ICFD Solver

Iñaki Çaldichoury, Facundo Del Pin, Rodrigo Paz

ICFD Solver Introduction

- **A CFD solver for incompressible flows (ICFD solver).**
- **Included with LS-DYNA double precision executables (starting R7).**
- **No extra licenses or tokens necessary**
- **Can be used as stand alone CFD solver or for coupled Fluid-structure interaction and conjugate heat.**
- **SMP and MPP versions available. Highly scalable in MPP.**
- **LSTC Objective to solve complex multiphysics problems within the same software package.**

ICFD Solver Introduction

Two main axes of development

Multi-physics :

Coupling with structure (FSI)
Coupling with thermal (Conjugate heat transfer)
Coupling with DEM

Classic CFD :

Turbulence models
Free surface problems/Wave Generator and Multiphase
Non-Newtonian fluids and Porous media
Steady State solver

ICFD Development status

Turbulence :

- RANS models added : Realizable k-epsilon, k-omega, SST, Spalart Almaras (R9)
- New tools developed to control boundary layer generation (R9)
- Steady state solver (Beta)
- DES turbulence model (Scheduled)

ICFD Development status

Keywords added or modified :

- ICFD_CONTROL_TURBULENCE
- ICFD_BOUNDARY_TURBULENCE
- ICFD_INITIAL_TURBULENCE
- MESH_BL
- ICFD_CONTROL_GENERAL/ICFD_CONTROL_STEADY

ICFD Development status

Thermal solver :

- Coupling with non linear thermal solver possible (for examples in cases that involve radiation) (Beta)
- New GMRES solver for conjugate heat transfer for calculation speed ups up to a factor ten on mid size problems (over 1M elements) (R9)
- Added temperature dependent viscosity laws to take into account solidification process in mold flow applications (R9)
- Added option to output heat transfer coefficient on surface in an ascii file (Beta)

ICFD Development status

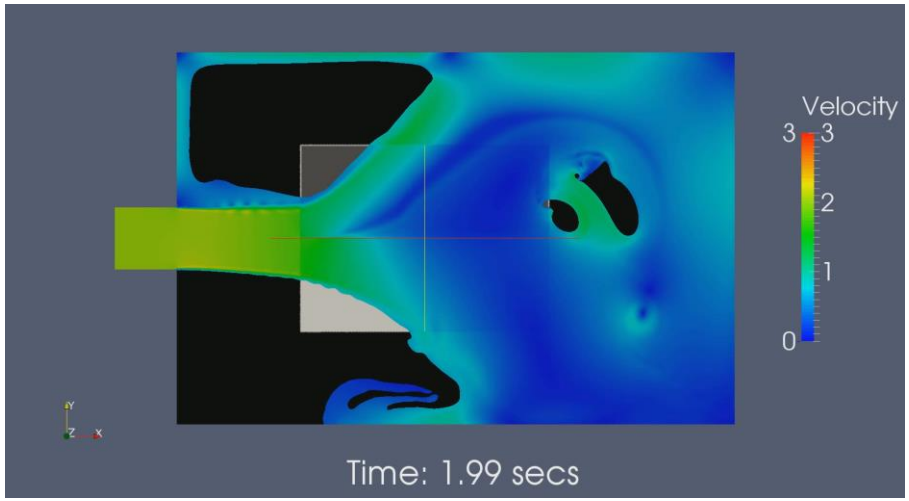
Keywords added or modified :

- CONTROL_THERMAL_SOLVER (solver type 17)
- ICFD_MAT
- ICFD_MODEL_NONNEWT
- ICFD_CONTROL_CONJ

ICFD Development status

Porous media:

- Anisotropic and isotropic porous media models (R8)
- Added a new Anisotropic Porous Media flow model (PM model ID=9): It uses a variable permeability tensor field which is result of a solid dynamic problem (Beta)



ICFD Development status

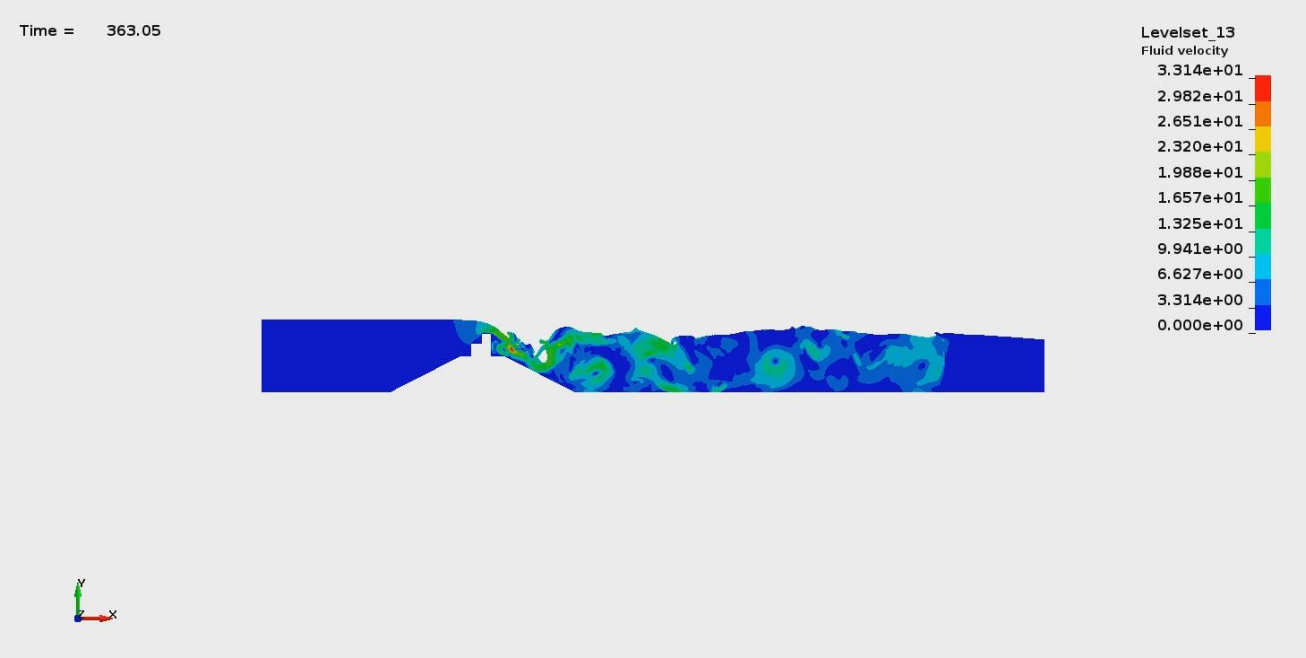
Keywords added or modified :

- ICFD_MAT
- ICFD_MODEL_POROUS

ICFD Development status

Free Surface and multiphase

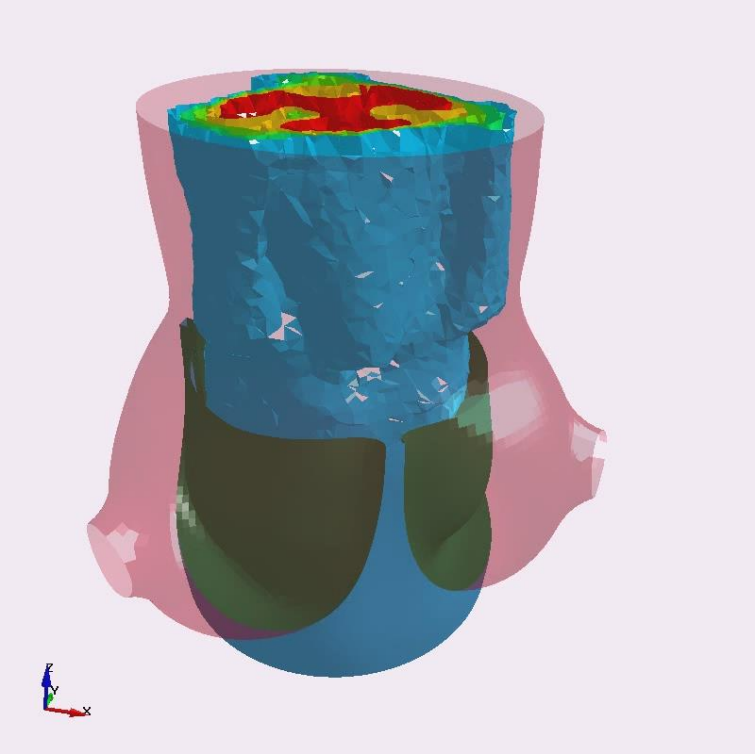
- Improvements on multiphase (Beta)
- Wave generator (Scheduled)
- Absorbing condition for Wave damping (Beta)



ICFD Development status

Blood mechanics:

- Added non-Newtonian models for blood flow (Carreau model for example) (R9)
- Added Windkessel boundary condition (Beta)



ICFD Development status

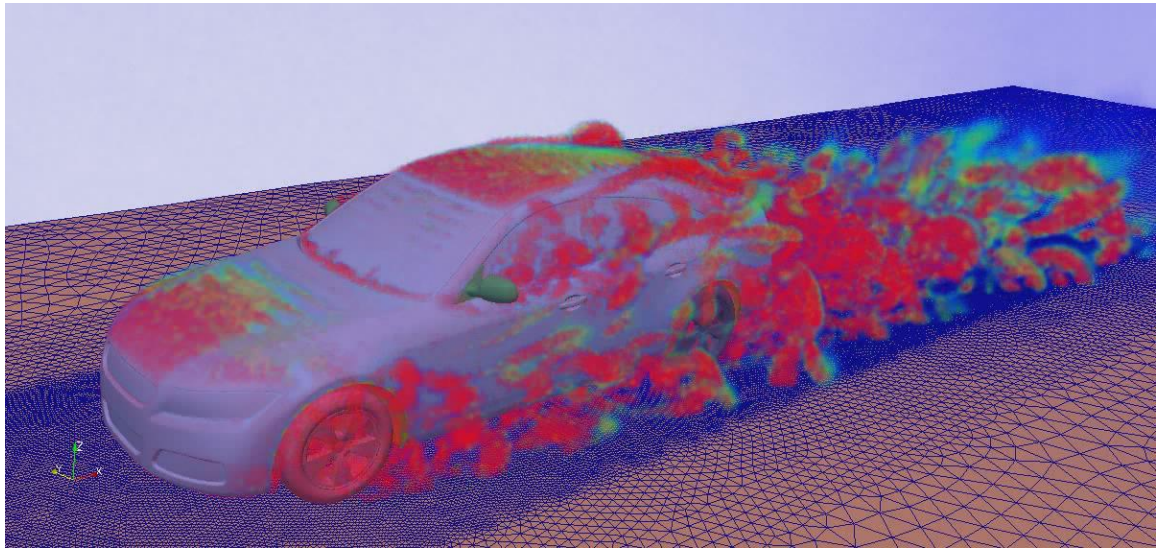
Keywords added or modified :

- ICFD_MODEL_NONNEWT
- ICFD_BOUNDARY_WINDKESSEL

ICFD Development status

Coupling:

- Coupling with DEM (R9)
- FSI : Added possibility to impose displacements on surface mesh to avoid having to use one way FSI coupling (R9)



ICFD Development status

Keywords added or modified :

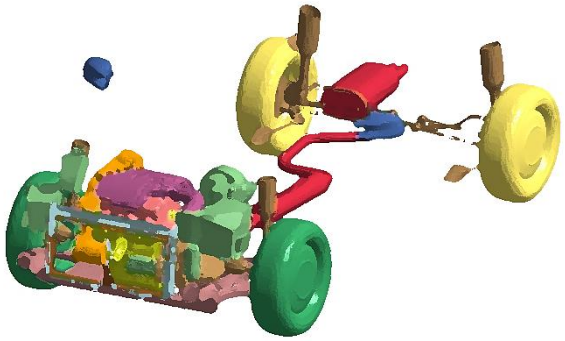
- ICFD_CONTROL_DEM_COUPLING
- ICFD_CONTROL_IMPOSED_MOVE

ICFD Development status

New LSPP GUI to accommodate for ICFD solver:

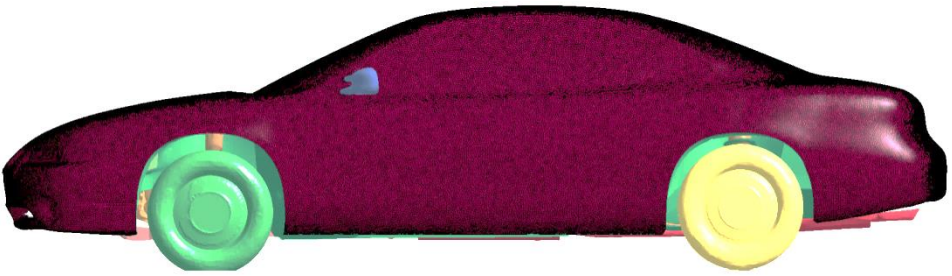
Youtubevideo

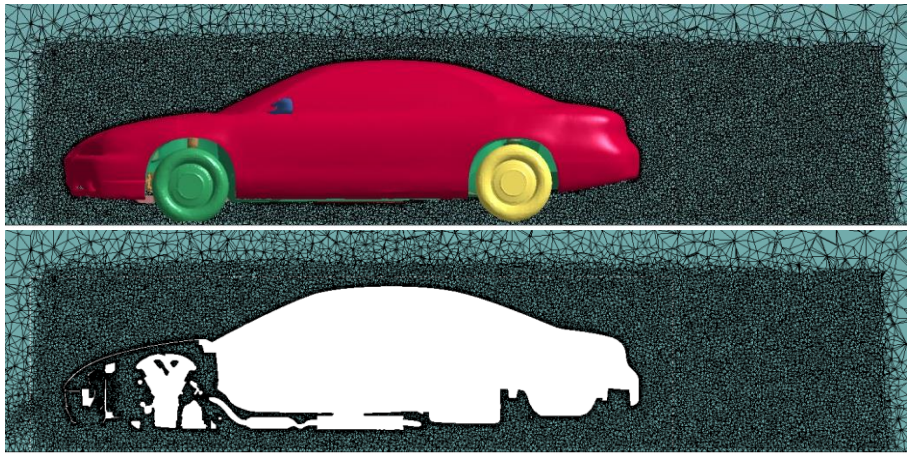
CFD analysis example on GM sedan model



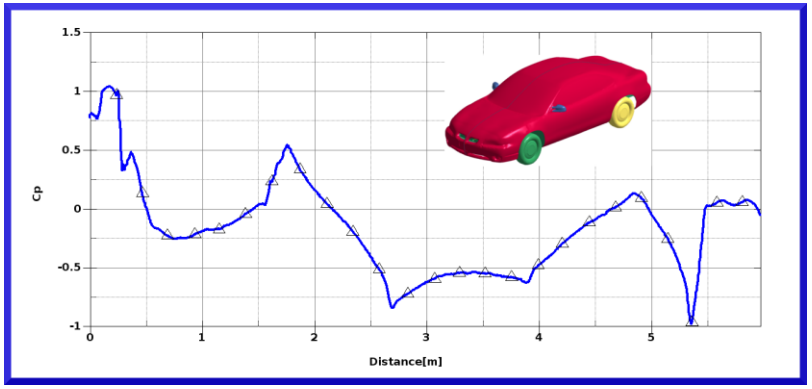
Original CAD geometry

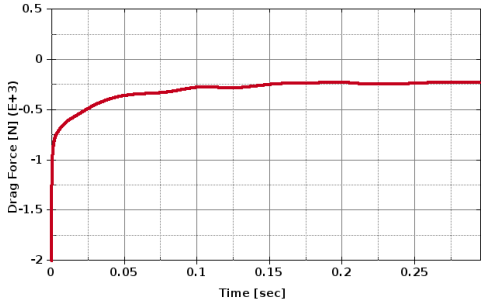
Surface mesh and automatic volume mesh

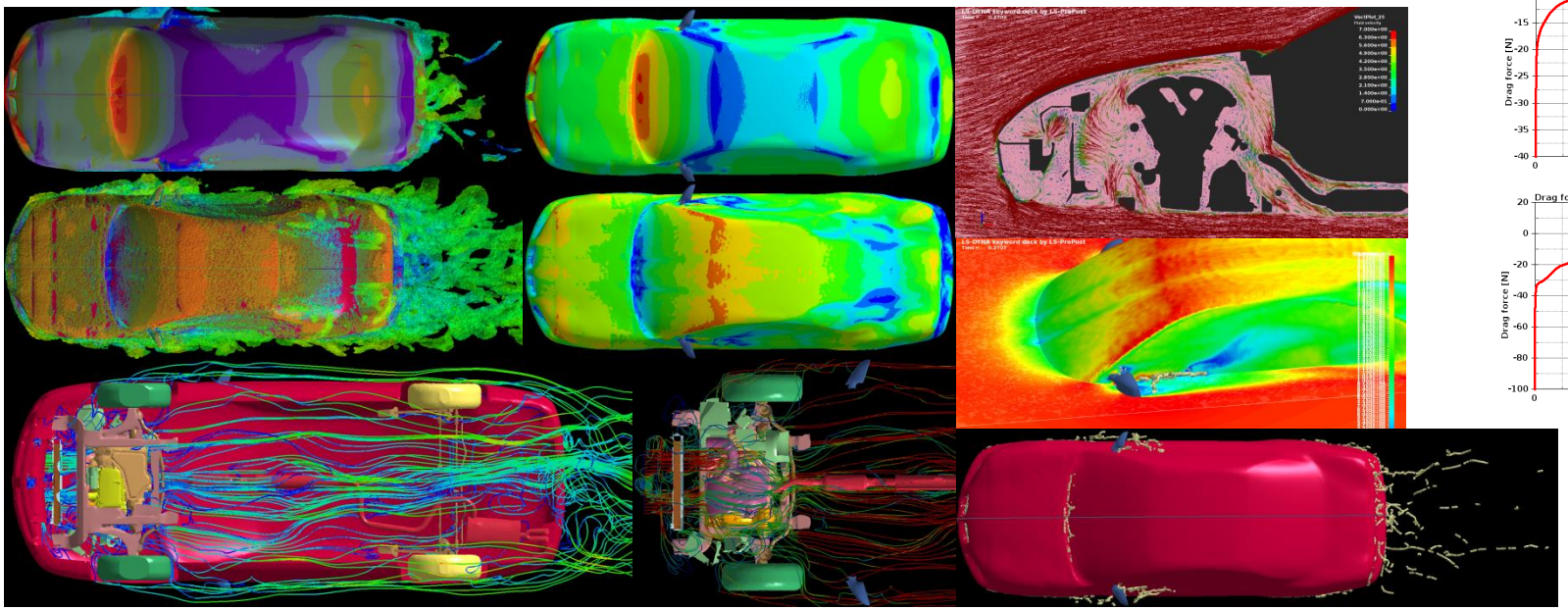


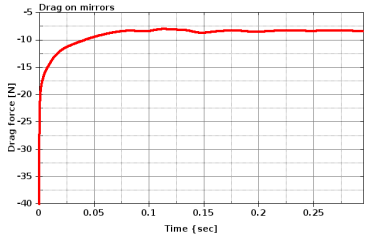


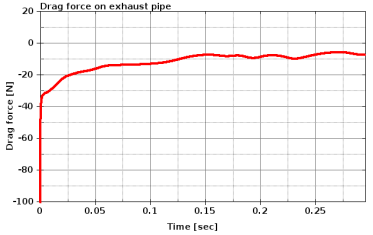
CFD analysis example on GM sedan model



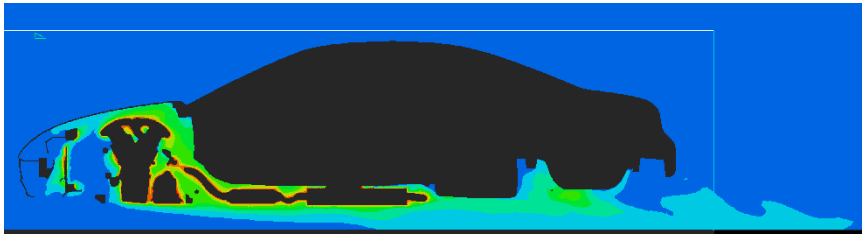




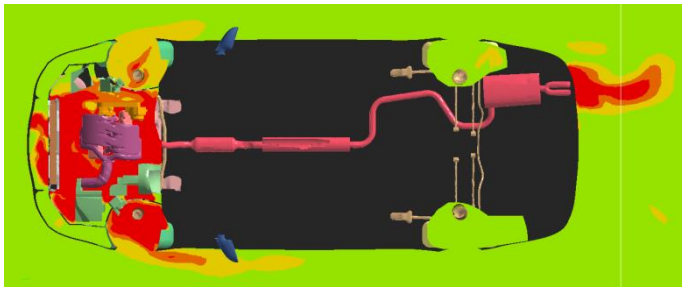


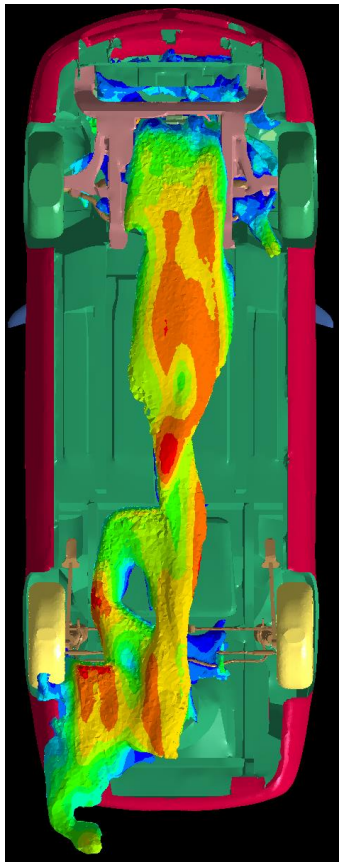


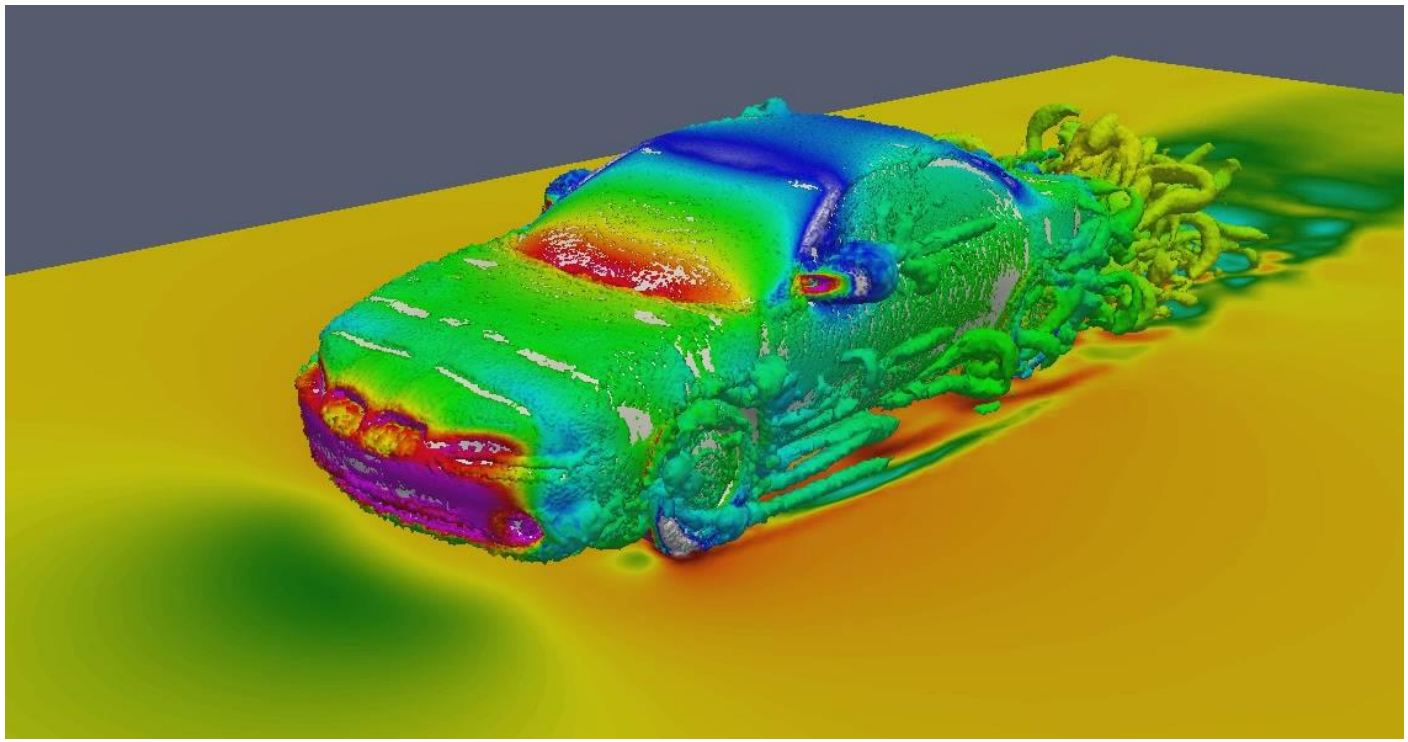
CFD analysis example on GM sedan model



Thermal analysis for engine and exhaust.

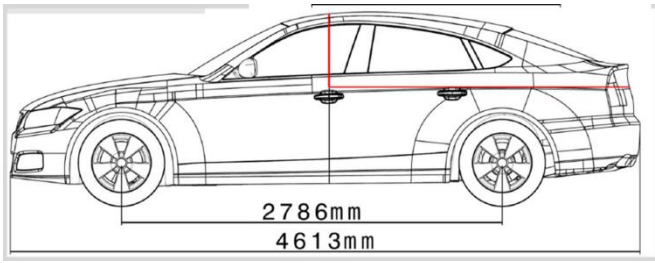


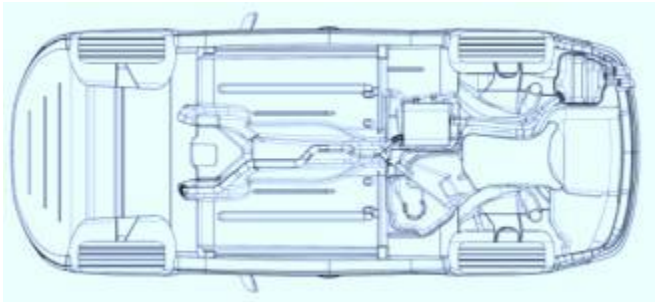




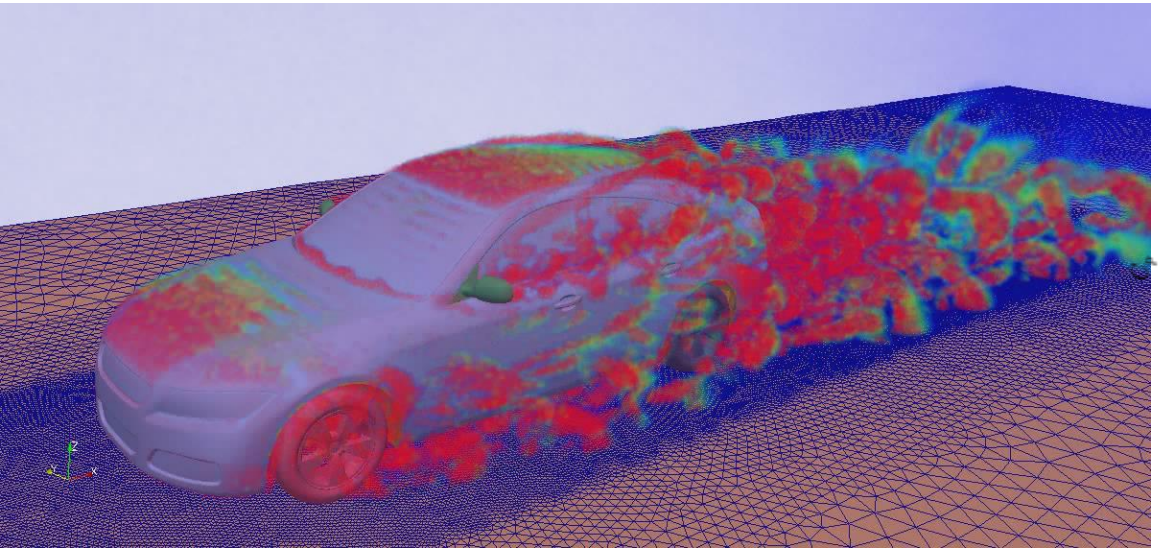
The DrivAer benchmark is part of the QA

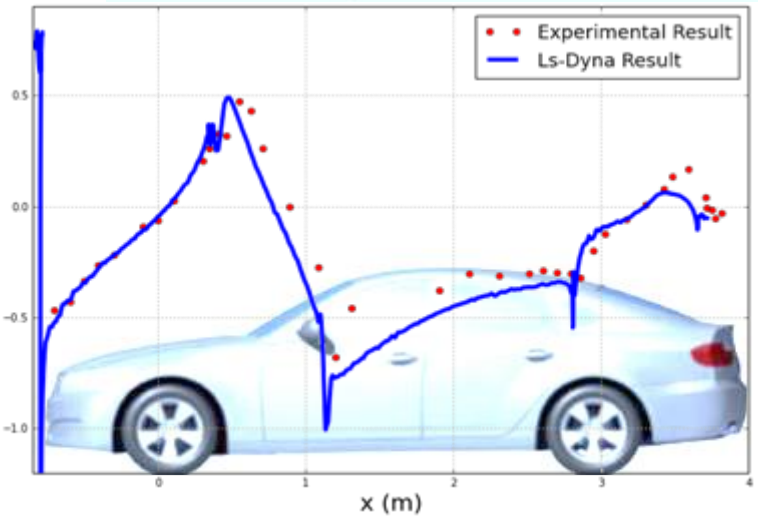
- Designed by TUM, Inst. For Aerodynamics.
- The objective is to perform automotive aerodynamics validation.
- It is a generic reference model with a modern car geometry.
- There is wind tunnel experimental data for comparison.
- LS-DYNA provides excellent agreement with the experimental data.



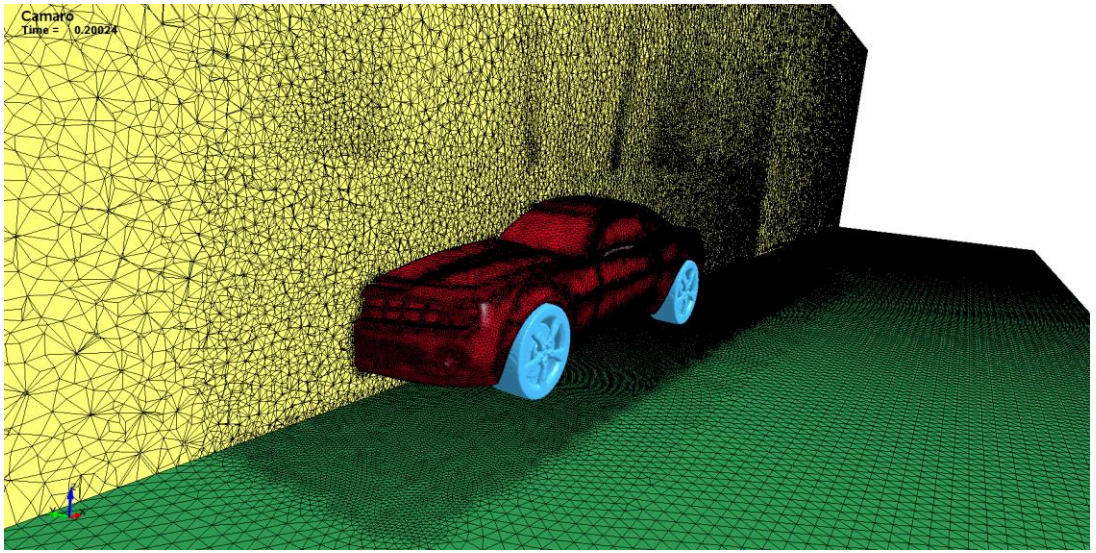


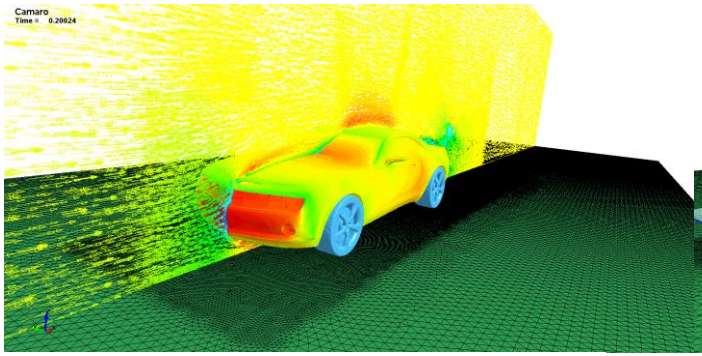
Configuration used in the study F_D_wM_wW
Fastback_Detailed underbody_with Mirrors_with Wheels

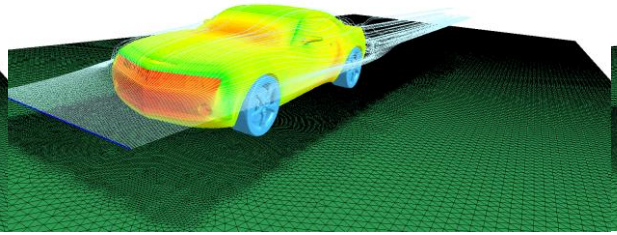


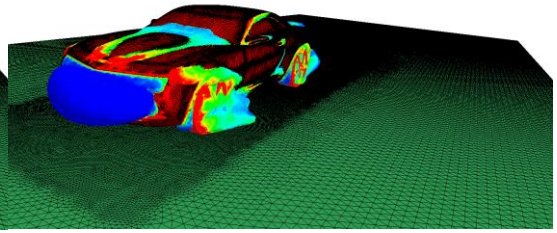


CFD+FSI analysis

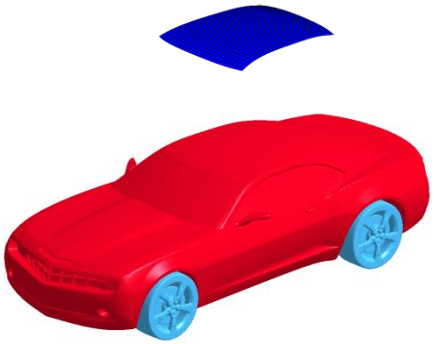


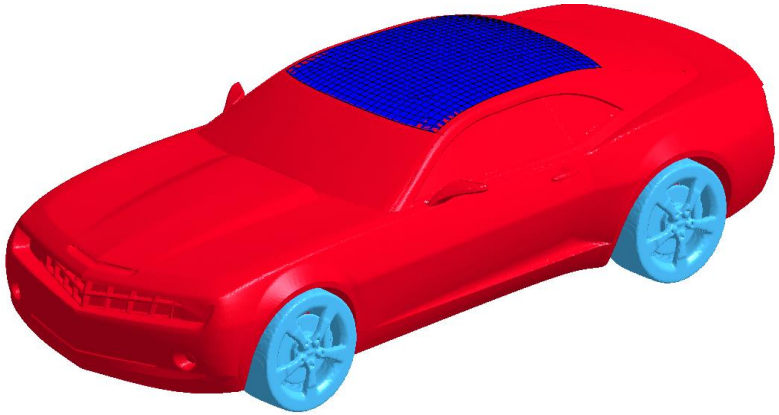


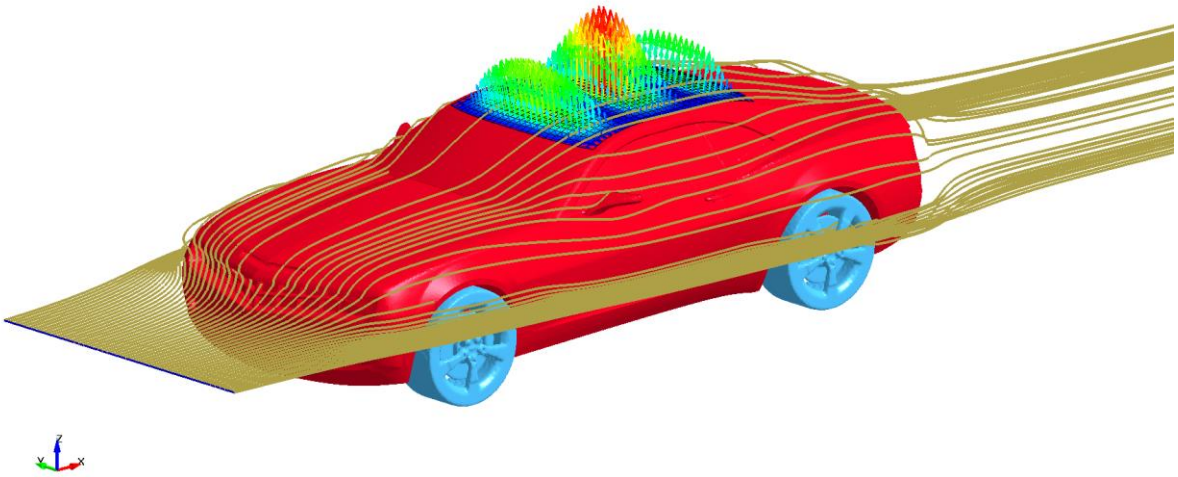




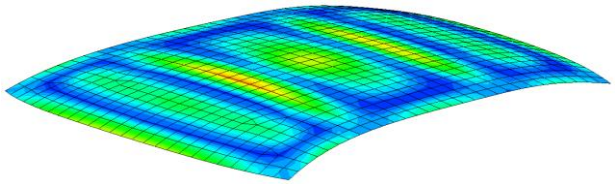
CFD+FSI analysis

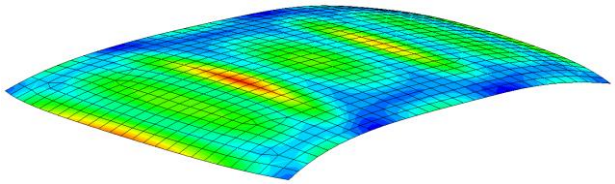






Structural pressure and Stress

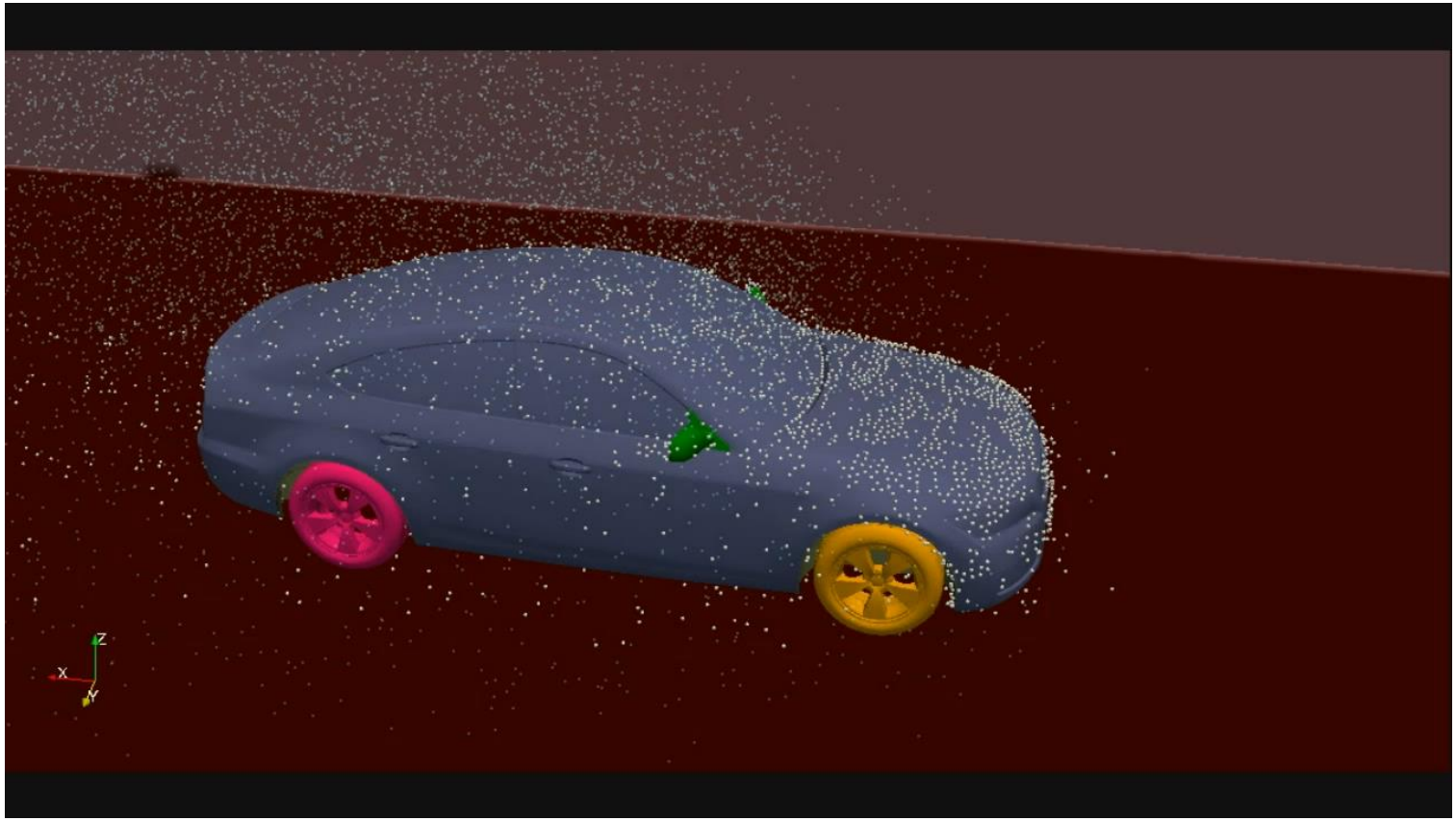




DEM coupling

- ✓ It was requested by the automotive industry to study mud and snow deposition on vehicles.
- ✓ It has potential in the area of drug delivery, erosion of river bed and some types of FSI by using the particle bonding feature.

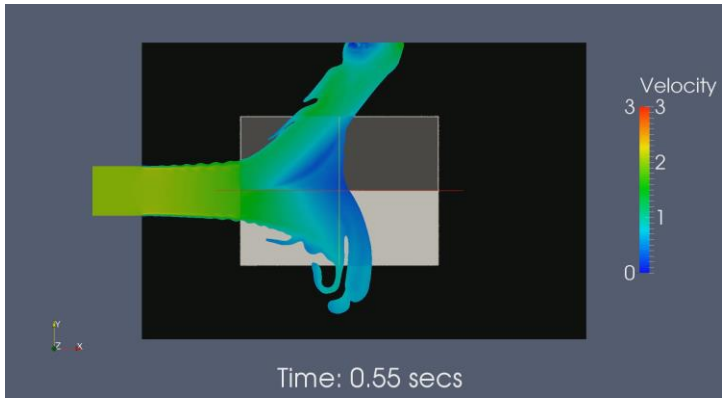
Example of application using DEM with capillary force coupled to a turbulent flow.



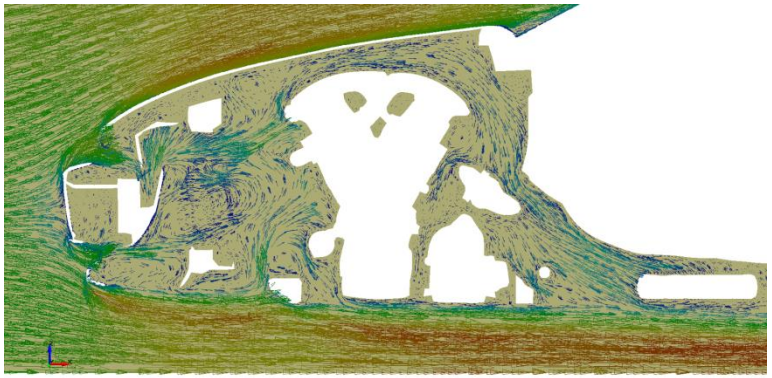
Flow in Anisotropic/Isotropic Porous Media

- ✓ Modeling of Newtonian/Non-Newtonian flows over the whole range of variation of porosity material parameters:
  {porosity → 0, permeability → 0} and {porosity → 1, permeability →∞},
  i.e. all Reynolds numbers.
- ✓ Heat Transfer and Free-Surface flow capabilities.

Mold filling with anisotropic material:

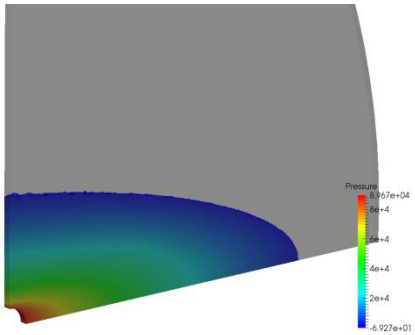


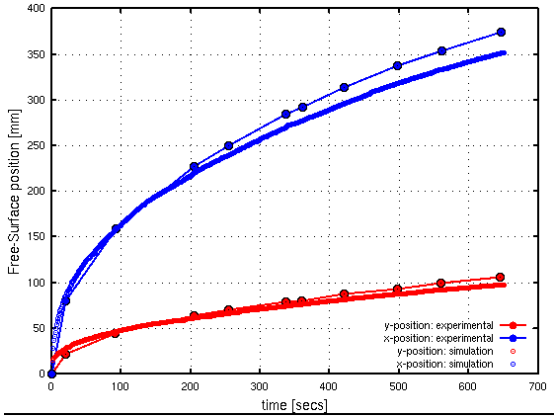
High Reynolds flow through a car radiator:



Pure Darcy-Forchheimer Anisotropic Porous Media Flow:

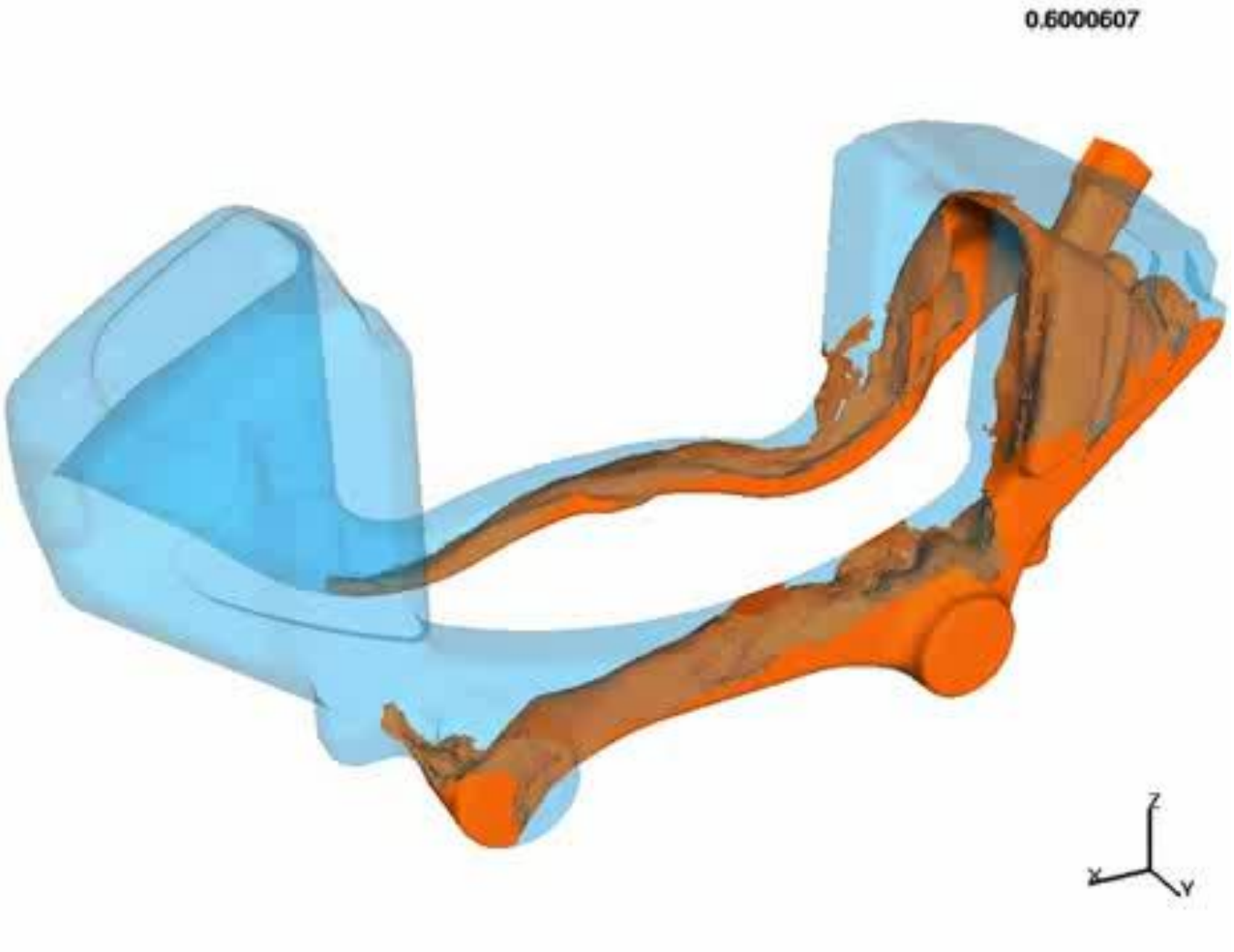
Validation: Sandia Nat. Lab. experiment SD06
(www.sandia.gov/wind/other/040076.pdf).





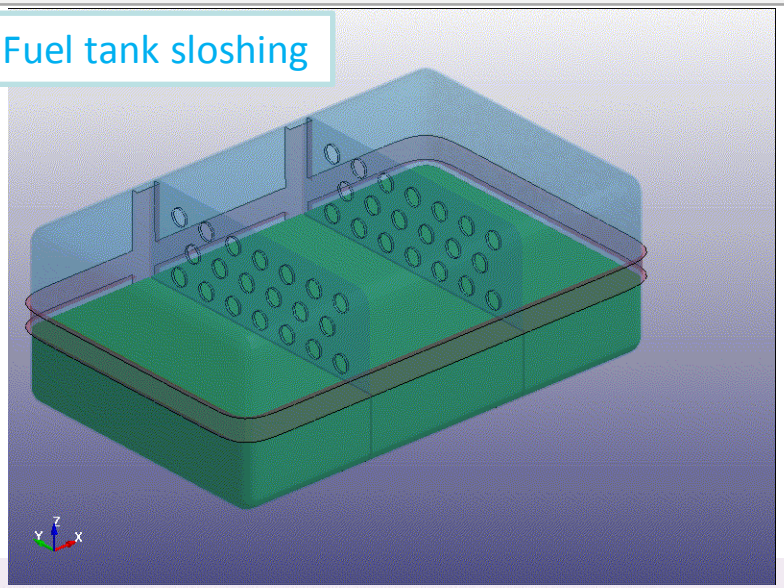
ICFD – Mold filling

Level set for mold filling applications
Viscosity function of temperature laws to simulate solidification process

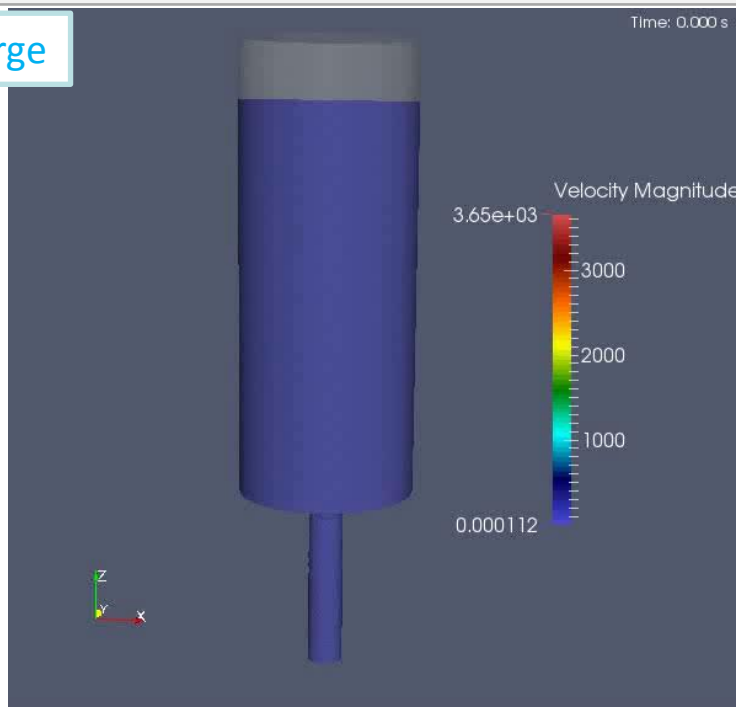


ICFD – Pressure forces on a body

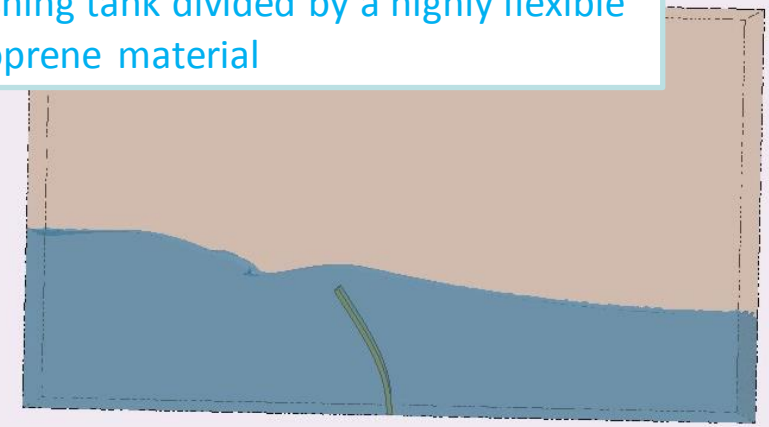
Fuel tank sloshing



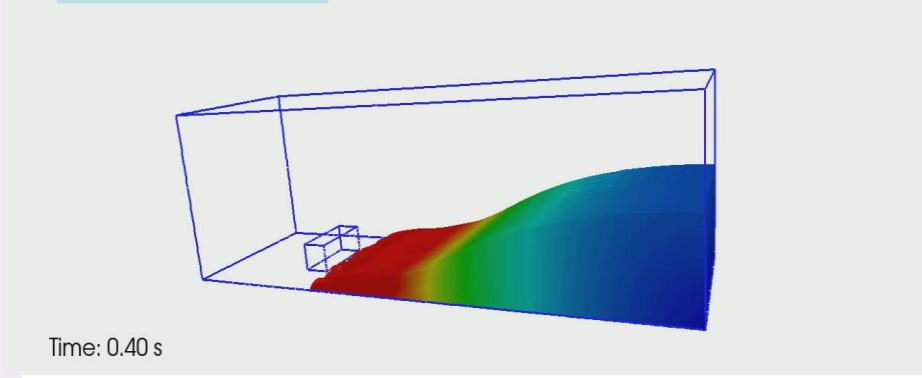
Liquid discharge



Sloshing tank divided by a highly flexible Neoprene material

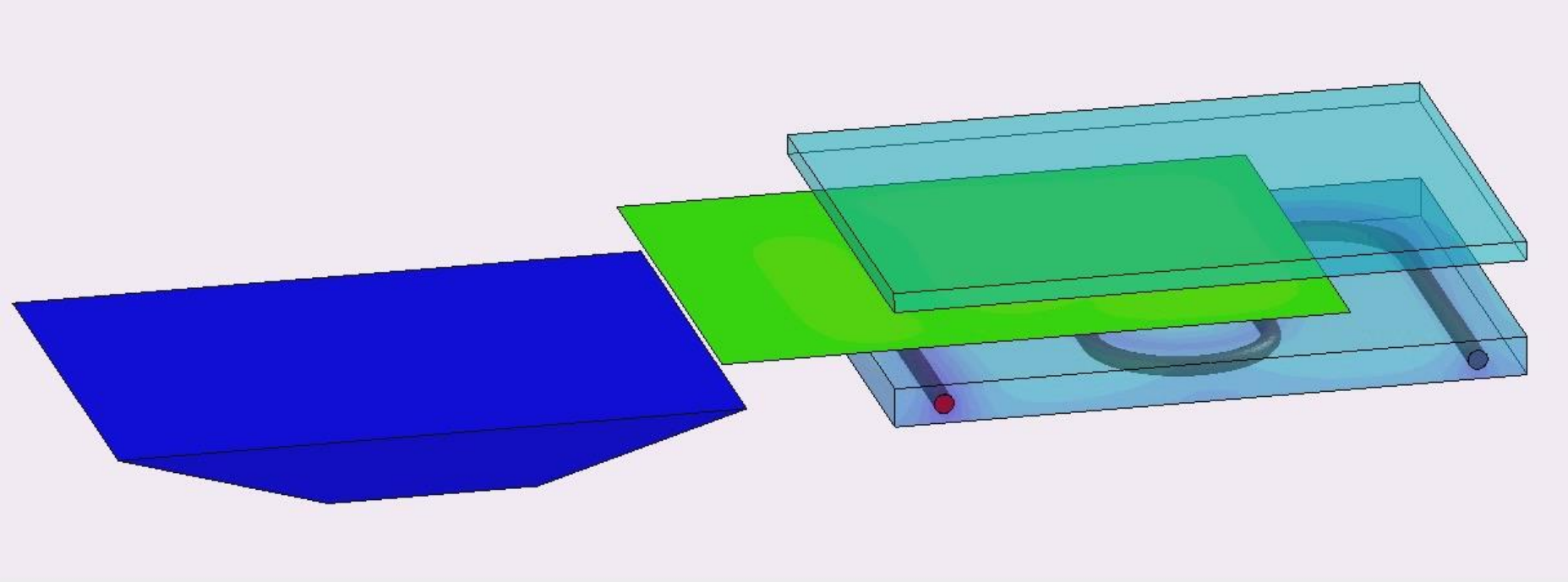


Wave Impact



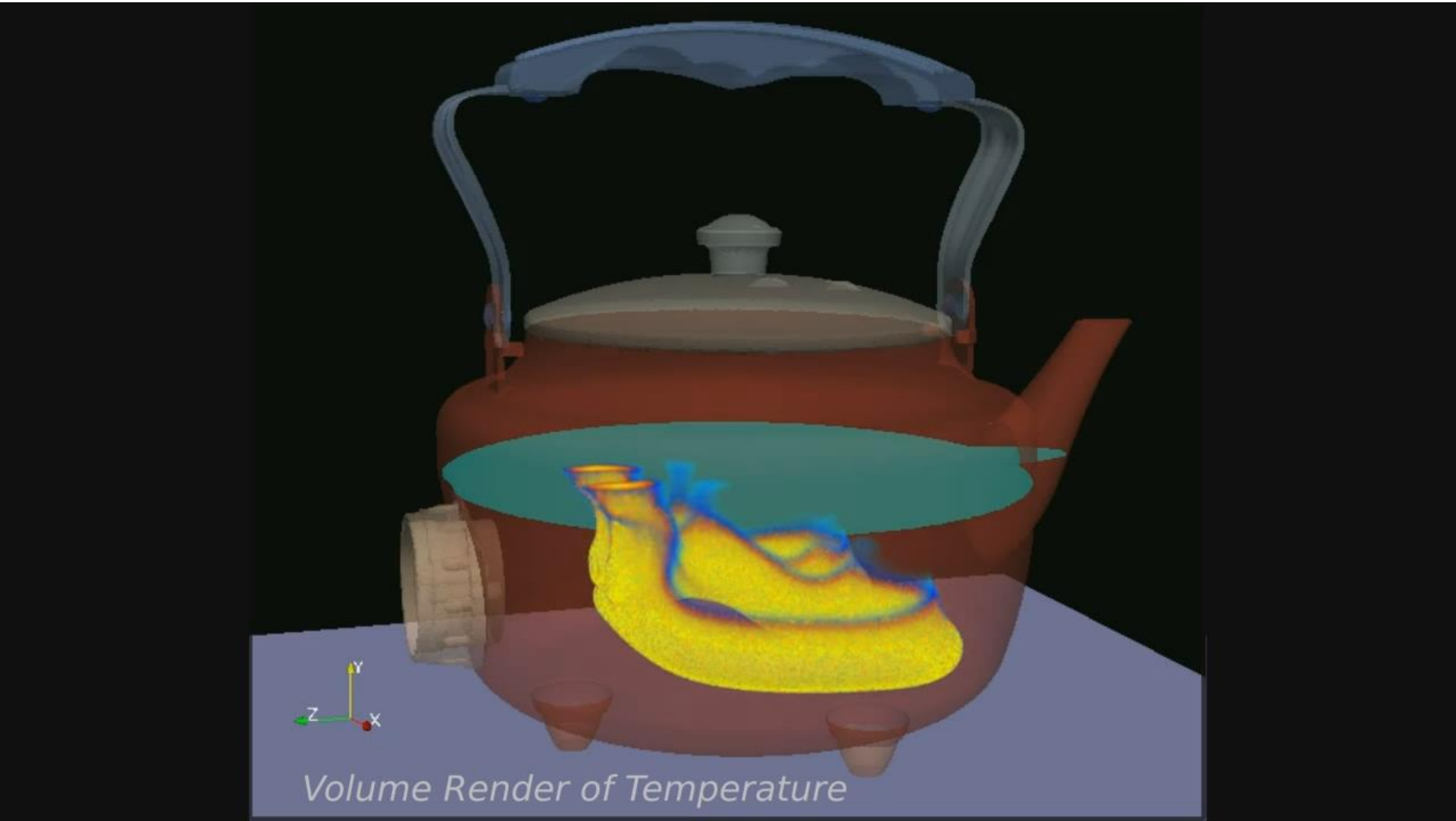
ICFD – Conjugate heat transfer

Stamping and subsequent cooling



ICFD – Conjugate Heat Transfer + EM

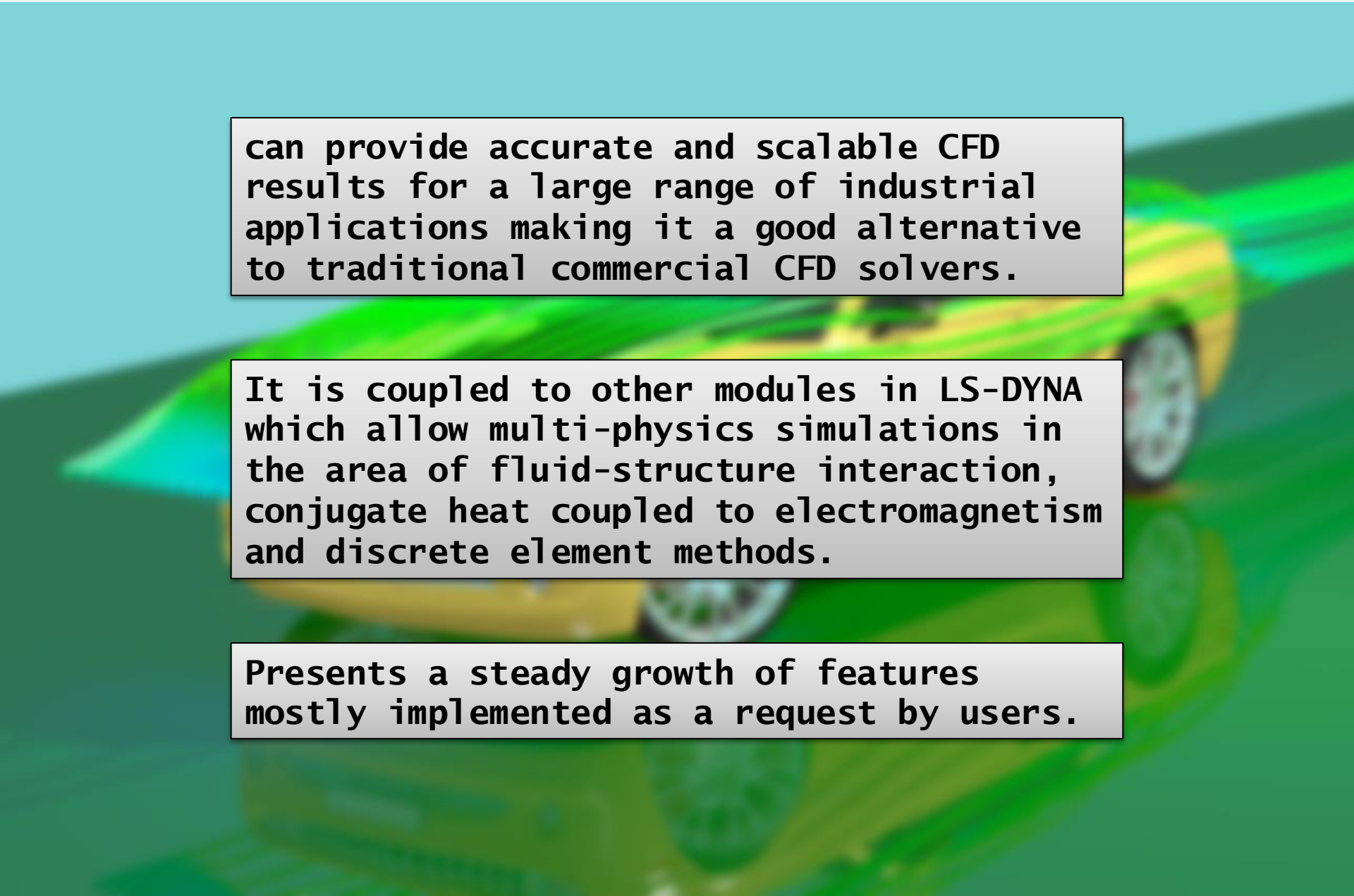
EM and conjugate heat transfer: An electric current connected to the coil generates heat which is transferred to the fluid. The heat induces a flow motion due to natural convection.



To Learn more

- **On the LSTC website, a special section is dedicated to the ICFD solver : http://www.lstc.com/applications/new_multiphysics**
- **The ICFD web menu includes a “Features” section, a “Test cases” Section, a “Gallery” and a “Documentation” section.**
- **In the “Test Cases” section, PDF documents are available describing some results obtained on reference validation cases (usually simple geometries and phenomena but good precision required)**
- **Feedback from users is welcomed.**

In summary the ICFD solver:



can provide accurate and scalable CFD results for a large range of industrial applications making it a good alternative to traditional commercial CFD solvers.

It is coupled to other modules in LS-DYNA which allow multi-physics simulations in the area of fluid-structure interaction, conjugate heat coupled to electromagnetism and discrete element methods.

Presents a steady growth of features mostly implemented as a request by users.