Exploring FSI Simulations with LS-DYNA ICFD: Background and Capabilities

Satish Kumar M
Application Engineer – II (Structural Mechanics)

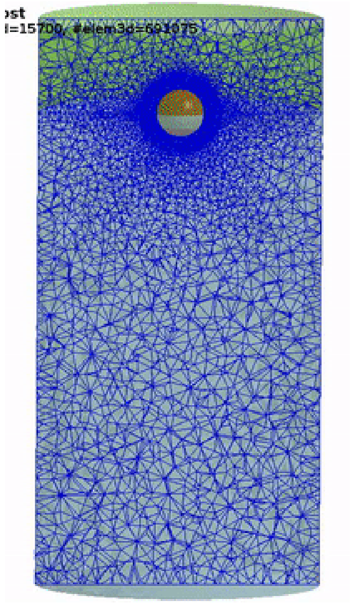
Facundo Del Pin
Senior Principal R&D Engineer (LS-DYNA)

Ansys

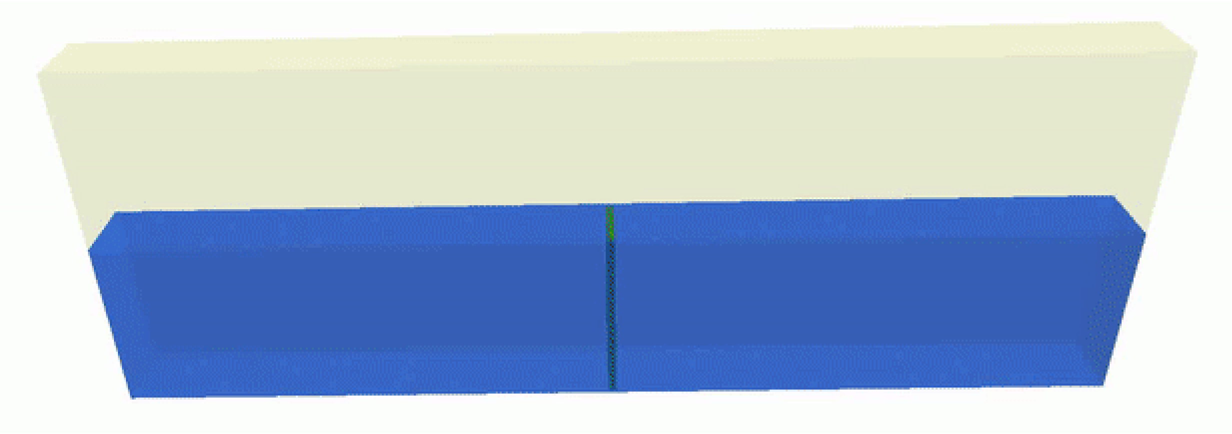
Contents

Overview

- LS-DYNA® is a general-purpose finite element program capable of simulating complex real-world problems. The code's origins lie in highly nonlinear, transient dynamic finite element analysis using explicit time integration.
- Over the years, many functionalities have been added (Implicit solver, Thermal solver, ALE and CESE solvers for fluids, SPH, Electromagnetics) constantly improving the software's Multiphysics capabilities.
- In accordance with this vision, the ICFD solver offers advanced tools to solve complex fluid structure and thermal interaction problems.

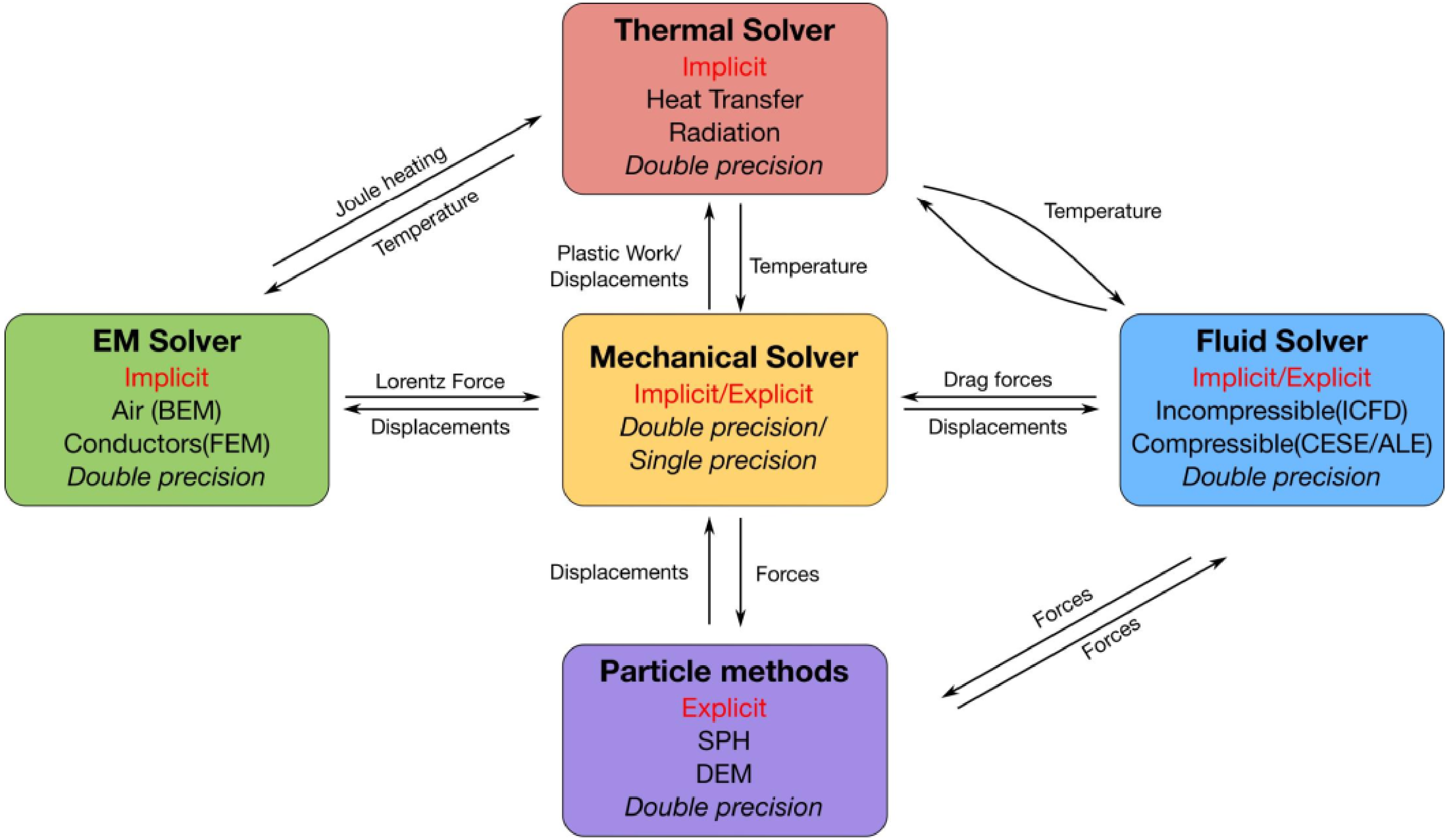


Ball submerging in a liquid.



Benchmark problem featuring a rolling tank with an elastic plate in between

LS-DYNA Multiphysics



History of LS-DYNA ICFD

R7 (2012-2013)

- Incompressible flow solver
- Automatic volume mesh generator
- Basic free surface and turbulent modeling
- Weak and strong FSI tools
- Conjugate heat transfer

R8 (2014-2015)

- Non inertial reference frame
- Addition of porous media
- Improvement on post treatment tools

R9 (2016)

- Addition of turbulence models (k-epsilon, k-omega, WALE)
- Addition of non newtonian models
- Non-Linear conjugate heat transfer
- Coupling with DEM

R10 (2017)

- Steady state solver
- Wave generator and wave damping
- Weak thermal coupling
- Linear fluid structure interaction tools

Current version is R14 (many more updates).

Release Notes: LS-DYNA R14 Release Notes

What is LS-DYNA ICFD?

- The LS-DYNA ICFD is an incompressible flow solver.
- It serves two purposes,
    - Standalone implicit flow solver
    - Flow-solver for FSI problems where the flow is in the incompressible regime.
- Commercial CFD solves for the below PDEs (aka. Navier Stokes Equations),

Conservation of momentum →

$$\frac{\partial \rho \vec{v}}{\partial t} + (\vec{v}.\vec{\nabla})(\rho \, \vec{v}\,) = -\vec{\nabla} p + \rho \vec{f} + \vec{\nabla}.\tau_{ij}$$

Conservation of mass →

$$\frac{\partial \rho}{\partial t} + \vec{\nabla}.(\rho v) = 0$$

Conservation of Energy →

$$\frac{\partial (\rho e)}{\partial t} + \vec{\nabla}.((\rho e + p)\vec{v}) = \vec{\nabla}.\tau_{ij}\,\vec{v} + \rho \vec{f}.\vec{v}$$

- But when it comes to incompressible flow some assumptions can be made which makes it possible to simplify the above equations.

What is LS-DYNA ICFD?

- A flow can be considered **incompressible** if **its Mach number is below** 0.3. Many flows meet this hypothesis:

Scenario	Mach Number
Ocean current speed	$M < 0.01$
Pipeline flow speed	$M < 0.05$
Typical German highway car speed	$M < 0.12$
Wind turbine (HAWT) survival speed	$M < 0.18$

- Neglecting temperature dependence, and assuming a constant fluid density throughout the domain, we get,

Conservation of mass → $\vec{\nabla}.(\vec{v}) = 0$

Conservation of momentum → $\rho \dfrac{\partial \vec{v}}{\partial t} + \rho(\vec{v}.\vec{\nabla})(\vec{v}) = -\vec{\nabla} p + \mu \Delta \vec{v}$

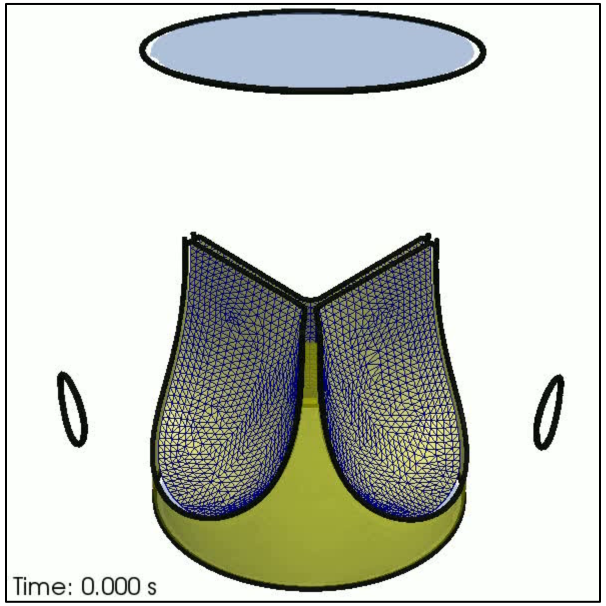
Three equations and four unknowns *has been reduced to* ***two equations and two unknowns.***

What is FSI?

Fluid–structure interaction (FSI) is the interaction of some movable or deformable structure with an internal or surrounding fluid flow.

Challenges in FSI Analysis:

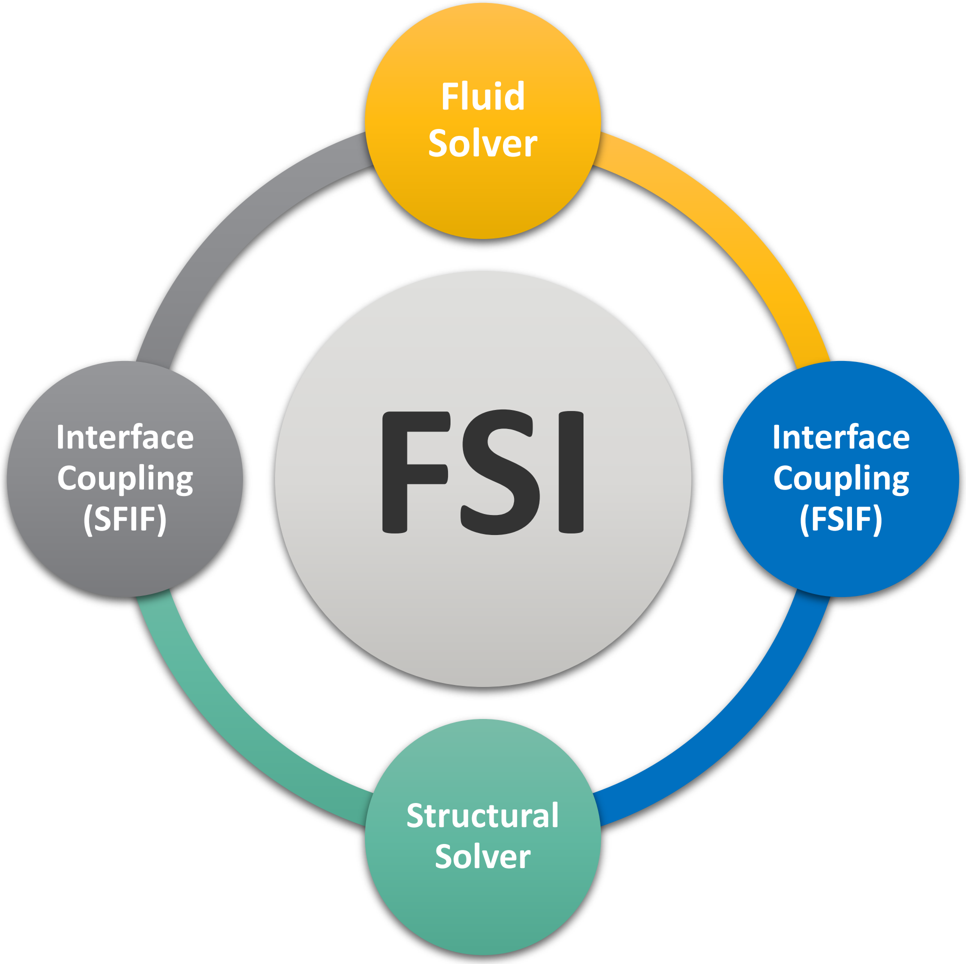
- Fluid dynamics.
- Structural dynamics.
- Coupling.
- Computational complexity.



Examples: FSI can be observed in various real-world scenarios, including:

- Aircraft wings or turbine blades interacting with airflow.
- Ships and offshore structures subjected to wave forces.
- Blood flow in arteries and veins interacting with blood vessel walls.

What is FSI?



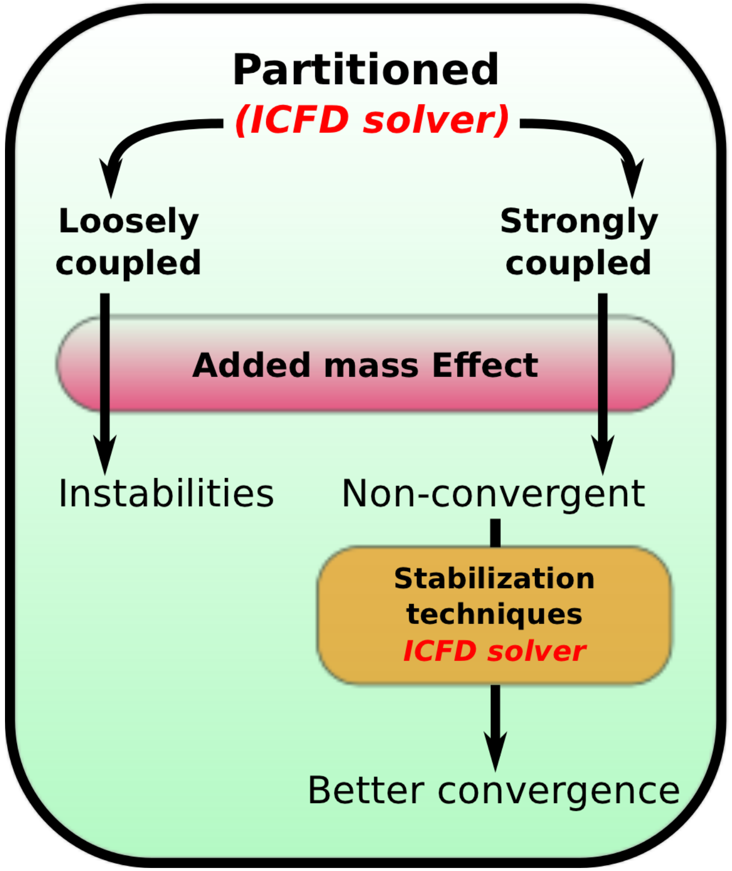
FSI Algorithm:

- Governing equations are solved simultaneously in a partitioned approach adding stabilization terms at the interface.
- Transfer of forces are accomplished using Interface coupling (FSIF and SFIF). These are computed as follows,

FSIF = $-\int p\, n\, dA$ (*pressure force*) + $\int \tau\, n\, dA$ (*shear force*)

SFIF = $-\int \sigma\, n\, dA$ (*stress tensor of the solid*)

How is FSI implemented on LS-DYNA ICFD?

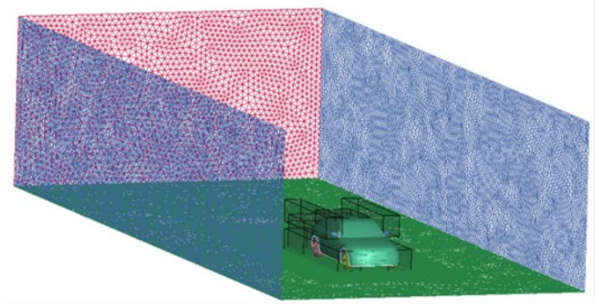


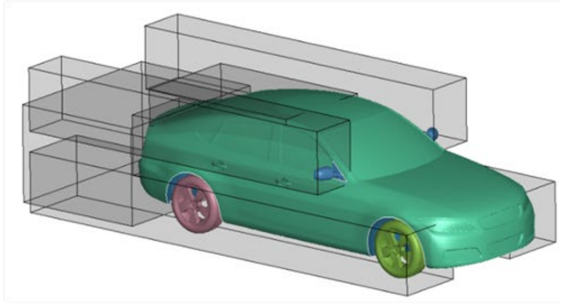
- LS-DYNA ICFD's FSI algorithm's objective is to use the structural solver's capabilities to the fullest.
- Hence, a **partitioned approach** is taken.
- Two kinds of coupling are available,
  - Strong Coupling (Implicit Structural Solver coupled with Implicit ICFD Solver).
  - Weak Coupling (Explicit Structural Solver coupled with an Implicit ICFD Solver).
- Both two-way coupling, and one-way coupling is possible.

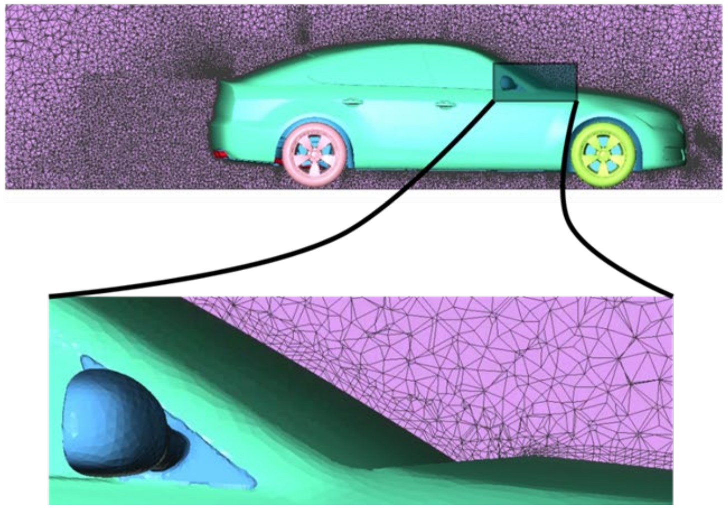
Capabilities of LS-DYNA ICFD

Volume Mesher

- Only the surfaces meshes have to be provided to define the geometry (No input volume mesh needed).
- In 3D, those surface meshes can be defined by Triangles or Quads. In 2D, beam-like elements are used.
- These surface meshes must be watertight, with matching interfaces and no open gaps or duplicate nodes!
- As an option, it is also possible for the user to build and use his/her own volume mesh (Tets only).

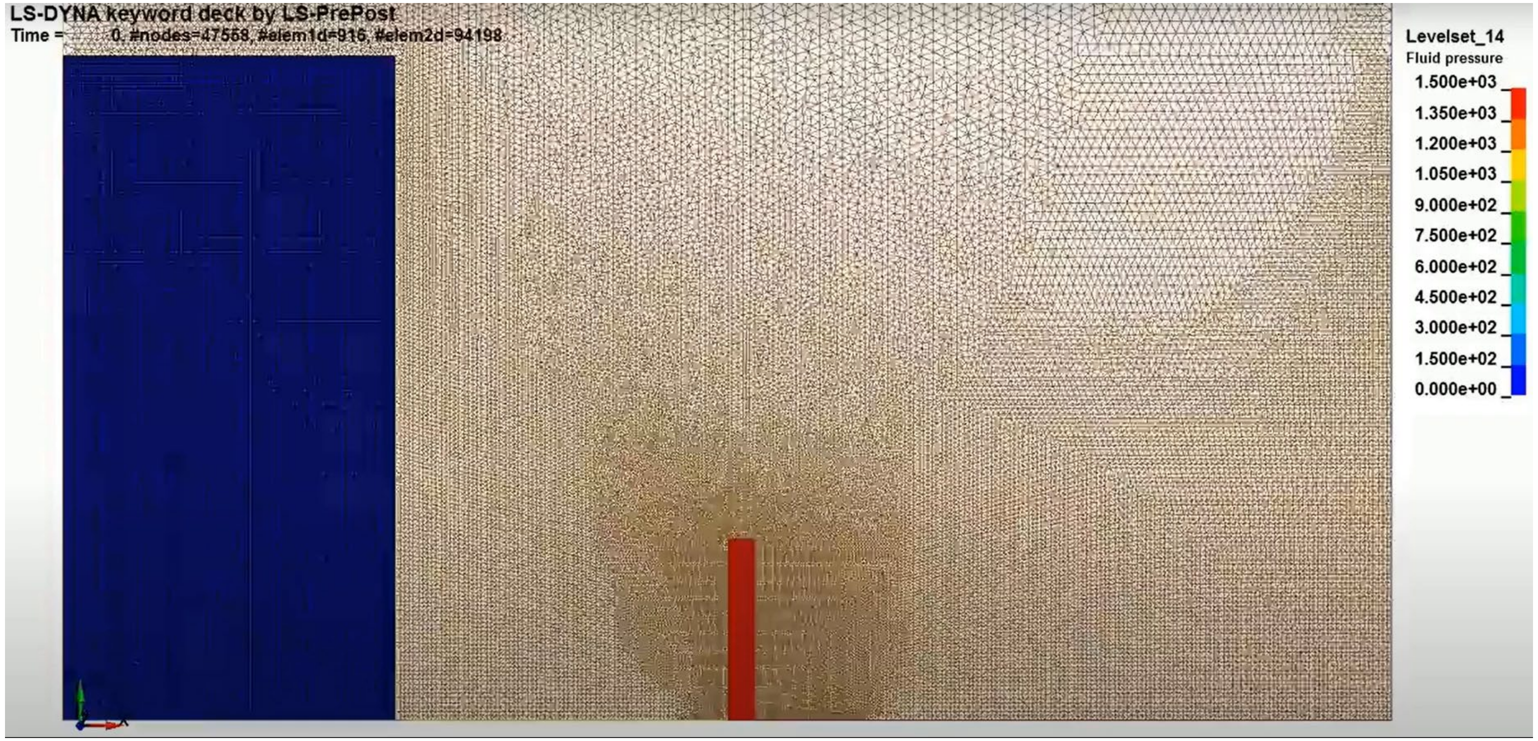






Capabilities of LS-DYNA ICFD

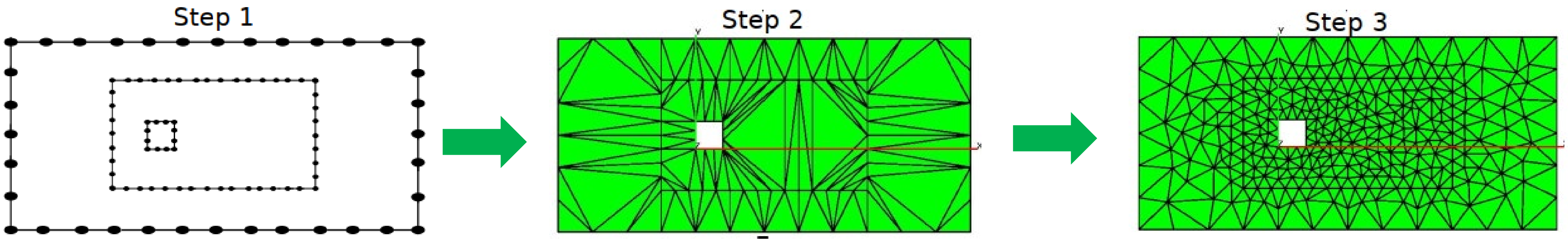
Adaptive Meshing



The solver uses an ALE approach for mesh movement which means that large deformations of the fluid mesh can occur

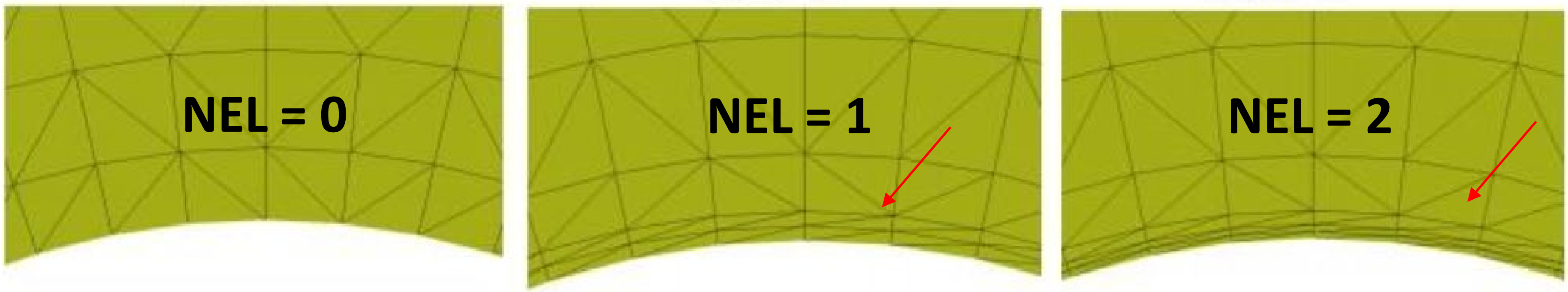
Capabilities of LS-DYNA ICFD

Local Mesh Refinement



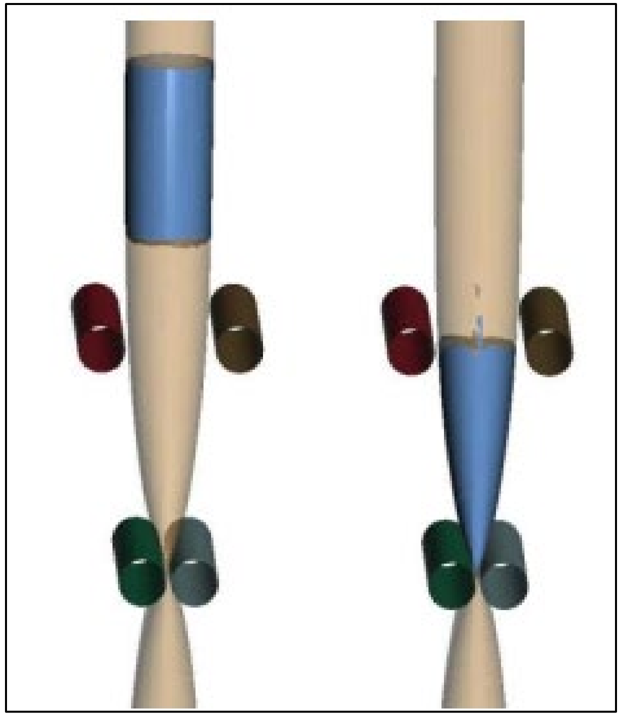
A user defined mesh size can be imposed on a volume domain.

Automatic Boundary Layer Mesh



Gap Closure Treatment

- In the case of body fitted meshes a complete blockage of flow through small gaps could be challenging.
- This is because the meshing algorithm will continue to insert elements in the gap region, preserving continuity of velocity and pressure and thus, transporting mass through the gap.
- In some scenarios, it may be desirable to completely block parts of the domain where boundaries are close enough.



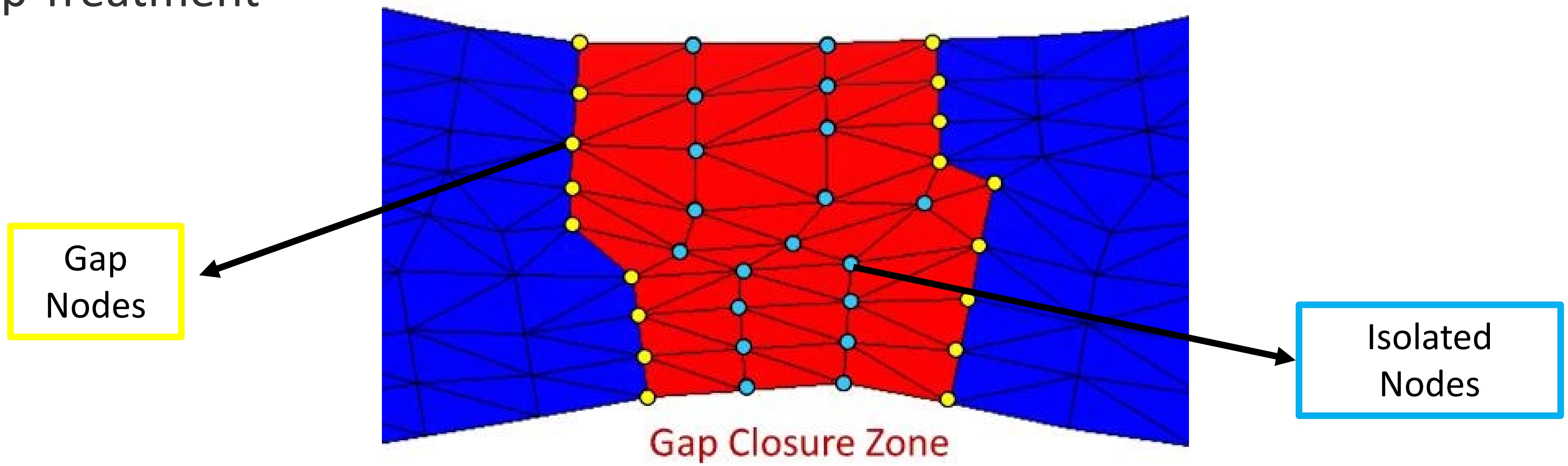
Flow through a thin channel

Gap Closure Treatment

- A new keyword *ICFD_CONTROL_GAP has been added.
- If the distance between contacting bodies is less than a threshold value, the flow is blocked at this region. This feature relies on,

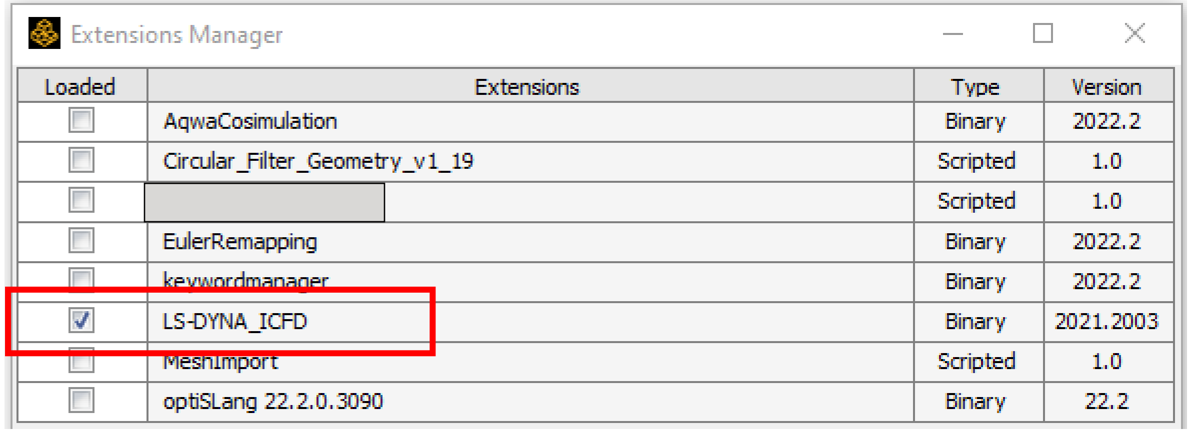
  1) Gap Detection

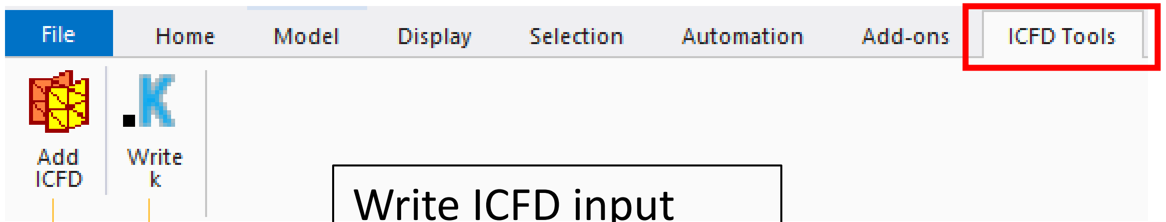
  2) Gap Treatment



LS-DYNA ICFD ACT

- LS-DYNA_ICFD extension requires at least version 2021R2.
- Install the binary from WB Project page – Extension > Install Extension > LS-DYNA_ICFD





Define ICFD cards to set the fluid domain, and parts.

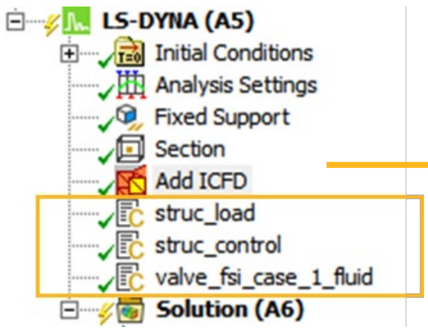
Write ICFD input deck in Solver Files directory.

Courtesy: Eric Stamper

Ansys

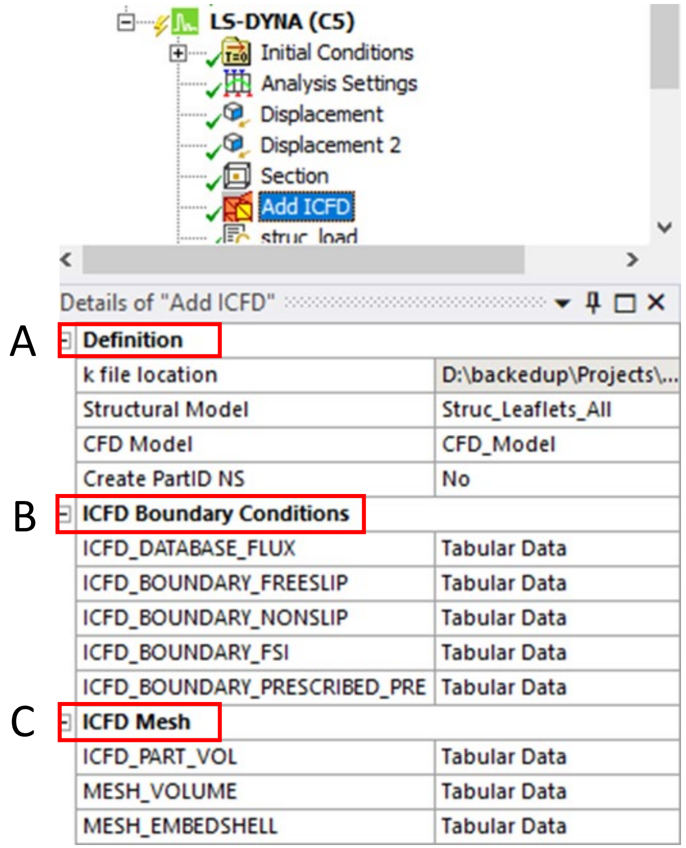
LS-DYNA ICFD ACT

- All the necessary LS-DYNA keywords that reference a Part Id have been added into the ACT.
- These include *ICFD_ and *MESH_ keywords.
- The user only needs to provide the relevant NS for each keyword.
- *ICFD_BOUNDARY_FSI performs the coupling.
- Any additional keywords can be added using command snippets.




```
*ICFD_CONTROL_OUTPUT
$#    msgl      outl     dtout   lsppout         -     itout
         2         7      0.01         0
*ICFD_CONTROL_TAVERAGE
$#      dt
       0.2
*ICFD_CONTROL_TIME
$#     ttm        dt       cfl    lcidsf     dtmin     dtmax    dtinit    tdeath
     0.8571.00000E-4
*ICFD_CONTROL_TURBULENCE
$#    tmod    submod      wlaw        ks        cs         -    lcids1    lcids2
         2         1
$#      cs

*ICFD_MAT
$#     mid       flg        ro       vis        st  stsflcid        ca
         1           0.001005  0.001020       0.0
```



Courtesy: Eric Stamper

Fluid Structure Interaction Simulation of Prosthetic Heart Valves

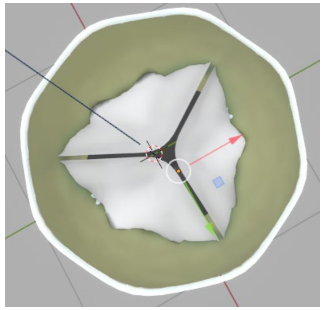
Goal

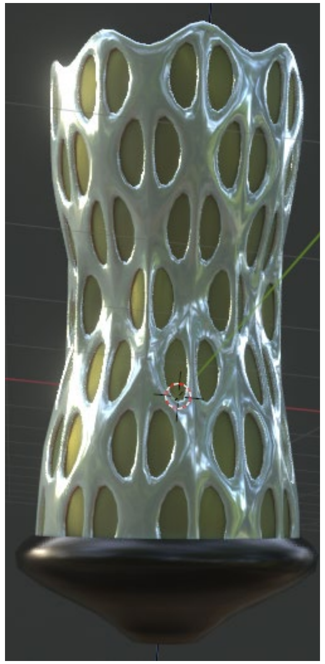
- Provide a workflow to simulate the full process of stent crimping, positioning and deployment of TAVR (Transcatheter aortic valve replacement).
- After deployment, an FSI approach is taken to study the hemodynamics and kinematics. And study the effect on stent due to the blood flow.

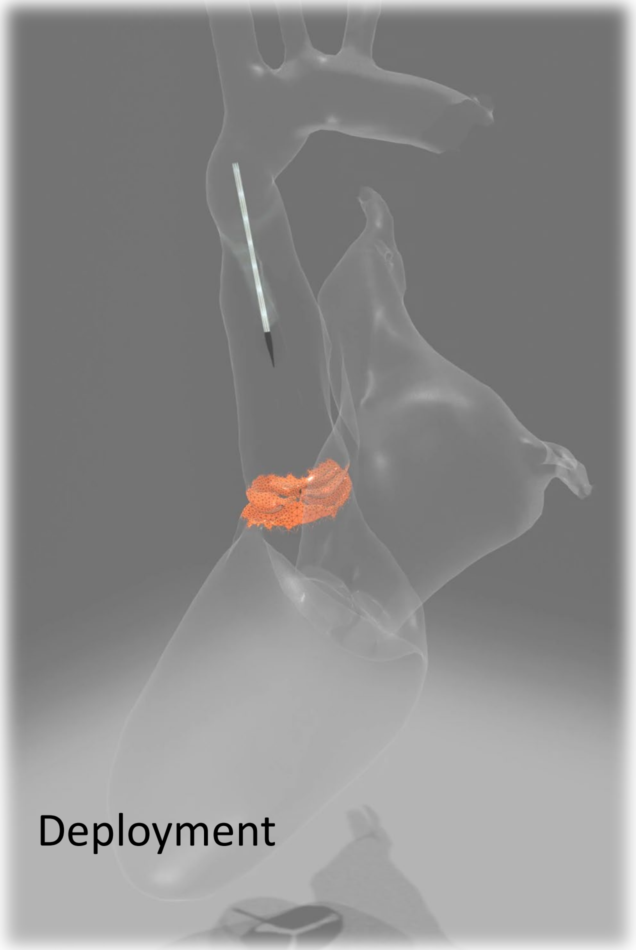
Solution

The complete workflow was designed in LS-DYNA as the solver works well to capture all the uncertainties in achieving the goals. The model was developed by dividing them into two different phases,

- The initial phase is to model the crimping, positioning, and deployment of TAVR using the explicit structural solver.
- Final state of the model from the structural analysis is taken and used for a CFD FSI simulation.

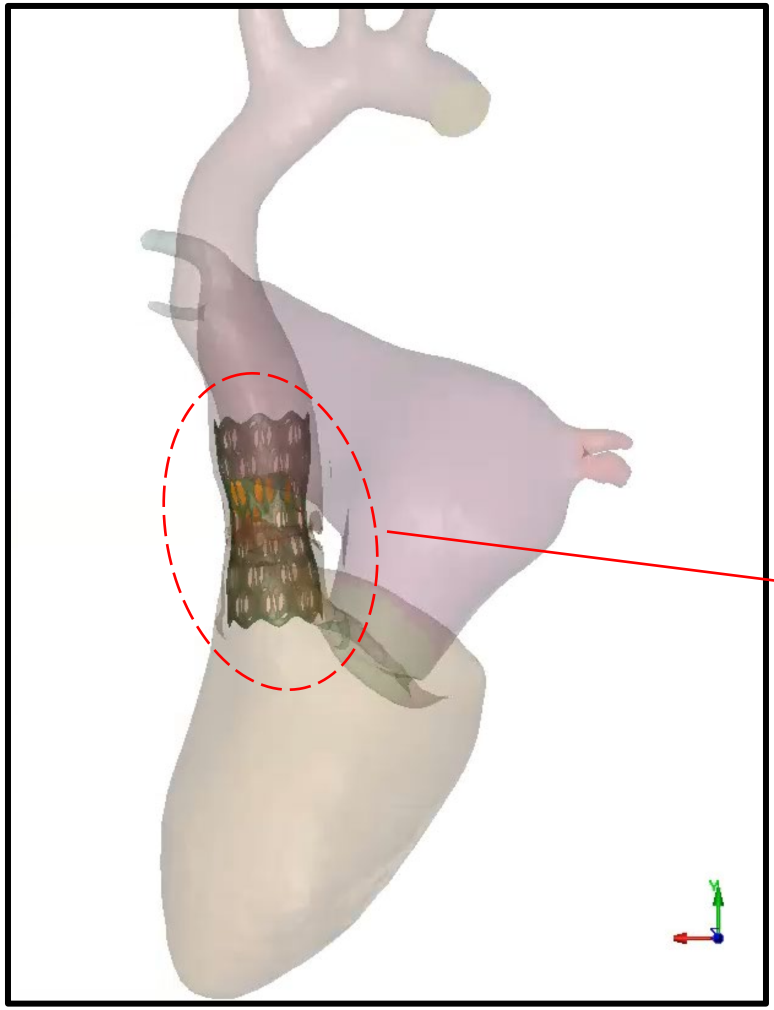






Courtesy: Facundo Del Pin

Fluid Structure Interaction Simulation of Prosthetic Heart Valves



Fluid-Structure Interaction simulation

Courtesy: Facundo Del Pin

When LS-DYNA ICFD FSI?

- LS-DYNA can be considered when the user expects to solve a problem that involves the below,
  - Large deformations.
  - Strong added mass effects, large density differences between fluid and structure.
  - High speeds.
  - Complex contacts.
- No EOS is needed which simplifies the problem a lot and allows to represent a wide range of fluids simply by two material parameters (ρ and μ).
- It is important to keep in mind that no pressure wave is solved. Therefore, applications involving explosions or sound cannot be solved under this hypothesis.

Best Practices

- Build complexity in a step-wise manner.
- Start with building the model as separate entities before moving towards coupling.
- Talk with domain experts.

Questions?