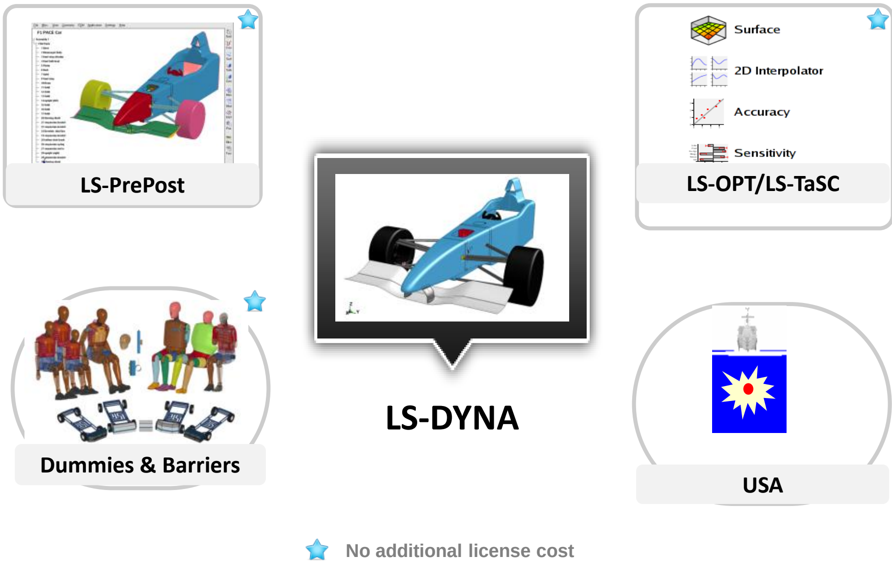
Recent Developments in LS-DYNA

LS-DYNA User Forum
September 25, 2013

LSTC
Livermore Software
Technology Corp.

Introduction

LSTC Products



LS-DYNA Application Areas

Development costs are spread across many industries

Automotive
- Crash and safety
- NVH
- Durability

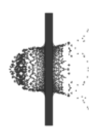
Aerospace
- Bird strike
- Containment
- Crash

Manufacturing
- Stamping
- Forging

Consumer Products

Structural
- Earthquake safety
- Concrete structures
- Homeland security

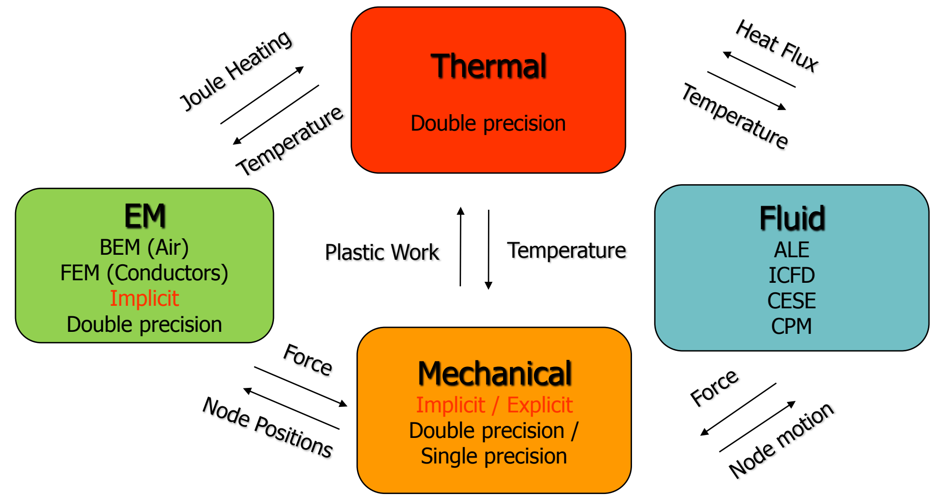
Electronics
- Drop analysis
- Package analysis
- Thermal



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

Accommodates Strongly Coupled Simulations

Multiple field equations are strongly coupled



One Code for Multiple Solutions

One Model

+

LS-DYNA

Multi-physics
- Structure + Fluid, ..

Multi-stage
- Implicit + Explicit ....

Multi-formulations
- linear + nonlinear +

Many Results

Manufacturing, Durability, NVH, Crash

Ongoing Developments

Ongoing Developments

1) SPH
2) Discrete Element Method (DEM)
3) LS-PrePost
4) Coupled Multi-physics Solvers

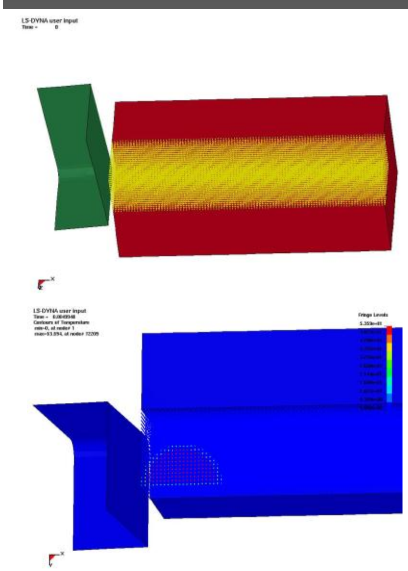
SPH Thermal Solver

- An explicit thermal conduction solver is implemented for SPH analysis.
- Following keywords and materials are supported:

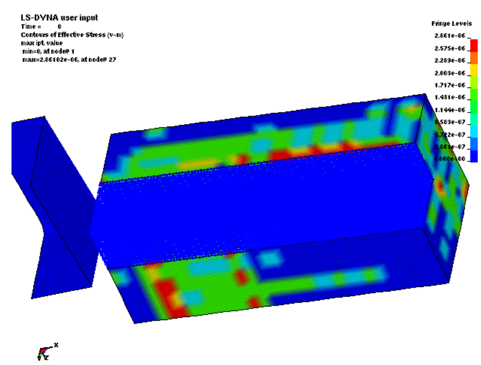
*INITIAL_TEMPERATURE_OPTION
*BOUNDARY_TEMPERATURE_OPTION
*BOUNDARY_FLUX_OPTION

*MAT_THERMAL_ISOTROPIC
*MAT_ADD_THERMAL_EXPANSION
*MAT_VISCOELASTIC_THERMAL
*MAT_ELASTIC_VISCOPLASTIC_THERMAL
*MAT_ELASTIC_PLASTIC_THERMAL

Metal Cutting with Heat

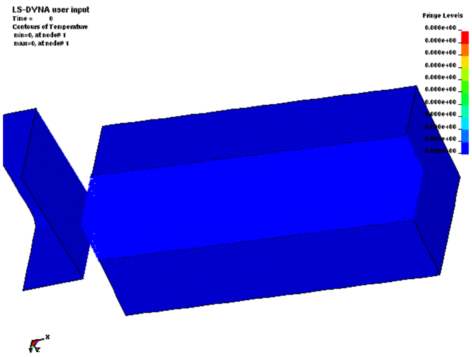


Setup

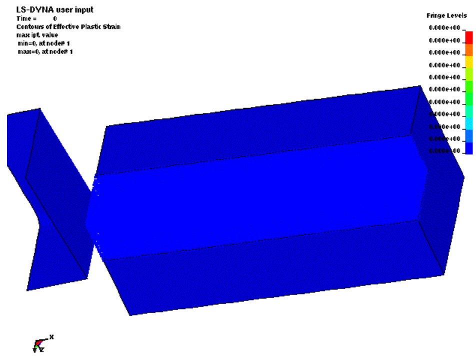


Contour plot of Von-Misses stress

Metal Cutting with Heat



Contour plot of Temperature



Contour plot of Plastic strain

Heat source: *BOUNDARY_FLUX
*MAT_JOHNSON_COOK (stress flow depends on the temperature)

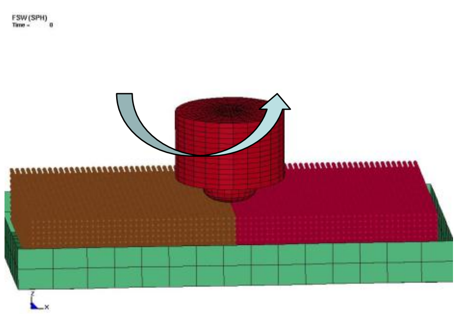
Friction Stir Welding with SPH

Tools: Rigid body

Working pieces: Johnson_cook material with viscoplasticity.
heat capacity=875 thermal conductivity=175

EQHEAT=1.0. FWORK=1.0
for heat source

ADD_THERMAL_EXPANSION
applied for the working pieces



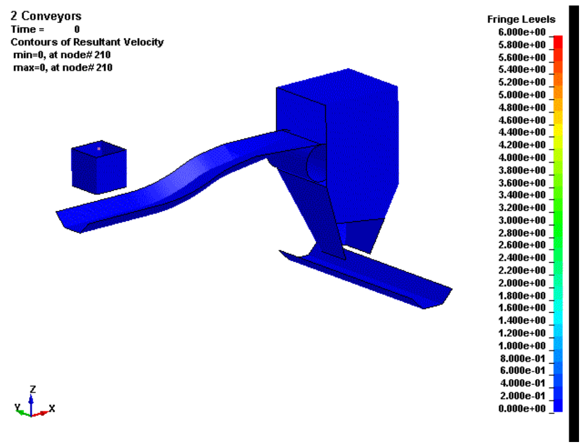
Courtesy Kirk A. Fraser at ROCHE

Discrete Element Sphere (DES)

**DEFINE_DE_INJECTION*

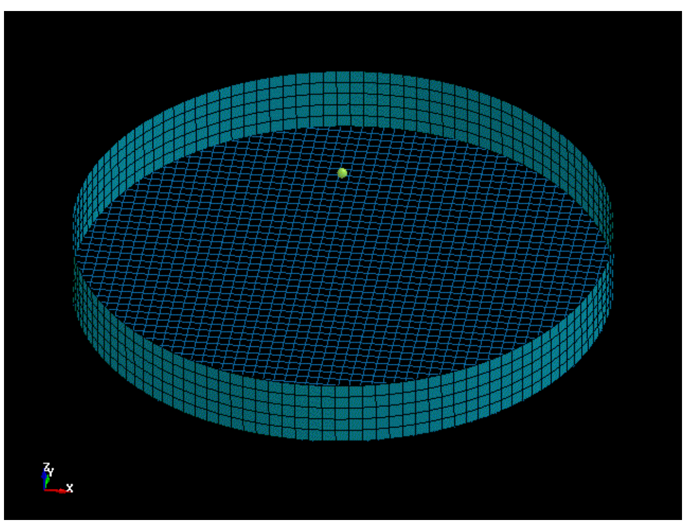
**DEFINE_DE_TO_SURFACE_COUPLING*

- Source of DES
- Generate traction force to simulate conveyor belt



Discrete Element Sphere (DES)

**DEFINE_DE_TO_BEAM_COUPLING*

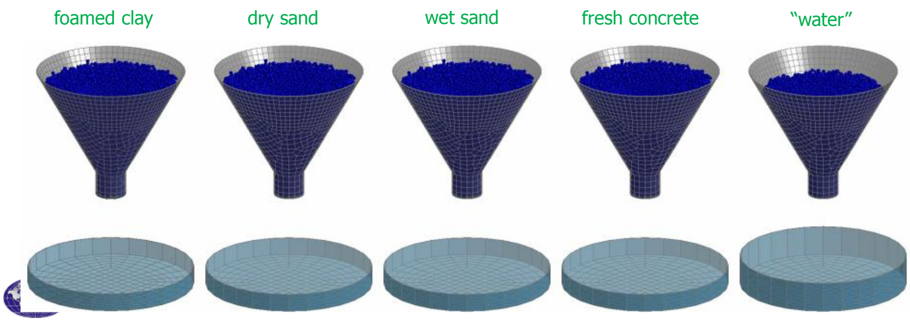


DEM– Funnel Flow

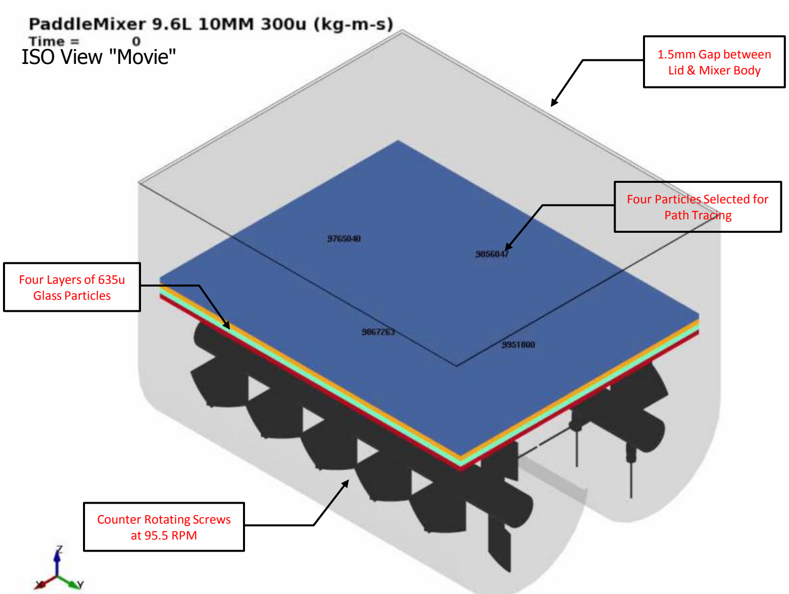
Variation of the parameters

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
```



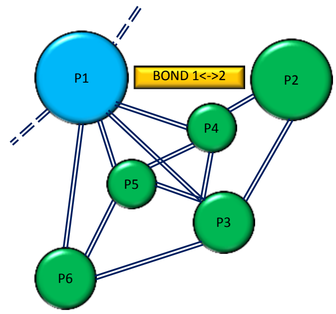
DEM for Paddle Mixer



DES Bond Model

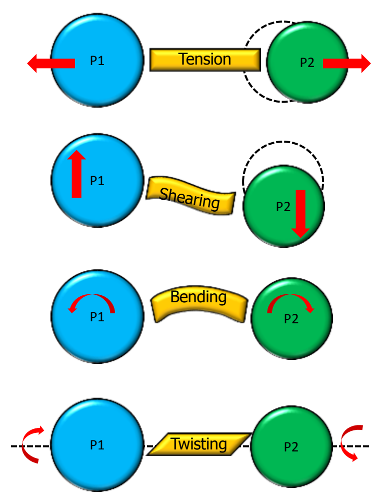
Extending into Continuum Mechanics

- 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.



DES Bond Model - Mechanical Behaviors

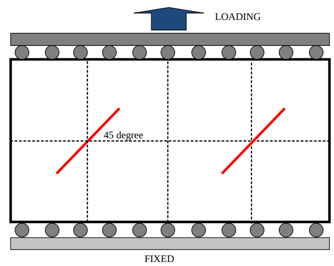
- Every bond is subjected to:
  - *Stretching*
  - *Shearing*
  - *Bending*
  - *Twisting*
- The breakage of a bond results in Micro-Damage which is controlled by the critical fracture energy value J_{IC}.



DES Bond Model

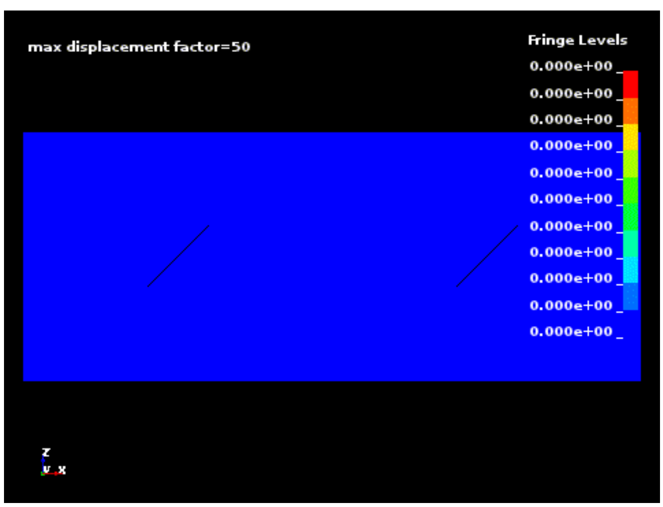
Verification Test of Quasi-Static Loading

- Two Inclined Cracks Under Slow Quasi–Static Loading
- A pre-cracked rectangular plate
  - Size: 100mm x 40mm
  - crack length: 14.1mm
- Material Properties
  - Density: 2,235 kg/m^3
  - Young's modulus: 65 GPa
  - Poisson ratio: 0.2
  - Critical Fracture Value: 204 J/m^2



Quasi-Static Loading

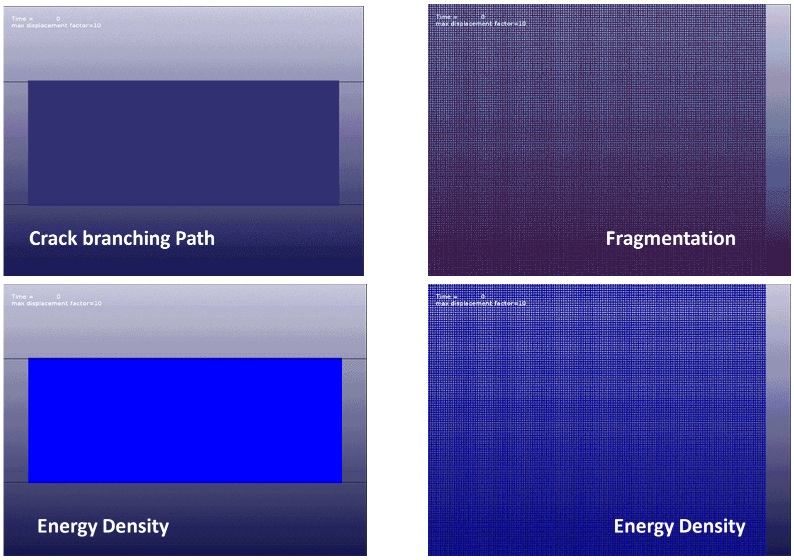
Propagation of Two Inclined Cracks Under Quasi-Static Loading



R=0.125 Total Strain Energy in each elements

Fragmentation Analysis

Dynamic Loading



LS-DYNA Multi-Physics Solvers

	ALE	SPH	DES	PGas
ALE		▲	▲	NA
SPH			■	NA
DES				●
Pgas				

● available in R7.0

● in house testing

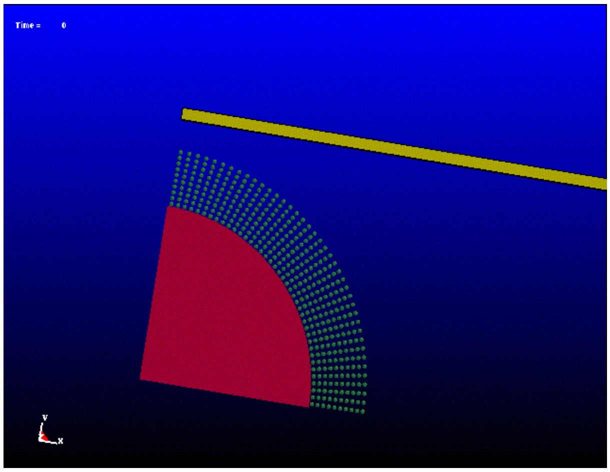
△ *ALE_COUPLING_NODAL

□ *DEFINE_SPH_TO_SPH_COUPLING
*DEFINE_SPH_DE_COUPLING

○ *PARTICLE_BLAST

LS-DYNA Multi-Physics Solvers

**ALE_COUPLING_NODAL*

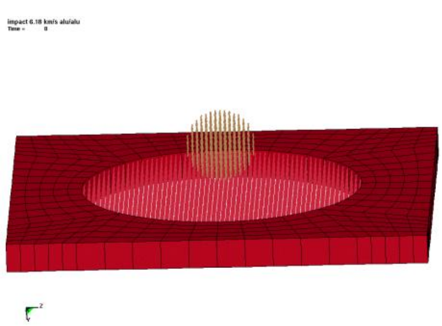


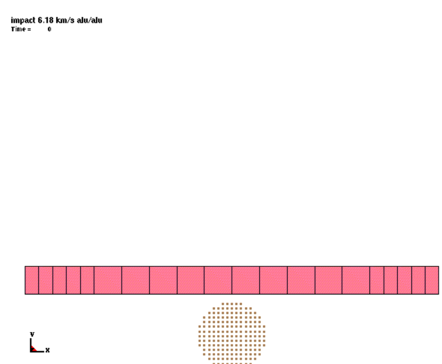
- Modeling explosion driven sands hitting on a plate
- Penalty method is under development

LS-DYNA Multi-Physics Solvers

**DEFINE_SPH_TO_SPH_COUPLING*
**DEFINE_SPH_DE_COUPLING*

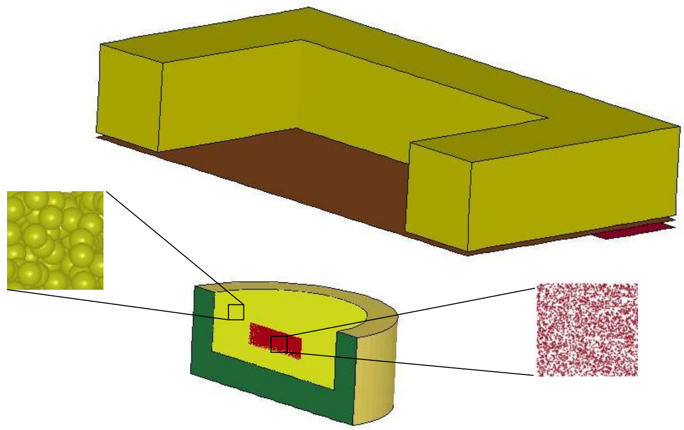
- Penalty based SPH to SPH/DE particle contact





LS-DYNA Multi-Physics Solvers

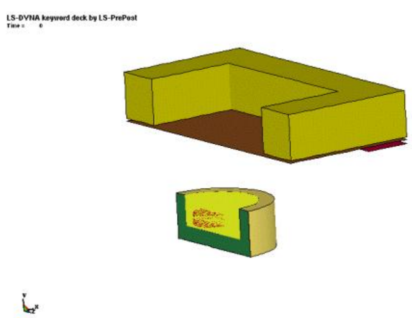
**PARTICLE_BLAST*

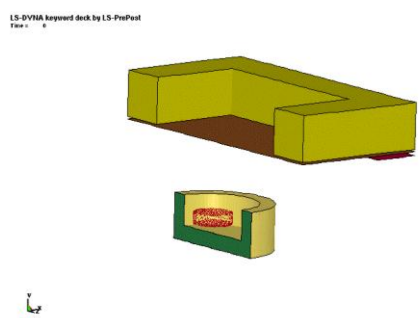


- Sand modeled with DES
- High explosive gas & air

LS-DYNA Multi-Physics Solvers

**PARTICLE_BLAST*





- Blast simulation with sand
- Blast simulation without sand

Fast Rendering in version 4.0/4.1

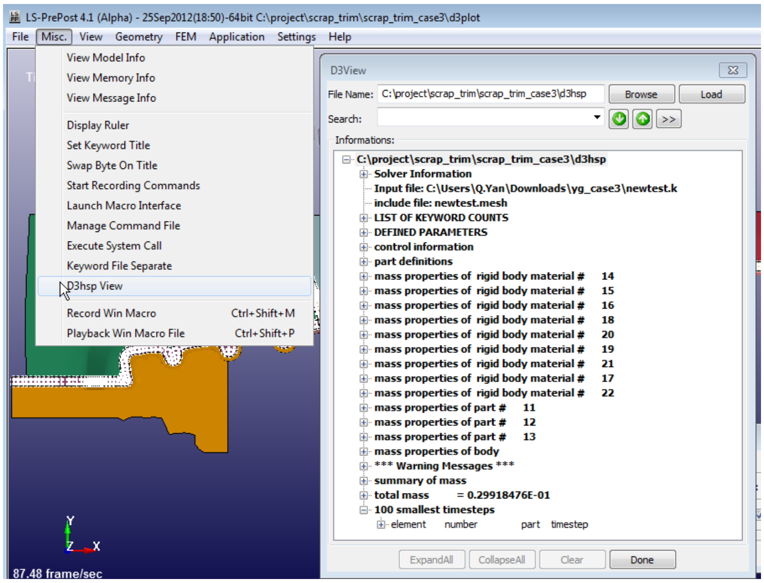
- Fast rendering is the default rendering mode for versions 4.0/4.1.
- If graphics hardware is not capable, “Normal Rendering” will be used automatically.
- To switch between “Fast rendering” and “Normal rendering” mode, enter ctrl-L twice before loading the data.
- Rendering mode will be memorized and recorded in the configuration file.
- For certain hardware and model size, fast render mode can result in 10x to 15x speed up.

D3HSP file viewing

Purpose: To look at the content of d3hsp file in an organized way

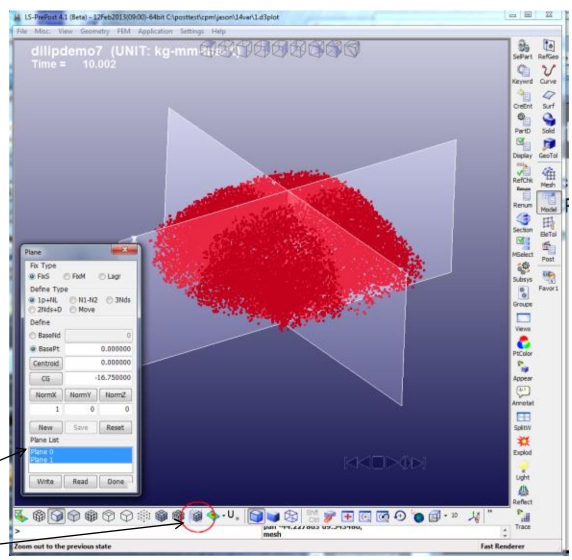
- d3hsp file contains a lot of information for the LS-DYNA run.
- LS-PrePost reads the information from this file and organizes them into a tree/list structure for easy reading.
- Key phase search is possible.
- Launch d3hsp view in misc pull-down menu.
- Only available in version 4.0 and later.

D3HSP file viewing



Cutting Plane for CPM (Particle)

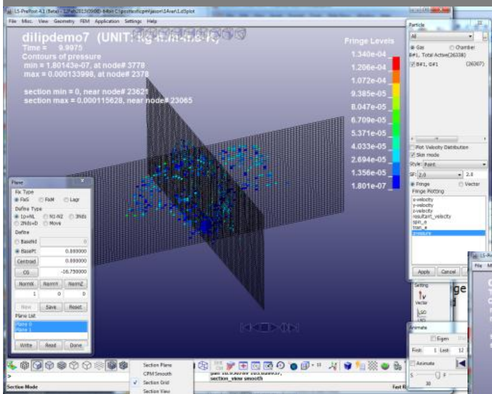
- A special new cutting plane interfaces has been developed for SPH, CPM (particle), DES, and CFD analyses.
- Multiple planes can be defined and visualized.



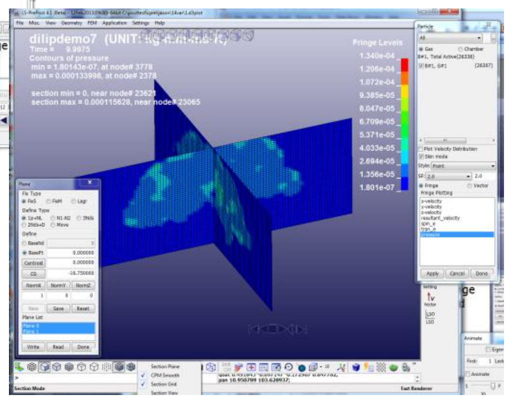
Multiple planes definitions

Click this icon to activate the plane interface

Cutting Plane for CPM (Particle)



Fringe particle data on cutting planes with grid.



Smoothed fringe particle data on multiple planes with grid.

Scripting Command Language

- Scripting Command Language (SCL) – is a C-like programming language to be executed within LS-PrePost.
- Executes LS-PrePost commands.
- Allows “if then else”, for, and while loop operations.
- Provides API (Application Programming Interface) to extract model and result data from LS-PrePost Data base.
- Operations can be done on extracted data to form new data.
  New data can be output to file or fringed on screen.
- Most suitable to perform same operations over different part of the model.
- Documentation and tutorial for Scripting Command Language will be available on LSTC ftp site soon.

Scripting Command Language

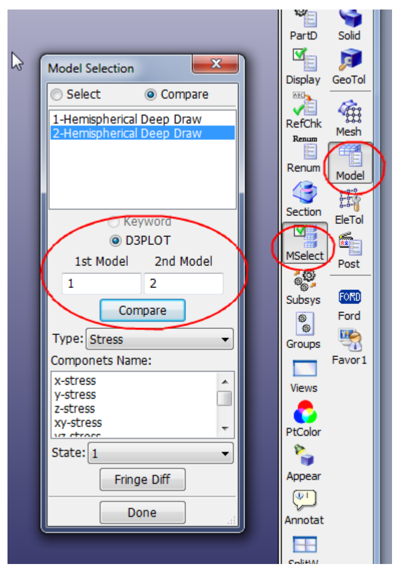
- To download the document on Scripting Command Language, go to the following directory:
  - ftp://ftp.lstc.com/outgoing/lsprepost/SCLexamples

- Lsppscripting.doc – describes how to use Scripting Command Language.

- SCL_Examples.zip – file contains examples scripts which demonstrate different operations.

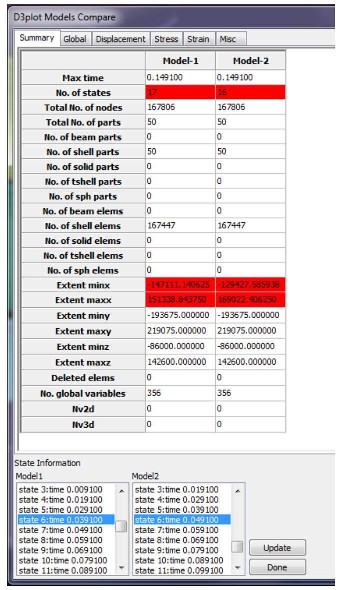
Model Compare for Post-Processing

- Read in 2 sets of d3plot files
- Model should be similar
- Go to Post->MSelect, and select 1st and 2nd model for comparison
- Click “Compare” and wait for the data processing (will take time depends on model size)
- Select State (time) for both models and then click “Update”
- Compare will show different data in the following categories:
  - Summary
  - Global
  - Displacement
  - Stress
  - Strain

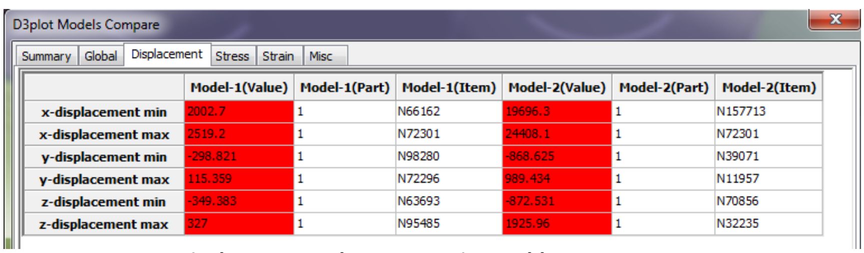


Model Compare for Post-Processing

- Summary – basic model information
- Global – global energy results
- Displacement – the 3 global min/max values of displacement with node IDs
- Stress – the six global min/max stress values with element IDs
- Strain – the six global min/max strain values with element IDs

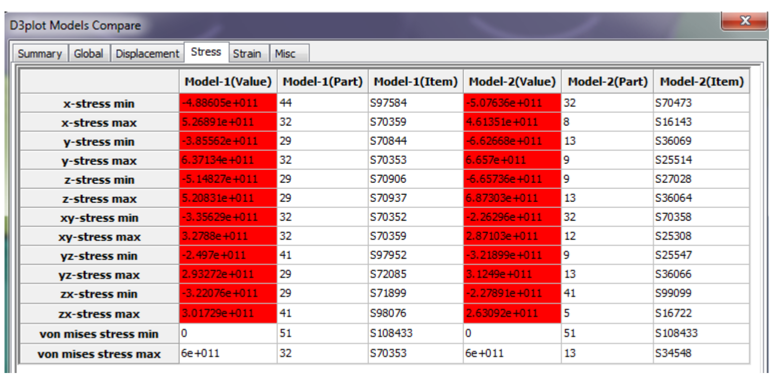


	Model-1	Model-2
Max time	0.149100	0.149100
No. of states	17	16
Total No. of nodes	167806	167806
Total No. of parts	50	50
No. of beam parts	0	0
No. of shell parts	50	50
No. of solid parts	0	0
No. of tshell parts	0	0
No. of sph parts	0	0
No. of beam elems	0	0
No. of shell elems	167447	167447
No. of solid elems	0	0
No. of tshell elems	0	0
No. of sph elems	0	0
Extent minx	-147111.140625	-129427.585938
Extent maxx	151338.843750	169022.406250
Extent miny	-193675.000000	-193675.000000
Extent maxy	219075.000000	219075.000000
Extent minz	-86000.000000	-86000.000000
Extent maxz	142600.000000	142600.000000
Deleted elems	0	0
No. global variables	356	356
Nv2d	0	0
Nv3d	0	0



	Model-1(Value)	Model-1(Part)	Model-1(Item)	Model-2(Value)	Model-2(Part)	Model-2(Item)
x-displacement min	2002.7	1	N66162	19696.3	1	N157713
x-displacement max	2519.2	1	N72301	24408.1	1	N72301
y-displacement min	-298.821	1	N98280	-868.625	1	N39071
y-displacement max	115.359	1	N72296	989.434	1	N11957
z-displacement min	-349.383	1	N63693	-872.531	1	N70856
z-displacement max	327	1	N95485	1925.96	1	N32235

Displacement values comparison table



	Model-1(Value)	Model-1(Part)	Model-1(Item)	Model-2(Value)	Model-2(Part)	Model-2(Item)
x-stress min	-4.88605e+011	44	S97584	-5.07636e+011	32	S70473
x-stress max	5.26891e+011	32	S70359	4.61351e+011	8	S16143
y-stress min	-3.85562e+011	29	S70844	-6.62668e+011	13	S36069
y-stress max	6.37134e+011	32	S70353	6.657e+011	9	S25514
z-stress min	-5.14827e+011	29	S70906	-6.65736e+011	9	S27028
z-stress max	5.20831e+011	29	S70937	6.87303e+011	13	S36064
xy-stress min	-3.35629e+011	32	S70352	-2.26296e+011	32	S70358
xy-stress max	3.2788e+011	32	S70359	2.87103e+011	12	S25308
yz-stress min	-2.497e+011	41	S97952	-3.21899e+011	9	S25547
yz-stress max	2.93272e+011	29	S72085	3.1249e+011	13	S36066
zx-stress min	-3.22076e+011	29	S71899	-2.27891e+011	41	S99099
zx-stress max	3.01729e+011	41	S98076	2.63092e+011	5	S16722
von mises stress min	0	51	S108433	0	51	S108433
von mises stress max	6e+011	32	S70353	6e+011	13	S34548

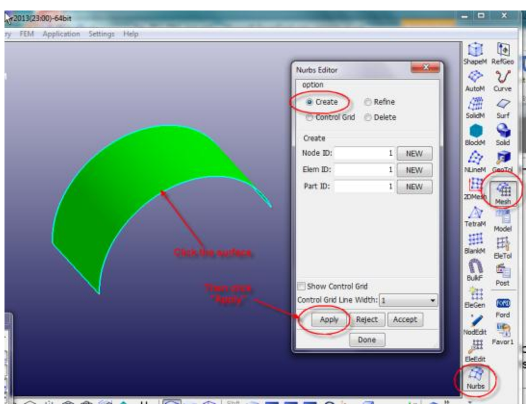
Stress values comparison table

ISO-Geometry Element

- Create *ELEMENT_NURB data from IGES or STEP geometry data.
- Read iso-geometry element (*ELEMENT_NURB) data.
- Read igaplot file for post-processing, igaplot is created by LS-DYNA when isogeometry element is presented in the keyword data.
- Current version of LS-DYNA create both igaplot file along with interpolated mesh for the NURBS element, the interpolated mesh is stored in the regular d3plot file.
- Fringe data can only be processed with the interpolated mesh for now. In the future, d3plot will not contain the interpolated mesh, LS-PrePost will fringe stress/strain data on the isogeometry element directly.

ISO-Geometry Element

- To Create iso-geometry element, go to Mesh->Nurbs->Create,

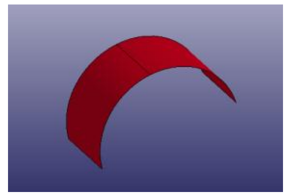


- Current development on capability to modify the isogemetry element within LS-PrePost. Allows user to refine the number of patches and modify the control points.

ISO-Geometry Element

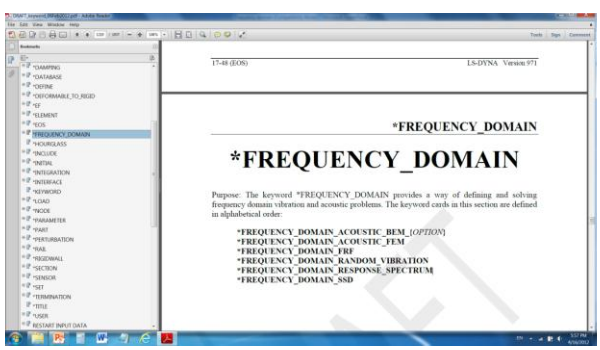
- The keyword data of iso-geometry element

```
$# LS-DYNA Keyword file created by LS-PrePost 4.1 (Beta) - 12Apr2013(23:00)
$# Created on Apr-21-2013 (00:35:25)
*KEYWORD
*ELEMENT_SHELL_NURBS_PATCH
$#   npeid       pid       npr        pr       nps        ps
         1         1         5         2         2         1
$#     wfl      form       int      nisr      niss     imass
         1         0         0         0         0         0
$#     rk1       rk2       rk3       rk4       rk5       rk6       rk7       rk8
     0.000     0.000     0.000  1.552261  1.552261  3.104522  3.104522  3.104522
$#     sk1       sk2       sk3       sk4       sk5       sk6       sk7       sk8
     0.000     0.000  1.000000  1.000000     0.000     0.000     0.000     0.000
$#      n1        n2        n3        n4        n5        n6        n7        n8
         1         2         3         4         5         0         0         0
         6         7         8         9        10         0         0         0
$#      w1        w2        w3        w4        w5        w6        w7        w8
  1.000000  0.713630  1.000000  0.713630  1.000000     0.000     0.000     0.000
  1.000000  0.713630  1.000000  0.713630  1.000000     0.000     0.000     0.000
*NODE
$#   nid               x               y               z      tc      rc
       1       -0.999313       -0.037062           0.000       0       0
       2       -0.962931       -1.018022           0.000       0       0
       3        0.018534       -0.999828           0.000       0       0
       4        1.000000       -0.981634           0.000       0       0
```



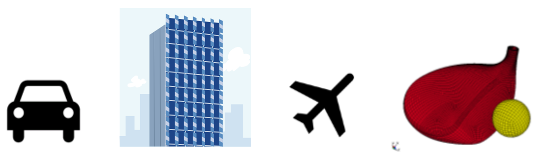
Frequency Domain Features

- BEM acoustics
- FEM acoustics
- Frequency response function
- Random vibration (fatigue)
- Response spectrum analysis
- Steady state dynamics

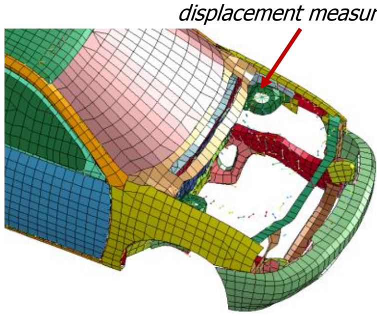


Application

- NVH of automotive and airplane
- Acoustic design and analysis
- Defense industry
- Fatigue of machine and engine
- Civil Engineering, Earthquake Engineering

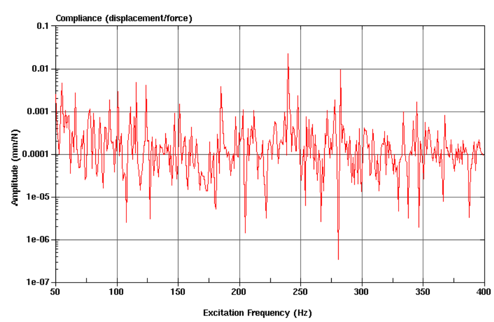


Car Body NVH



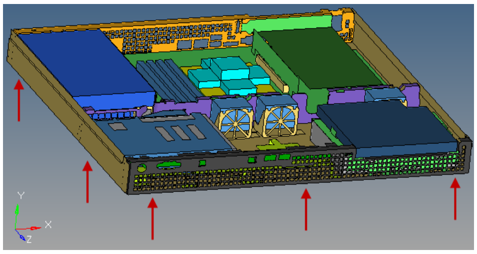
FRF formulations

FRF	Input	Output
Accelerance	Force	Acceleration
Effective Mass	Acceleration	Force
Mobility	Force	Velocity
Impedance	Velocity	Force
Dynamic Compliance	Force	Displacement
Dynamic Stiffness	Displacement	Force



Random Vibration

A cluster server is analyzed by LS-DYNA to understand the location of vibration damage under standard random vibration condition.



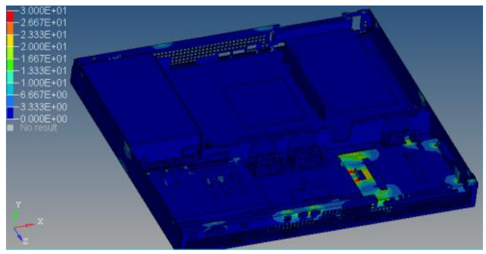
Input PSD

GRMS=1.63 g^2/Hz	
Hz	g^2/Hz
10	0.001
20	0.003
40	0.003
80	0.02
120	0.02
200	0.0015
500	0.0015

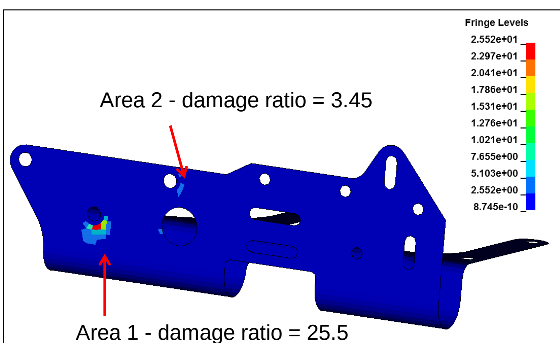
It is found that the 3σ Von-Mises stress is less than the yield stress of the material (176 MPa).

Maximum values

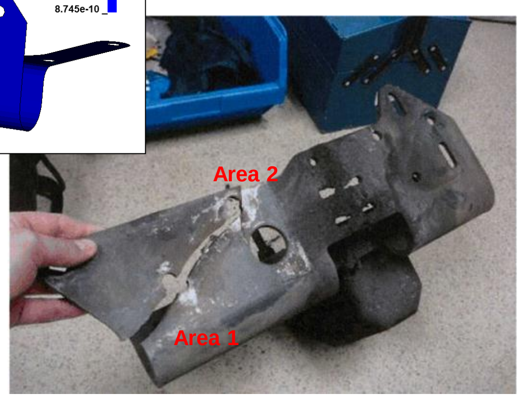
	1σ	3σ
$S_{v\text{-}m}$ (MPa)	41.2	123.6



Random Vibration Fatigue



1. *Steinberg's 3 band technique*
2. *Dirlik method*
3. *Narrow band method*
4. *Wirsching method*
5. *Chaudhury and Dover method*
6. *Tunna method*
7. *Hancock method*



Initial Damage in Random Fatigue

FREQUENCY_DOMAIN_RANDOM_VIBRATION_*FATIGUE

Card1	1	2	3	4	5	6	7	8
Variable	MDMIN	MDMAX	FNMIM	FNMAX	RESTRT	MFTG	RESTRM	**INFTG**
Type	I	I	F	F	I	I	I	I
Default	1		0.0		0	0	0	0

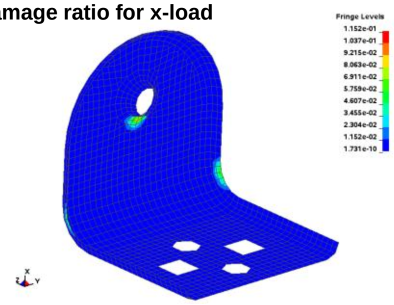
Define Card 7 if option FATIGUE is used and INFTG=1.

Card7	1	2	3	4	5	6	7	8
Variable	**FILENAME**							
Type	C							
Default	d3ftg							

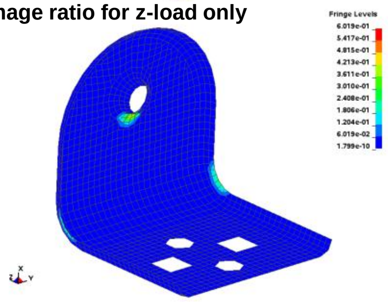
VARIABLE	DESCRIPTION
INFTG	Flag for including initial damage ratio. EQ.0: no initial damage ratio, EQ.1: read existing d3ftg file to get initial damage ratio.
FILENAME	Path and name of existing binary database (by default, D3FTG) for initial damage ratio.

Example: Initial Damage

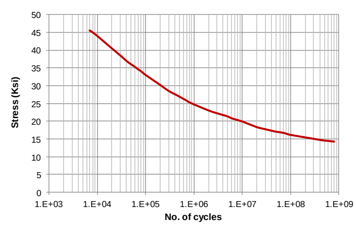
Damage ratio for x-load



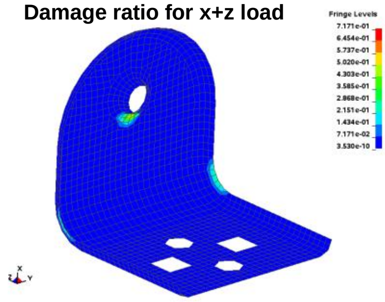
Damage ratio for z-load only



Time of exposure: 4 hours



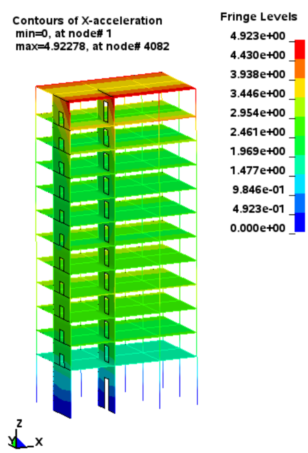
Damage ratio for x+z load



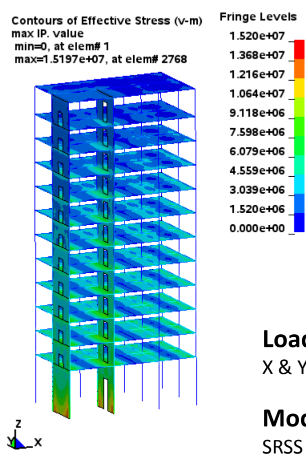
Dirlik method is used (MFTG = 2).

Response Spectrum

x acceleration



Von Mises stress



Loading
X & Y acceleration spectra

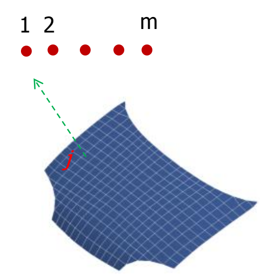
Mode combination
SRSS

Acoustic Transfer Vector

FREQUENCY_DOMAIN_ACOUSTIC_BEM_*ATV

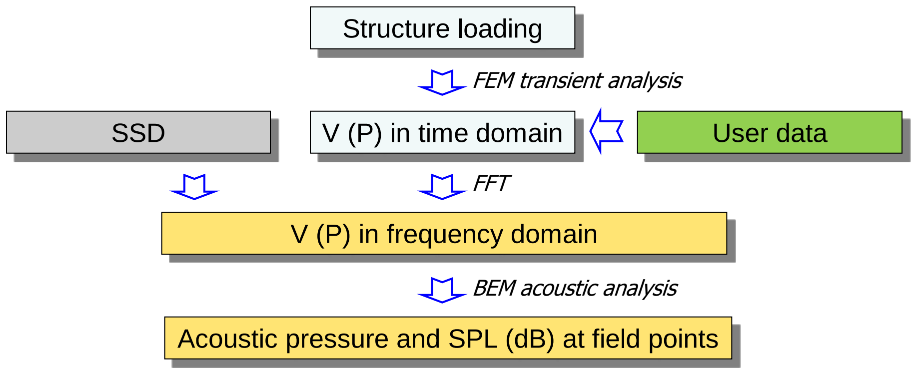
- Acoustic Transfer Vector is obtained by including the option **ATV** in the keyword.
- It calculates acoustic pressure (and sound pressure level) at field points due to **unit normal velocity** of each surface node.
- ATV is dependent on structure model, properties of acoustic fluid as well as location of field points.
- ATV is useful if the same structure needs to be studied under multiple load cases.

$$\begin{Bmatrix} P_1 \\ P_2 \\ \vdots \\ P_i \\ \vdots \\ P_m \end{Bmatrix} = \begin{bmatrix} \Omega_{1,1} & \Omega_{1,2} & \cdots & \Omega_{1,j} & \cdots & \Omega_{1,n} \\ \Omega_{2,1} & \Omega_{2,2} & \cdots & \Omega_{2,j} & \cdots & \Omega_{2,n} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ \Omega_{i,1} & \Omega_{i,2} & \cdots & \Omega_{i,j} & \cdots & \Omega_{i,n} \\ \vdots & \vdots & \ddots & \vdots & \ddots & \vdots \\ \Omega_{m,1} & \Omega_{m,2} & \cdots & \Omega_{m,j} & \cdots & \Omega_{m,n} \end{bmatrix}_{m\times n} \begin{Bmatrix} V_1 \\ V_2 \\ \vdots \\ V_j \\ \vdots \\ V_n \end{Bmatrix}$$



Need to be computed only once *Change from case to case*

Various Input for Acoustics



*FREQUENCY_DOMAIN_ACOUSTIC_BEM

Card 4	1	2	3	4	5	6	7	8
Variable		NBC	RESTRT	IEDGE	NOEL	NFRUP		
Type		I	I	I	I	I		
Default		1	0	0	0	0		

RESTRT — Restart options:
EQ.3: LS-DYNA reads in user provided velocity history, saved in ASCII file "bevel".

Acoustics: Boundary_Acoustic_Mapping

*BOUNDARY_ACOUSTIC_MAPPING

Purpose: Define a set of elements or segments on structure for mapping structural nodal velocity to boundary of acoustic volume.

Card	1	2	3	4	5	6	7	8
Variable	SSID	STYP						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
SSID	Set or part ID
STYP	Set type: EQ.0: part set ID, see *SET_PART, EQ.1: part ID, see *PART, EQ.2: segment set ID, see *SET_SEGMENT.

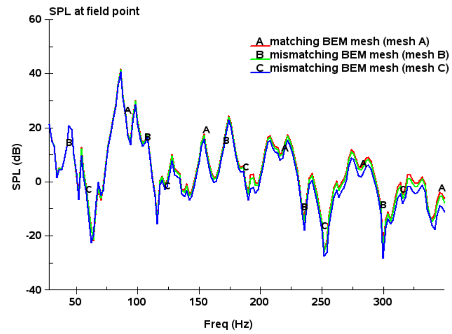
Example: Boundary_Acoustic_Mapping

Mesh A: 20 × 30 (600) Mesh B: 15 × 20 (300) Mesh C: 7 × 11 (77)

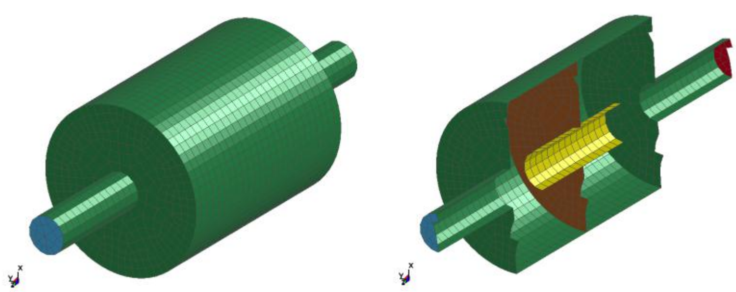
Acoustic mesh same as structure surface mesh

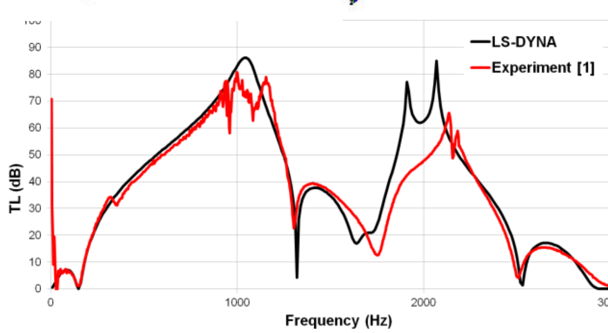
CPU time (Intel Xeon 1.6 GHz)

Mesh A	16 min 34 sec
Mesh B	10 min 10 sec
Mesh C	6 min 49 sec



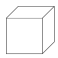
Acoustic Application: Muffler Transmission Loss



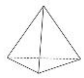


FEM Acoustics

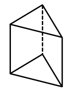
Hexahedron



Tetrahedron

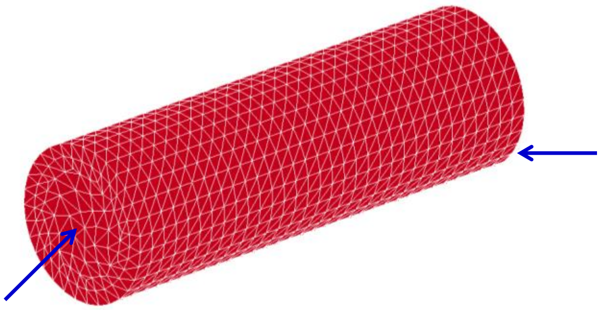


Pentahedron



Example

Solving an interior acoustic problem by indirect / collocation BEM and FEM.



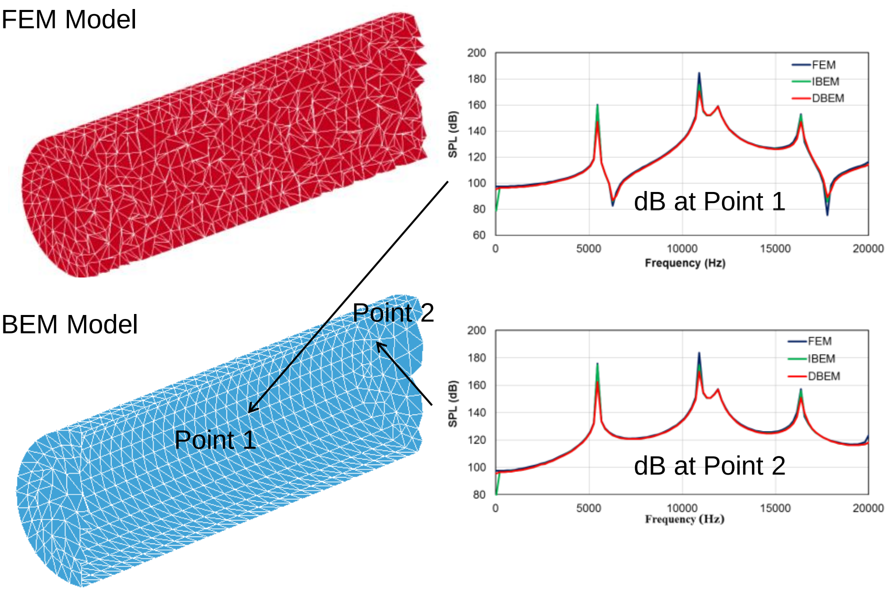
This edge is fixed in x, y, z translation dof.

Nodal force 0.01N is applied for frequency range of 10-20000 Hz.

Validation by Different Methods

FEM Model

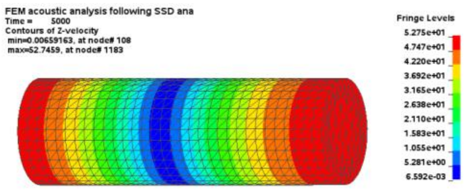
BEM Model



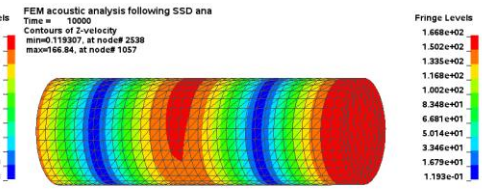
FEM Acoustics: Post Processing

Acoustic pressure distribution

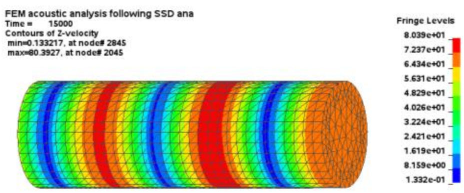
(by d3acs)



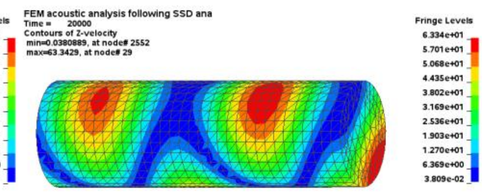
f =5000 Hz



f =10000 Hz



f =15000 Hz



f =20000 Hz

Incompressible ICFD solver (*ICFD)

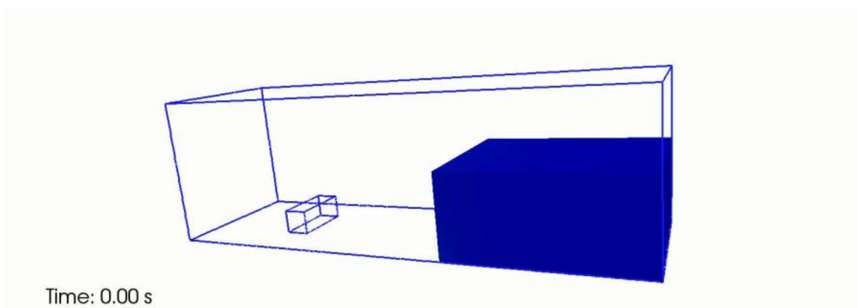
- A solver for problems that involve incompressible flows
- It is coupled to other features of LS-Dyna for multiphysics analysis
- Highly scalable in MPP
- Large database of validations problems
- Steady growth

Examples of application

- Free Surface Flow
- External/Internal Aerodynamics
- Conjugate heat transfer
- Conjugate heat transfer + Electromagnetism
- Fluid Structure interaction

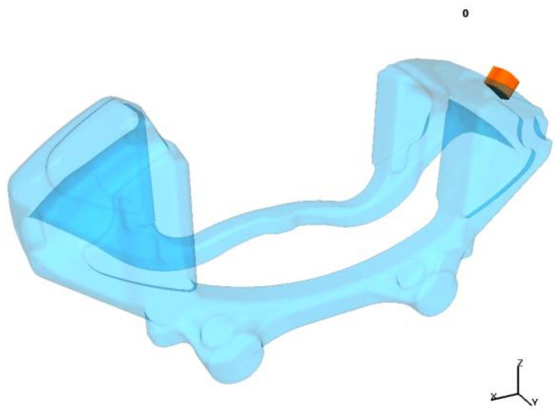
Examples of application

Free Surface flow: Impact force calculation



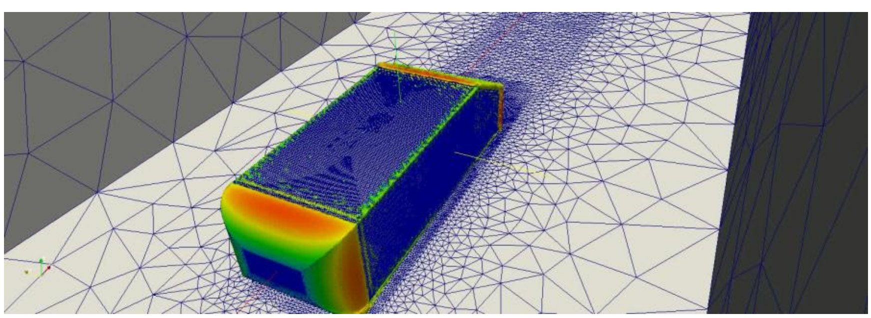
Examples of application

Free Surface flow: Mold filling simulation



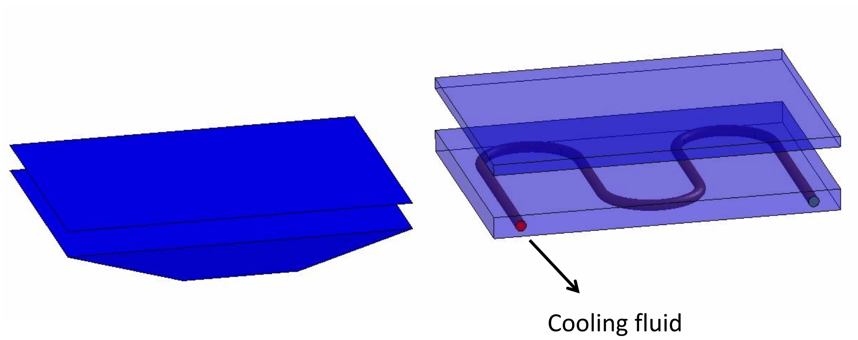
Examples of application

External Aerodynamics: Drag prediction for ground vehicles



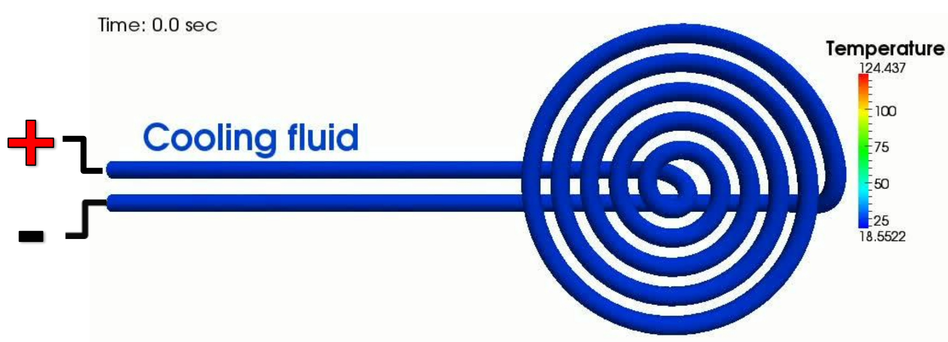
Examples of application

Conjugate Heat Transfer: Stamping



Examples of application

Conjugate Heat Transfer: Coupled to Electromagnetism for cooling

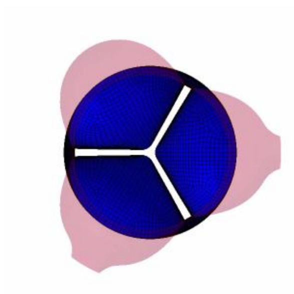


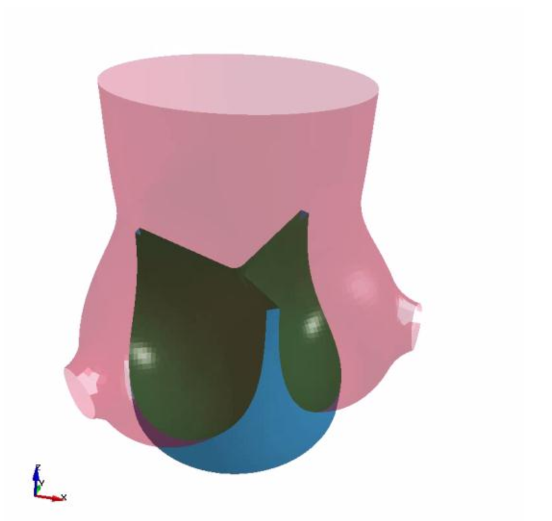
Courtesy of: Miro from the Institute for composite in Kaiserslautern

Examples of application

Fluid Structure Interaction

- Shell solid elements
- Implicit solid mechanics
- Strong FSI coupling

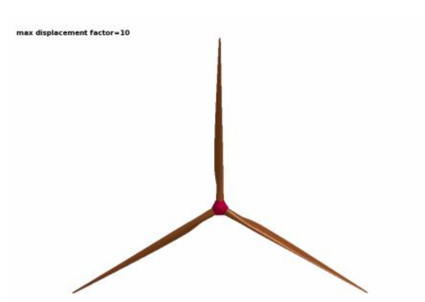




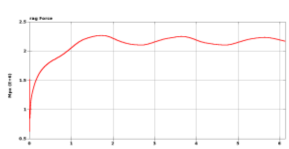
Courtesy of: Hossein Mohammadi, Mcgill University

Examples of application

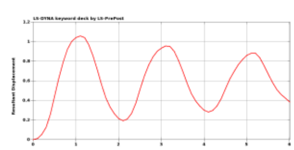
Fluid Structure Interaction: Torque and tip displacement



Torque



Blade Tip Displacement

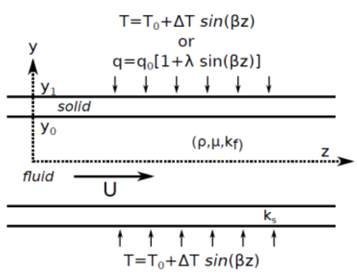


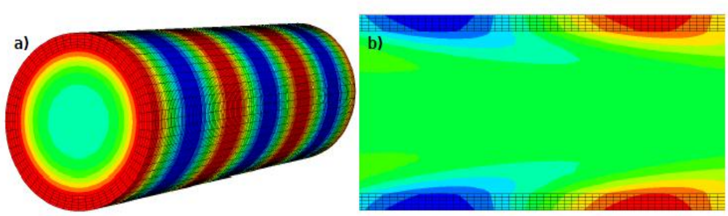
New Developments

- **Verification of conjugate heat transfer problems using analytical solutions.**
- New turbulent inflow and new turbulence model WALE.
- Improved performance and scalability.
- Numerous new LSPP tools.
- Export CFD output in *vtk* format.
- Non-Newtonian flow.

New Developments

Excellent agreement with analytical results.



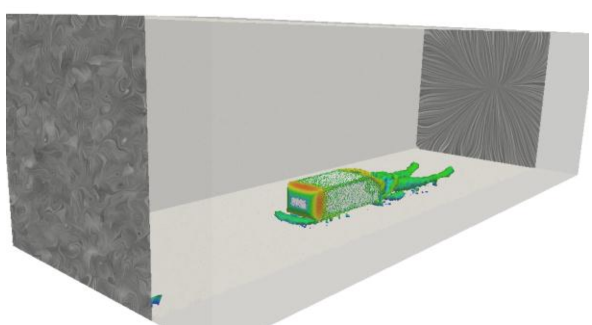


New Developments

- Verification of conjugate heat transfer problems using analytical solutions.
- **New turbulent inflow and new turbulence model WALE.**
- Improved performance and scalability.
- Numerous new LSPP tools.
- Export CFD output in *vtk* format.
- Non-Newtonian flow.

New Developments

Better approximation for problems where the inflow is turbulent.

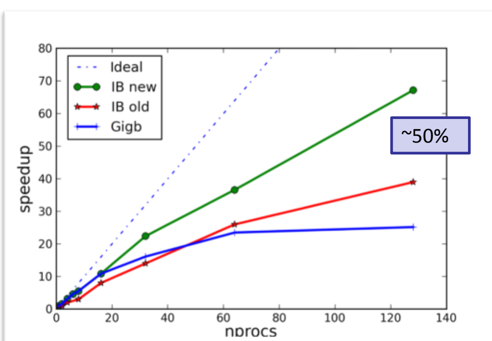


New Developments

- Verification of conjugate heat transfer problems using analytical solutions.
- New turbulent inflow and new turbulence model WALE.
- **Improved performance and scalability.**
- Numerous new LSPP tools.
- Export CFD output in *vtk* format.
- Non-Newtonian flow.

New Developments

Cluster: 12 cores per node Intel CPU, Mellanox InfiniBand.
Problem: Flow over a wall mounted cube. 1.5M elements, Re=40,000, LES



Sweet spot: 10,000 elem/core

New Developments

- Verification of conjugate heat transfer problems using analytical solutions.
- New turbulent inflow and new turbulence model WALE.
- Improved performance and scalability.
- **Numerous new LSPP tools.**
- Export CFD output in *vtk* format.
- Non-Newtonian flow.

New Developments

- New Surface Meshing tool for graded meshes.
- Visualization of free surfaces using iso-volumes.
- Post-processing of LSO output.
- Coloring of Iso-surfaces with some other field.
- New interfaces for cut-planes, etc.

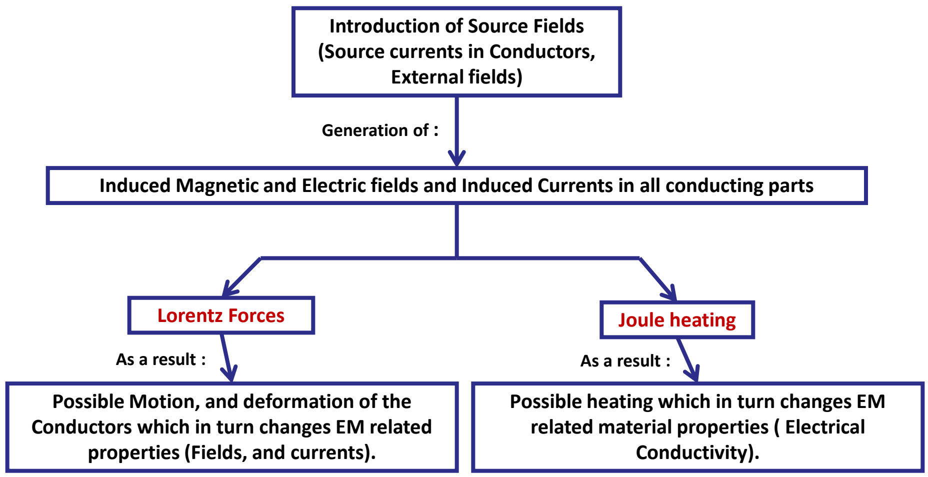
New Developments

- Verification of conjugate heat transfer problems using analytical solutions.
- New turbulent inflow and new turbulence model WALE.
- Improved performance and scalability.
- Numerous new LSPP tools.
- Export CFD output in *vtk* format.
- Non-Newtonian flow.

Future Developments

- Improve the support for multi-phase flows.
- Add porous media simulation.
- Improve the control over the boundary layer mesh generation.

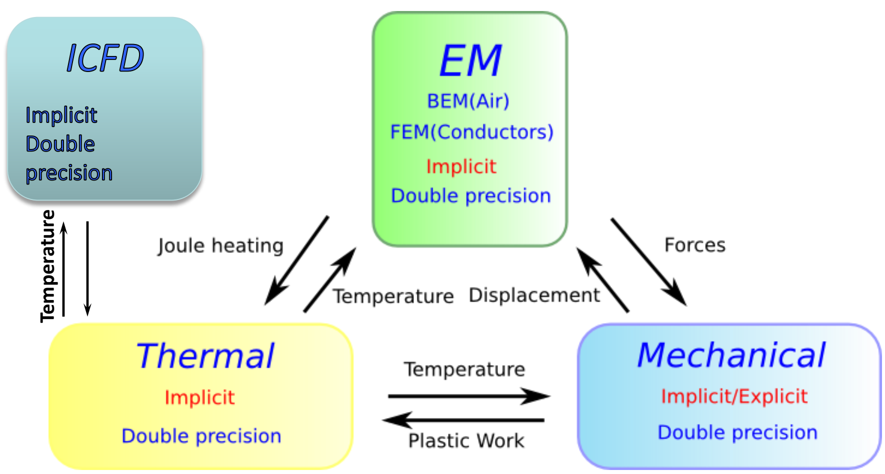
The EM Electromagnetics solver (*EM)



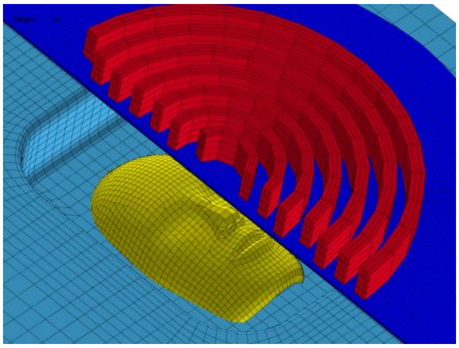
Consequence: Solver coupling needed

Coupling with other LS-DYNA Solvers

- **Scope of the new R7 solvers : to be coupled with LS-DYNA solvers in order to solve complex multiphysics problems.**



Electromagnetics for Magnetic Metal Forming

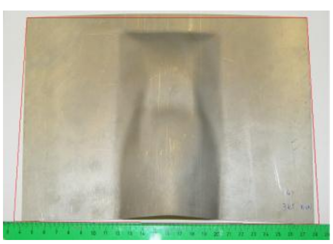


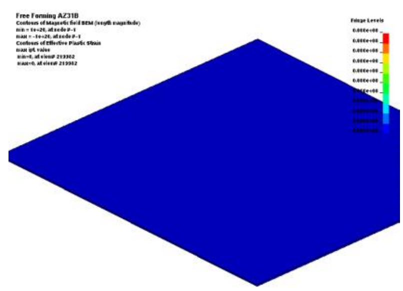
MMF: High velocity forming process

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

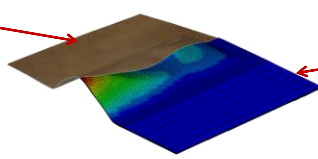
In collaboration with:
G. Mazars & G. Avrillaud:
Bmax, Toulouse, France

Electromagnetics for Magnetic Metal Forming

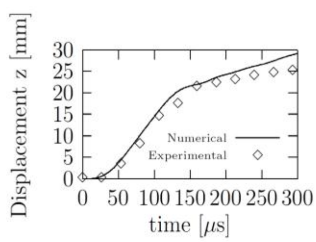




Experimental result

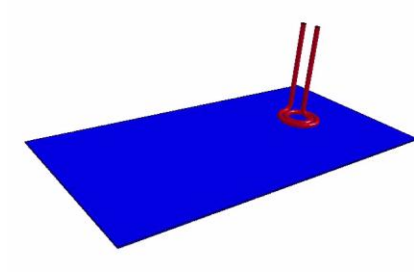


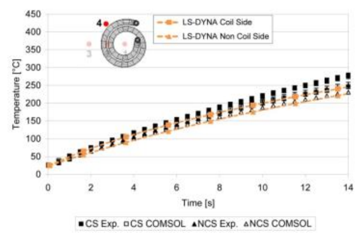
Numeric result



In collaboration with:
Ibai Ulacia, University of Mondragon &
Gipuzkoa, Basque country

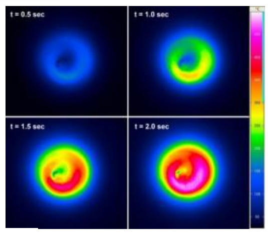
Electromagnetics for Inductive Heating Problems



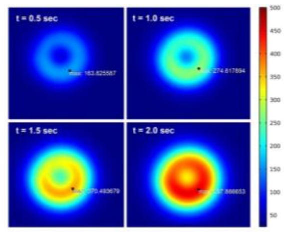


Heating by Electromagnetic Induction

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



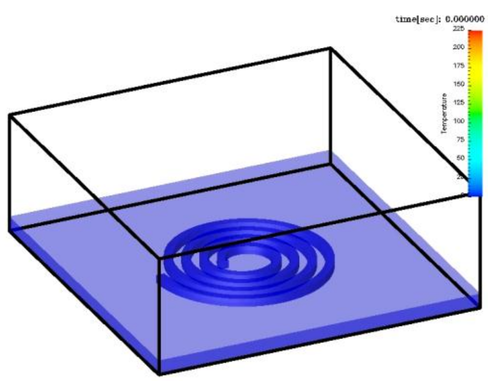
Thermal images from experience



LS-DYNA temperature fringes

EM Coupled with ICFD for Immersion

- 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.

Advancement Status

- All EM solvers work on solid elements (hexahedral, tetrahedral, wedges) for conductors.
- Shells can be used for insulator materials.
- MPP and SMP available.
- The EM fields as well as EM force and Joule heating can be visualized in LS-PREPOST :
  - Fringe components
  - Vector fields
  - Element histories
- LSO can be used for certain time histories.
- Website available for more information.

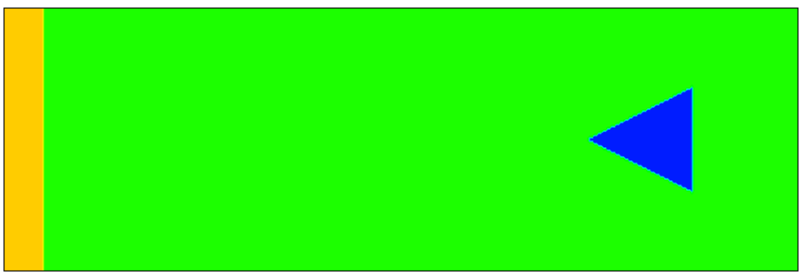
The CESE Compressible CFD Solver (*CESE)

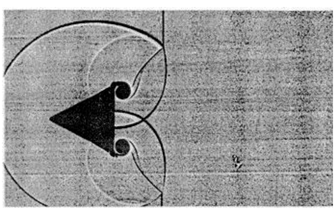
- *The CESE method is a high-resolution and genuinely multidimensional compressible flow solver for solving conservation laws using the Conservation Element/Solution Element (CE/SE) method.*
- Unique features include:
  - A unified treatment of space and time.
  - The introduction of the conservation element and the solution element as a vehicles for enforcing space-time flux conservation, locally and globally.
  - A novel shock capturing strategy without a *Riemann* solver.
  - Unlike conventional schemes, flow variables and their spatial derivatives are solved simultaneously.
- In addition to complex flow problems such as shock/pressure wave interaction, gaseous detonation, and cavitating flows, the CESE solver has been coupled with the solid structure solver (for FSI problems) & the solid thermal solver.
- The stochastic particle and the chemistry solvers are also coupled into the CESE solver.

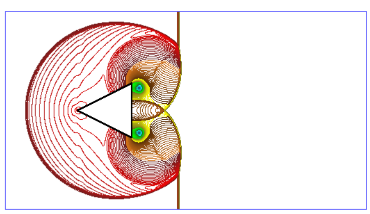
CESE Solver

- **Main Features:**
  - 3D solver as well as a 2D planar and a 2D axisymmetric solver.
  - Automatic coupling with structural and thermal LS-DYNA solvers. (embedded/immersed boundary approach or moving/fitted mesh)
  - Cavitation model.
  - Coupled stochastic particle & chemistry solvers.
- **Applications:**
  - Compressible flows (M>=0.3), especially subsonic & supersonic flows with shock waves.
  - Shock/acoustic wave interaction.
  - Cavitating flows.
  - Conjugate heat transfer problem.
  - Stochastic particle flows: fuel sprays, dusty & aerosol flows.
  - Chemically reacting flows: gaseous detonation and high-speed combustion.

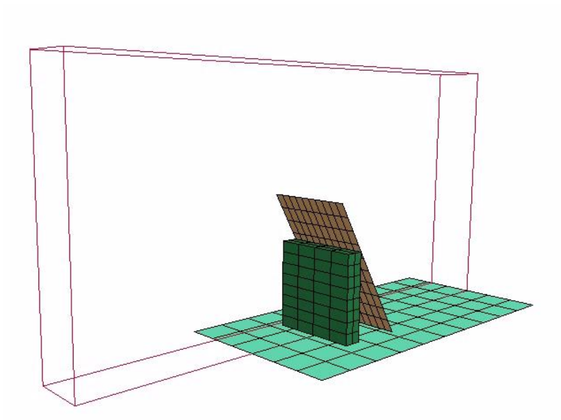
Moving Wedge & Shock Interaction

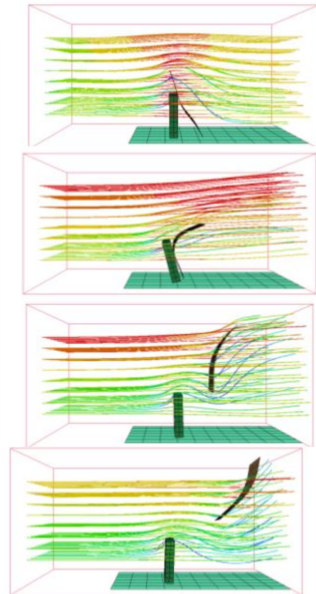




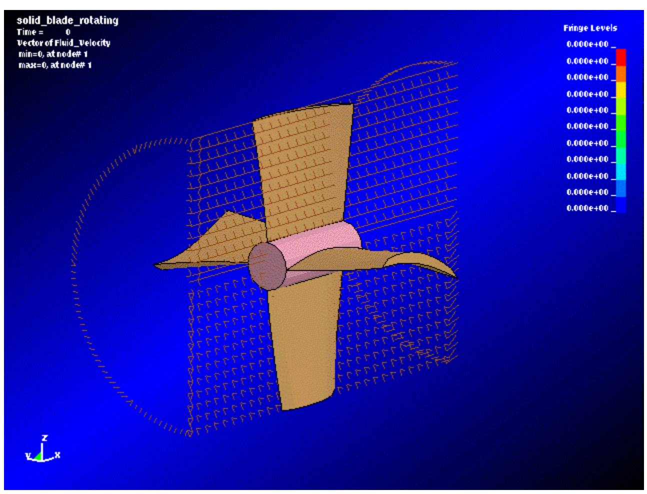


FSI problems



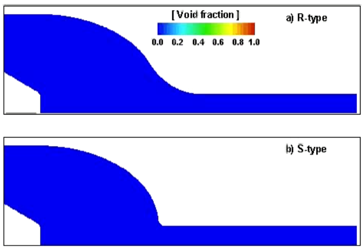


Turbomachines

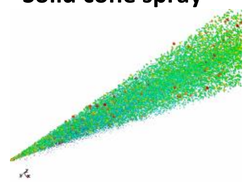


Cavitation and Sprays

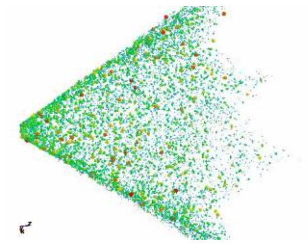
- **Cavitating flow**



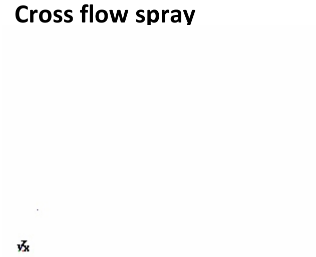
- **Solid cone spray**



- **Hollow cone spray**



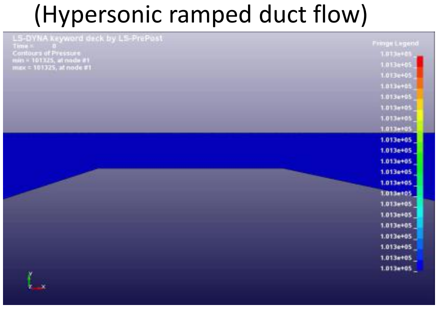
- **Cross flow spray**



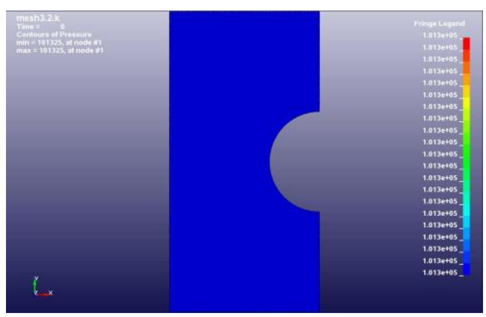
Chemically Reacting Flows

- **Detailed model.**
- **5 species:** *O2,,N2,O,N,NO*
- **11 reaction steps**
- **Initial mixture:** $O_2 + 3.76N_2$
- **N - S solver.**

(Hypersonic ramped duct flow)



(Hypersonic blunt body detached shock)



Summary

- LSTC is working to be the leader in scalable, low cost, large scale, multi-physics simulations, leading to solutions to a variety of problems with a single universal numerical model. To make this possible:
  - LS-PrePost, LS-Opt, and LS-TaSC are continuously improving and gaining more usage within the LS-DYNA user community
  - LSTC is providing dummy, barrier, and head form models to reduce customer costs.
  - The incompressible flow solver is fully coupled to heat transfer and structures for FSI simulations
  - The electromagnetics solver is coupled to heat transfer and structural elements for fully coupled simulations
  - Coupling between ALE methodology, SPH, discrete elements, and the airbag particle method will lead to new application areas in the future and improve current methodologies

Future

- New features and algorithms will be continuously implemented to handle new challenges and applications
  - Electromagnetics,
  - Acoustics,
  - Compressible and incompressible fluids
  - Element technology: isogeometric, Cosserat, higher-order quadratic/cubic
  - Multi-physics, isogeometric, and higher order element contact
  - Discrete element methodology for modeling granular materials
  - Simulation based airbag folding and THUMS dummy positioning underway
- Multi-scale capabilities are under development
  - Implementation underway (New approach which is more user friendly)
- Hybrid MPI/OPENMP developments are showing significant advantages at higher number of processors (>10,000) for both explicit and implicit solutions.

Thank You