Recent Developments and Roadmap Part 3: Incompressible CFD

12th International LS-DYNA User's Conference

June 5, 2012

LSTC
Livermore Software Technology Corp.

Outline

- Introduction
- Recent developments

LS-PrePost	Mr. Philip Ho
Dummies	Dr. Christoph Maurath
Incompressible CFD	Dr. Facundo Del Pin
Electromagnetics	Dr. Pierre L'Eplattenier
ALE, DEM, SPH, Particle	Dr. Jason Wang

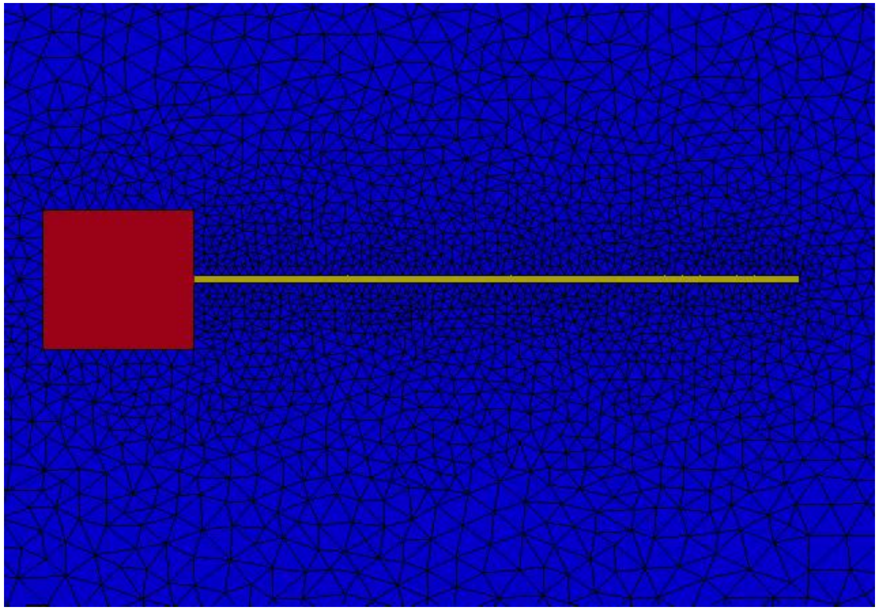
- Conclusions

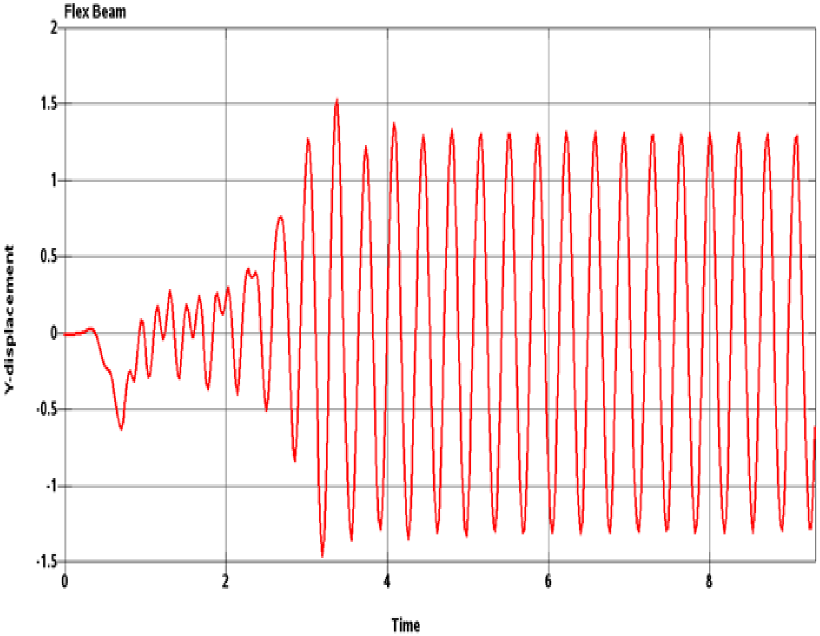
Incompressible CFD

Dr. Facundo Del Pin

Introduction to the Incompressible CFD solver

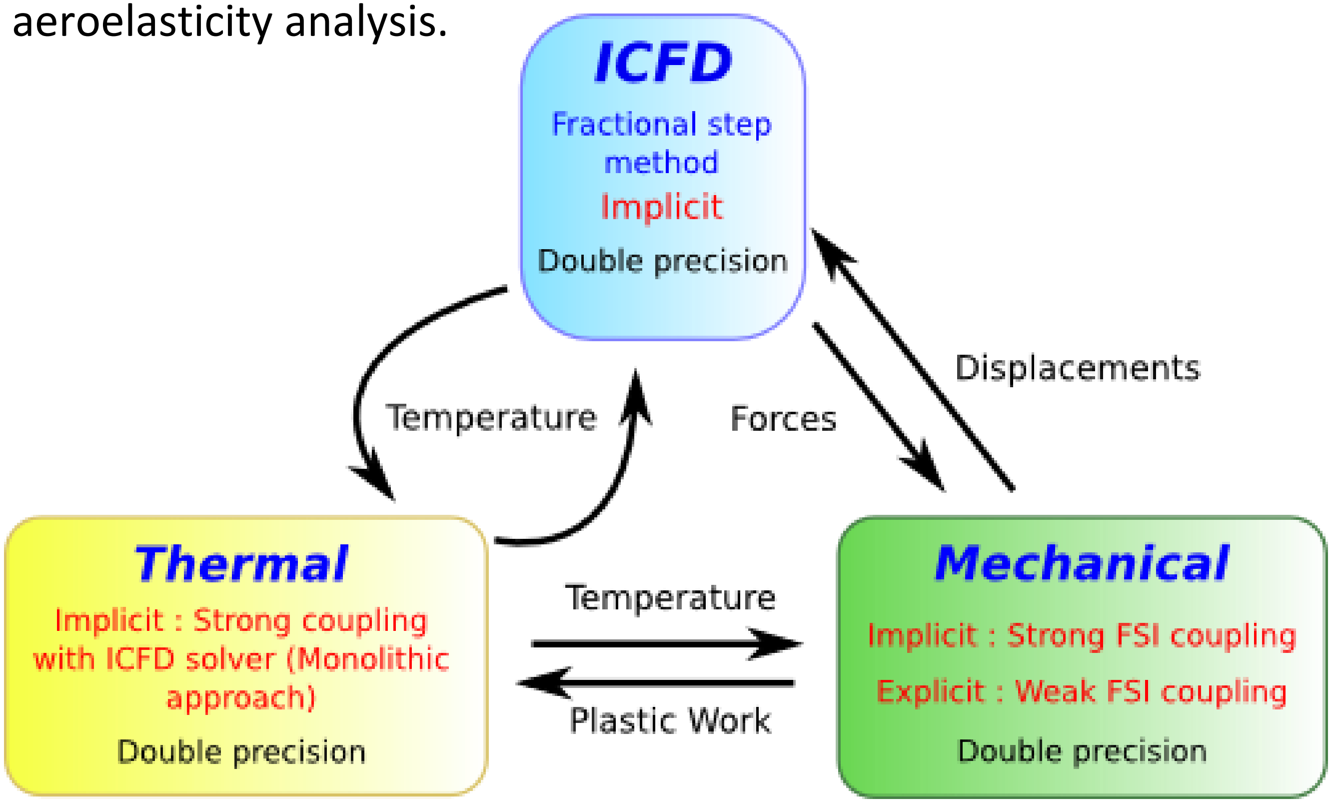
- Stabilized **Finite Element formulation** for the **Fluid Mechanics Navier-Stokes** equations
- **Free surface** capabilities and multi-phase approximations,
- Can run as a **stand alone implicit CFD** solver or be coupled with the structural (**FSI problems**) and thermal solvers of LS-DYNA,
- **ALE approach for mesh movement**, all **FSI boundaries are Lagrangian** and deform with the structure.





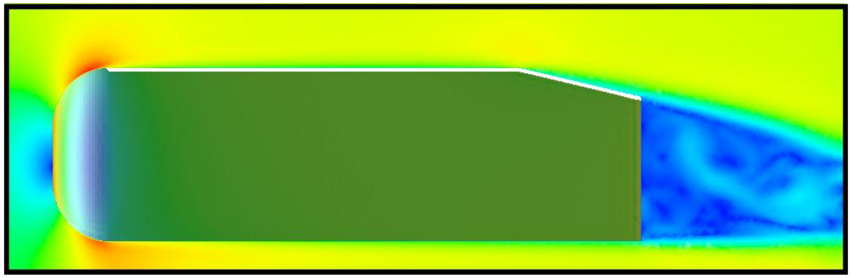
Coupling with other LS-DYNA solvers

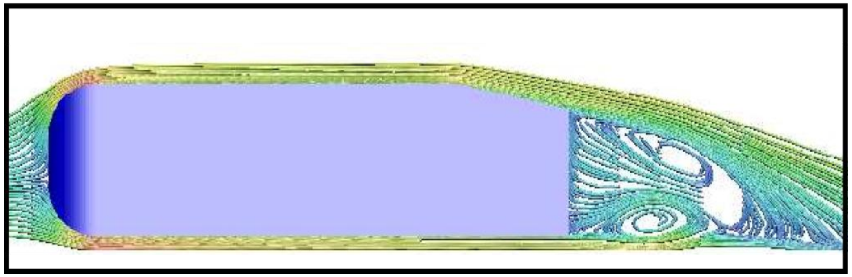
- Scope of the new 980 solvers to be coupled with LS-DYNA solvers in order to solve complex **fluid-structure** or **thermal** problems,
- **Strong coupling** is available for implicit mechanics. More robust but more costly,
- **Loose coupling** for explicit mechanics. Less robust and less costly. Suitable for simpler couplings.
  - E.g. aeroelasticity analysis.



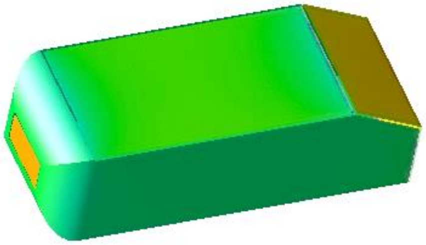
Applications : Aerodynamics

Ahmed bluff body example
Benchmark problem

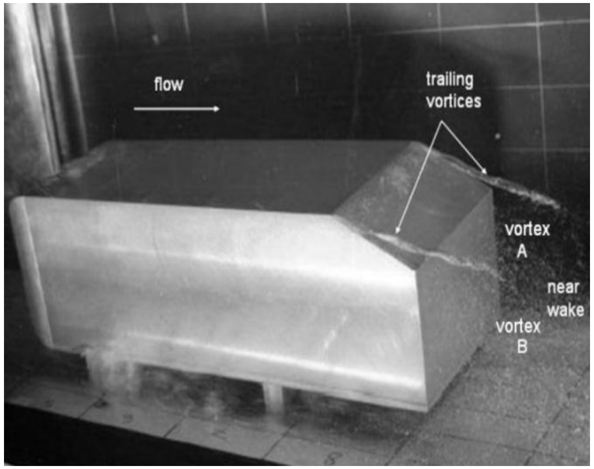




Drag calculation and Study of vortex structure,
Turbulence models available for solving
Can run as a CFD problem with static body or be transformed in a FSI problem with moving body (Eg : pitch or yaw movement).



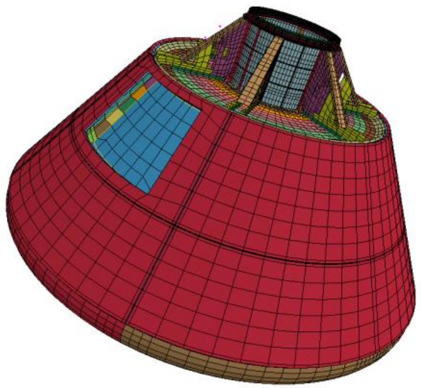
time[sec]: 0.000000

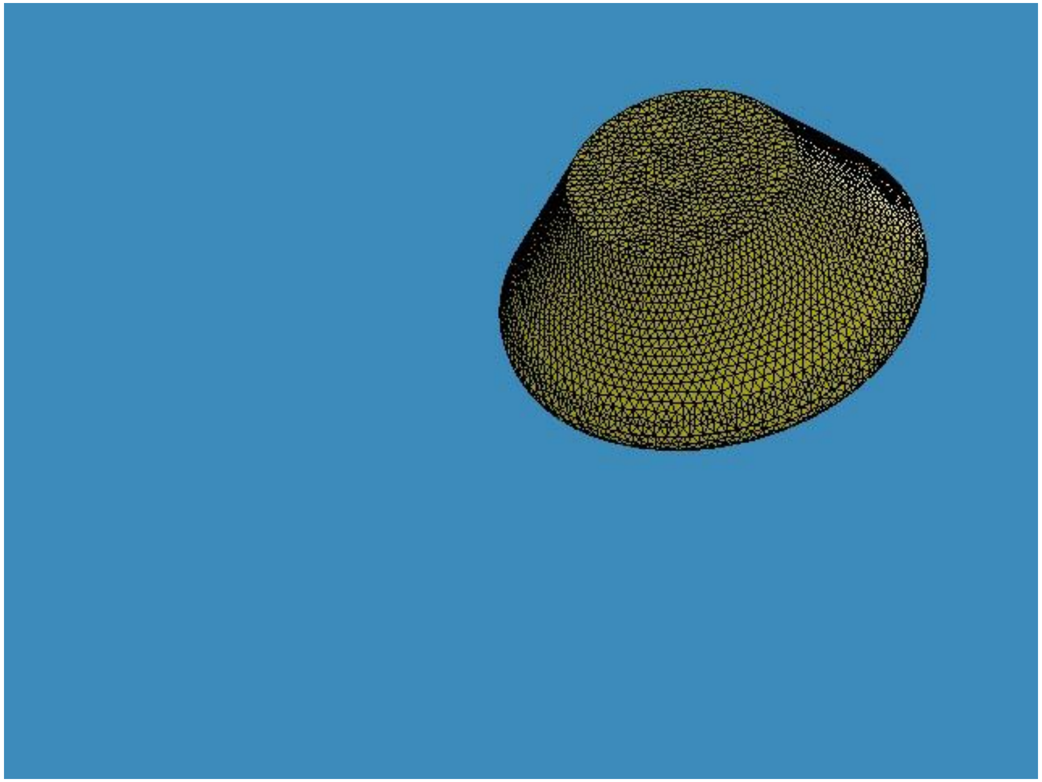


Applications : Slamming

Space Capsule impact on water (Slamming problem) :

Derived from Orion water landing module
/awg.lstc.com LS-DYNA Aerospace Working Group,
NASA NESC/GRC





Free fall impact / Strong FSI coupling

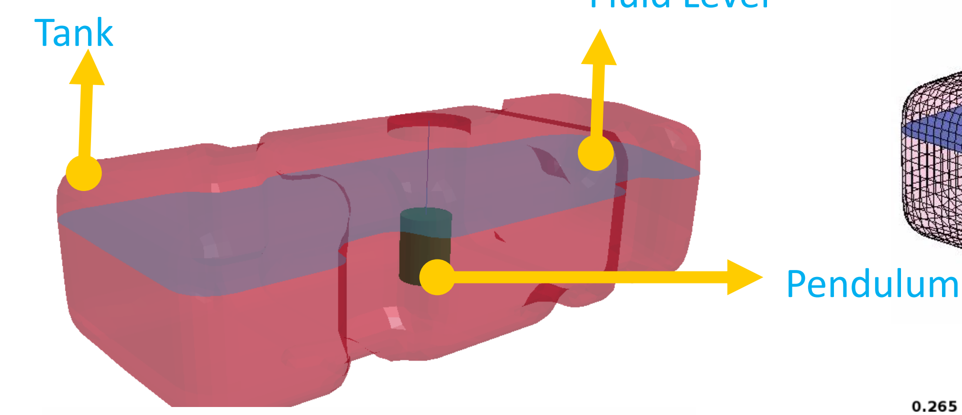
Proof of feasibility using ICFD solver

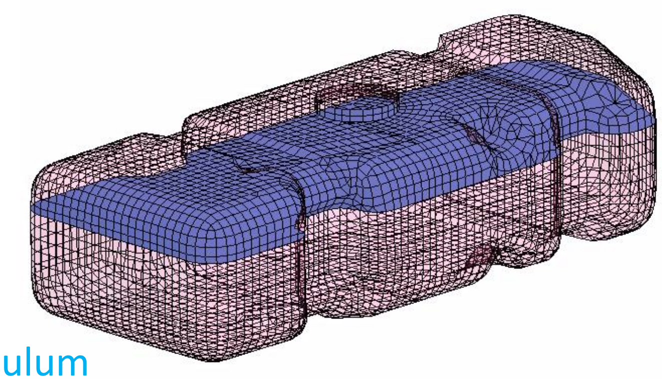
May be applied to similar Slamming problems

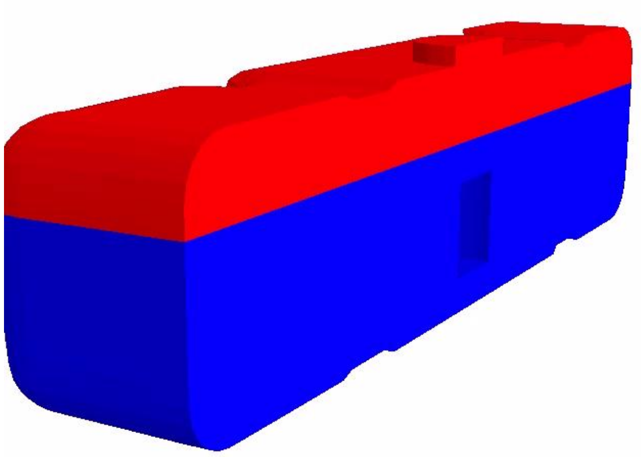
Sloshing

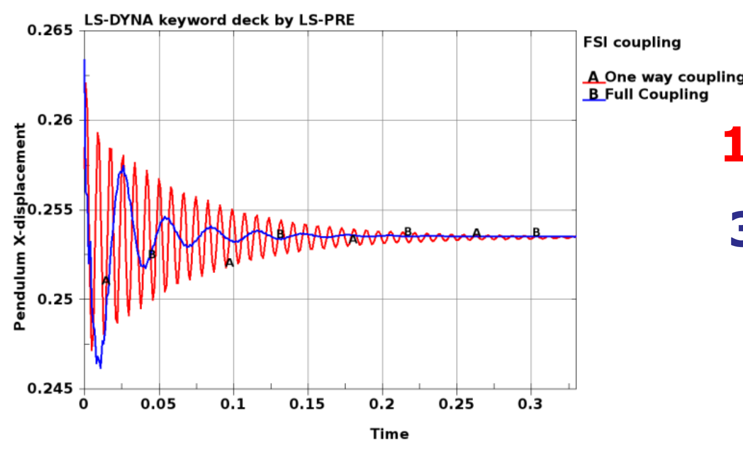
Water Tank example :

Moving Water Tank coming to a brutal halt,
Sloshing occurring,
Study of pendulum oscillations.





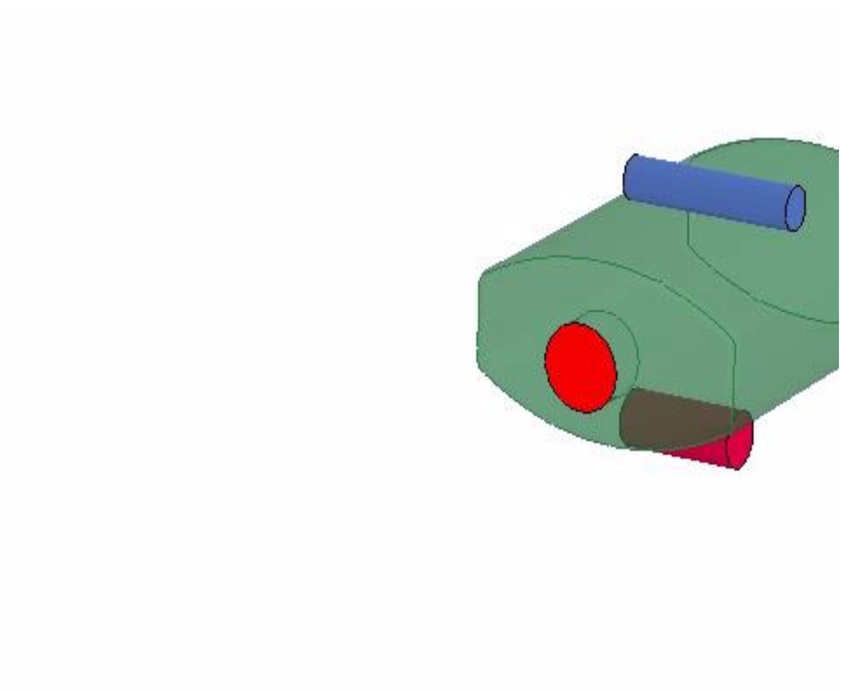




125 Hz

35 Hz

Strong FSI Coupling



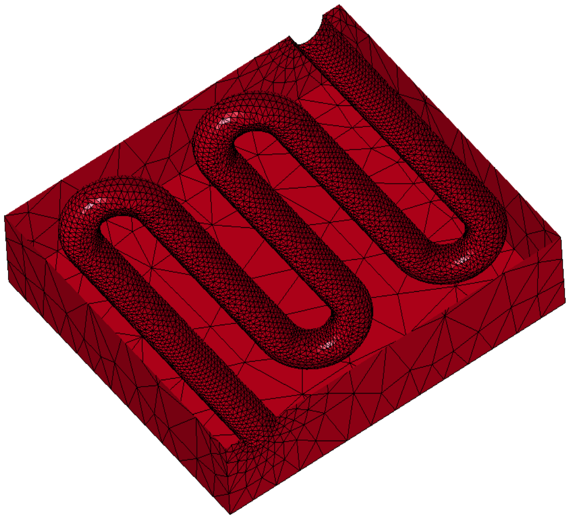
High viscosity Liquid coming out of bottle due to finger pressure

Simulation of synthetic heart valve. Density of Solid and Fluid (blood) very close: complex FSI strong coupling case.

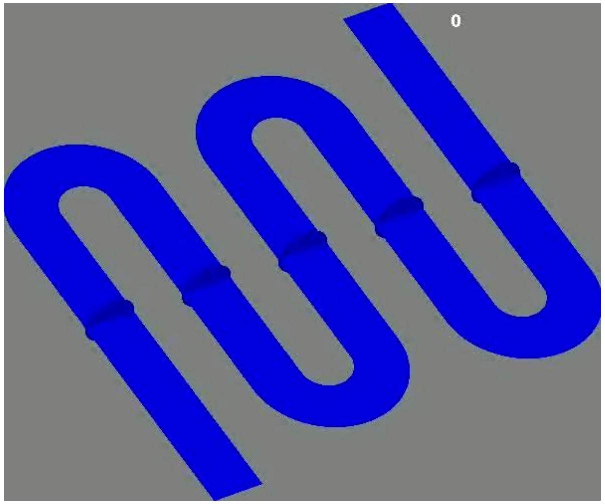
Conjugate Heat Transfer

Monolithic strong coupling between the solid and fluid thermal solvers providing good stability

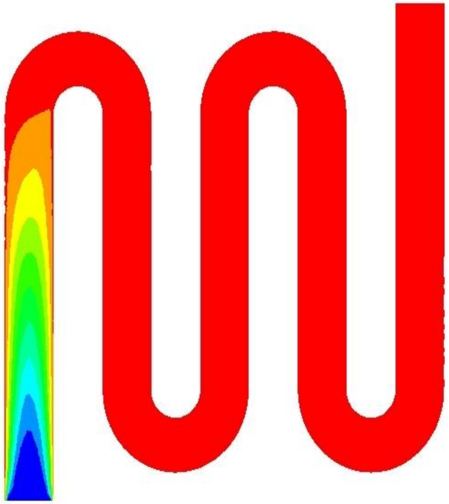
Thermal pipe flow example



Solid Mesh



Fluid velocities and Conjugate heat analysis (cut plane of fluid vel.)



Steady state fluid temperature (cut plane through the section)

Range of Applications

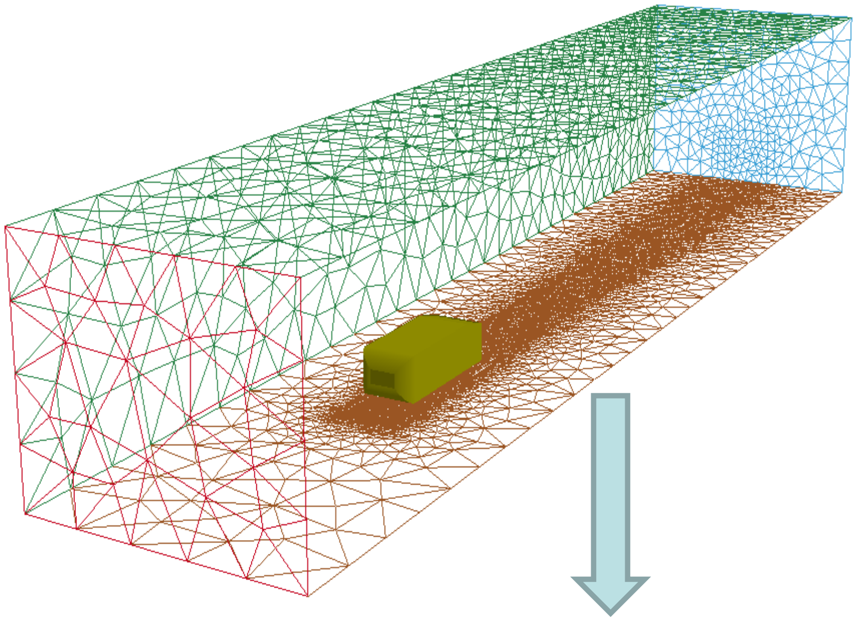
- **Most flows that we encounter in our daily activities are incompressible,**
- **Low Mach number (*Ma < 0.3*).**
  In Air : *Vel < 230 MPH , 370 KPH*.

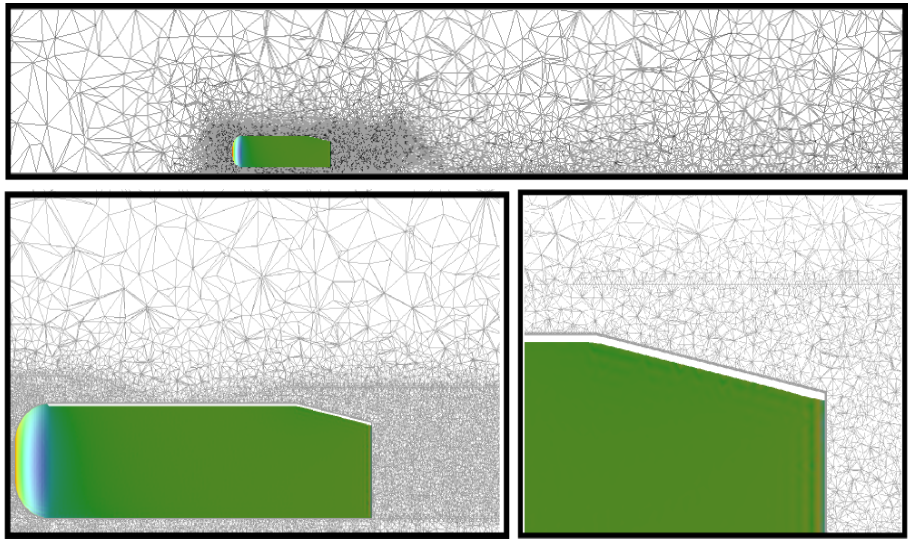
Examples:

Ground vehicle aerodynamics, Free surface and Multi phase flows, Wind turbines, Human body, Ship hydrodynamics, etc.

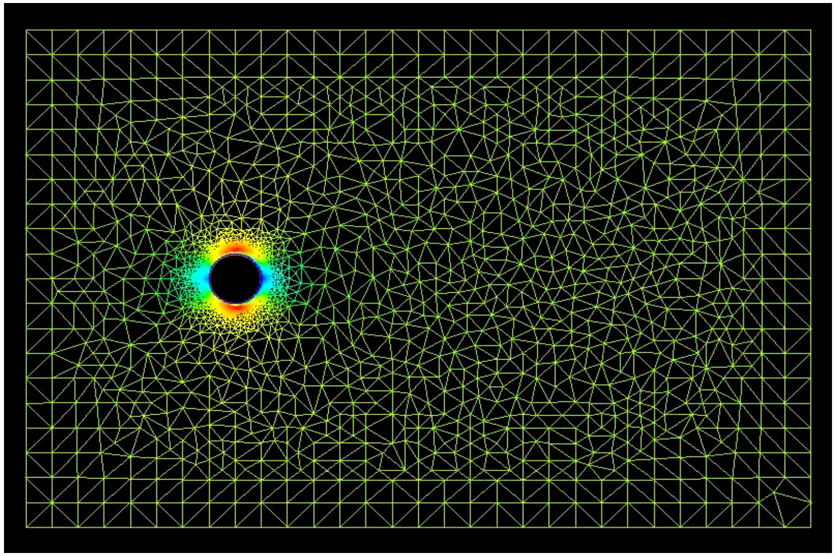
Automatic Fluid Volume Mesher

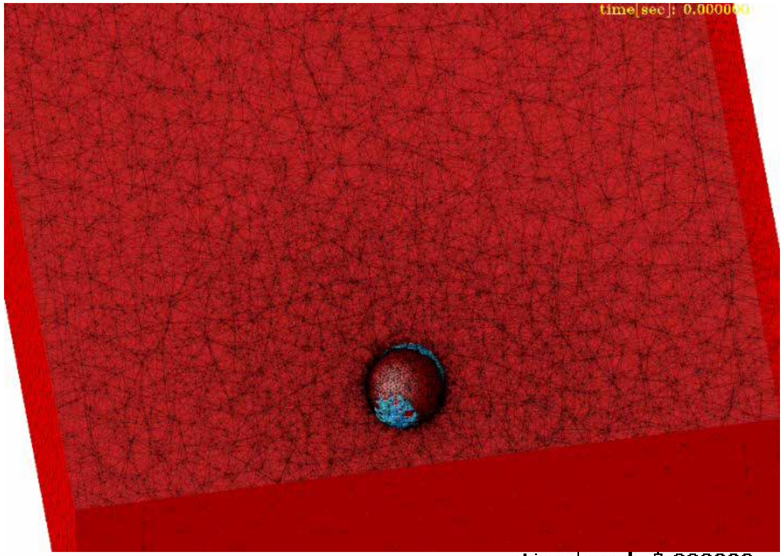
- In complicated geometries meshing for CFD problems could be a time consuming process for any commercial software
- **Simplification of the pre-processing stage**,
- Possibility to **specify local mesh size** for better resolution,
- Possibility to add **Boundary layer mesh**.
- Error estimators may be used to **automatically adapt the mesh**.



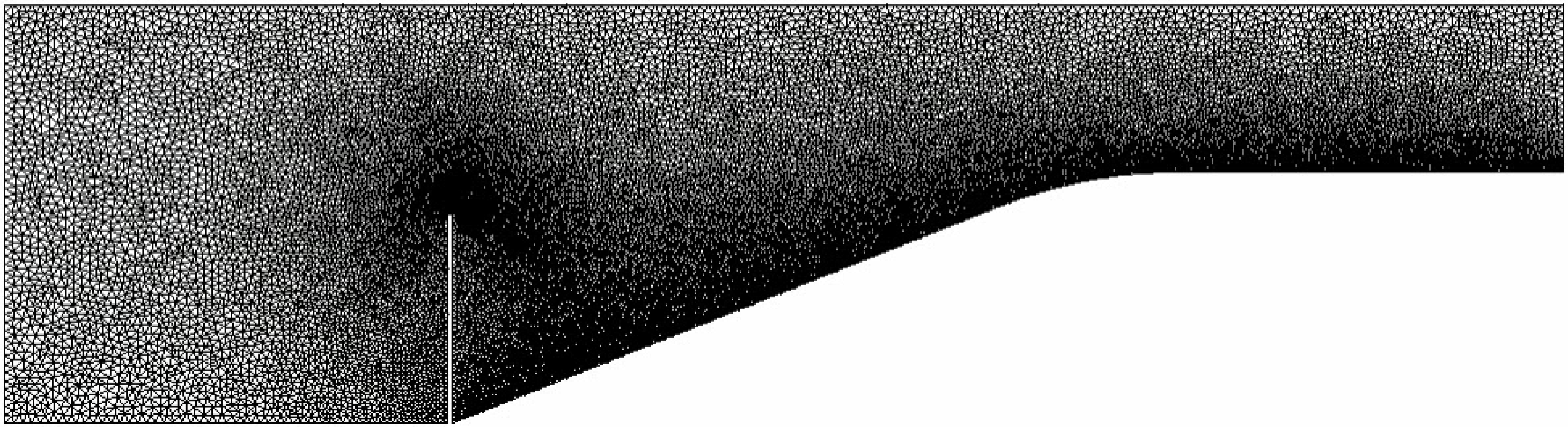


Error Control and Adaptive Re-Meshing



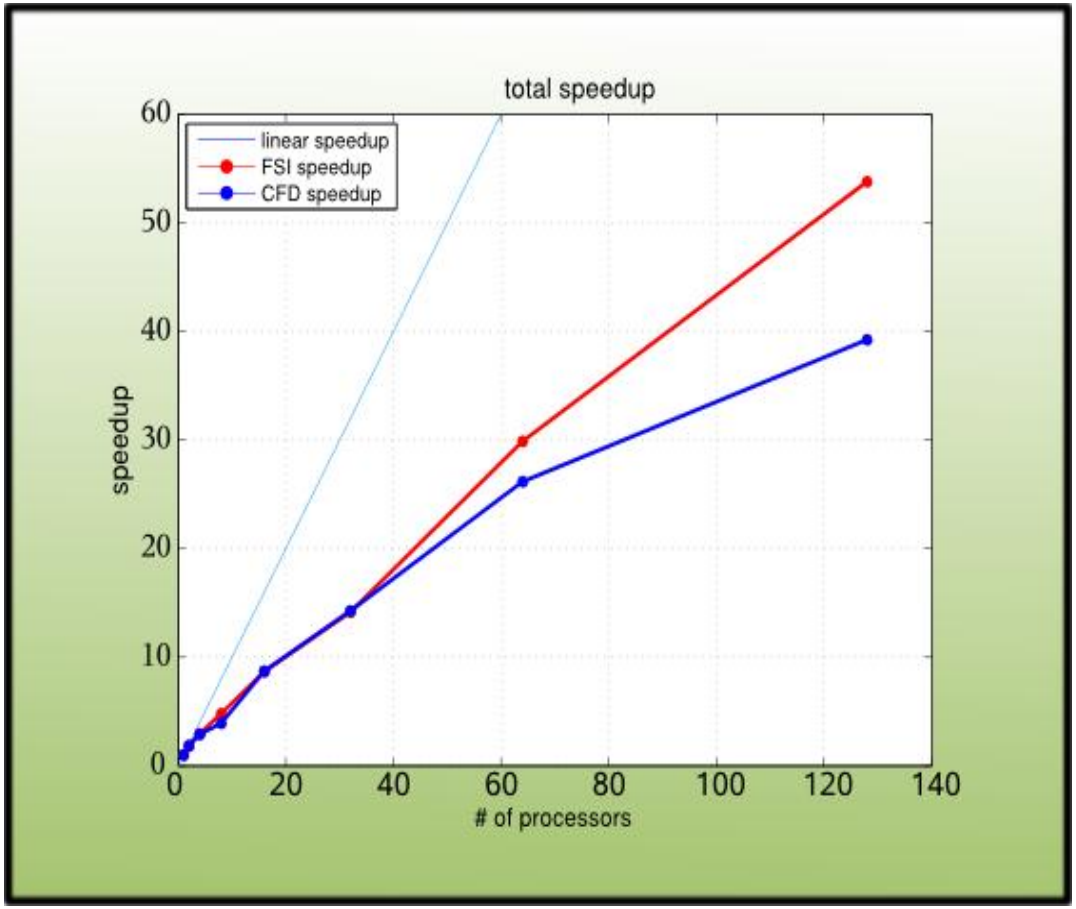


time[sec]: 0.000000



MPP Scalability: Real Car Model

The results show a **speedup of 40 for 128 cpus in the CFD** only case (2.1 M elements) and a **speedup of 55 for 128 cpus in the case of FSI** (3.6 M elements).



For the next development cycle further improvements will be implemented

Incompressible CFD Roadmap

Validation/Benchmarking process under way (problems will include FSI, Conjugate Heat transfer cases as well as more Aerodynamics and Free surface analyzes).

Additional post treatments and Tools with LSPP 3.2.

See Website for additional documentation

http://www.lstc.com/applications/icfd

Thank You !