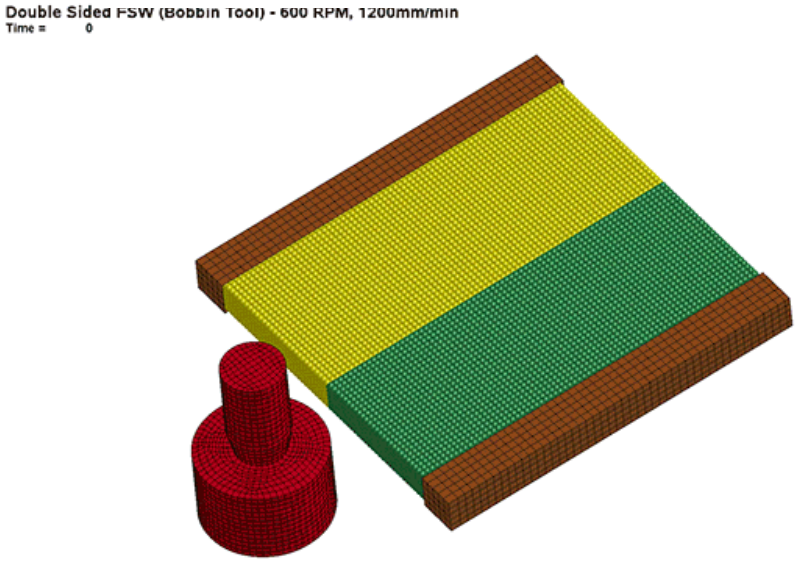
1. **Friction Stir Welding**
2. **SPH to SPH contact**
3. **SPH active region and new bucket sort**

# Friction Stir Welding

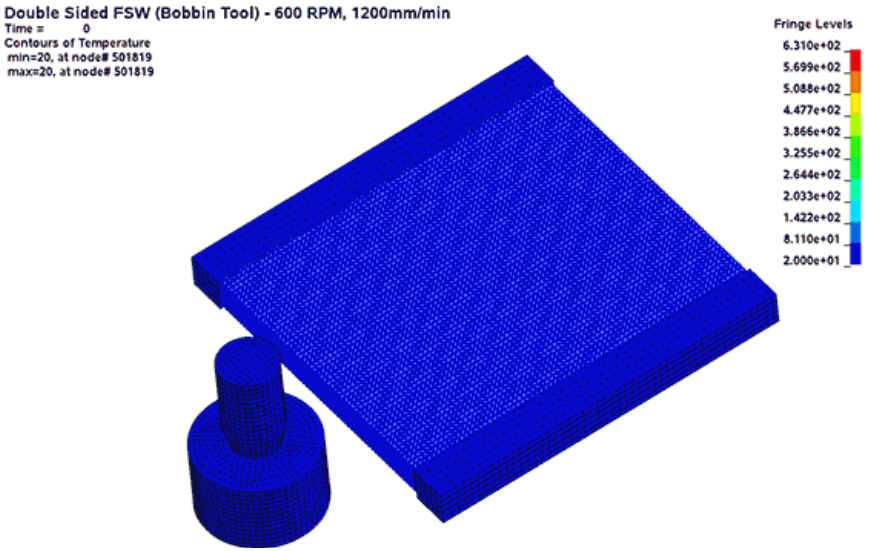



***Extended SPH thermal solver for SPH form 7 and 8***

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




***Material Mixing***



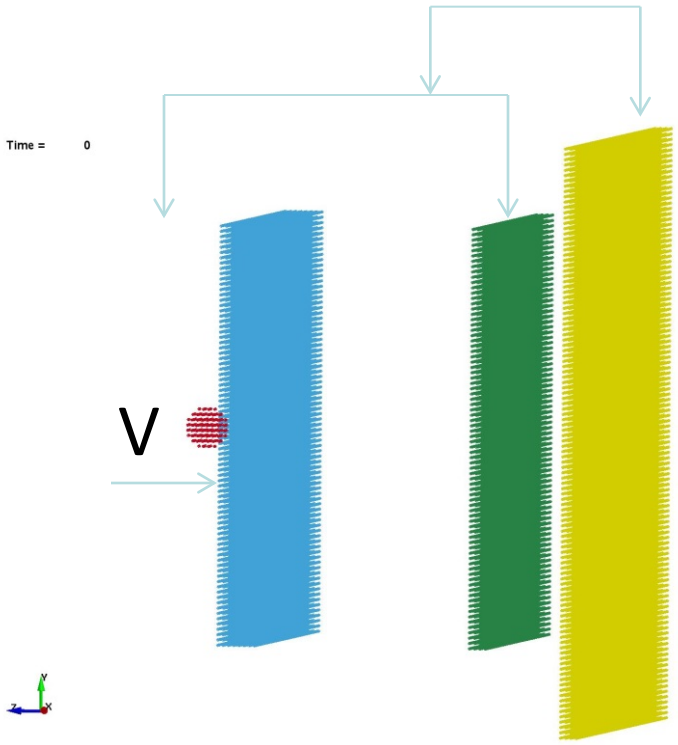

***Temperature Contours***

# SPH Interaction

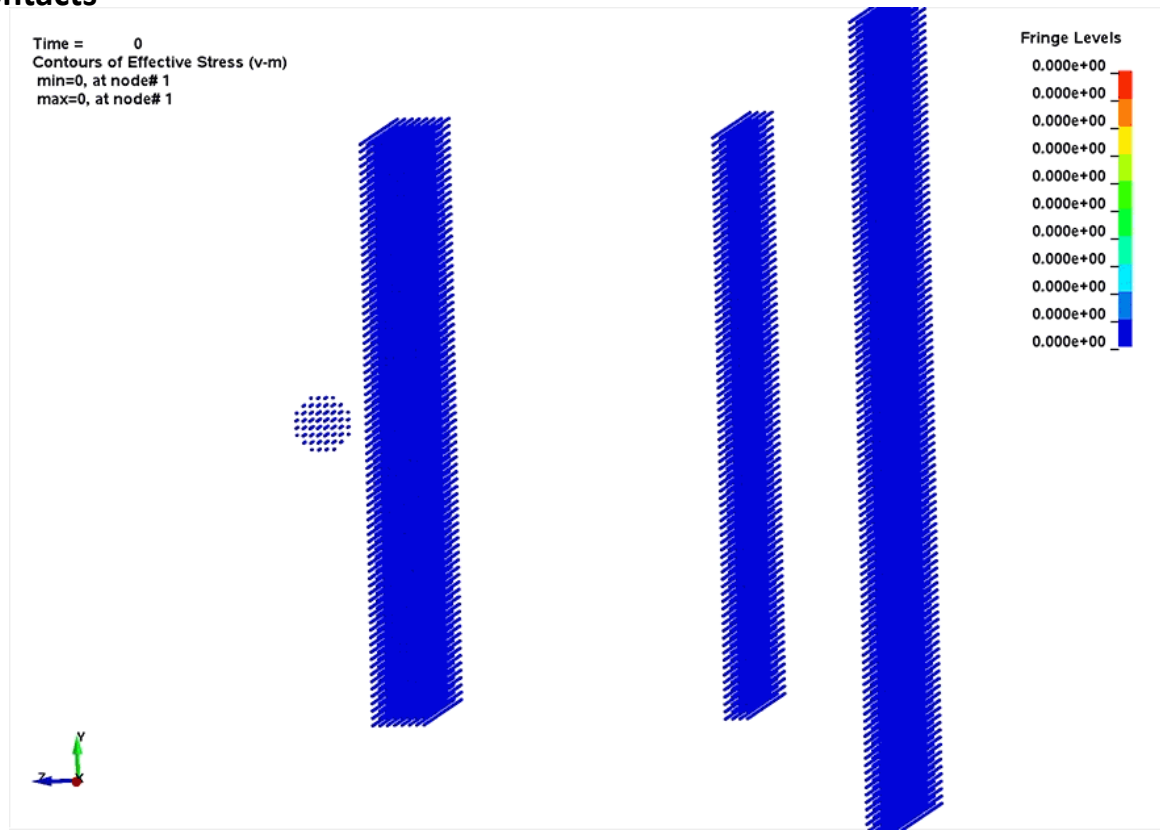
**Multiple impacts with Keyword: SECTION_SPH_INTERACTION**

**Define the different type of interactions between SPH parts.**

**Metals with Standard interactions**

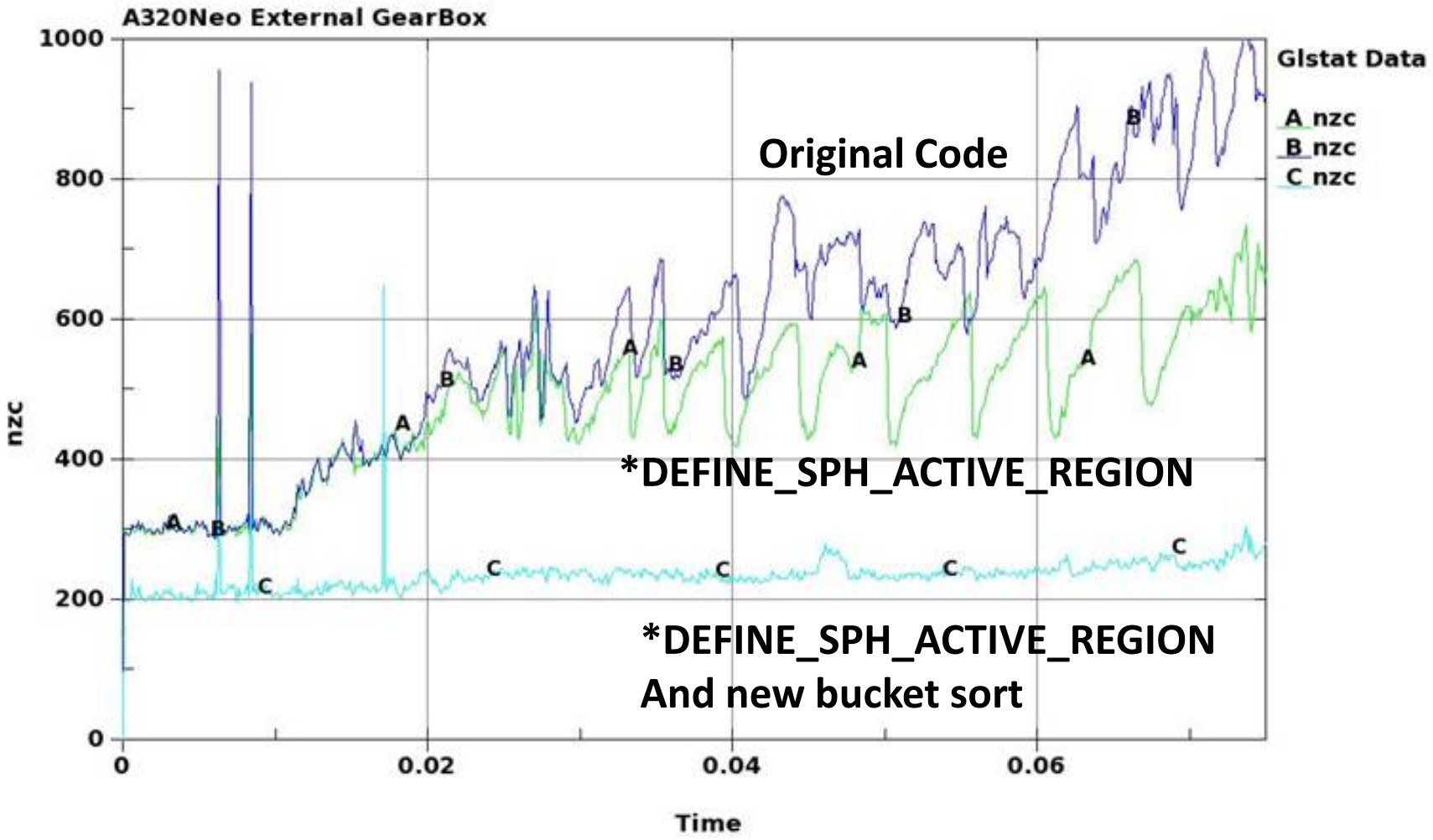



**Ceramic with node to node contacts**



# SPH Active Region & Better Bucket Sort

Time per Zone Cycle

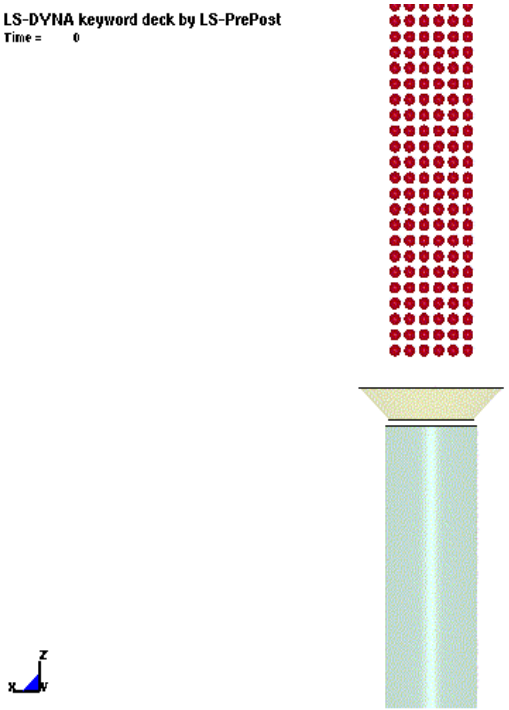


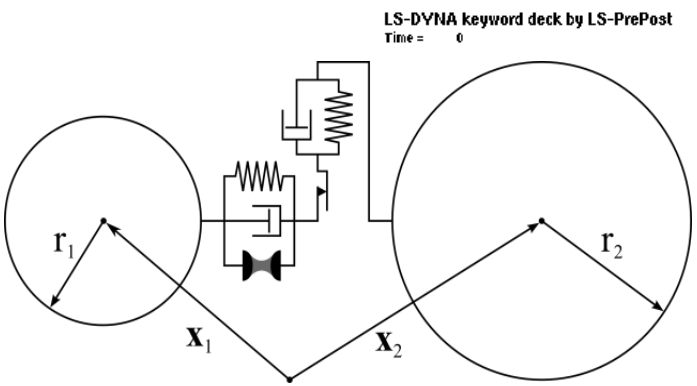
# Discrete Element Method (DEM)

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

# Discrete Element Method (DEM)

## Discrete Element Sphere



**Dry Particle**

**Wet Particle**

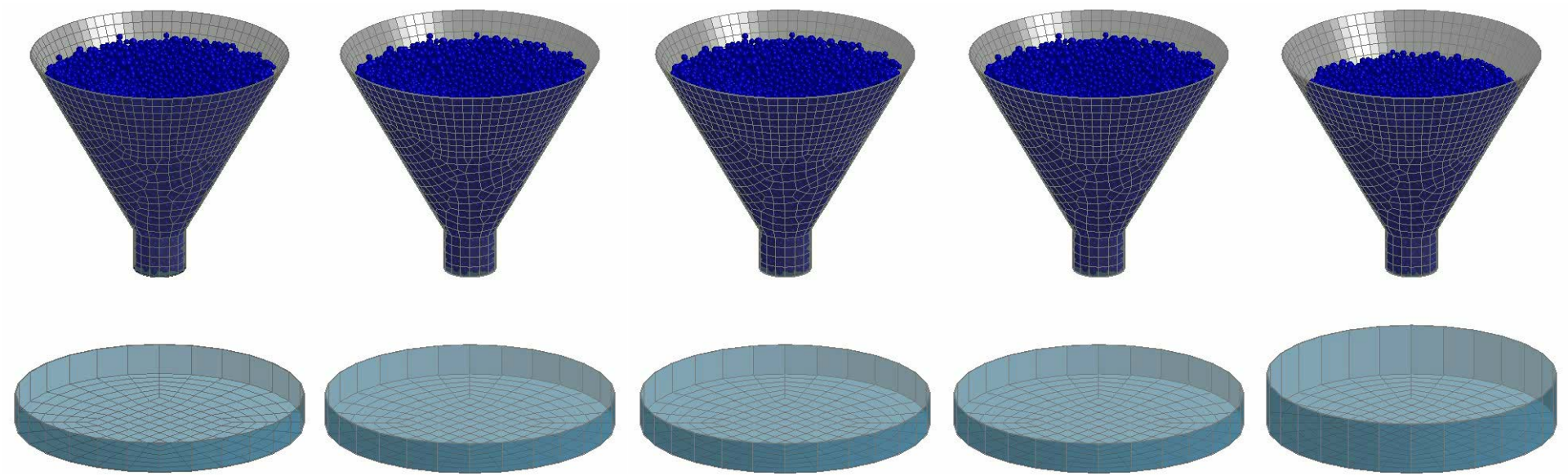
# Discrete Element Method (DEM)

## Discrete Element Sphere

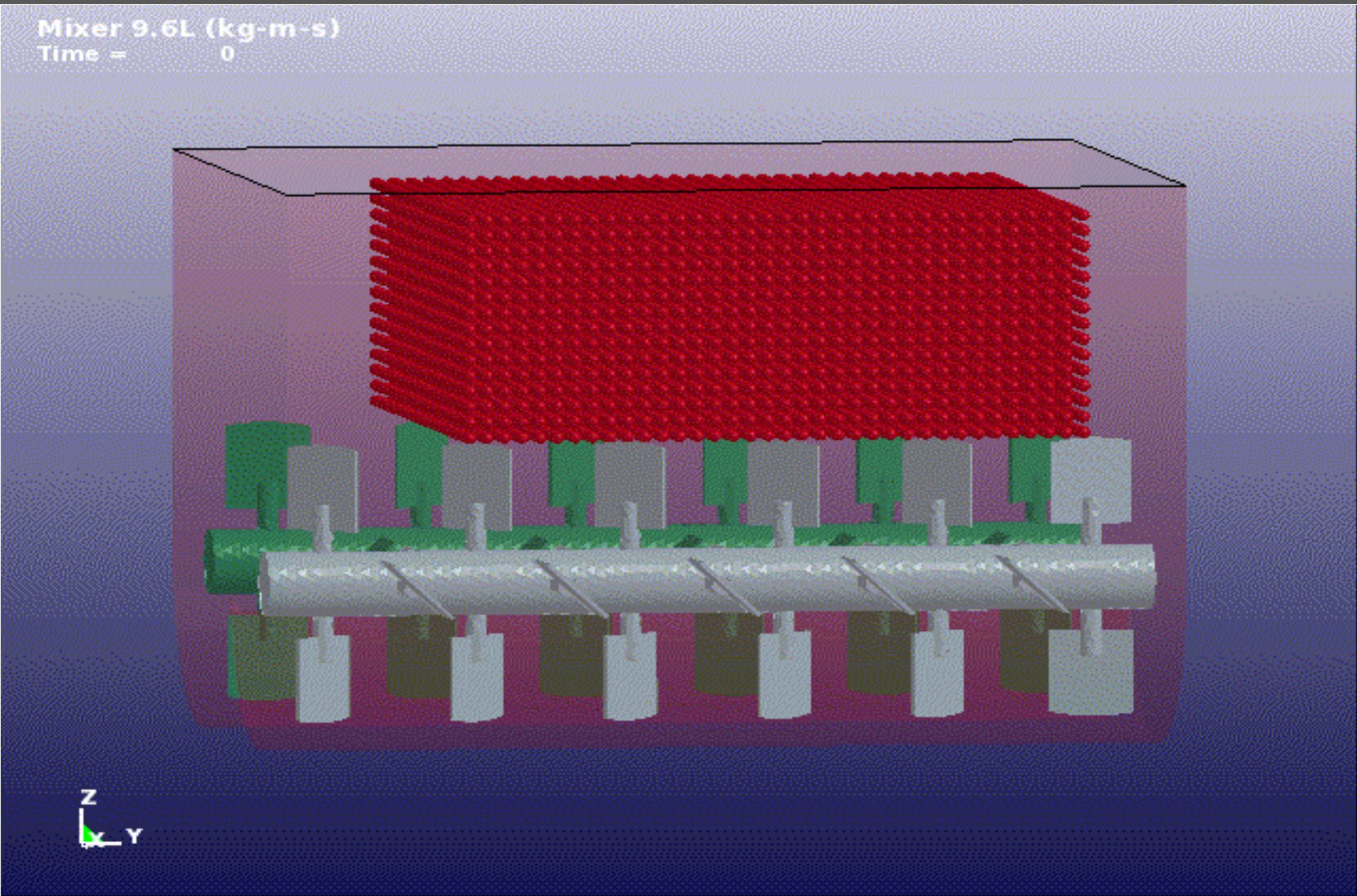
**Variation of the parameters Courtesy of Dr.-Ing. Nils Karajan, Dynamore GmbH**

| $ | 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 |

**foamed clay** **dry sand** **wet sand** **fresh concrete** **“water”**

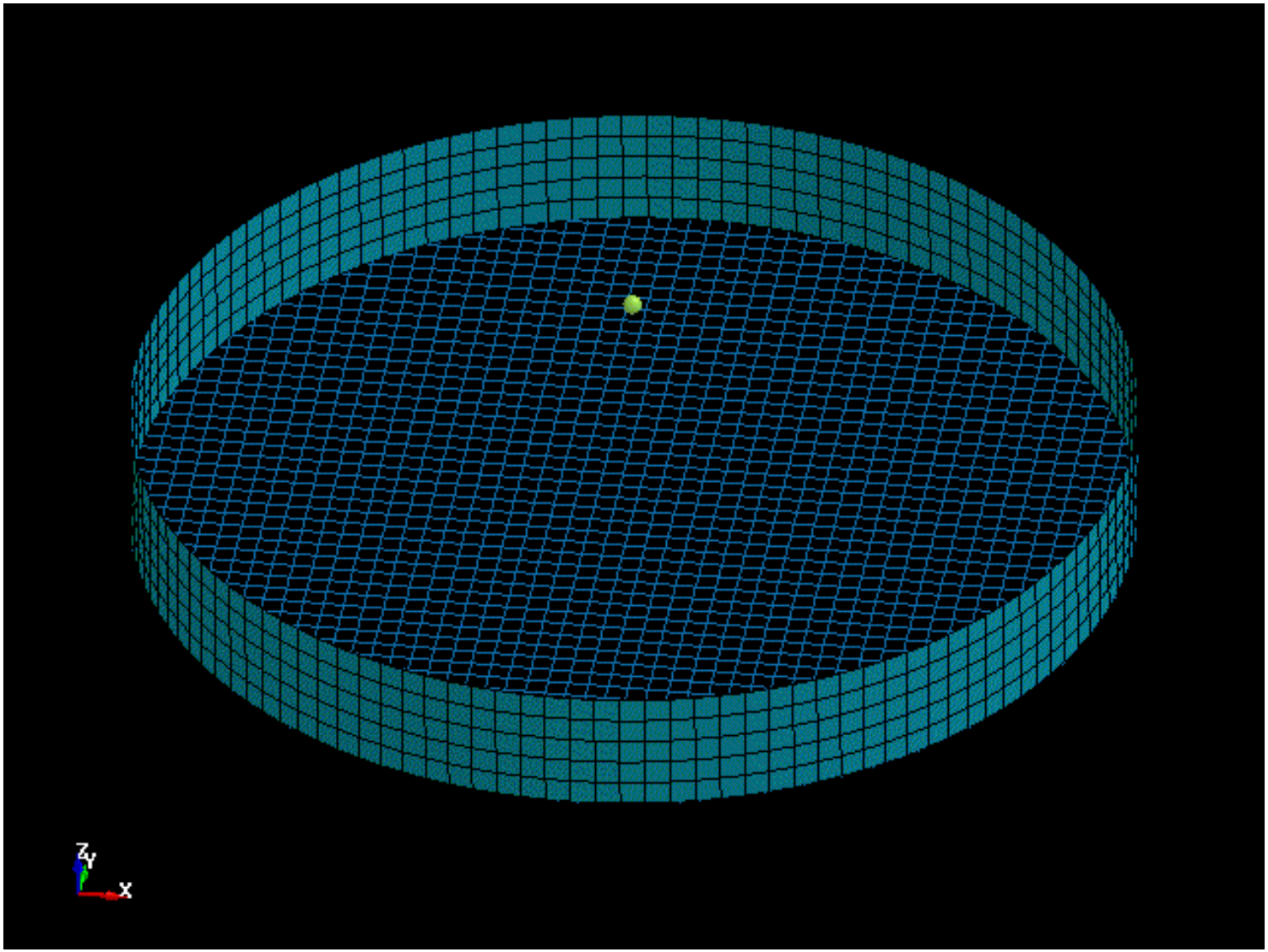


# DEM Mixer



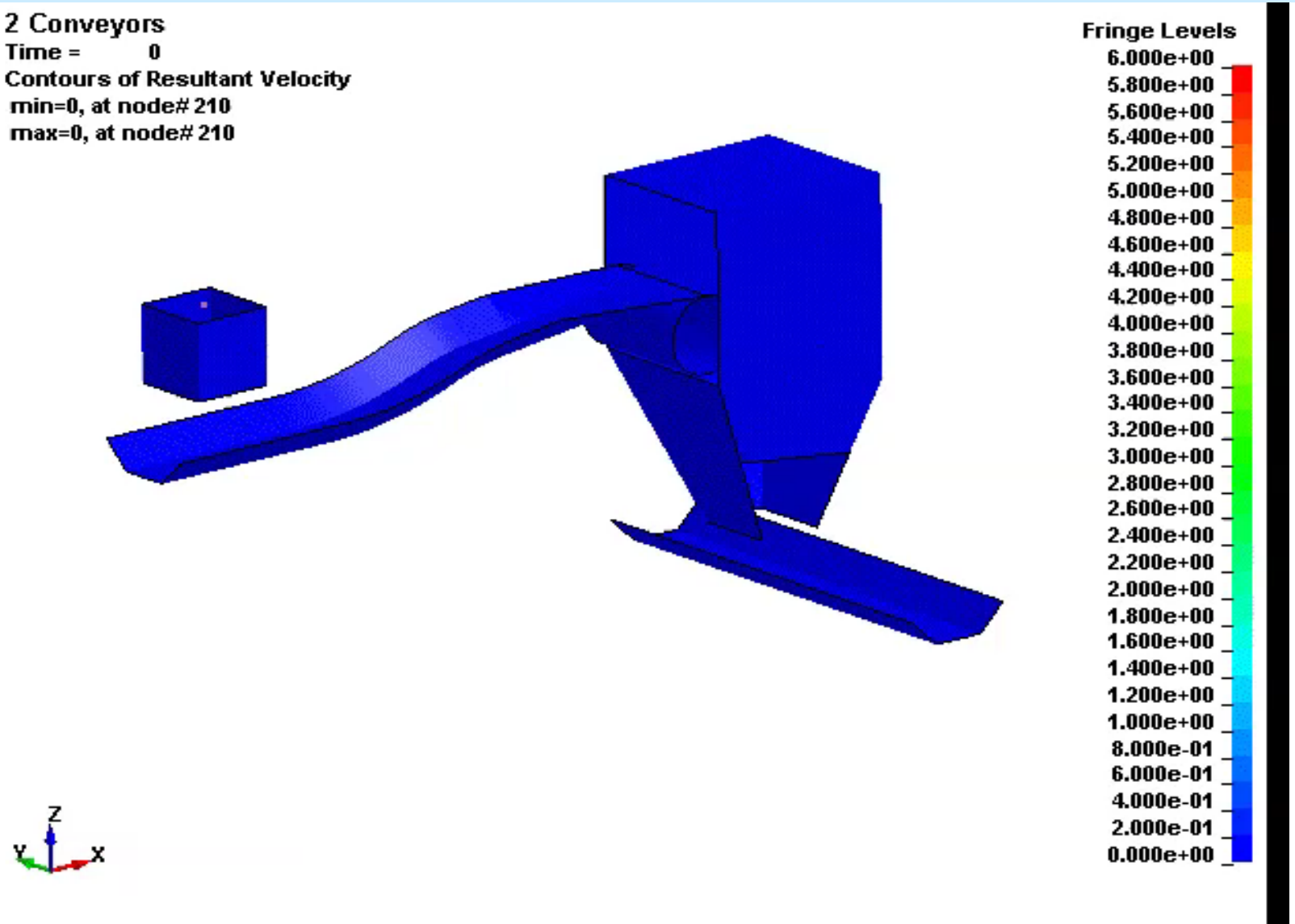
# Node to Beam Coupling

*DEFINE_DE_TO_BEAM_COUPLING



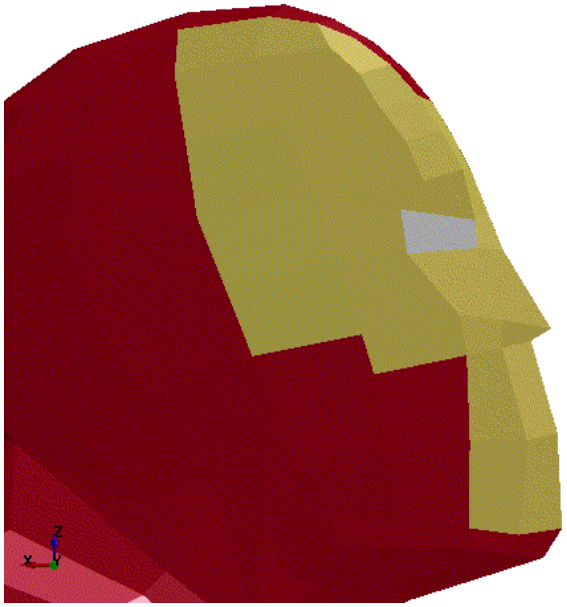
# Node to Surface Coupling

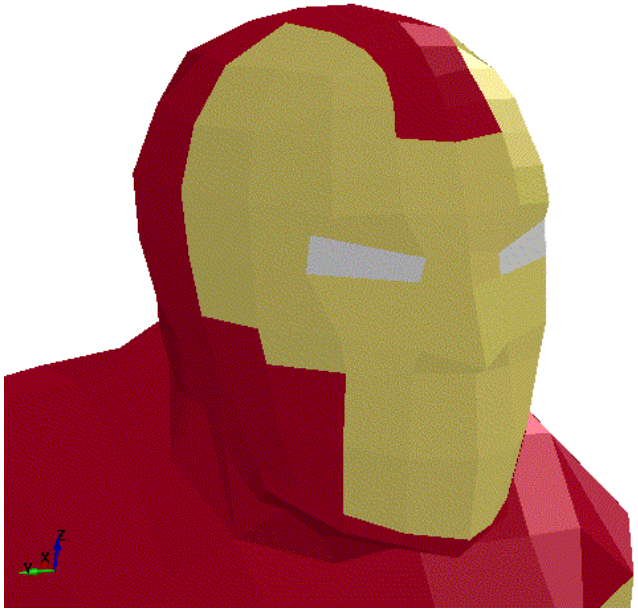
*DEFINE_DE_TO_SURFACE_COUPLING



# Tied Node to Surface Coupling

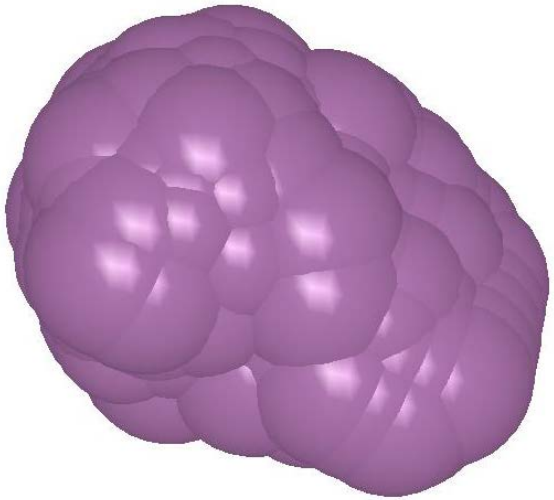
*DEFINE_DE_TO_SURFACE_TIED

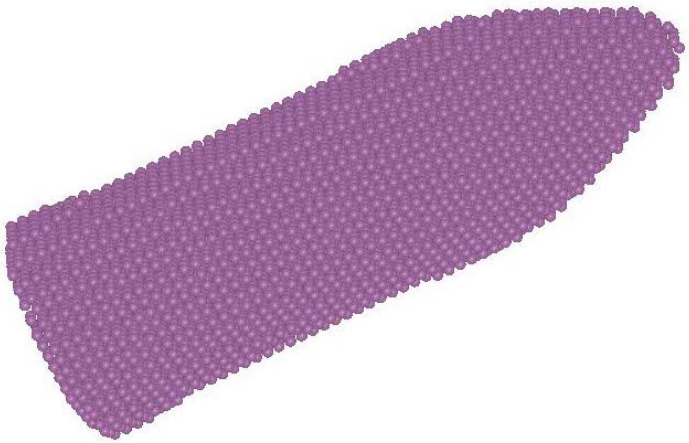


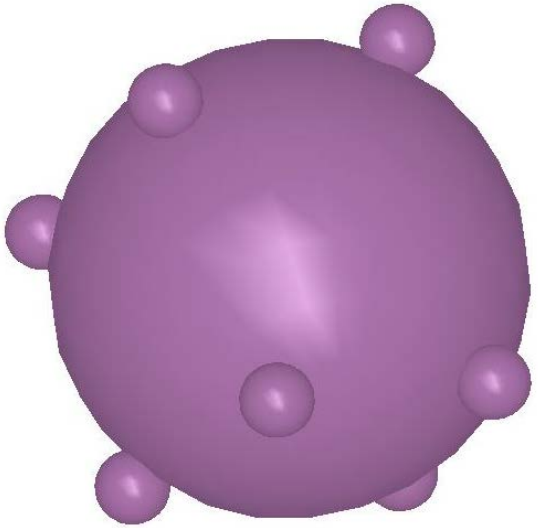


**Throwing a pie in the face, Courtesy of Kazuya, Lancemore**

# Other Shapes



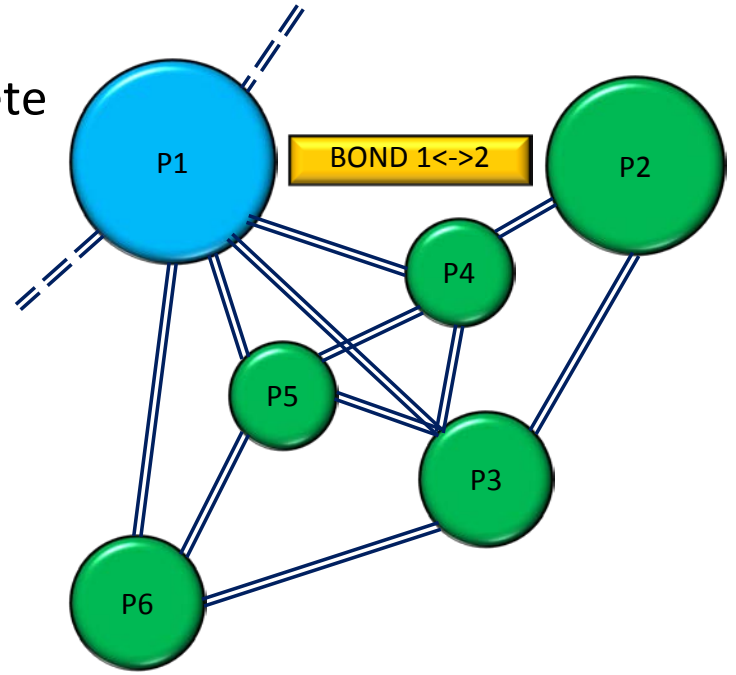




# Discrete Element Sphere with Bond Model

**Extending 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

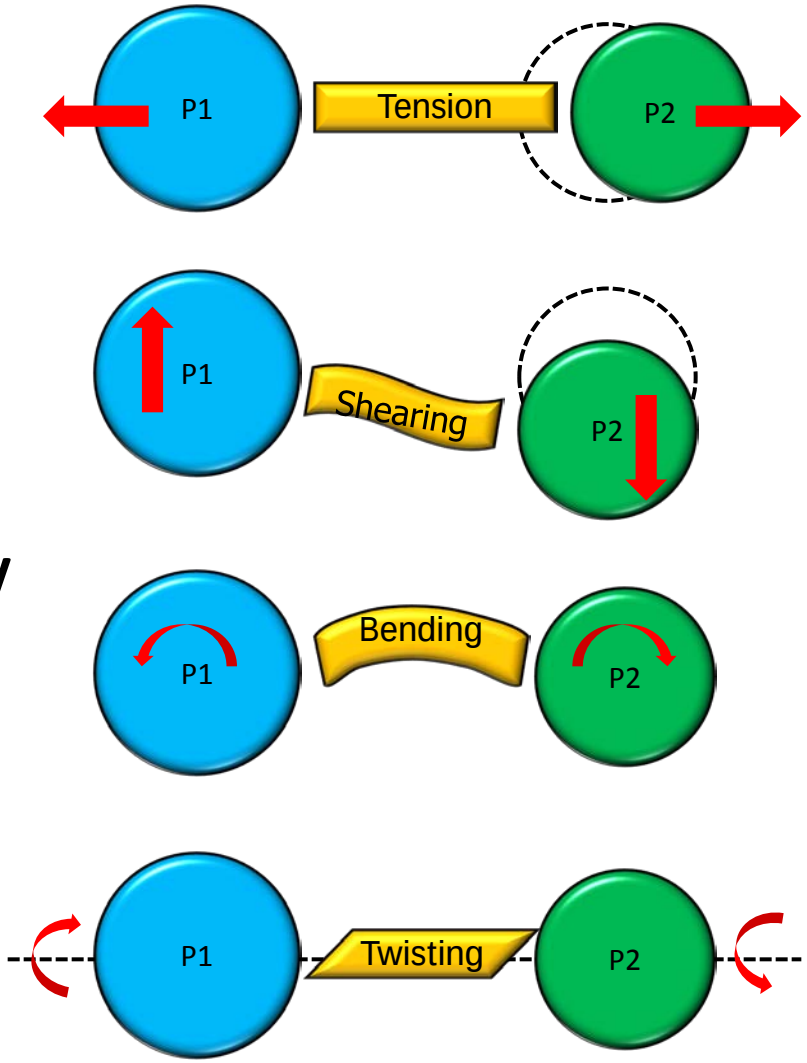


# DEM Bonds

- 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

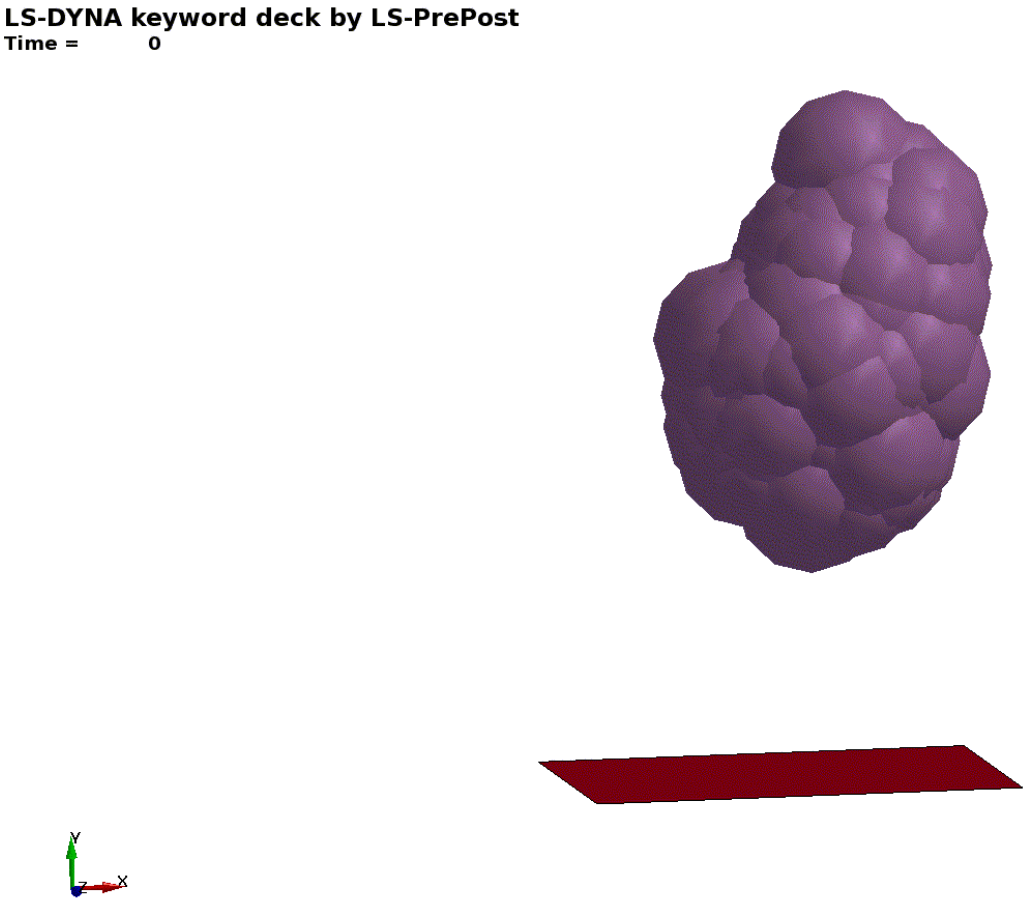
# DE Bond TYPE I

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



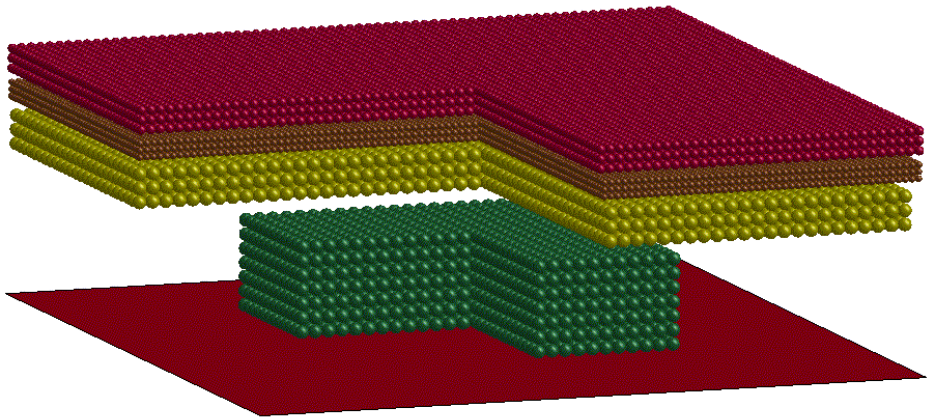
# DEM Bond TYPE I

- **Form different shapes of particles using DEM**



# DEM Bond TYPE I

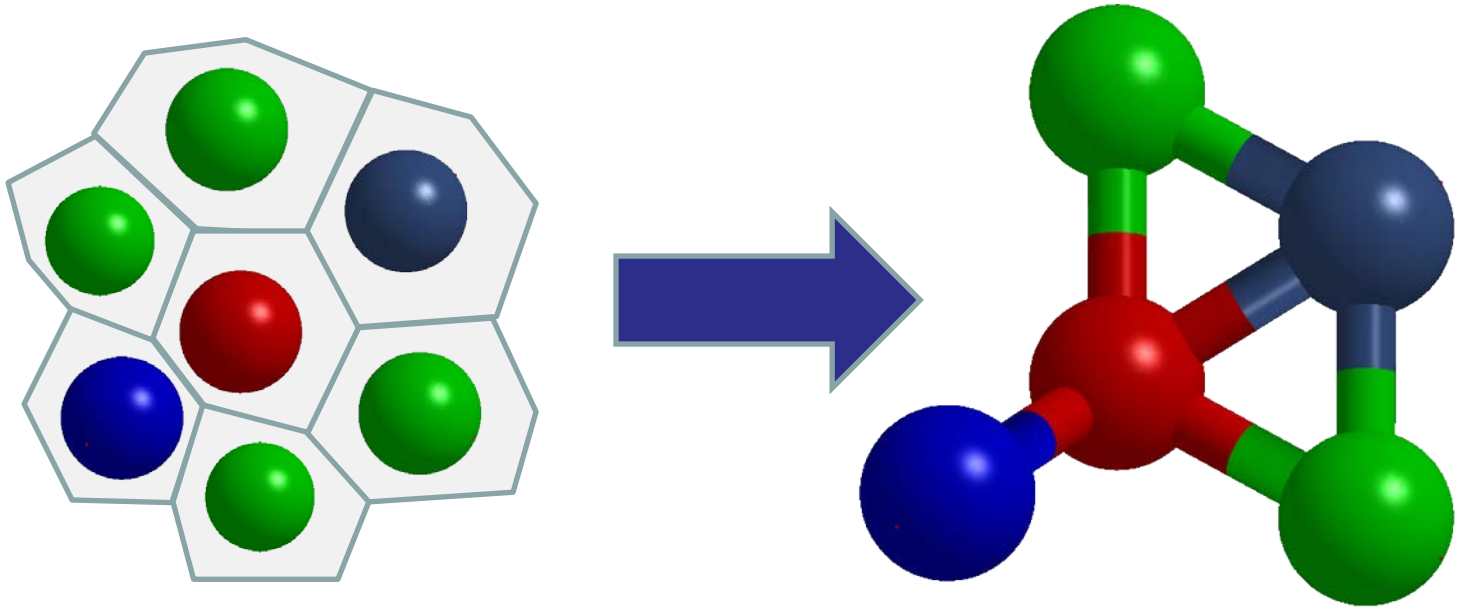
**DE Paste (microgm-micron-sec)**
**Time = 0**



# DEM TYPE II

**Heterogeneous BOND(HBOND)**

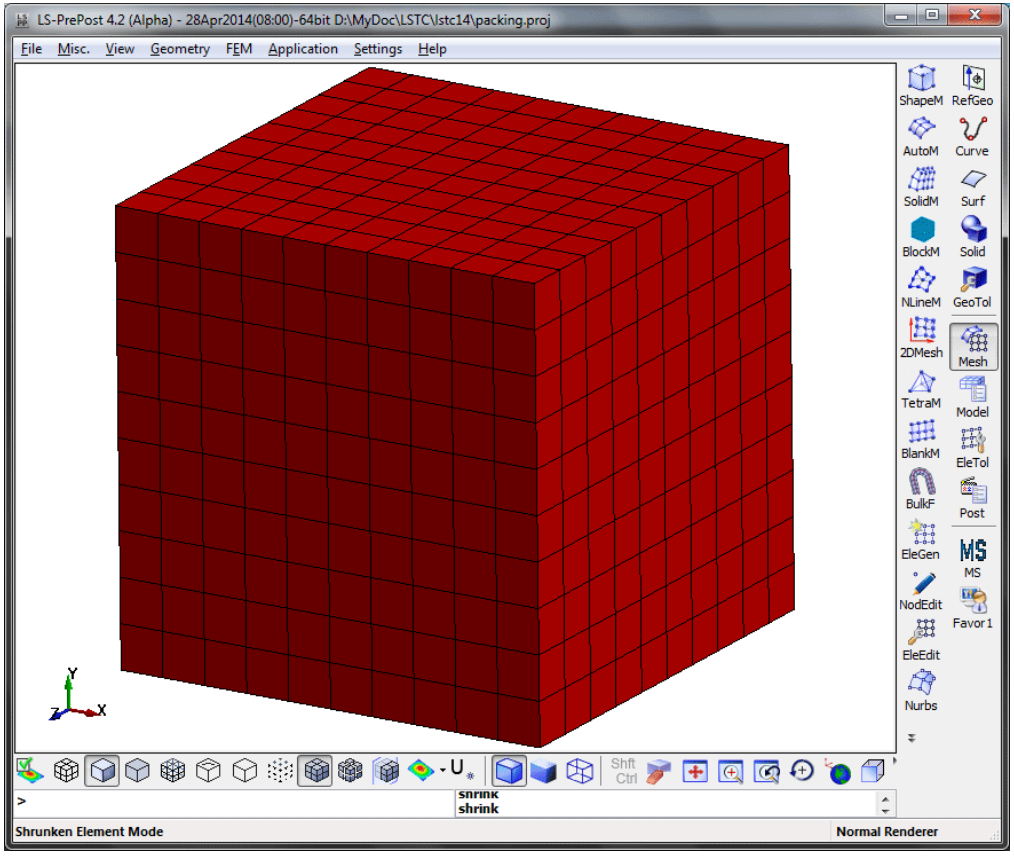
**Continuum Particle Method**



- ***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

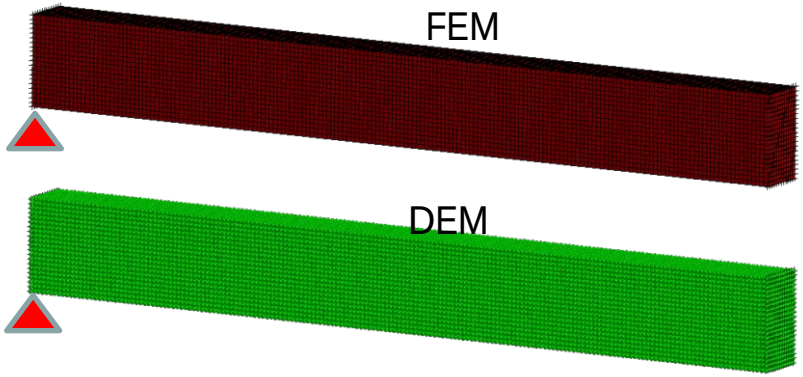
**From “Continuum” to “Discrete”**

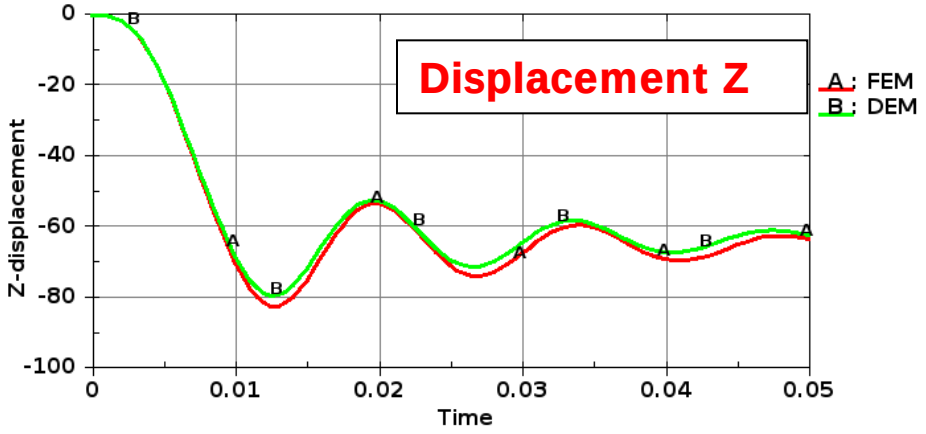



- One model setup
- One solver
- Same material models
- Multi-physics:
  *Continuum Mechanics*
  *Damage Mechanics*
  *Fracture Mechanics*
  *Discrete Mechanics*
- Built-in self contact
- Coupling with other FEM and particle methods

# HBOND Verification

## A simply-supported beam under a body force



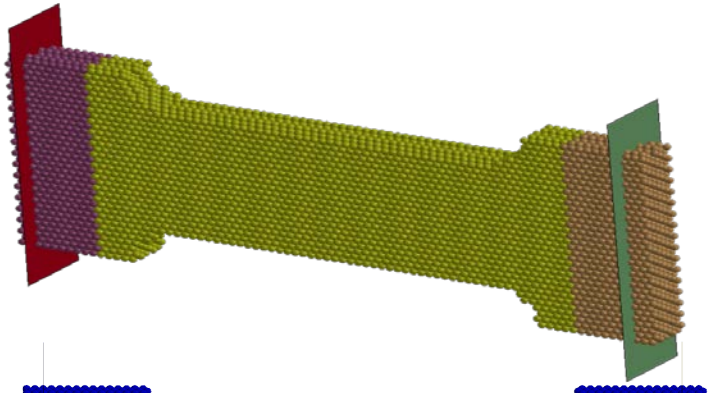



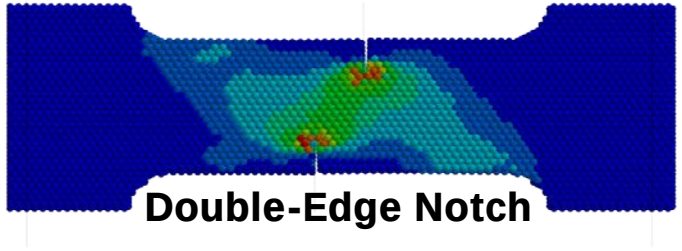

- **FEM & DEM models are created for one half beam with the symmetric boundary conditions in the middle.**
- **The displacements & stresses obtained by the DEM are very close to those by the FEM.**
- **No boundary effects.**

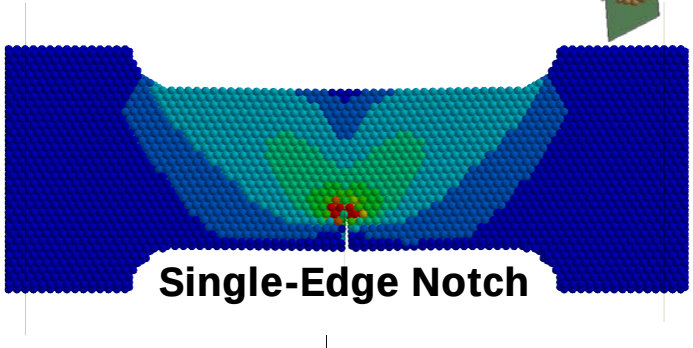
**Tension Stress X**

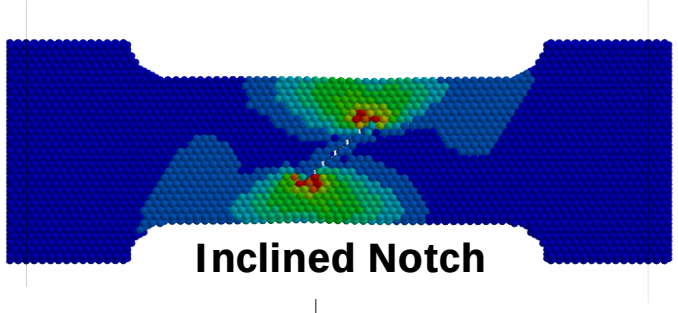
# HBOND Verification

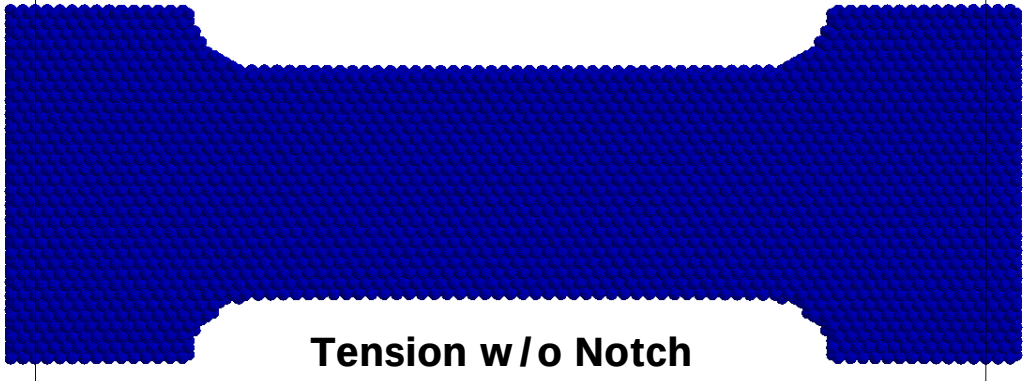
## Specimen under Tension





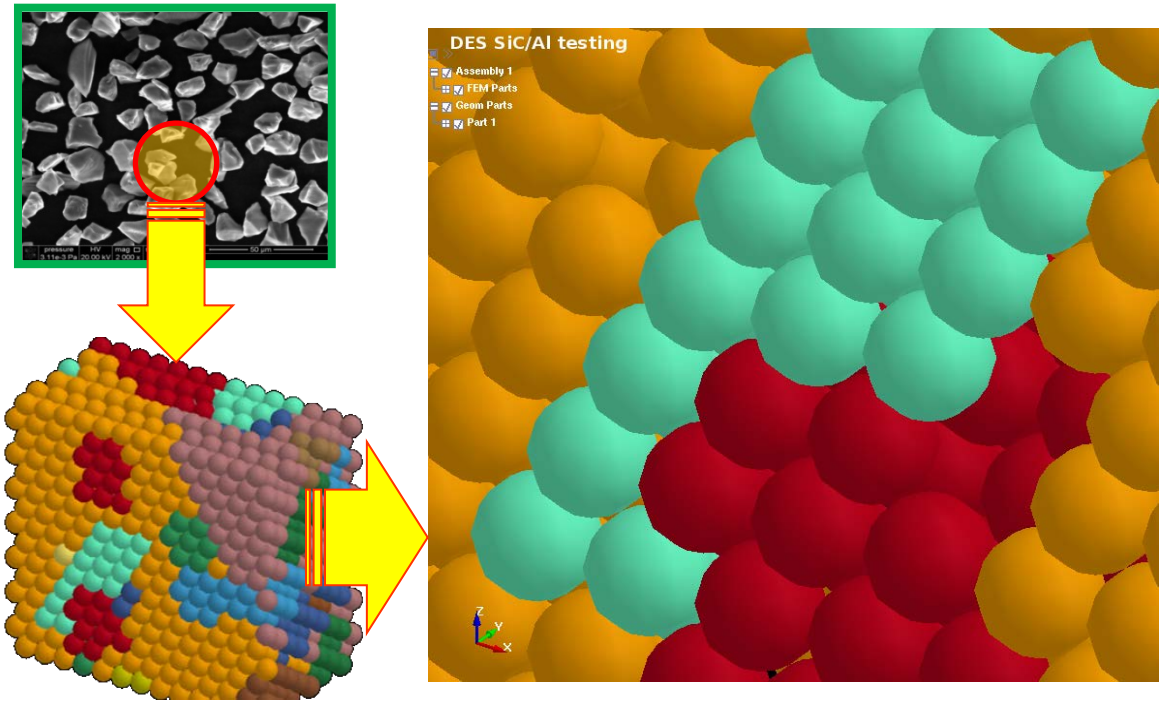



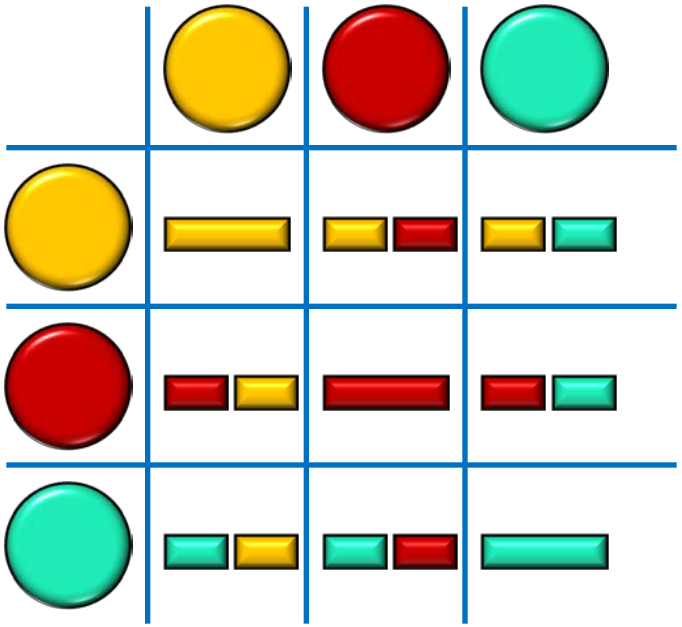



# HBOND Micro-Mechanics

***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.

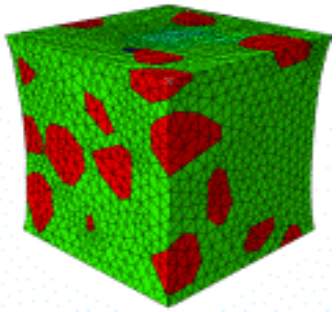


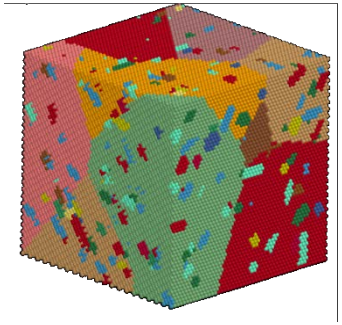
**Various heterogeneous bonds**

# SiC/Al Metal Matrix Composite

## DEM for Material Design

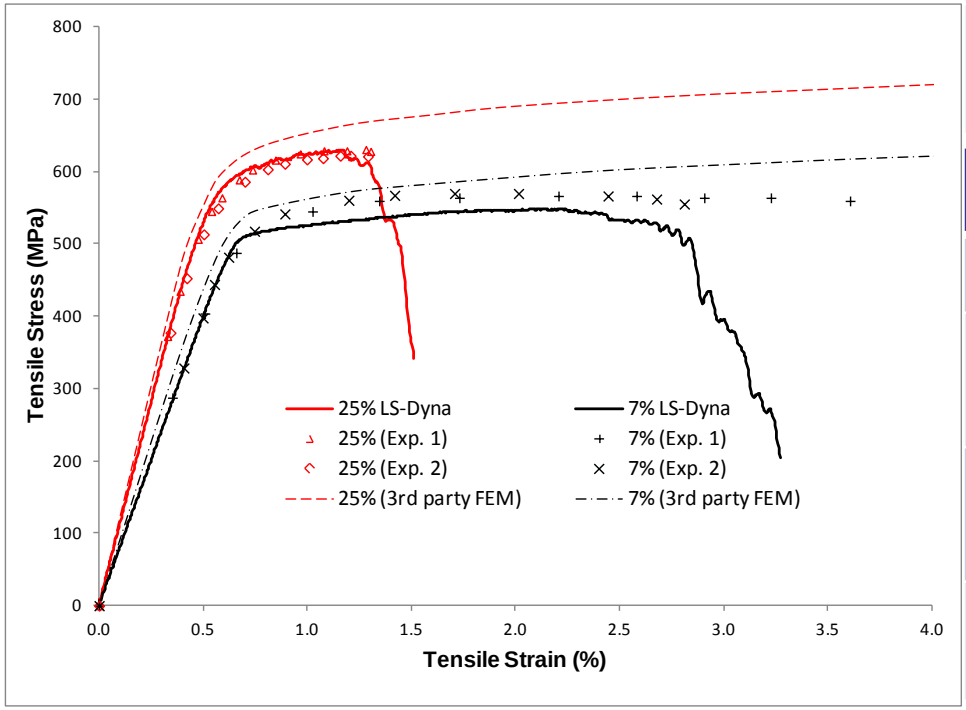


3rd party FEM



LS-DYNA DEM

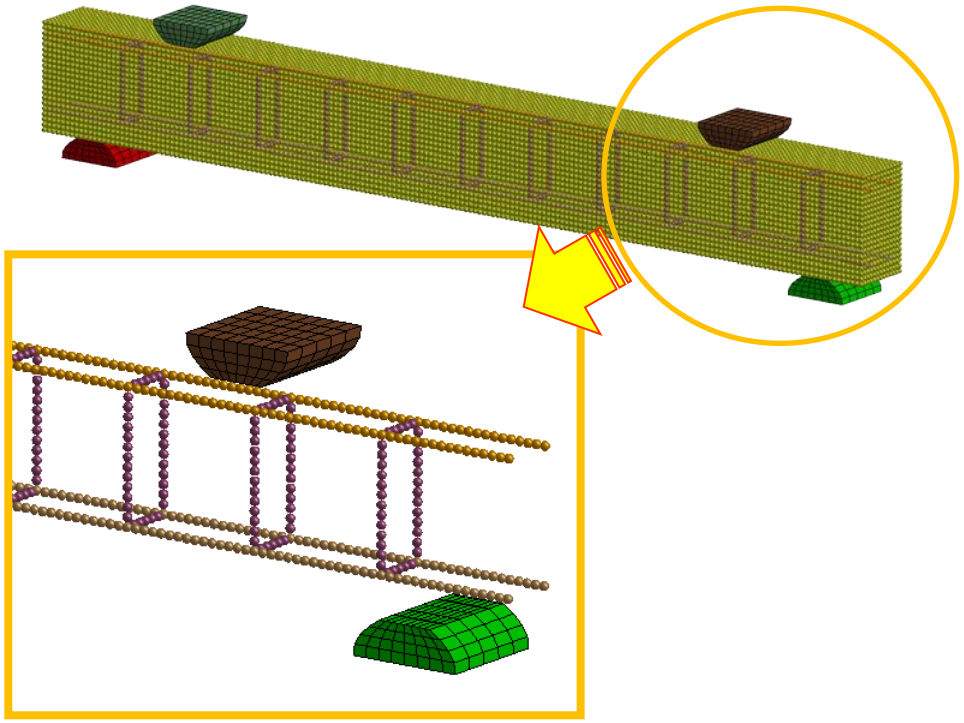
| Material Properties | Al | SiC |
|---|---|---|
| Density: [kg/m$^3$] | 2,700 | 3,100 |
| Young's modulus [GPa] | 71.7 | 427 |
| Poisson's ratio: | 0.33 | 0.17 |
| Failure Energy Rate: [kN/m] | 40 | 15 |
| Average Particle size: [um] | - | 13 |

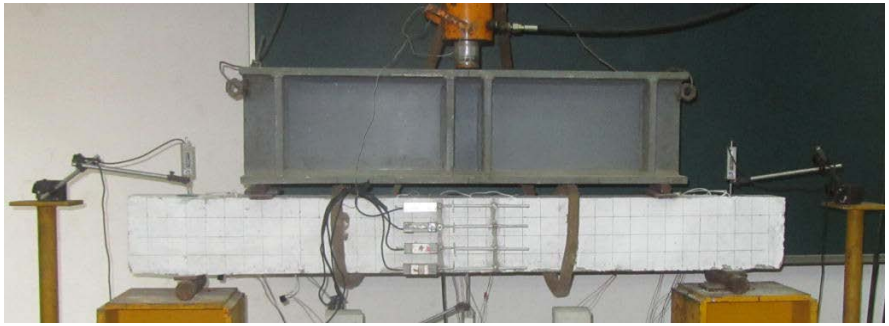


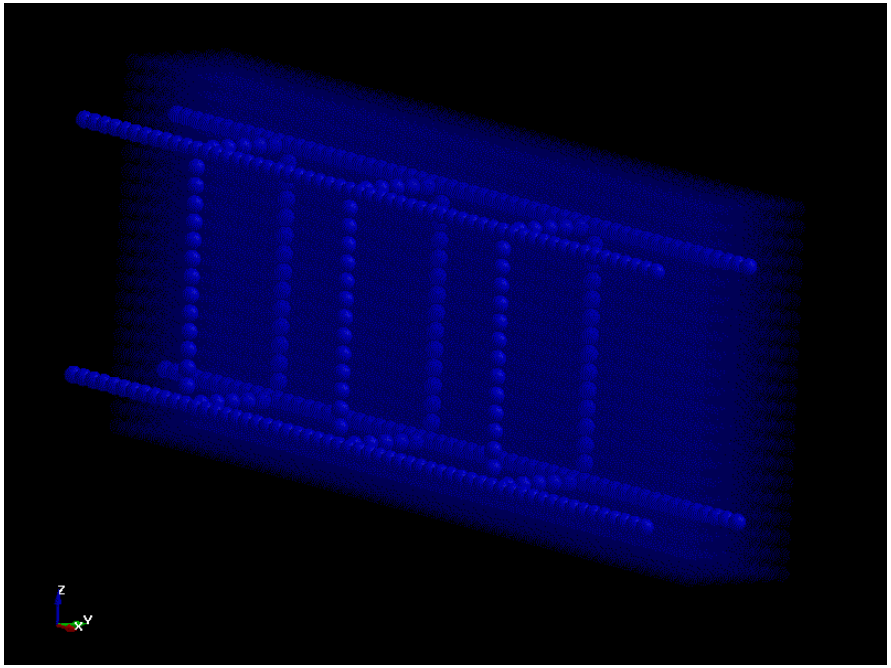

| LS-DYNA Results | 7% vol | | 25% vol | |
|---|---|---|---|---|
| | Exp. | Num. | Exp. | Num. |
| Young's modulus [GPa] | **84.2** | **80.6** | **113.3** | **113.5** |
| Tensile strength [MPa] | **568.6** | **545.0** | **623.6** | **641.3** |
| Limit strain | **1.8%** | **1.9%** | **1.2%** | **1.4%**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

# HBOND

## A Reinforced Bar under Four-Point Bending





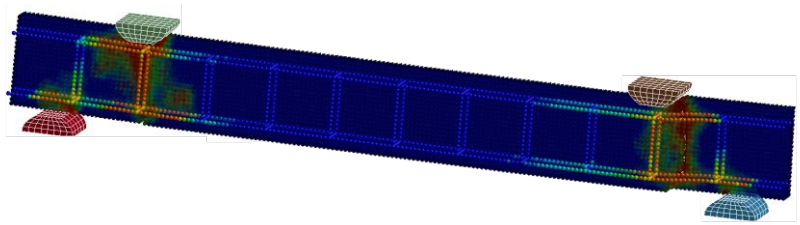


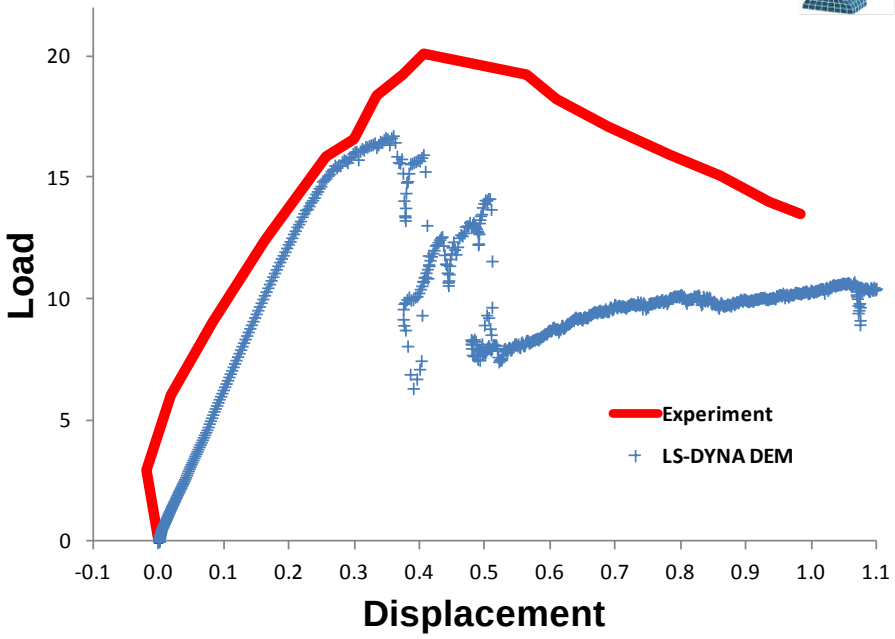
***DEFINE_DE_HBOND** bonds all parts.

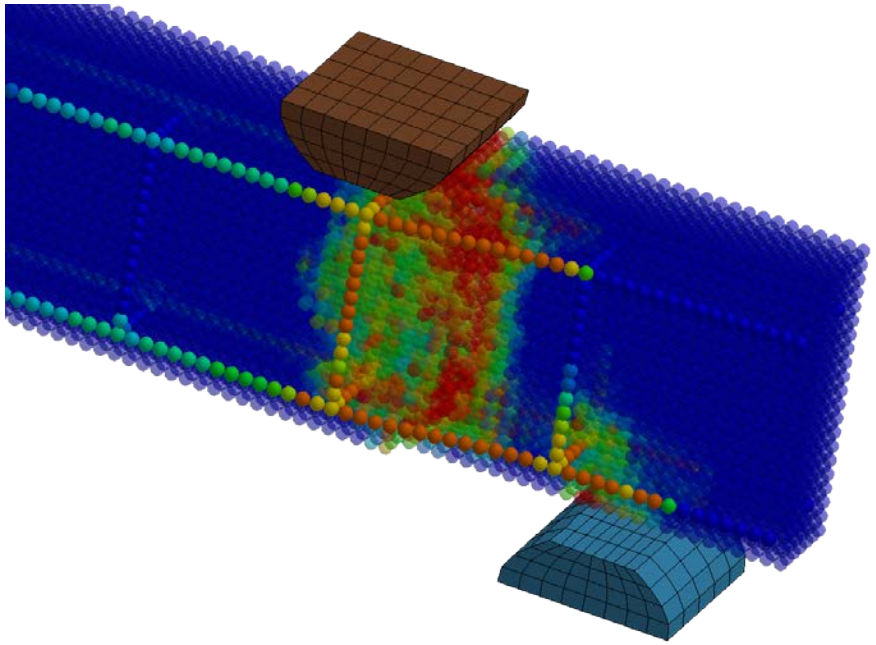
***INTERFACE_DE_HBOND** defines different de-bonding criteria between parts

**De-bonding process**

# Comparison between DEM & Experimental Results



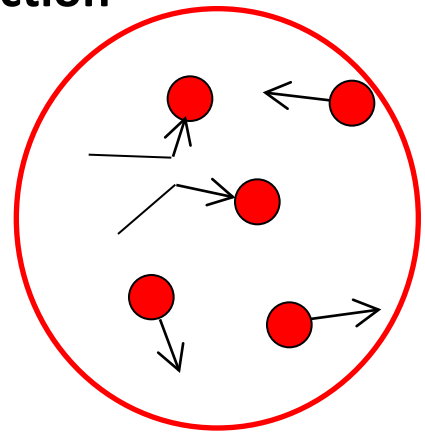


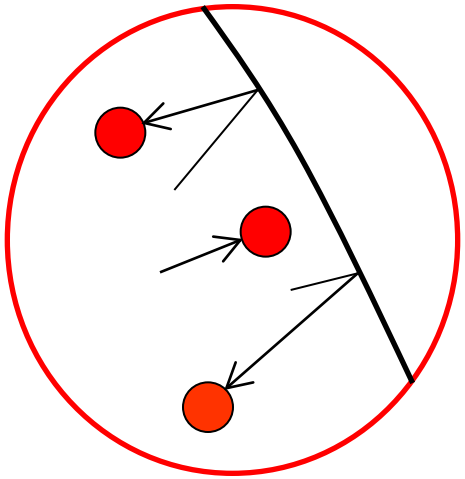



# Coupled Multi-Physics Solvers

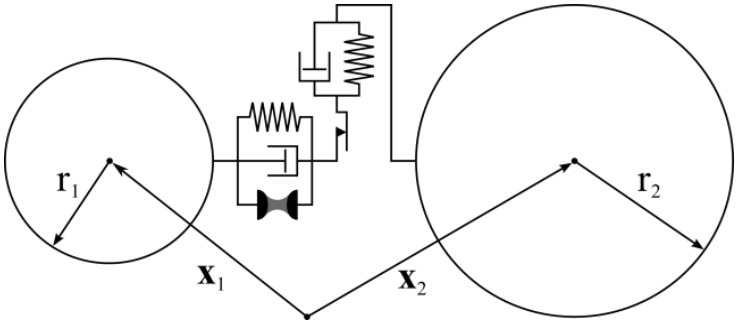
**Particle-particle interaction**

**Particle Gas**



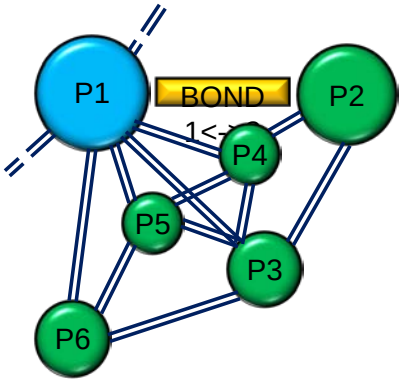


**DEM**



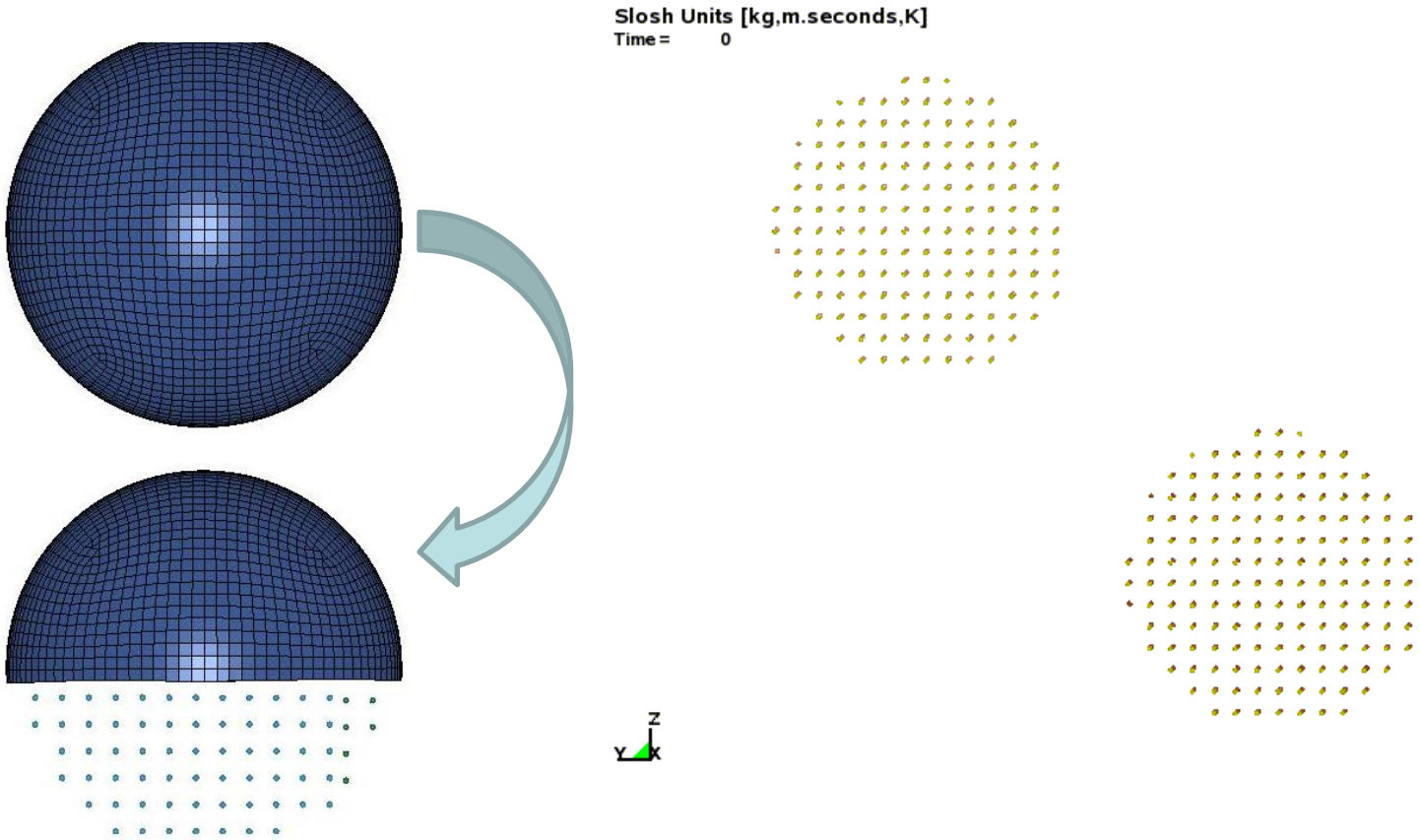

1) **Node to Node**
2) **Node to Beam**
3) **Node to Segment**
4) **Node to Volume**

**DEM bond**



# Node to Node Coupling

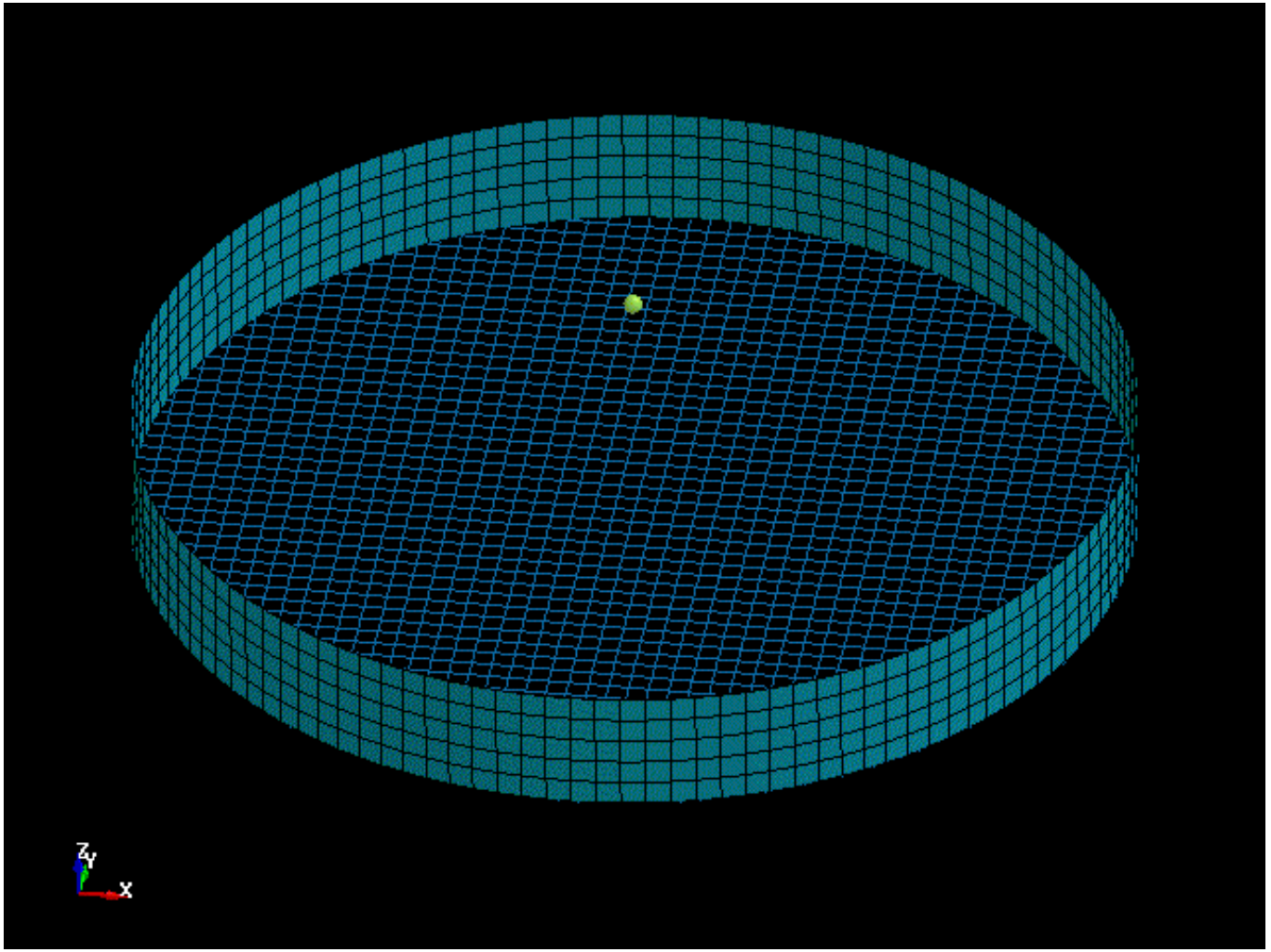
## SPH to SPH Contact




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

# Node to Beam Coupling

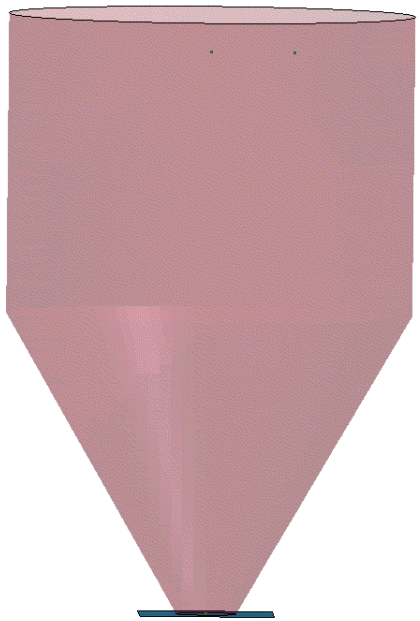
**DES to Beam Contact**



# Node to Surface Coupling

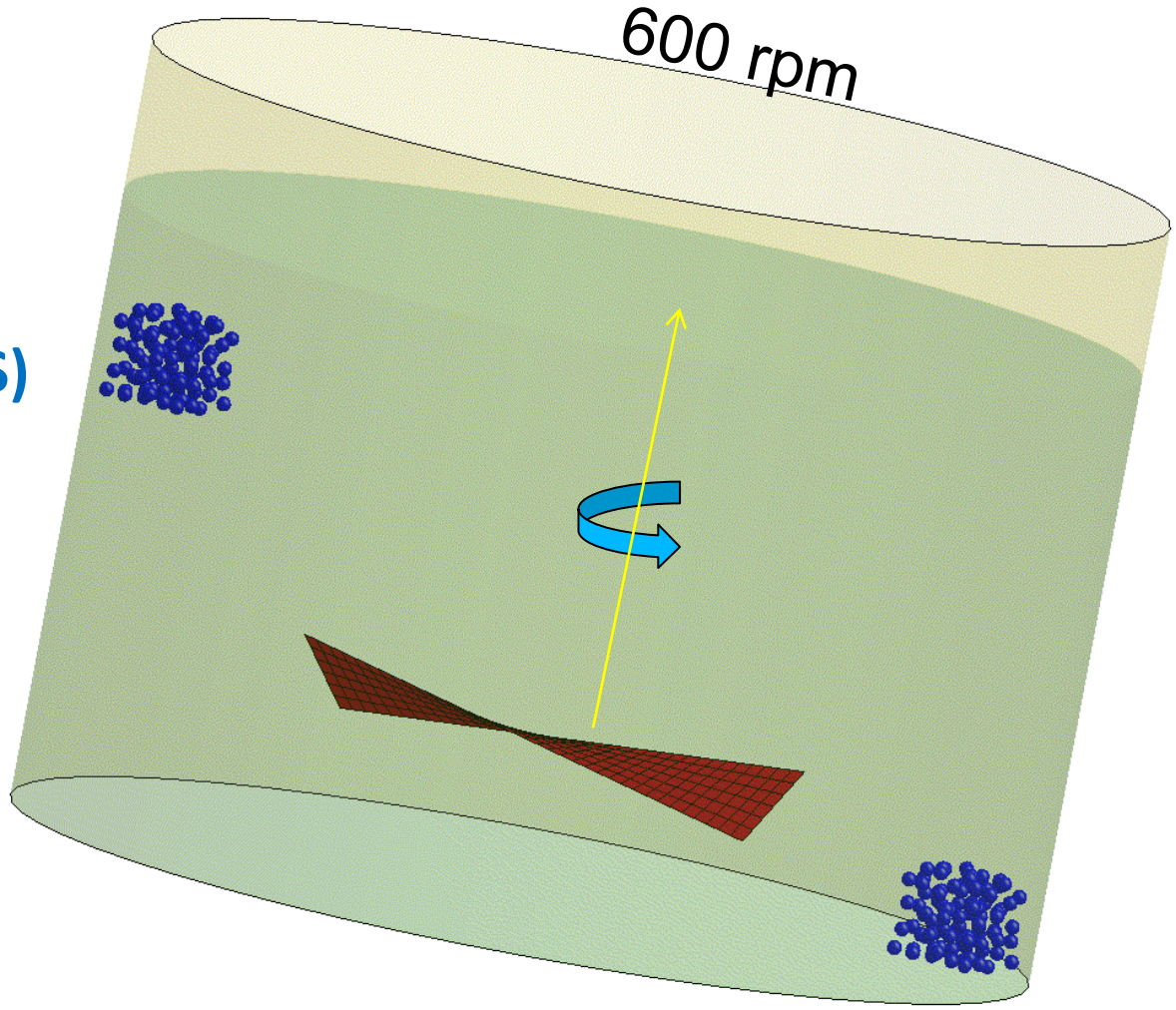
## DES to Segment Contact

Particles (kg-m-s)
Time = 0



# Node to Volume Coupling

**DES to ALE Contact**




**Powder(DES)**

**Water(ALE)**

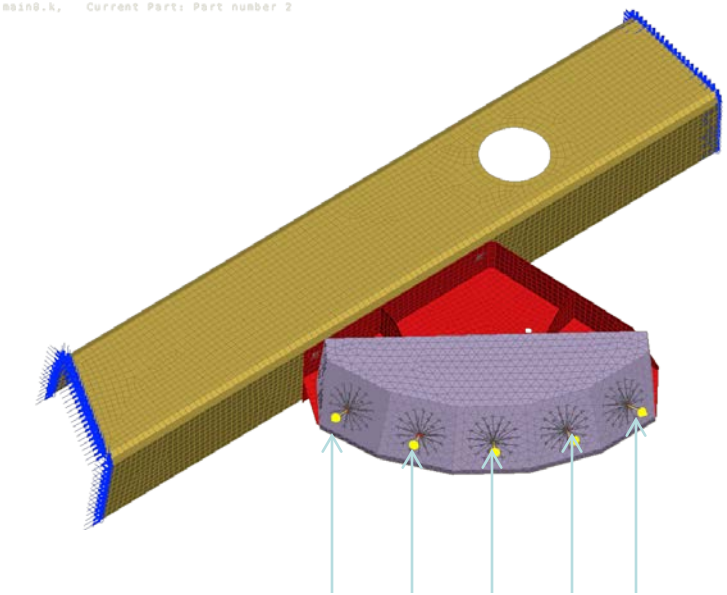
# Modal Dynamics

# Modal Dynamics

- Modal Dynamics in LS-DYNA has been enhanced in LS971 R71 with new features and keywords.
  - *CONTROL_IMPLICIT_MODAL_DYNAMICS
    - To activate modal dynamic analysis
  - *CONTROL_IMPLICIT_MODAL_DYNAMICS_MODE
    - To select a subset of modes to use in the analysis
  - *CONTROL_IMPLICIT_MODAL_DYNAMICS_DAMPING
    - To specify damping coefficients for modal damping.
- Good for model analysis
  - with multiple load cases
  - where the response is well captured with a small number of modes
  - linear behavior in the elements and materials

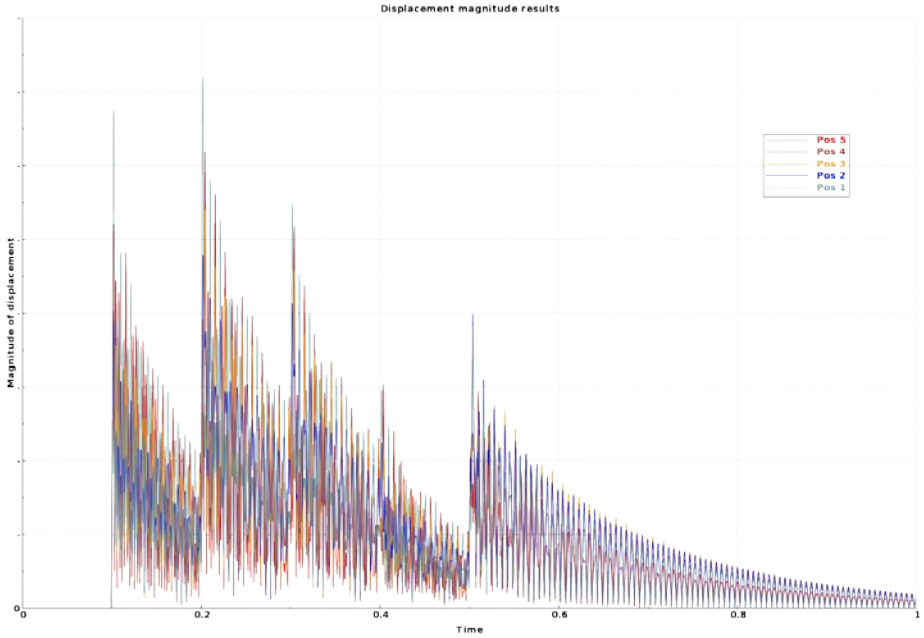
# Modal Dynamics Example

- Instrument Bracket from BAE Systems Hägglunds



5 Impulse forces

Response for 5 impulse forces



# Modal Dynamics Example

- An instrument bracket is subjected to impulsive loading.
- Force spikes are applied sequentially to five points.
- Response is computed using Transient modal dynamics.
- 30 eigenmodes extracted up to 1370 Hz.
- Constant modal damping used, 1 % of critical.
- 4 CPU cores used in all cases.
- Transient Explicit: 3h 39 min (single precision)
- Transient non-linear implicit: 4 h 26 min
- Transient modal dynamics:  3 min 33 sec  -- 60 Times faster!

# LS-TaSC

# LS-TaSC

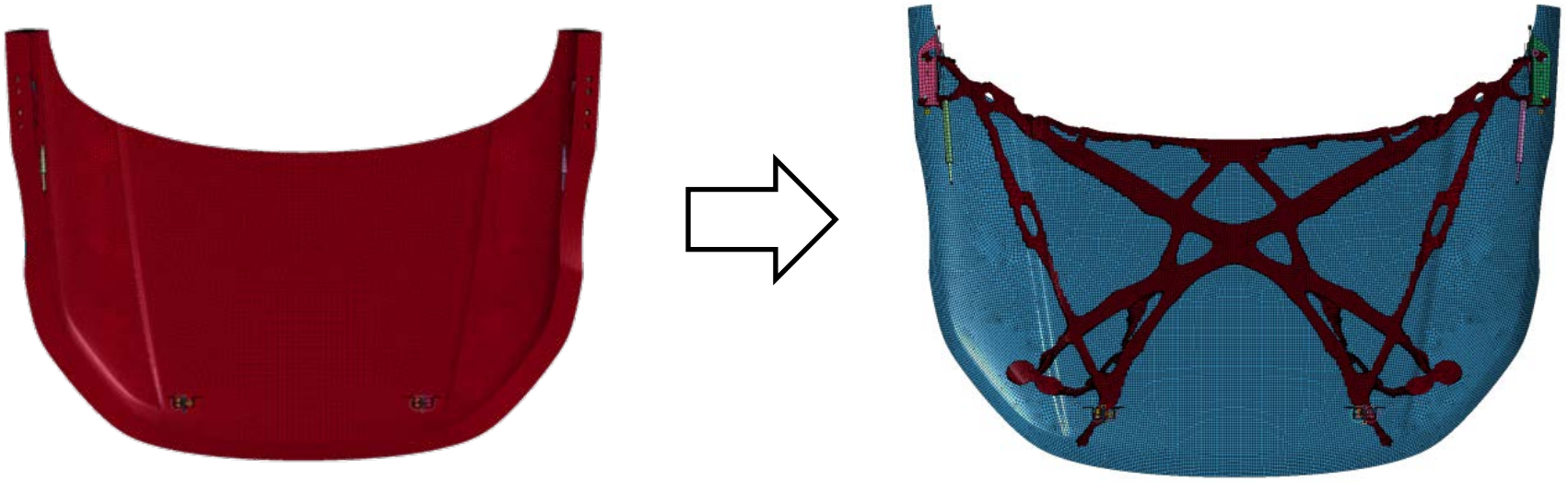
- Topology and Shape Computations using LS-DYNA.
- LS-TaSC goals:
- Solving for large, highly nonlinear problems. The methodology is valid for linear.
- Imbedded in the design process

# Version 3.0

- Major New Features
- Free surface design of solids
  - Geometry definitions
  - Automatic mesh smoothing
- Integration with LS-PrePost
- Several minor features.

# Topology design of a bonnet

Multi-criteria problem: One design for three nonlinear load cases, each with constraints.
During the design process, the algorithm will remove 90%-99% of the structure. Mesh refinement is therefore required.
Example courtesy of JRL.

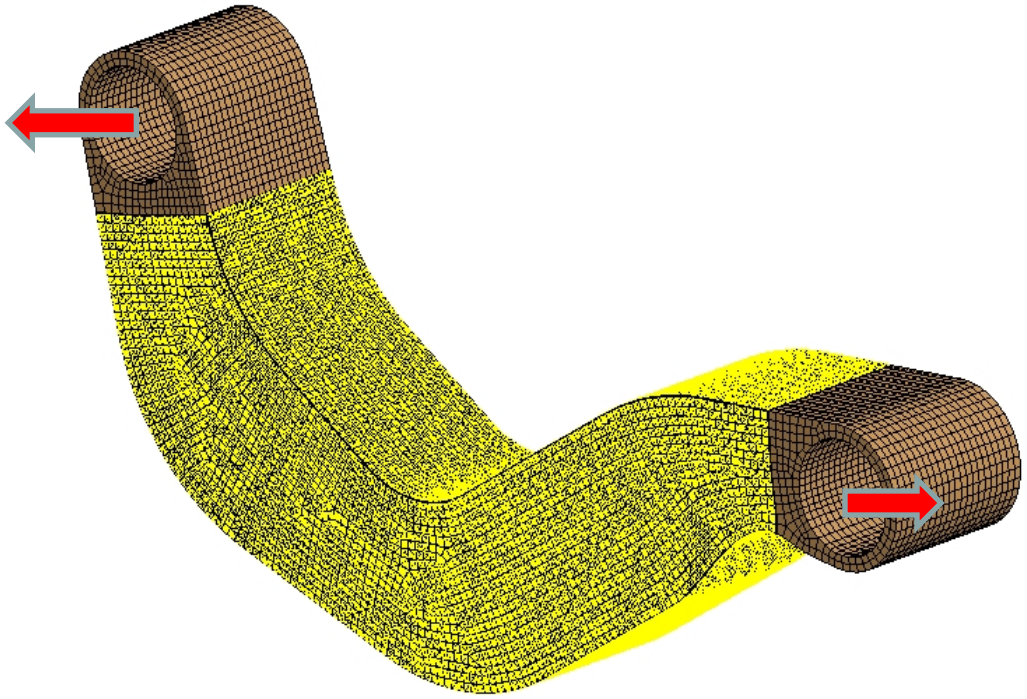


# Free surface design

- Redesign of the surface of solids to have a uniform stress (removing the stress concentrations).
- Geometry definitions are allowed.
- Very quick to set up the design problem.

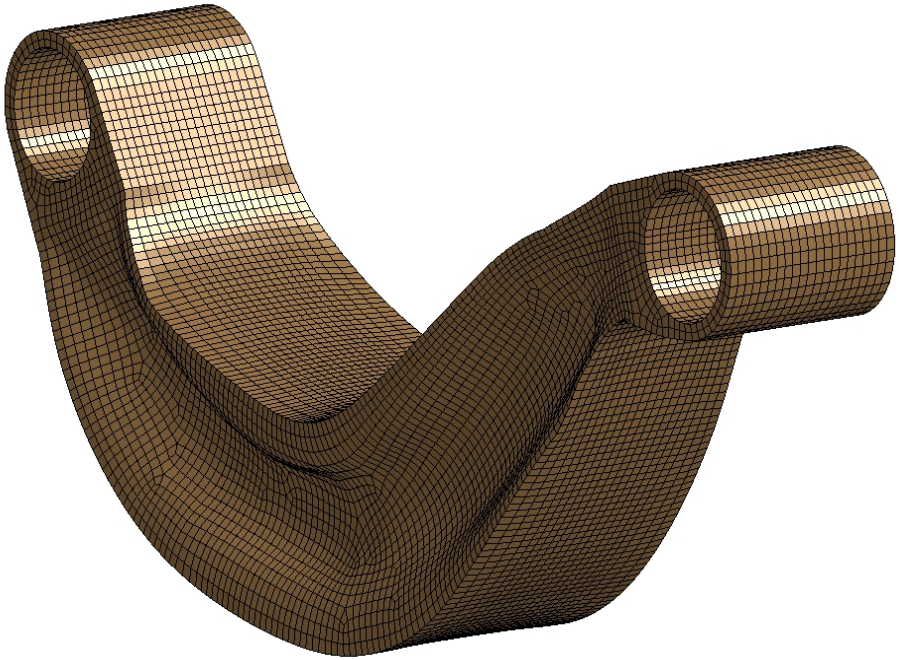
# Component surface design

- Component example with the yellow surface selected for redesign. Setting up a problem consists of defining a surface.

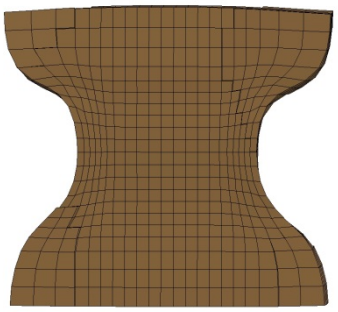


# Component shape design

- Design at iteration 16 shows the flanges that are created.

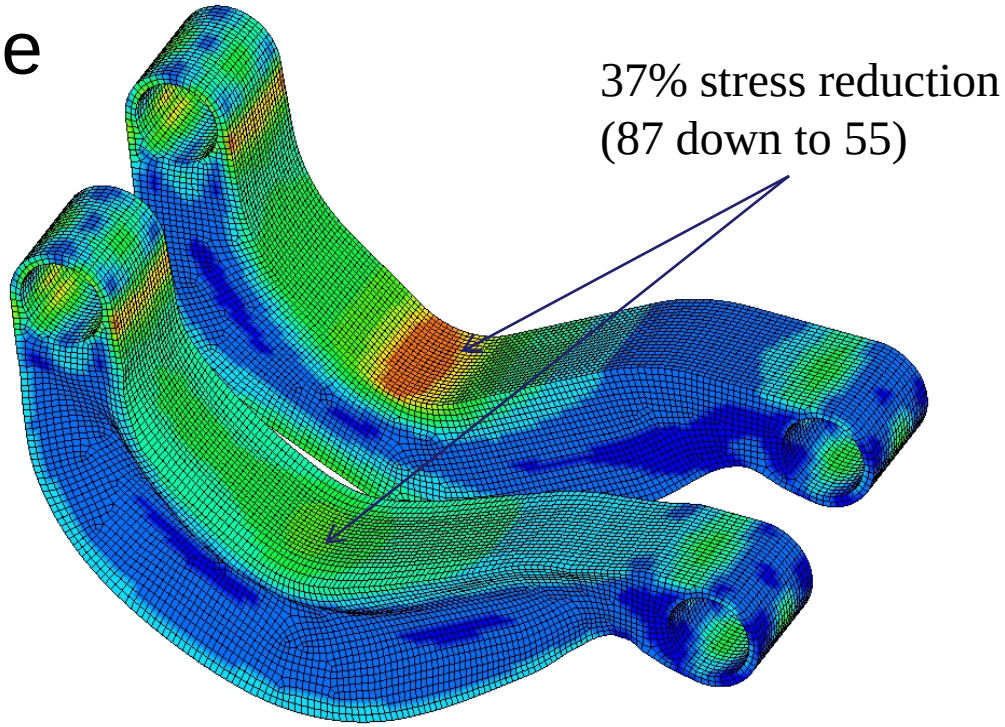


Cross-section

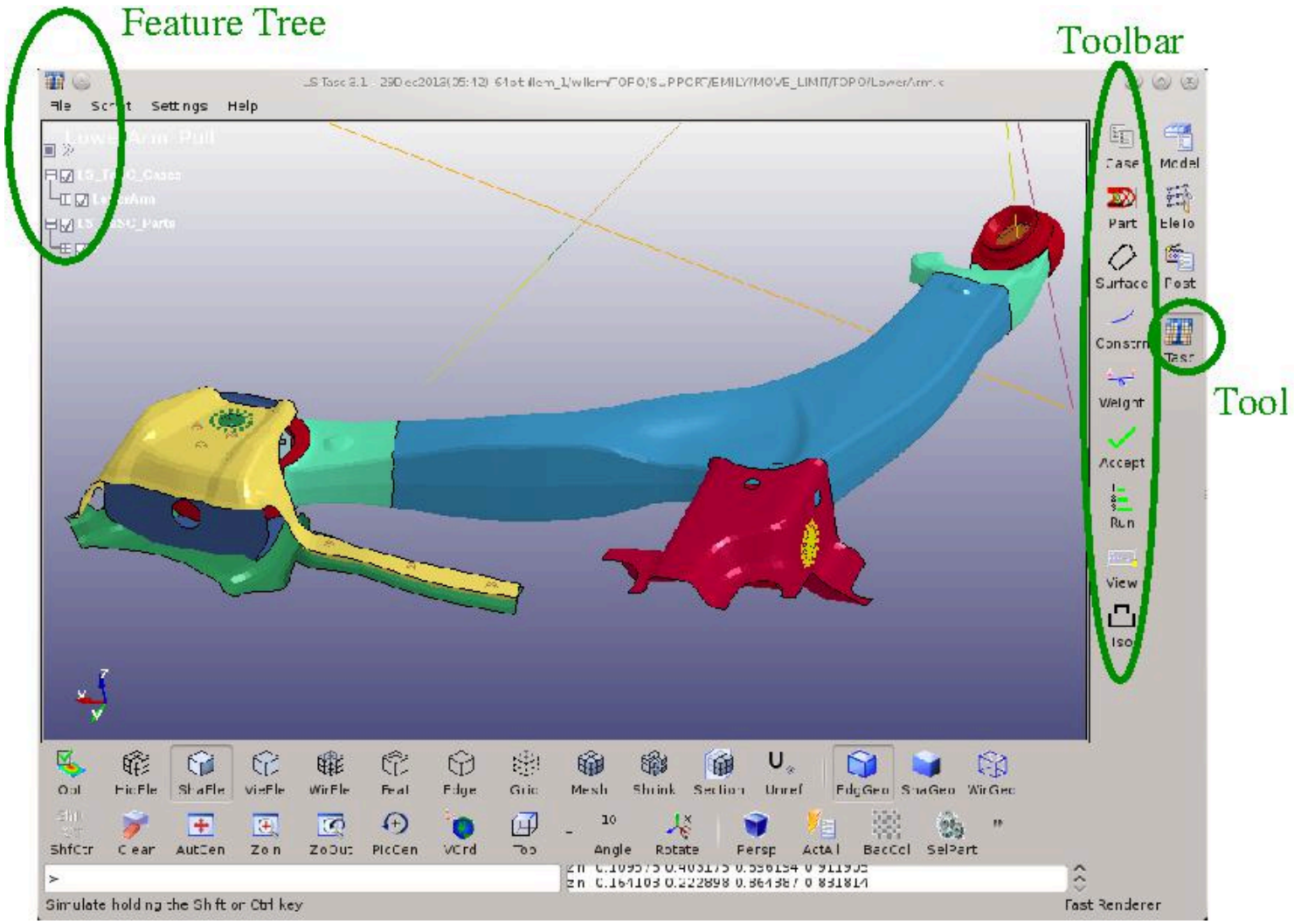


# Component shape design

- Comparing the Von Mises stress at iteration 16 with the original design shows a 37% improvement.



# LS-PrePost Integration



# LS-PrePost
# ICFD Capabilities

# 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.

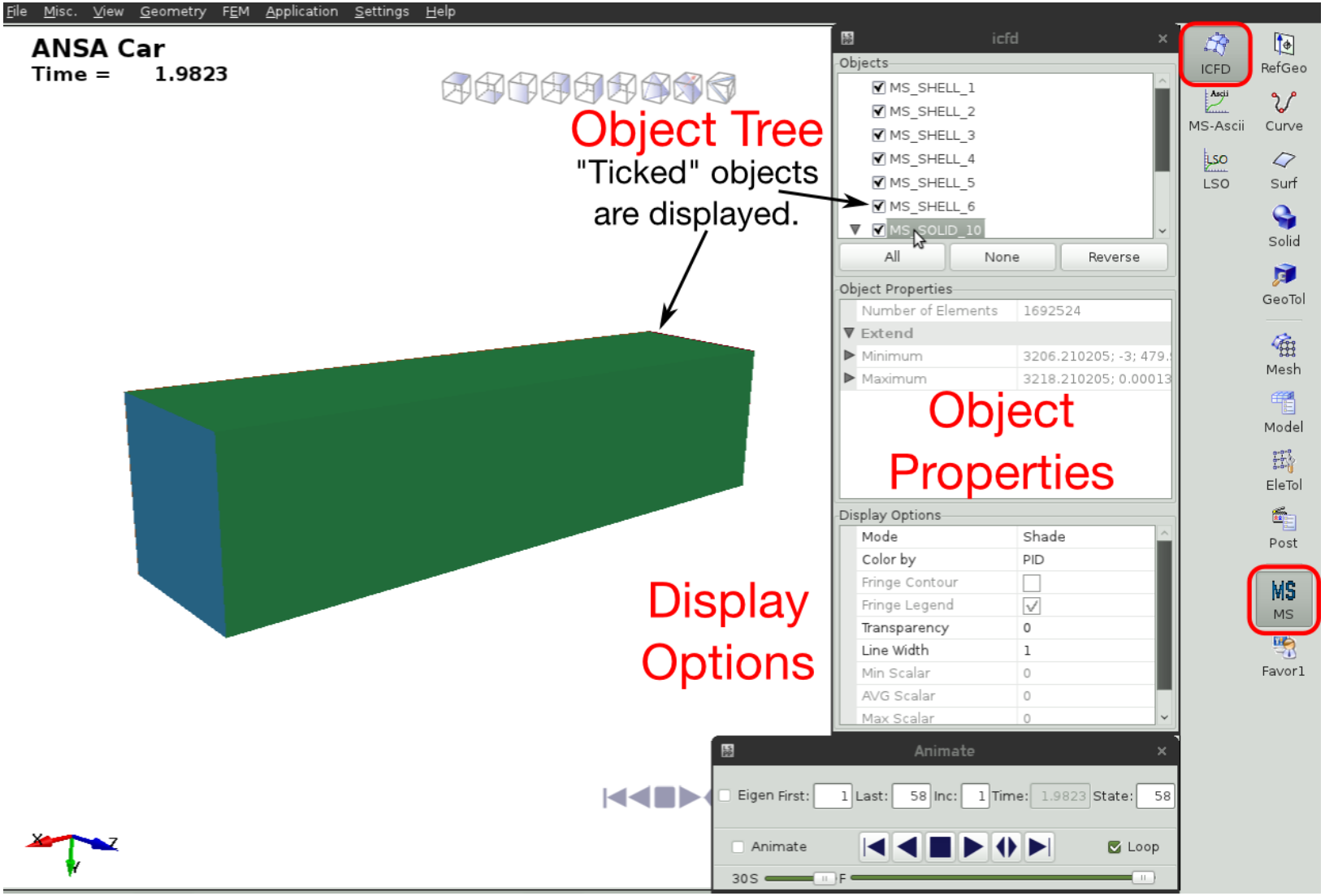
- Currently, LS-PrePost offers some tools in order to post treat the results from the ICFD solver based on its solid mechanic counterpart.

- 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.

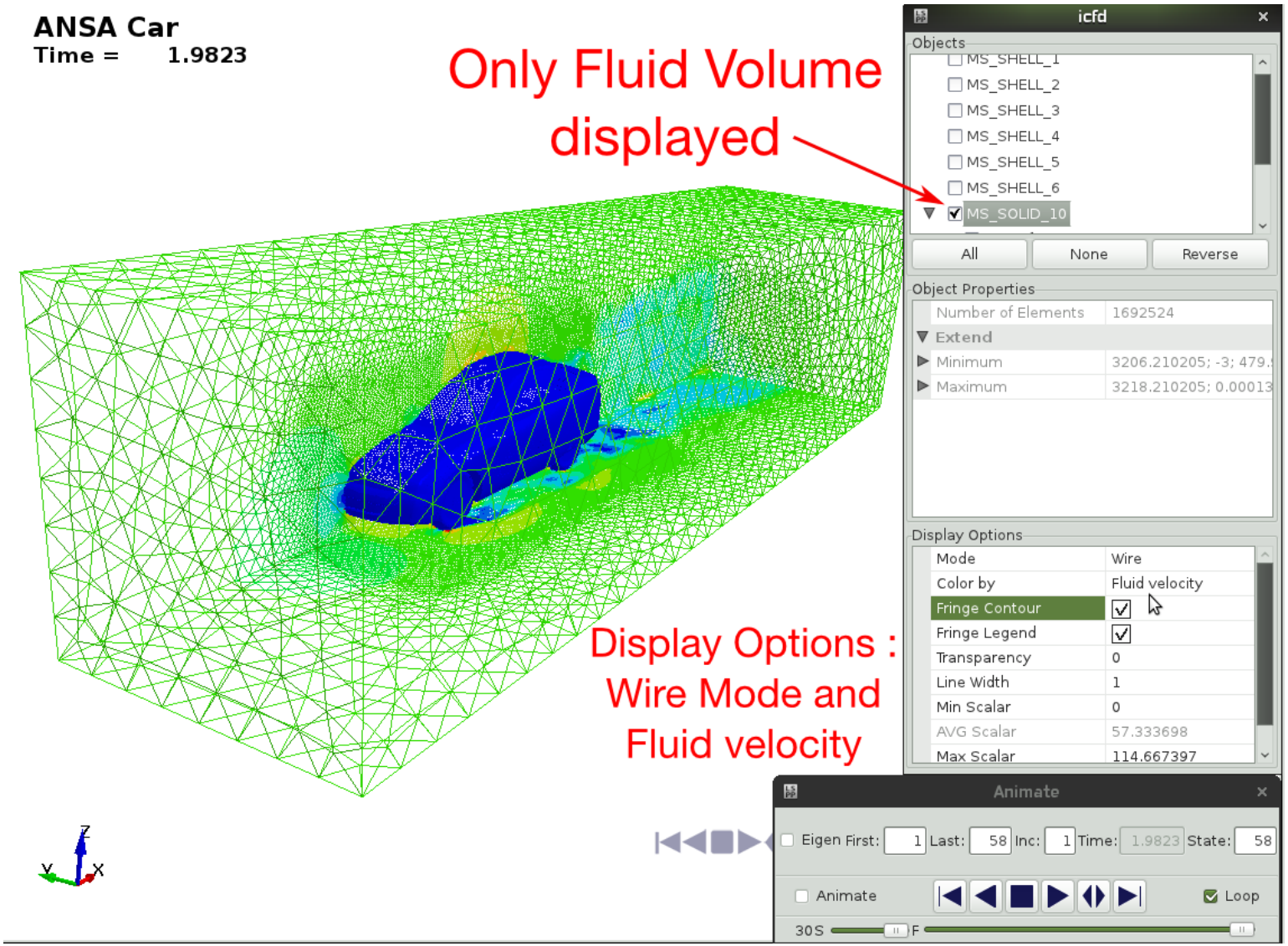
# ICFD Post Processor Introduction

## Object oriented structure:



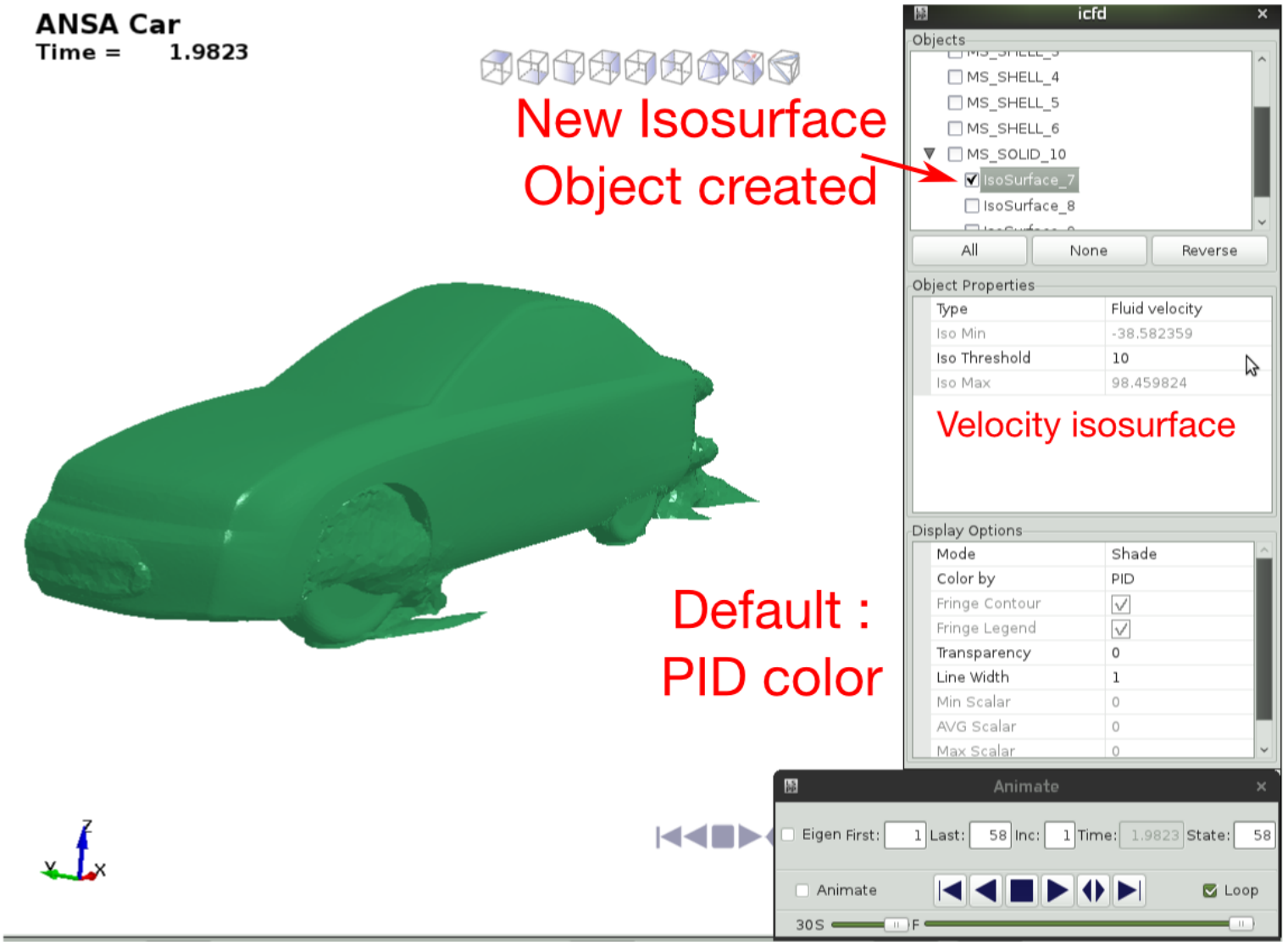
# ICFD Post Processor Introduction

## Object oriented structure:



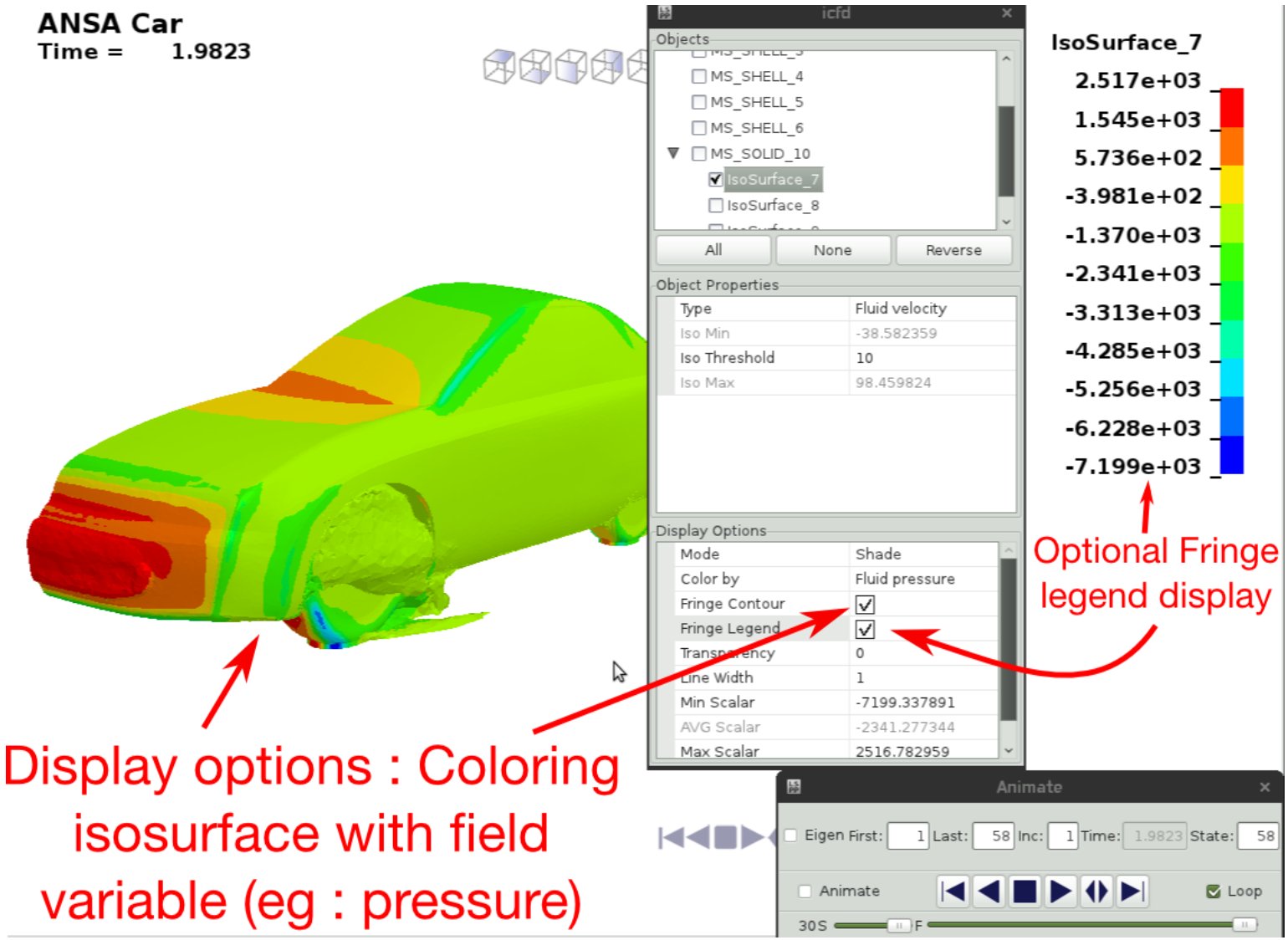
# ICFD Post Processor Introduction

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



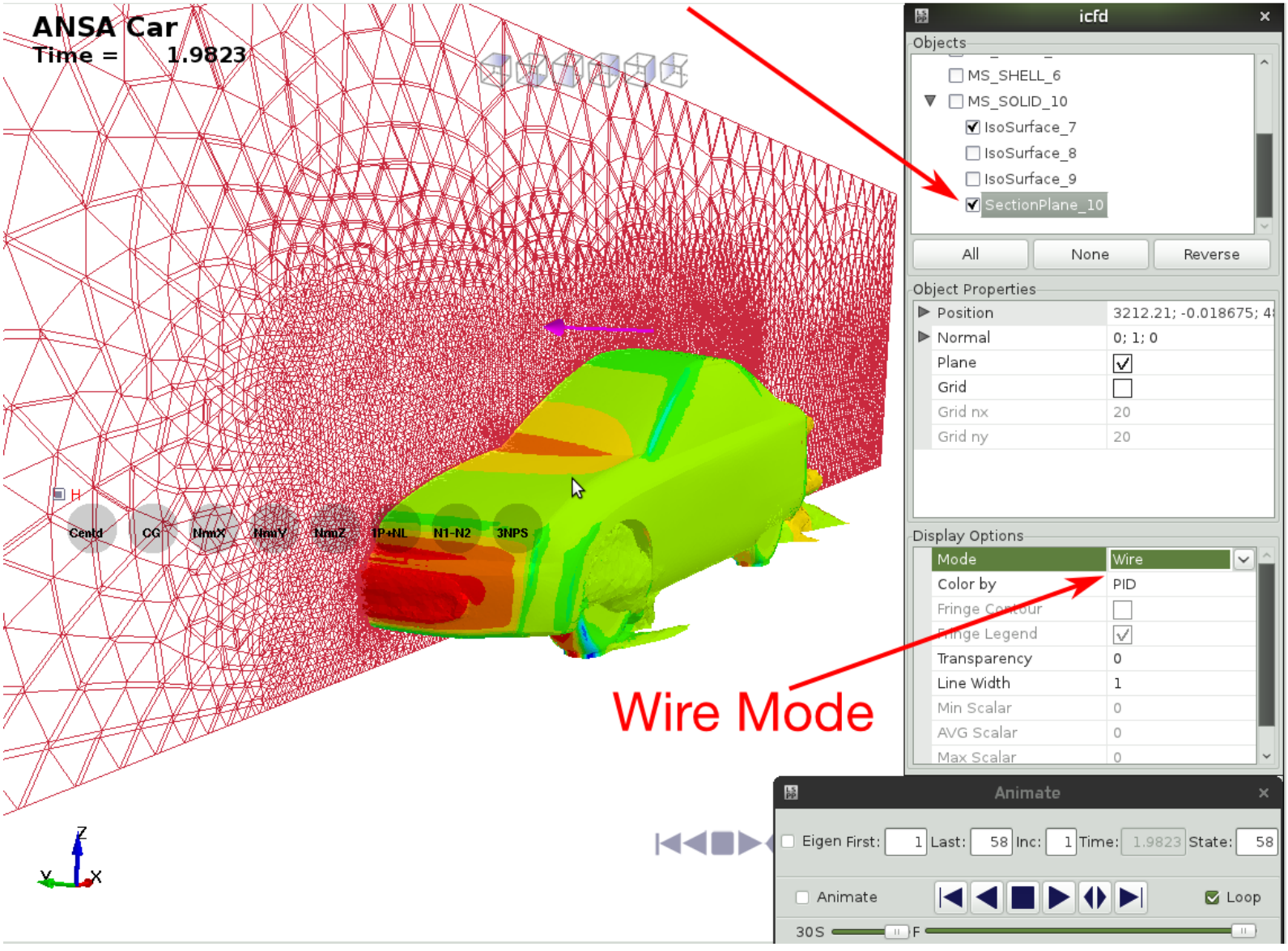
# ICFD Post Processor Introduction

## Modifying newly created object:

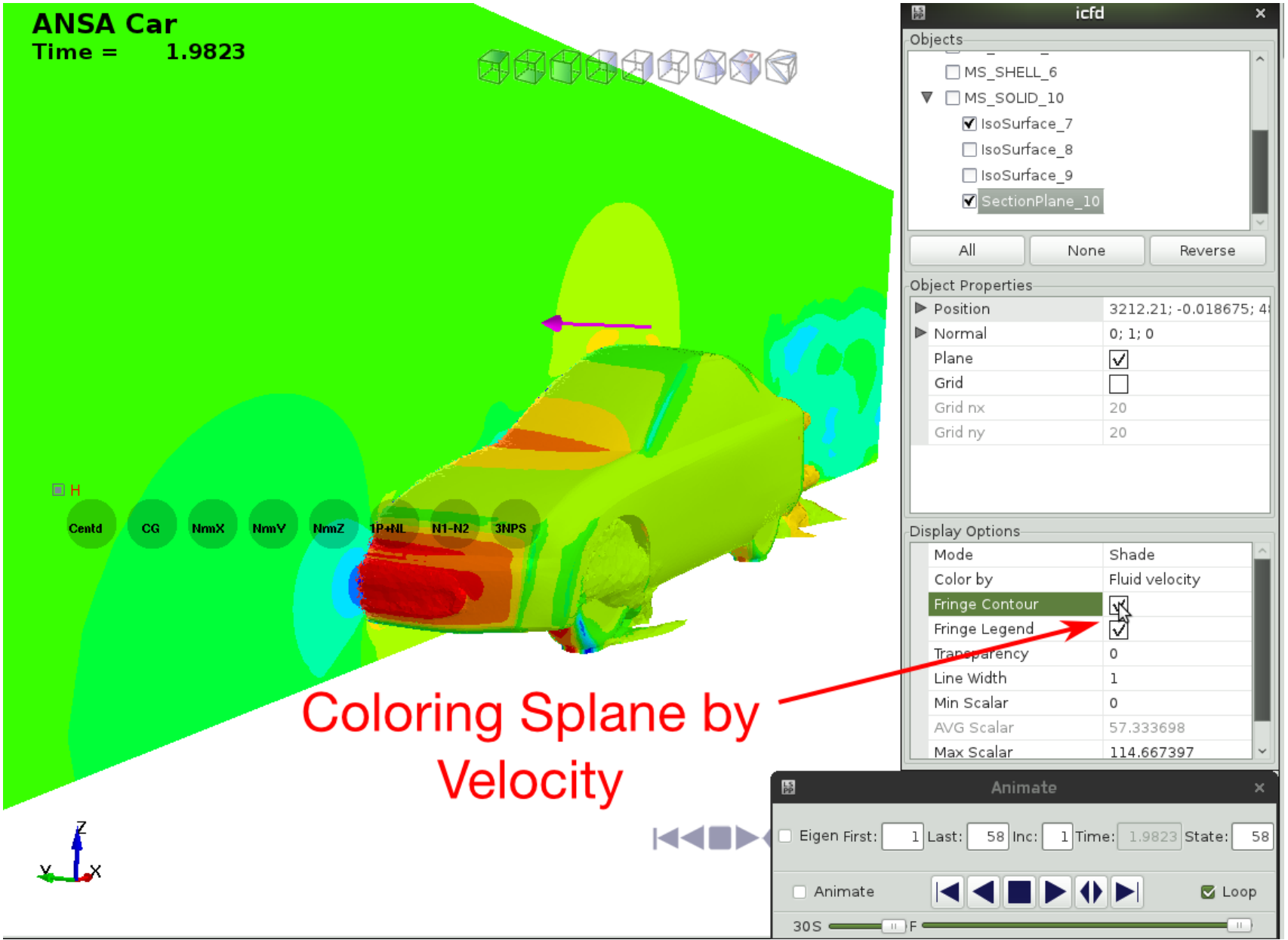


# ICFD Post Processor Introduction

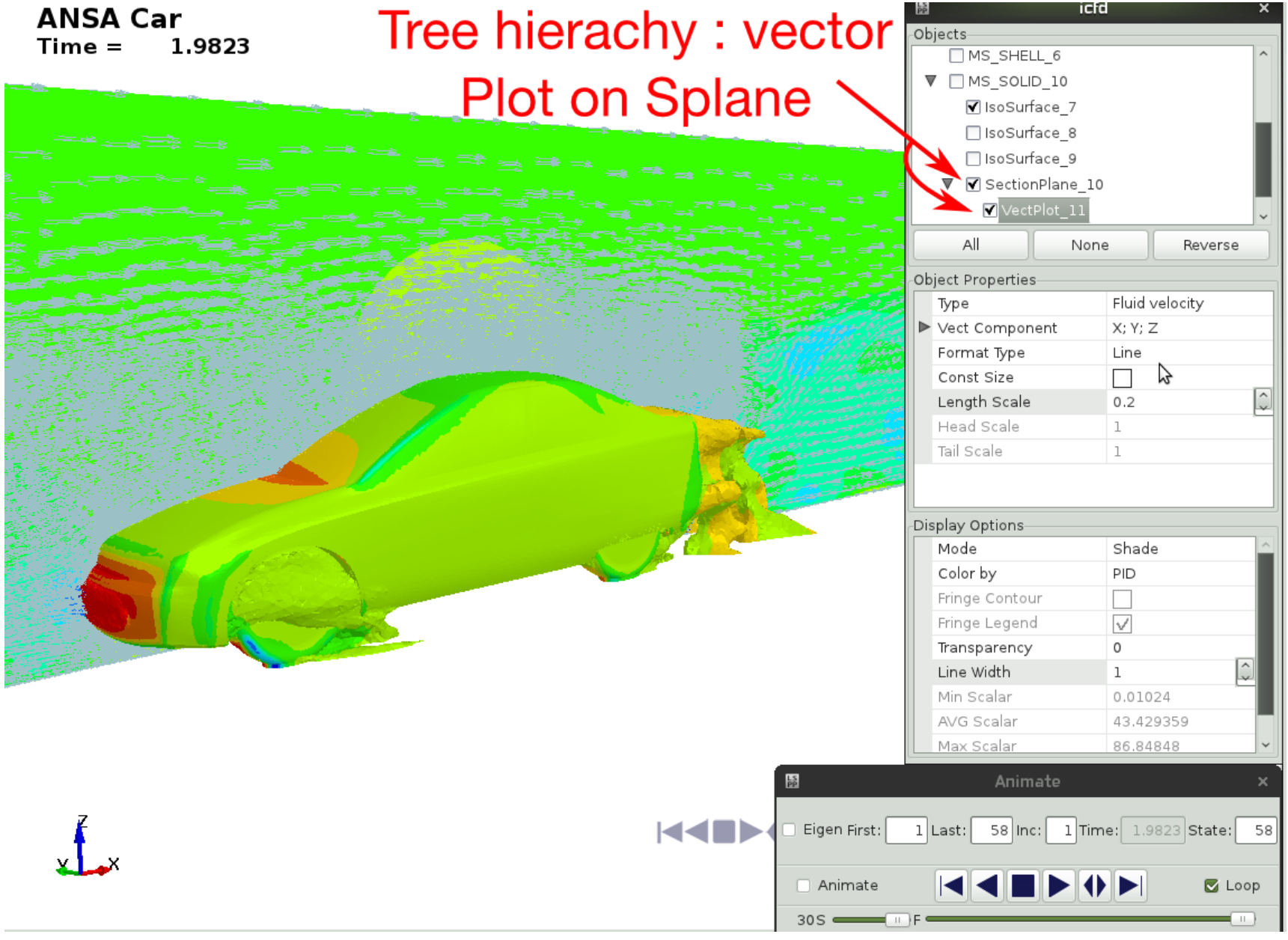
New object created : Splane



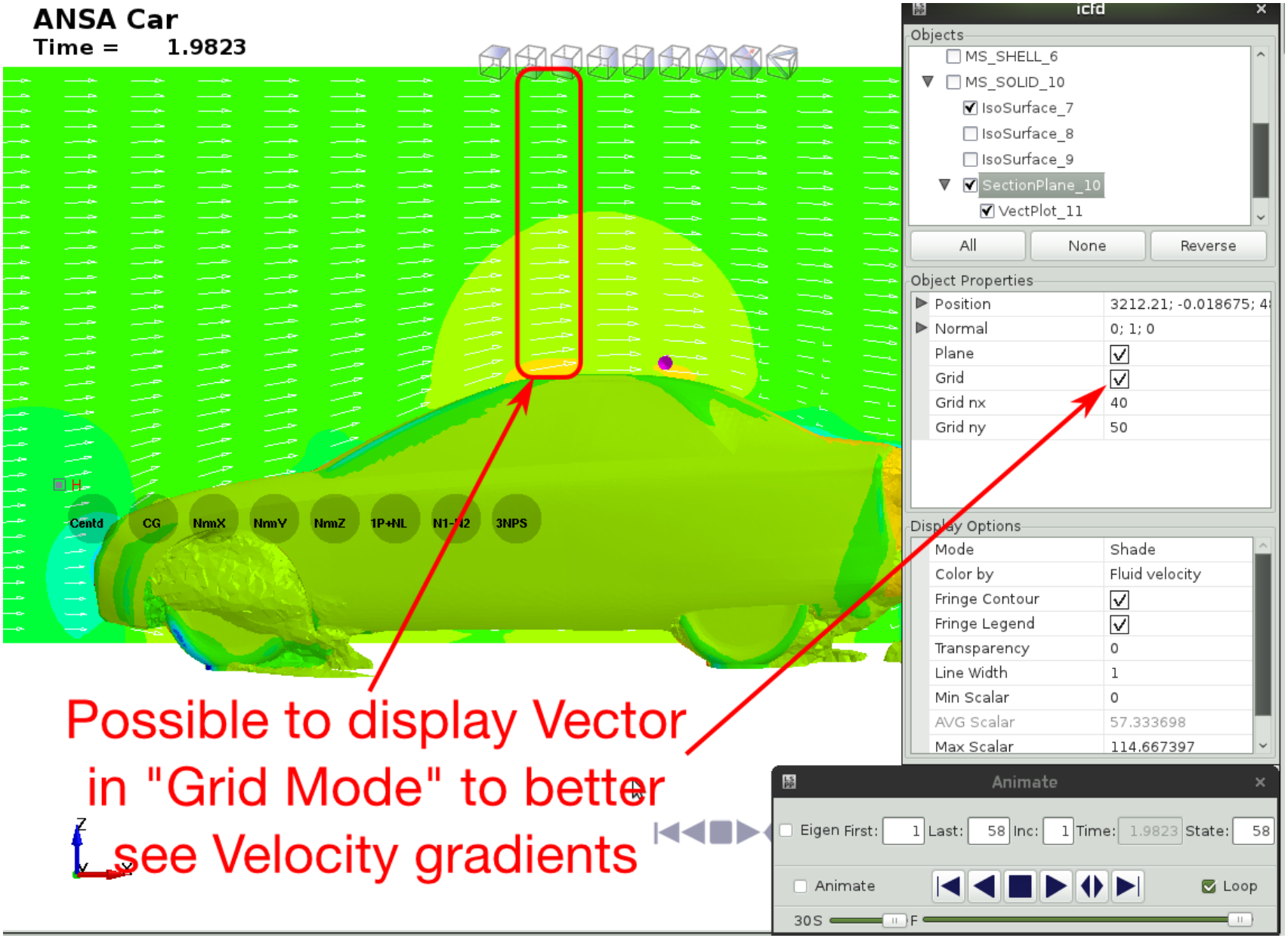
# ICFD Post Processor Introduction



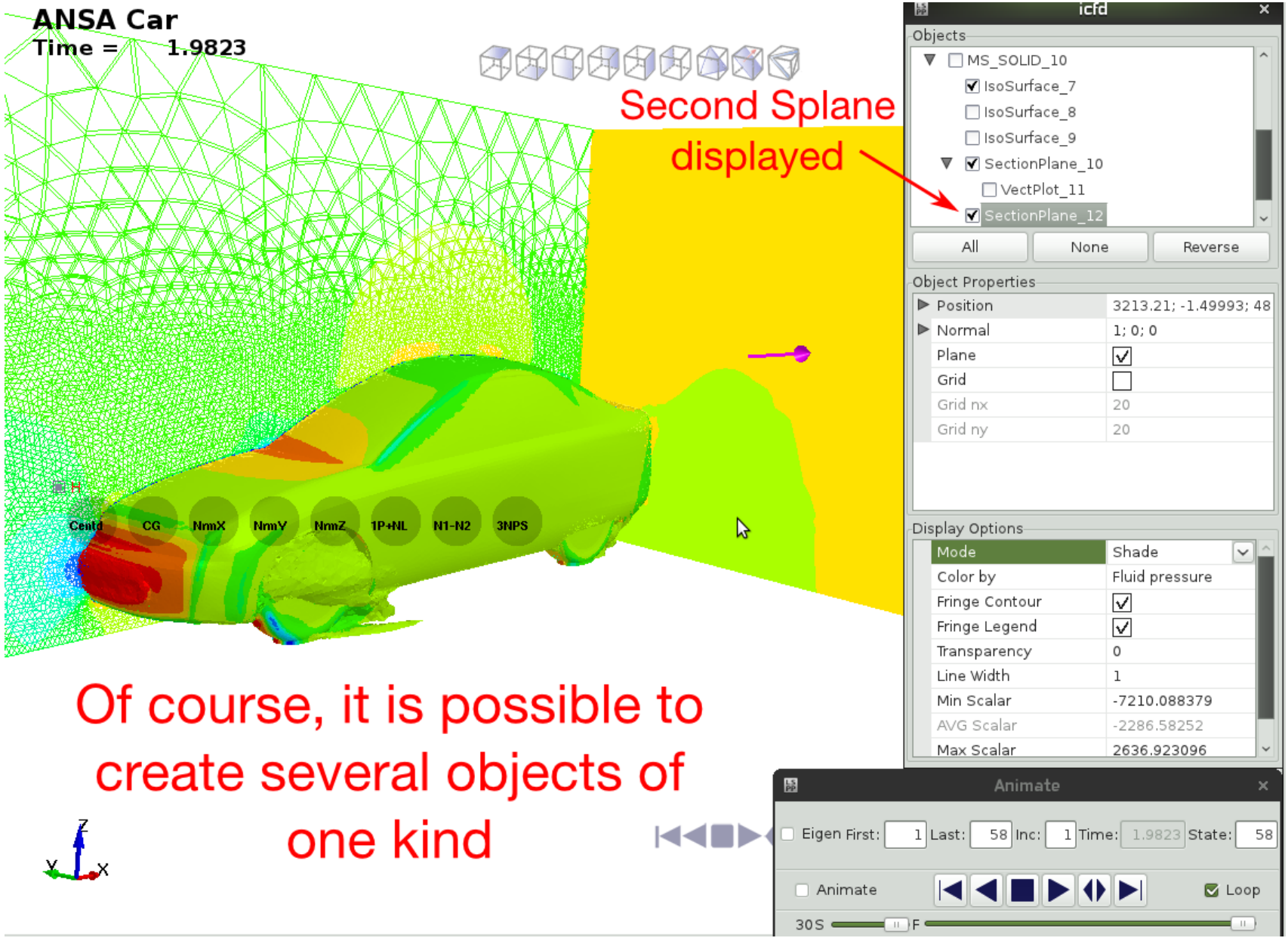
# ICFD Post Processor Introduction



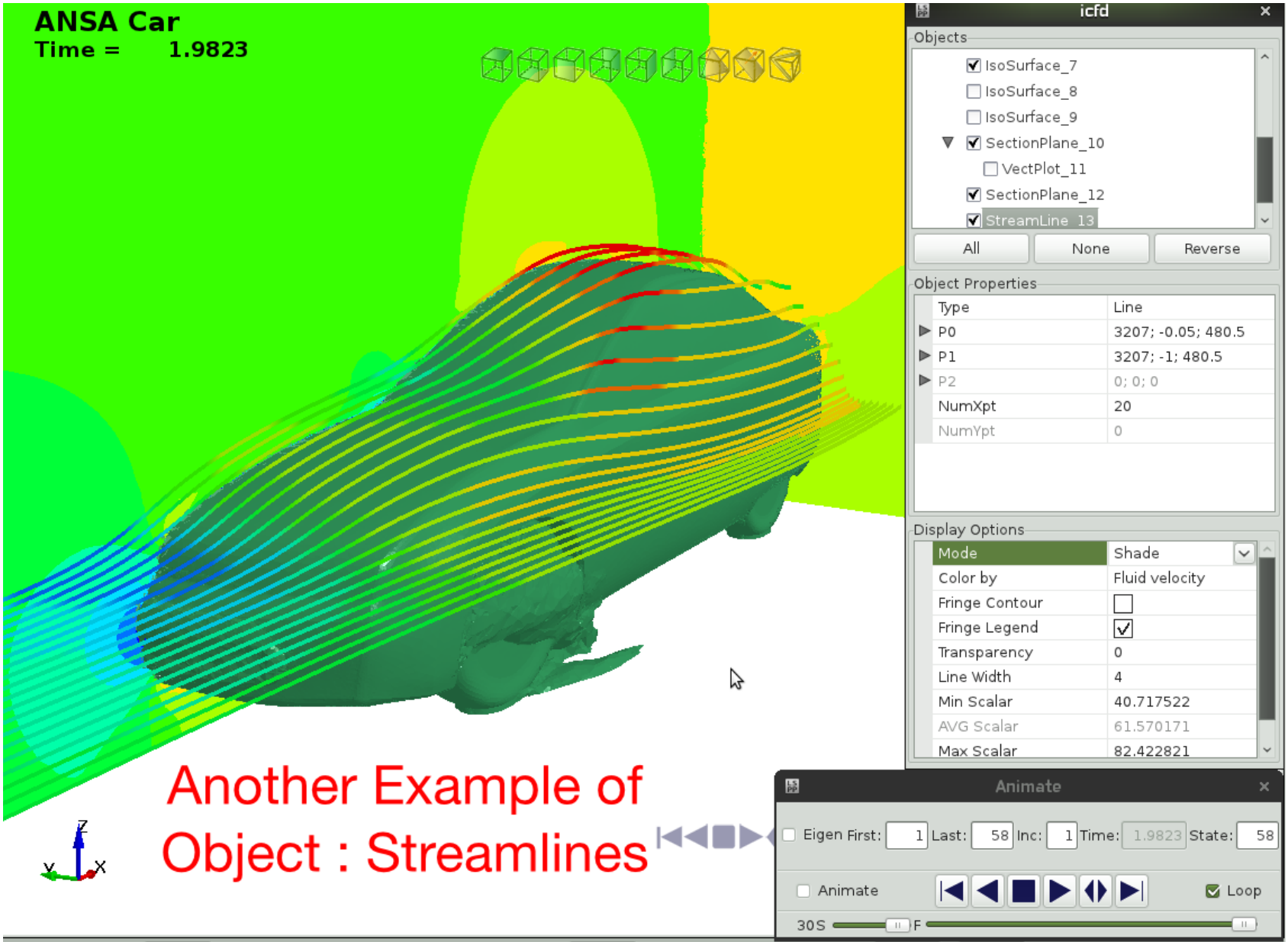
# ICFD Post Processor Introduction



# ICFD Post Processor Introduction

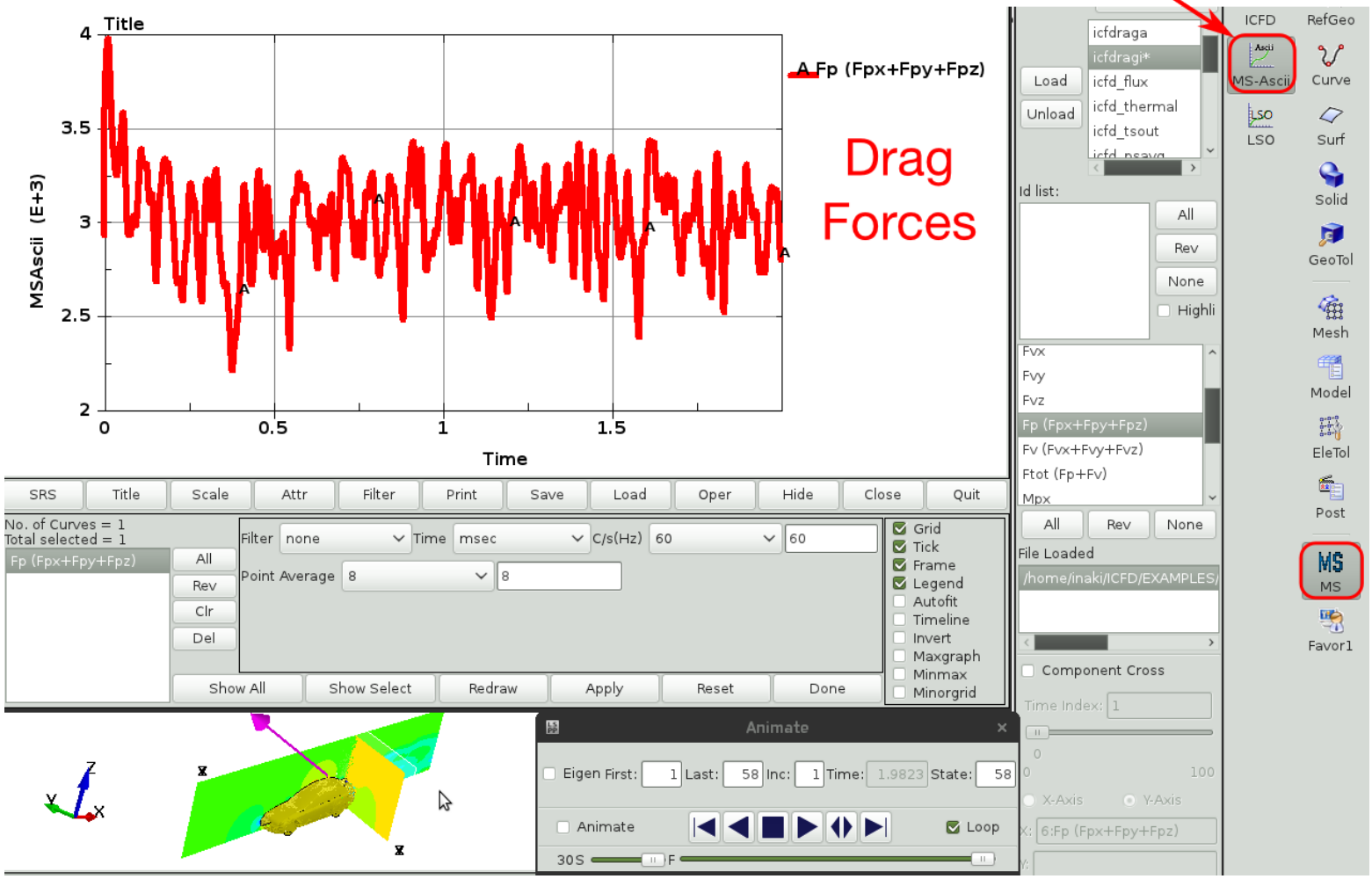


# ICFD Post Processor Introduction



# 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)



# 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.

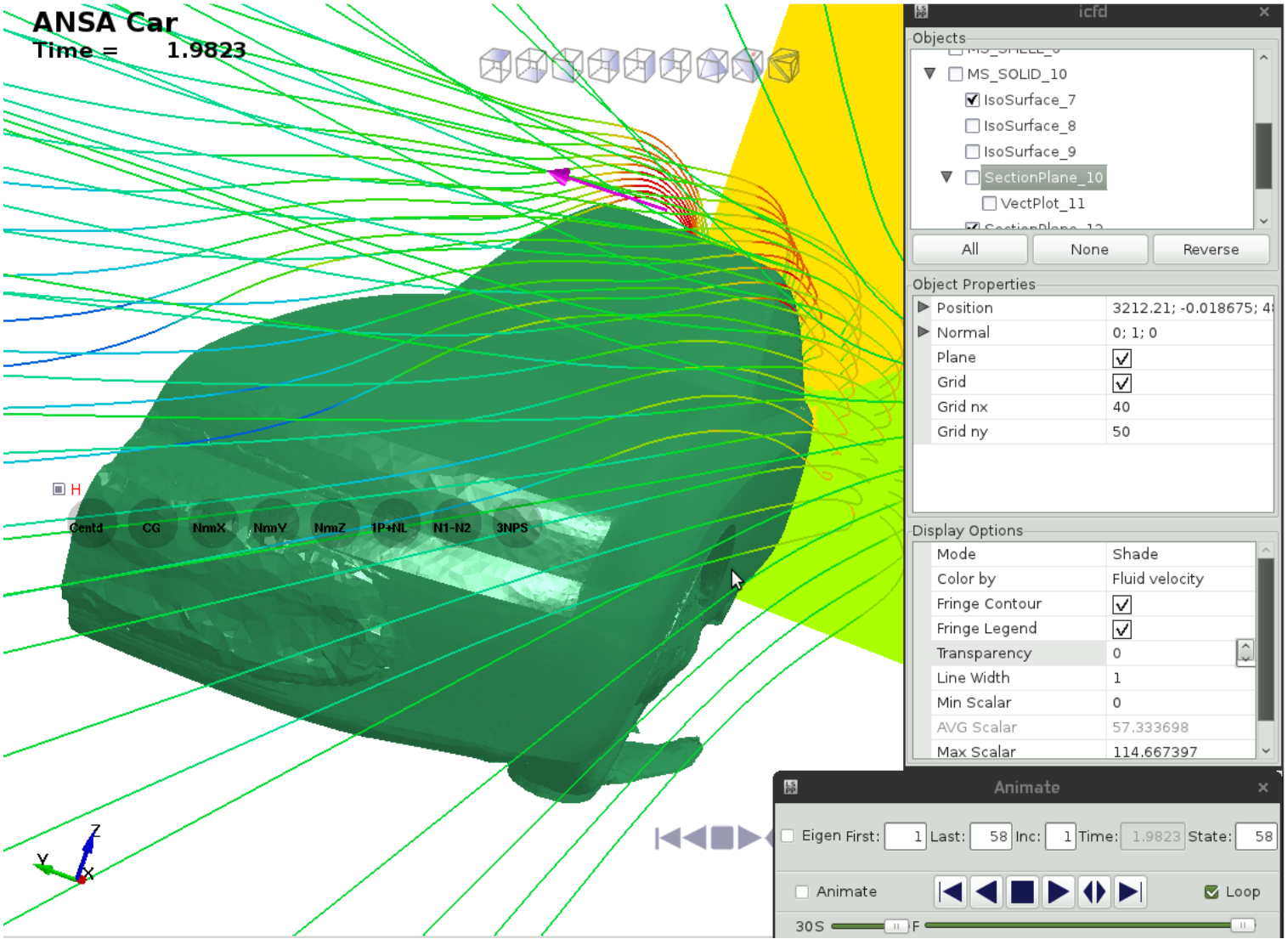
- More object types may be implemented in the future. A project is underway in collaboration with the University of San Diego in order to offer further advanced tools for qualitative post treating of the results.

- 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 alpha stage but can already be used for users eager to beta test its current functionalities.

# ICFD Post Processor Introduction

## Thank you for your attention !



# 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
  - New developments:
    - Combined implicit and explicit running together
    - Linear analysis combined with h-adaptivity

# 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
  - Discrete element methodology for modeling granular materials
  - Simulation based airbag folding and THUMS dummy positioning underway
- Multiscale capabilities are under development
  - Implementation underway (New approach which is more user friendly)
- Hybrid MPI/OPENMP developments are showing significant advantages at high number of processors for both explicit and implicit solutions

Thank You !