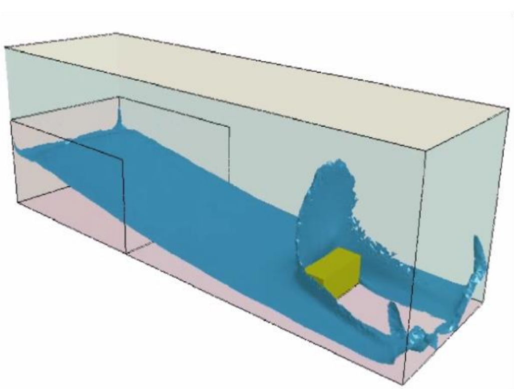
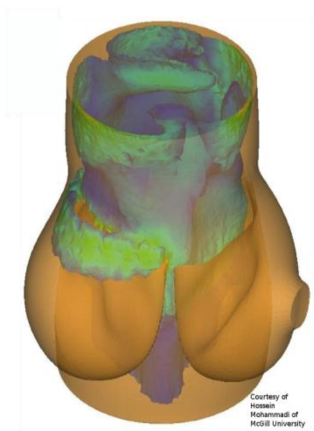
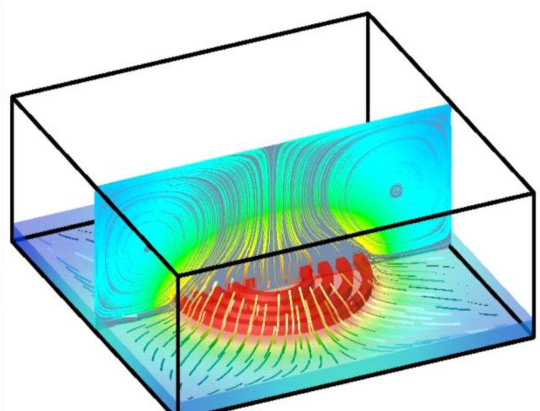
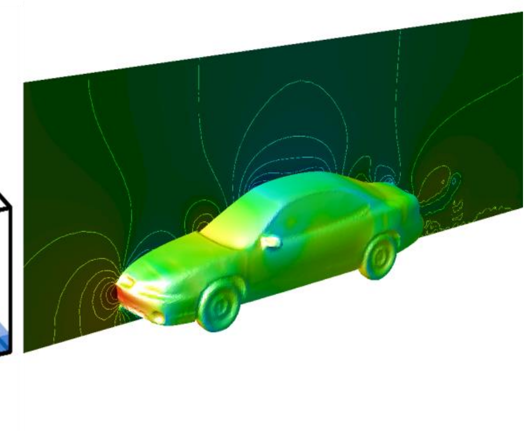
Incompressible CFD Module Presentation

Facundo Del Pin, Iñaki Çaldichoury, Rodrigo Paz









Introduction

1.1 Background

1.2 Main characteristics and features

1.3 Examples of applications

- LS-DYNA® is a **general-purpose** finite element program capable of simulating complex real world problems. It is used by the **automobile**, **aerospace, construction, military**, manufacturing, and bioengineering industries. LS-DYNA® is optimized for shared and distributed memory Unix, Linux, and Windows based, platforms, and it is fully QA'd by LSTC. The code's origins lie in **highly nonlinear, transient dynamic finite element analysis using explicit time integration.**

- Some of LS-DYNA® main functionalities include :
  - Full **2D** and **3D** capacities
  - **Explicit/Implicit** mechanical solver,
  - Coupled **thermal** solver,
  - Specific methods : **SPH, ALE, EFG**….,
  - **SMP** and **MPP** versions.

- The new release version pursues the objective of LS-DYNA® to become a strongly coupled **multi-physics solver** capable of solving complex real world problems that include several domains of physics.
- Three main new solvers will be introduced. Two fluid solvers for both compressible flows (CESE solver) and incompressible flows (ICFD solver) and the Electromagnetism solver (EM).
- This presentation will focus on the **ICFD solver**.
- The **scope** of these solvers is not only to solve their particular equations linked to their respective domains but to fully make use of LS-DYNA® capabilities by **coupling** them with the existing **structural** and/or **thermal** solvers.

Introduction

1.1 Background

1.2 Main characteristics and features

1.3 Examples of applications

- Double precision
- Fully **implicit**
- **2D** solver **/** **3D** solver
- **SMP** and **MPP** versions available
- Dynamic memory handling
- Can run as stand alone **CFD solver** or be coupled with LS-DYNA solid and thermal solvers for **FSI** and **conjugate heat transfer**.
- New set of keywords starting with ***ICFD** for the solver
- New set of keywords starting with ***MESH** for building and handling the fluid mesh

- The flow is considered **incompressible**.
- The **fluid volume mesh** is made out of **Tets** (triangles in 2D) and is **generated automatically**.
- Several meshing tools are available.
- For **FSI** interaction, loose or **strong coupling** is available.
- The solver is coupled with the thermal solver for solids for **conjugate heat transfer problems**.
- A level-set technique is used for **free surface flows**.
- **Non-Newtonian flows** models are available.
- The **Boussinesq model** is available for natural convection flows.
- Basic **turbulence models** are available.

Introduction

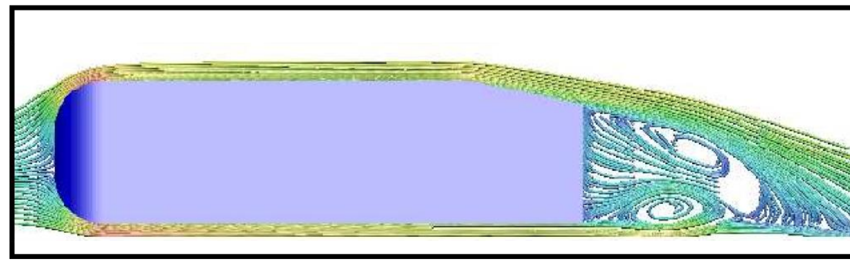
1.1 Background

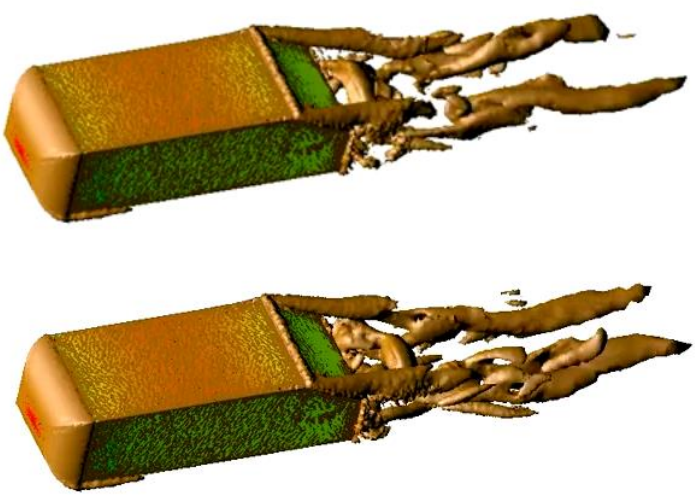
1.2 Main characteristics and features

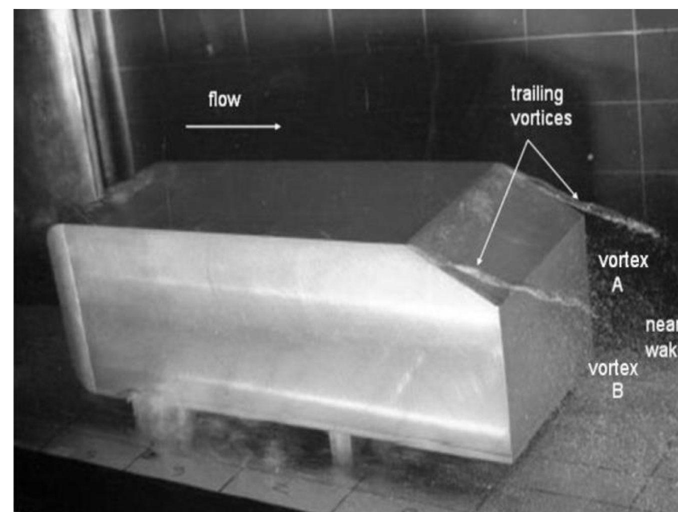
1.3 Examples of applications

Ahmed bluff body example (Benchmark problem) :

- **Drag calculation and Study of vortexes 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).**

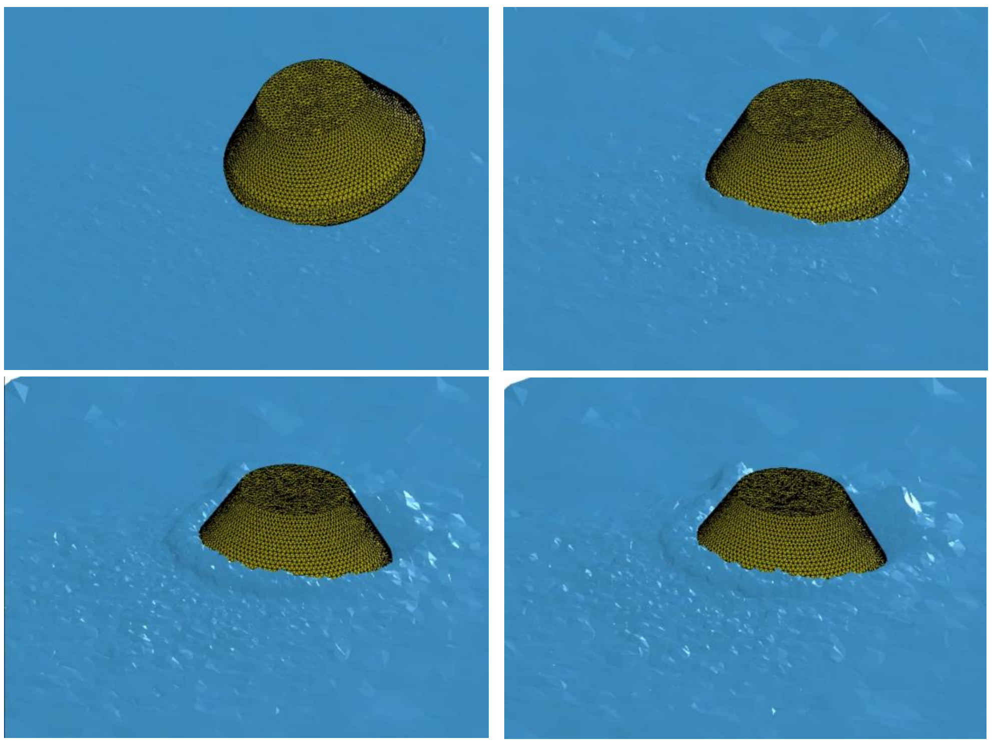


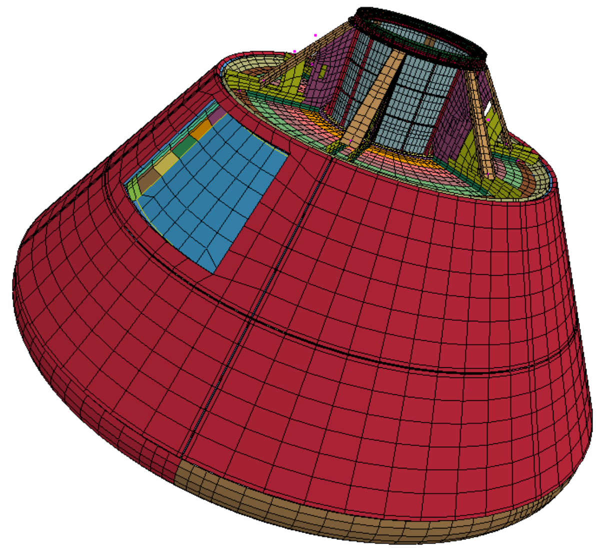




Space Capsule impact on water (Slamming problem) :

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

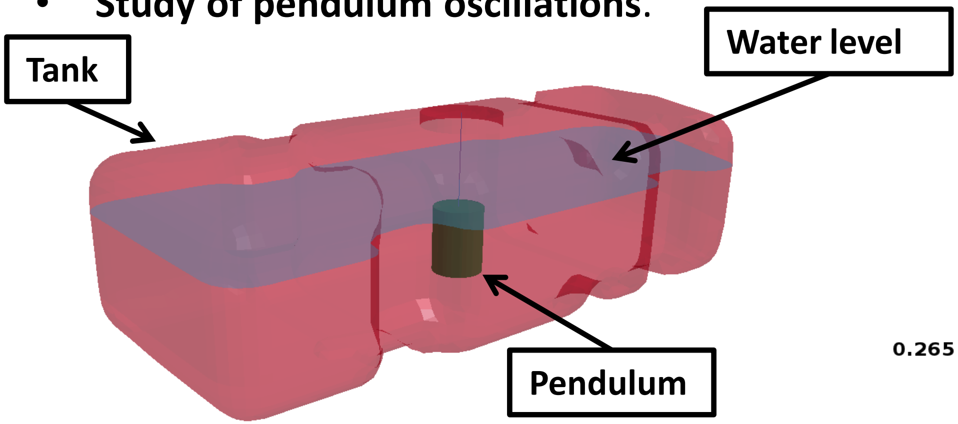


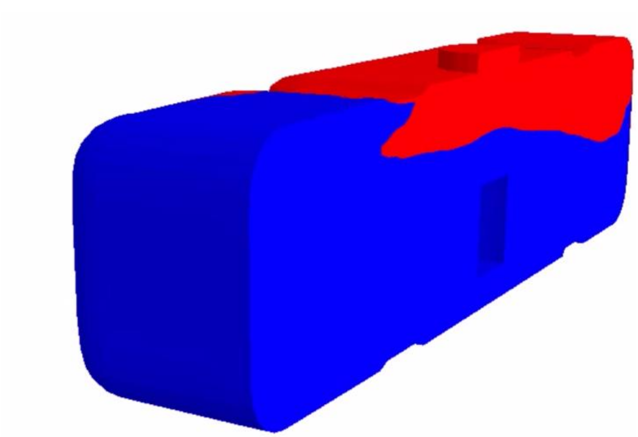


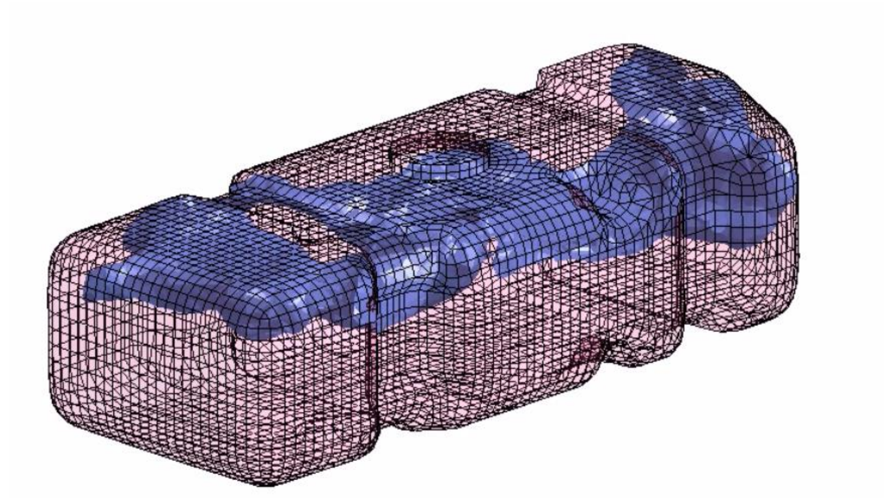
- Level Set Free surface problem
- Free fall impact / Strong FSI coupling,
- Proof of feasibility using ICFD solver,
- May be applied to similar Slamming problems.

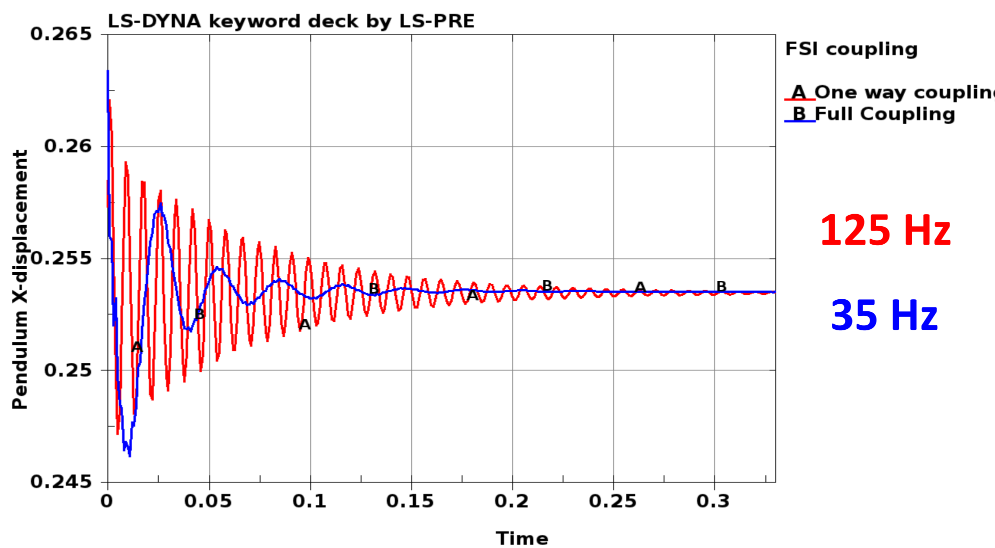
Water Tank example :

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



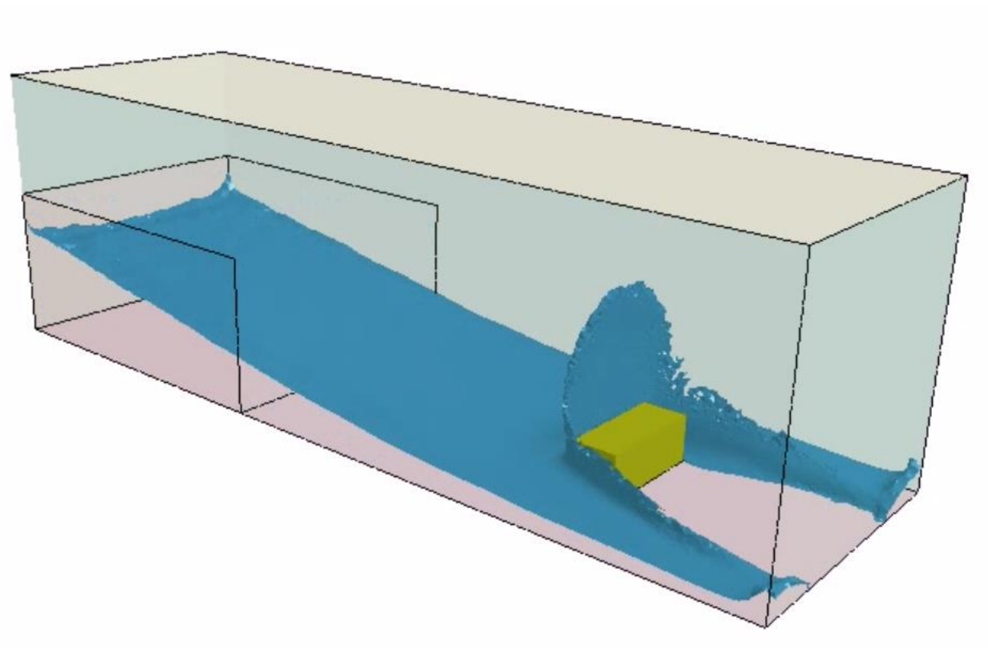


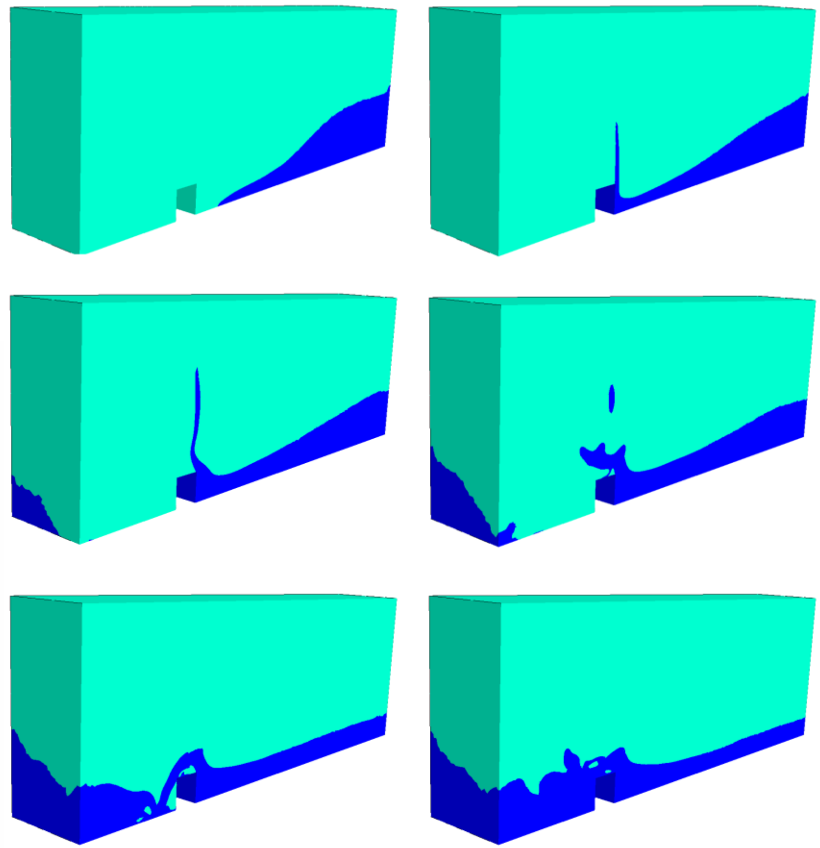




Wave impact on a rectangular shaped box:

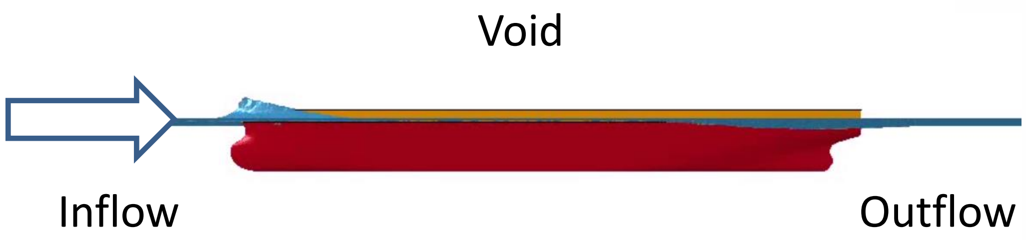
- Used to predict the force of impact on structure,
- The propagation of the wave shape can also be studied,
- Will be used and presented as a validation test case in the short term future.

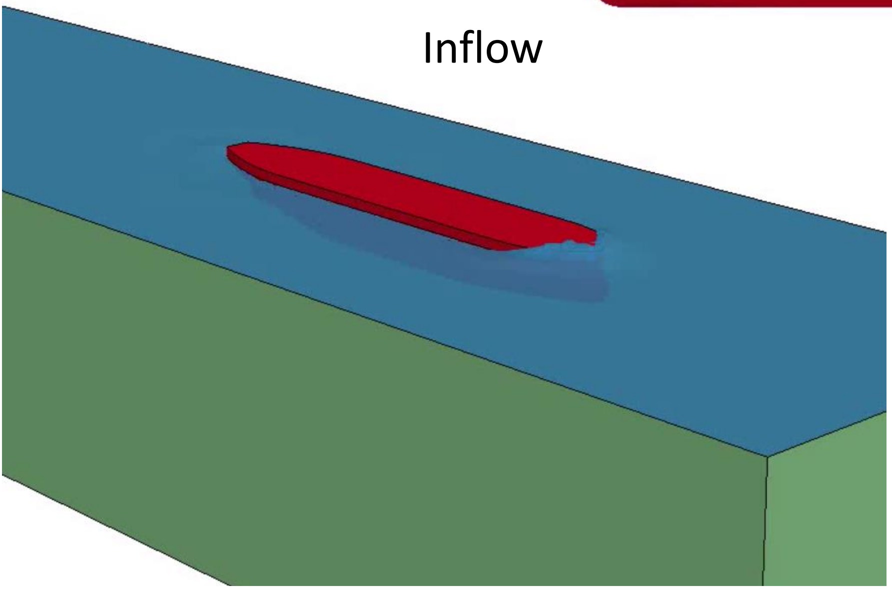




Hydrodynamics:

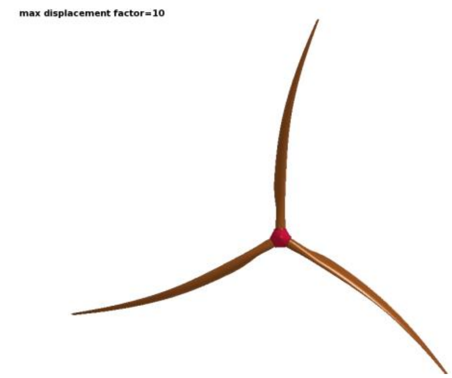
- **Complex Free surface problems that use Source and Sink terms, Strong FSI coupling, Dynamic remeshing and boundary layer mesh.**

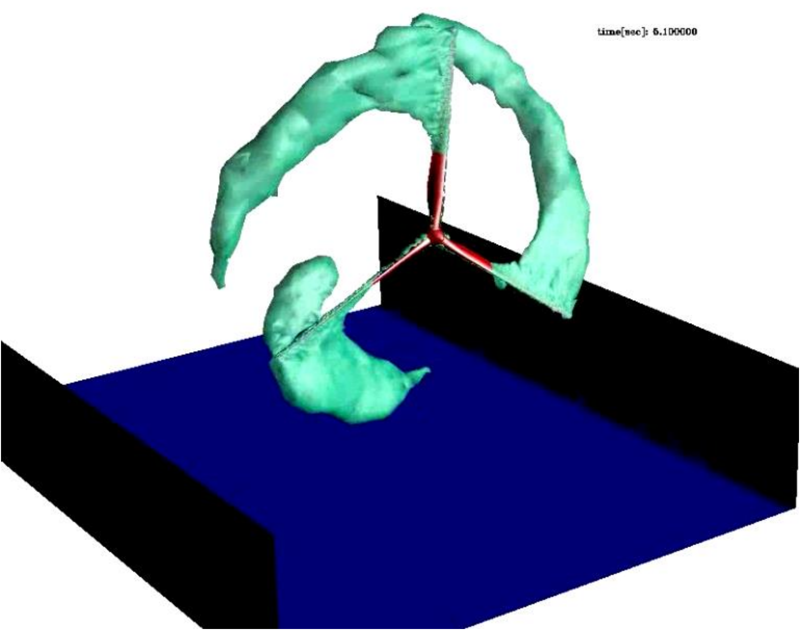


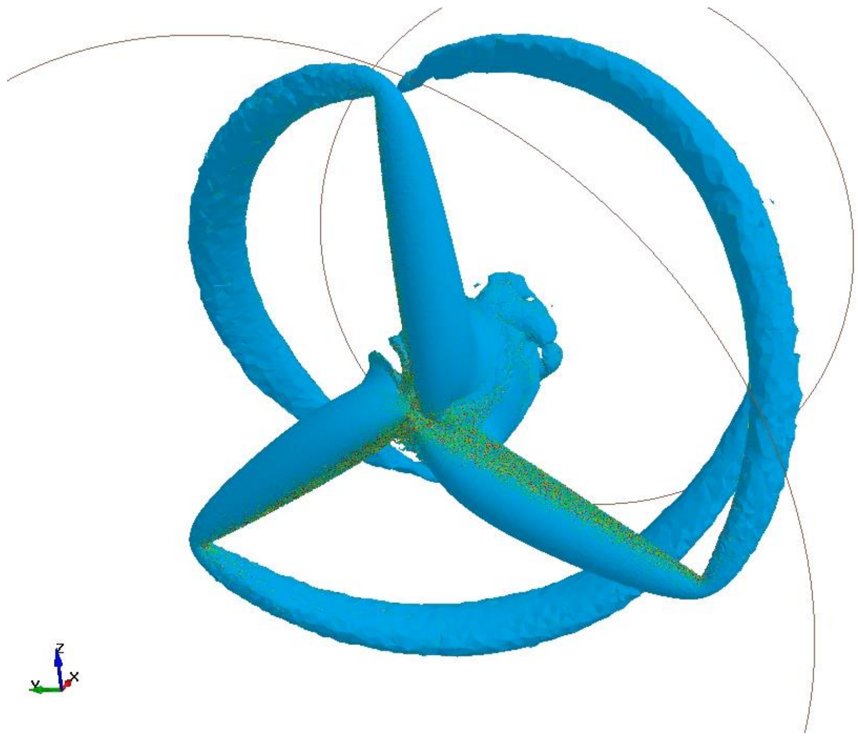


Wind turbines:

- **The aerodynamics of rotating wind turbines (VAWT or HAWT) can be studied through a FSI analysis.**
- **A non inertial reference frame feature can be used for rotating results.**

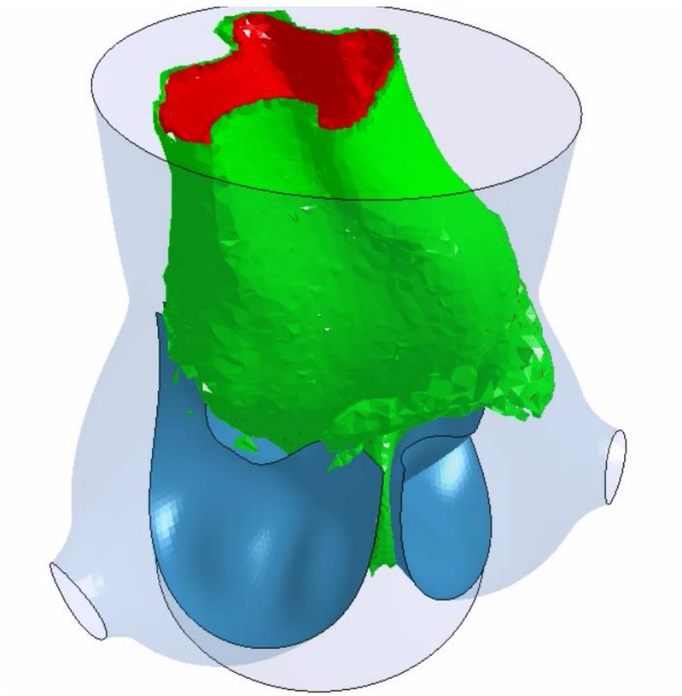


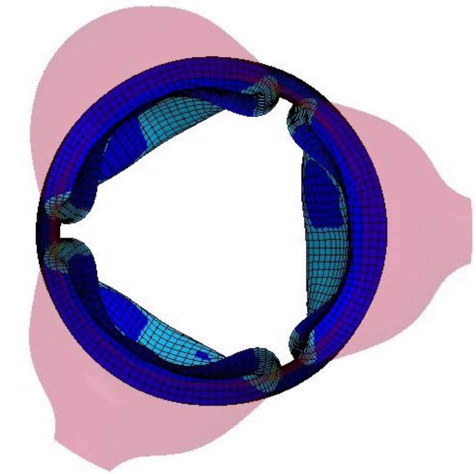


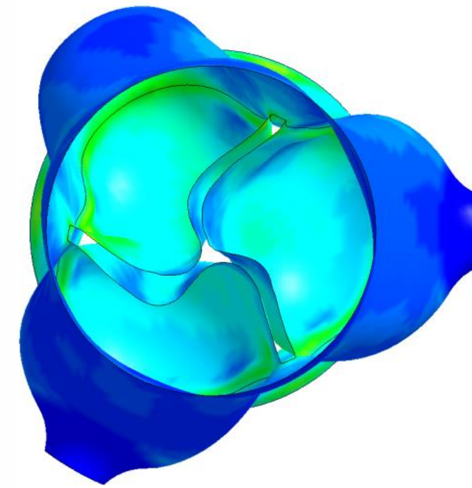


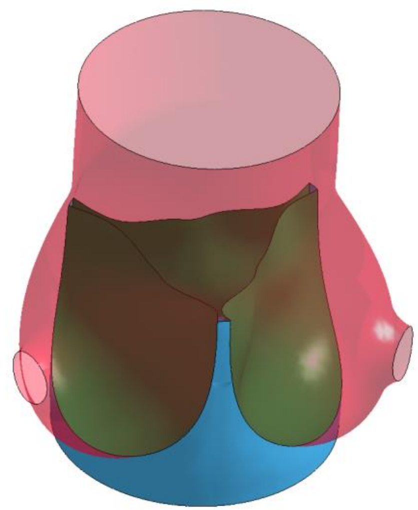
Heart valve FSI case:

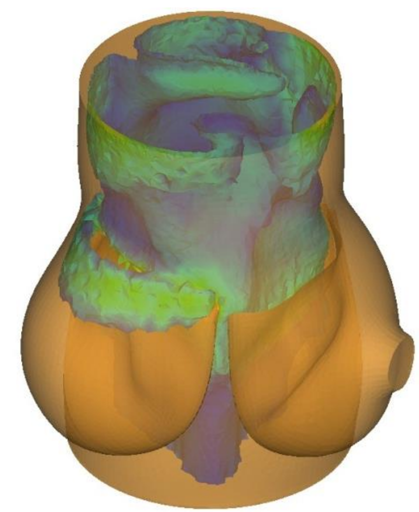
- **Blood and solid densities close. Large deformations of the solid. Strong FSI coupling mandatory.**
- **Courtesy of Hossein Mohammadi of McGill University, Montréal.**





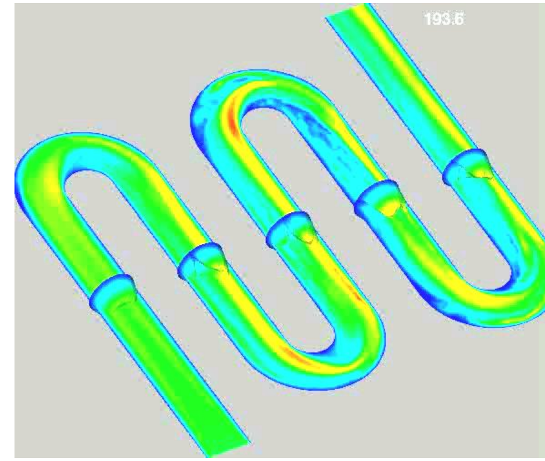


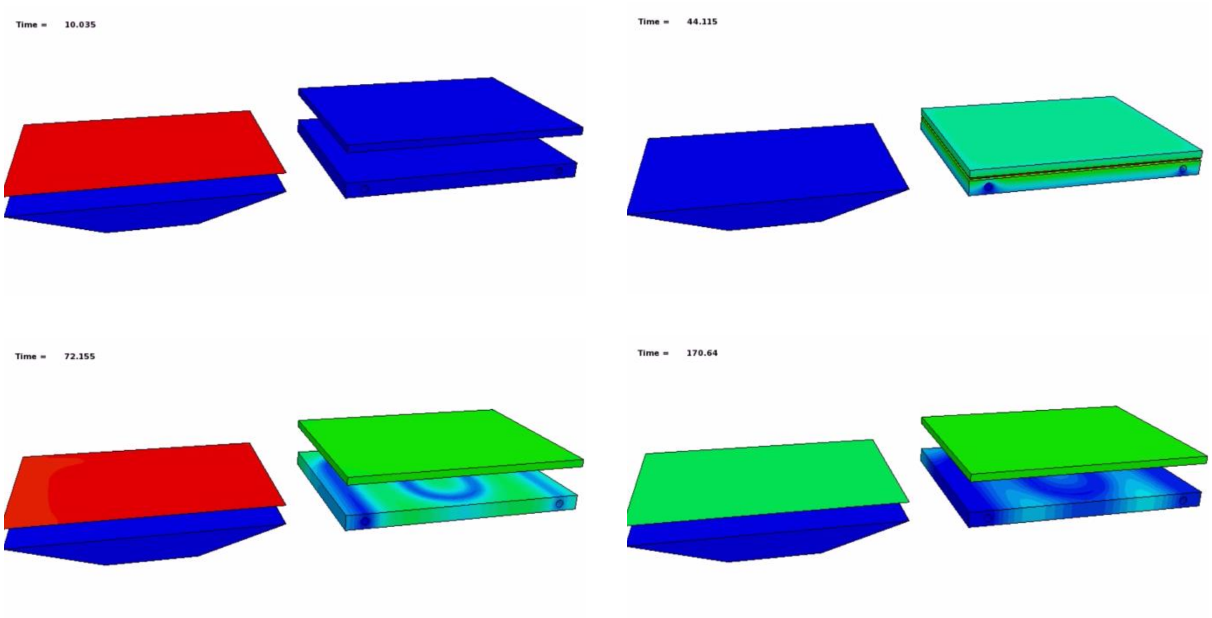


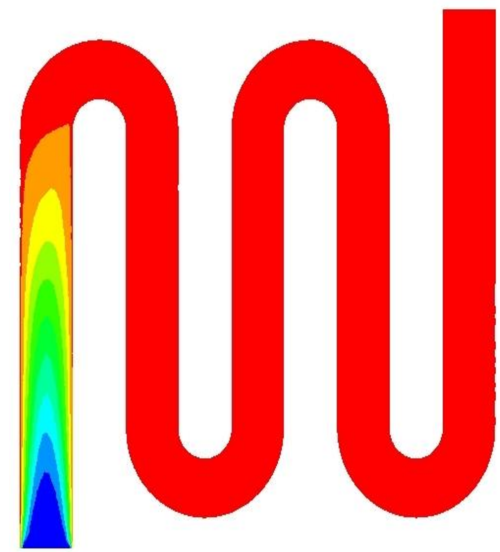


Conjugate Heat Transfer application:

- **Stamping process,**
- **Coupled fluid-structure and thermal problem,**
- **The fluid flowing through the serpentine is used to cool the dye and the workpiece.**

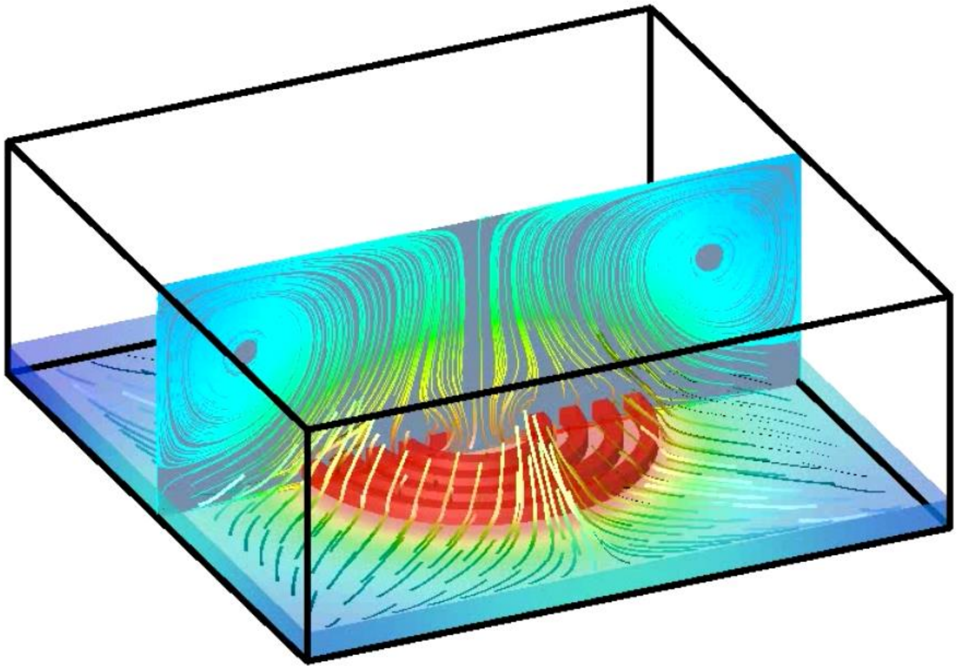


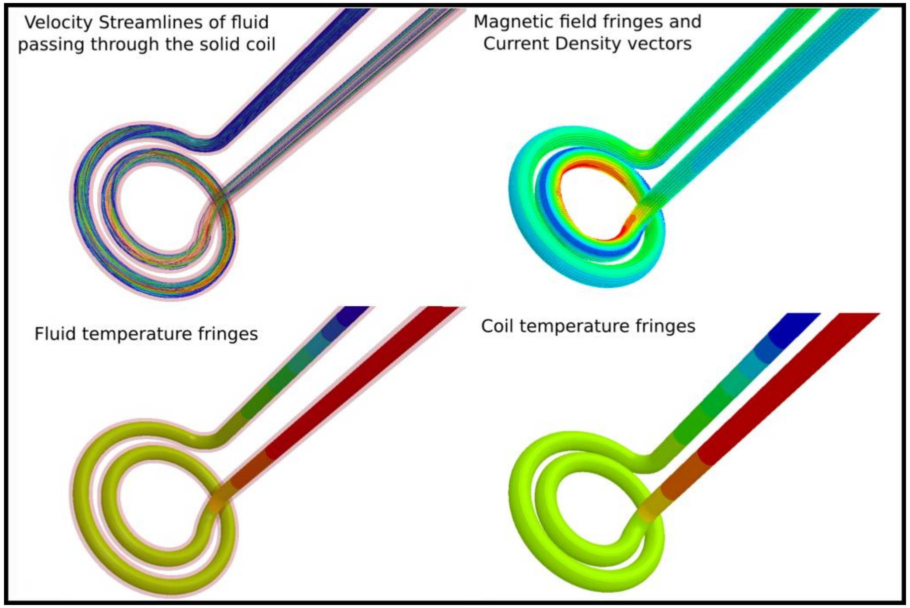




Coupled Thermal Fluid and EM problems:

- Coils being heated up due to Joule effect
- The heated coil can be used to warm up a liquid or a coolant can be used to cool them off
- Multiphysics problem involving the EM-ICFD and Solid thermal solvers.





- Courtesy of Miro from the Institute for composite in Kaiserslautern

Solver features

2.1 Focus on the incompressible hypothesis

2.2 Focus on the fluid volume mesh

2.3 Focus on the FSI coupling and thermal coupling

2.4 What's new ?

- Incompressible Navier-Stokes Momentum equations for Newtonian fluids (in Cartesian coordinates and in 2D) :

Velocity X-Component **Pressure**

$$\rho \frac{\partial \boxed{u}}{\partial t} + u\frac{\partial u}{\partial x} + \mathrm{v}\frac{\partial u}{\partial y} = -\frac{\partial \boxed{P}}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) + \rho f_x$$

Velocity Y-Component

$$\boxed{\rho}\frac{\partial \boxed{v}}{\partial t} + u\frac{\partial v}{\partial x} + \mathrm{v}\frac{\partial v}{\partial y} = -\frac{\partial P}{\partial y} + \boxed{\mu}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) + \rho \boxed{f_y}$$

Fluid Density **Fluid Dynamic Density** **Exterior Volume Forces**

- Two equations
- Three unknowns (U, V and P)

- Need for a third equation to close the system

- Differential form of continuity (**conservation of mass**) equation:

$$\boxed{\frac{\partial \rho}{\partial t} + \left(\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y}\right) = 0}$$

- If ρ is a constant (**incompressible flow hypothesis**), then the continuity equation reduces to :

$$\boxed{\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0} \quad \text{or} \quad \boxed{div\,\vec{V} = 0}$$

- No further need for any equation of state or other equation. **Uncoupled from energy equations or temperature**. Temperature is solved through its own system of equations (Heat equation)

Why do CFD solvers often use the incompressible hypothesis ?

- **No** need to define an equation of state (**EOS**) to close the system. This way, a large number of fluids and gazes can be easily represented in a simple way.

- Valid if **Mach number M<0.3**. A few Mach number values encountered in industrial applications are :

  - Ocean surface current speed : M<0.003
  - Pipeline flow speed : M<0.03
  - Typical highway car speed : M<0.12.
  - Wind turbine (HAWT) survival speed : M<0.18
  - TGV maximum operating speed : M<0.27

- **A wide range of applications** meet this condition.

Why do incompressible CFD solvers often run using an implicit scheme?

- For low speeds fluid mechanics, there is a desire to be **independent** from any time step **constraining CFL** condition especially in cases where a very fine mesh is needed in order to capture some complex phenomena.

- By running in implicit, the solver can use **time step values a few times higher than the CFL condition**.

- **Remark** : Since no pressure wave is calculated in an incompressible flow, the CFL condition reduces to :

$$\Delta t_{CFL} = \frac{l_e}{u_e}$$

With l_e, the characteristic mesh size and U_e the flow velocity.

Incompressible flow solvers	Compressible flow solvers (CESE, ALE)
Drag studies around bluff bodies (cars, boats…)	High speed flows (airbag deployments)
Flows in pipes/tubes…	Supersonic flows (Shock waves)
Slamming, Sloshing and Wave impacts.	Explosives and Chemical reactions
Transient and steady state analysis	Rapid and brief phenomena

Solver features

2.1 Focus on the incompressible hypothesis

2.2 Focus on the fluid volume mesh

2.3 Focus on the FSI coupling and thermal coupling

2.4 What's new

- Why does the fluid solver use Tets (triangles in 2D) to generate the fluid volume mesh ?

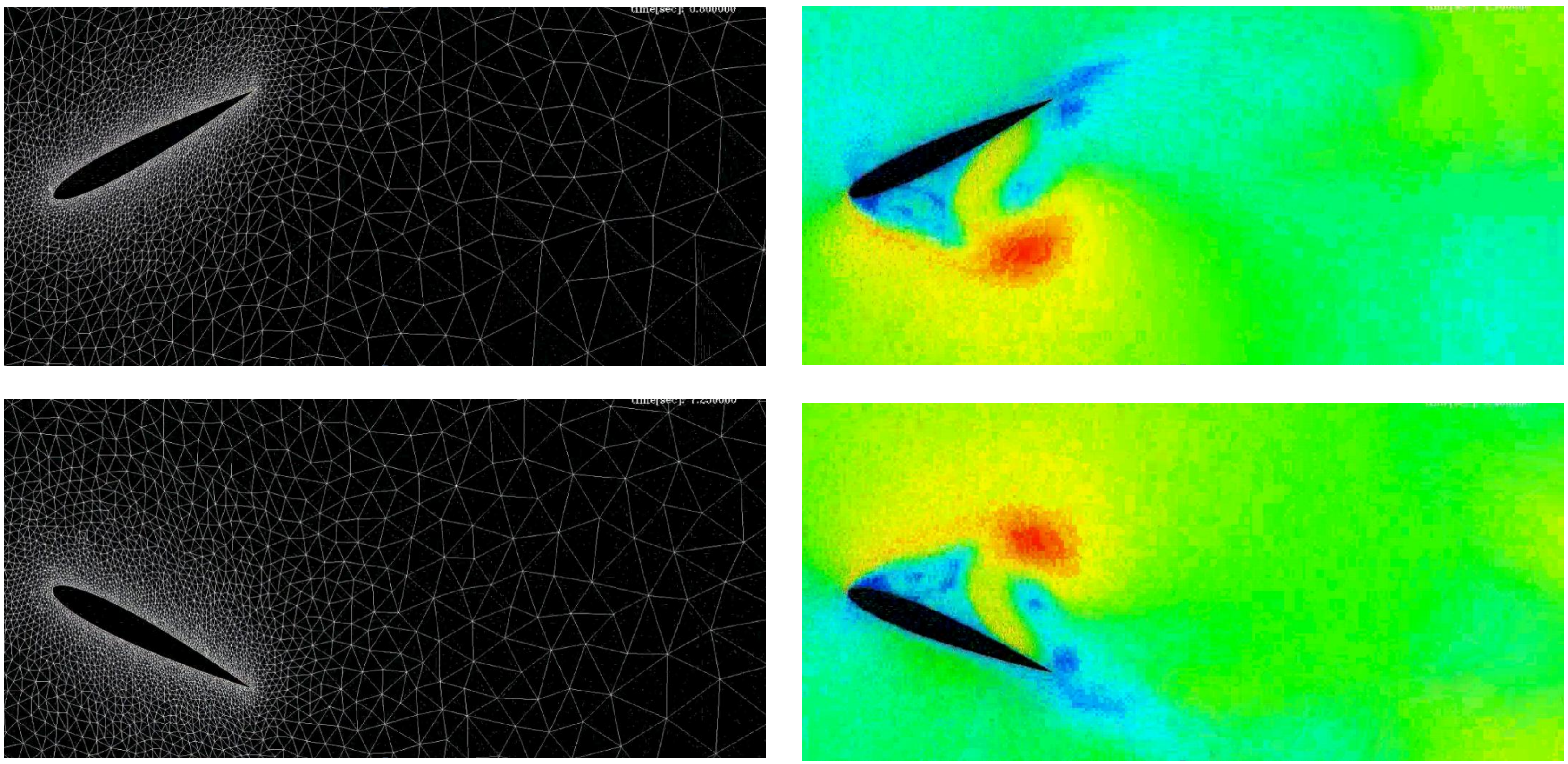
- For fluid mechanics , using tets and unstructured meshes provides a certain number of advantages that can prove to be decisive for a fluid mechanic solver :

  - **Automatic : It is possible to automatically generate a volume mesh which greatly simplifies the pre-processing stage, the file handling and reduces the source of errors that the user could introduce by building his own mesh.**

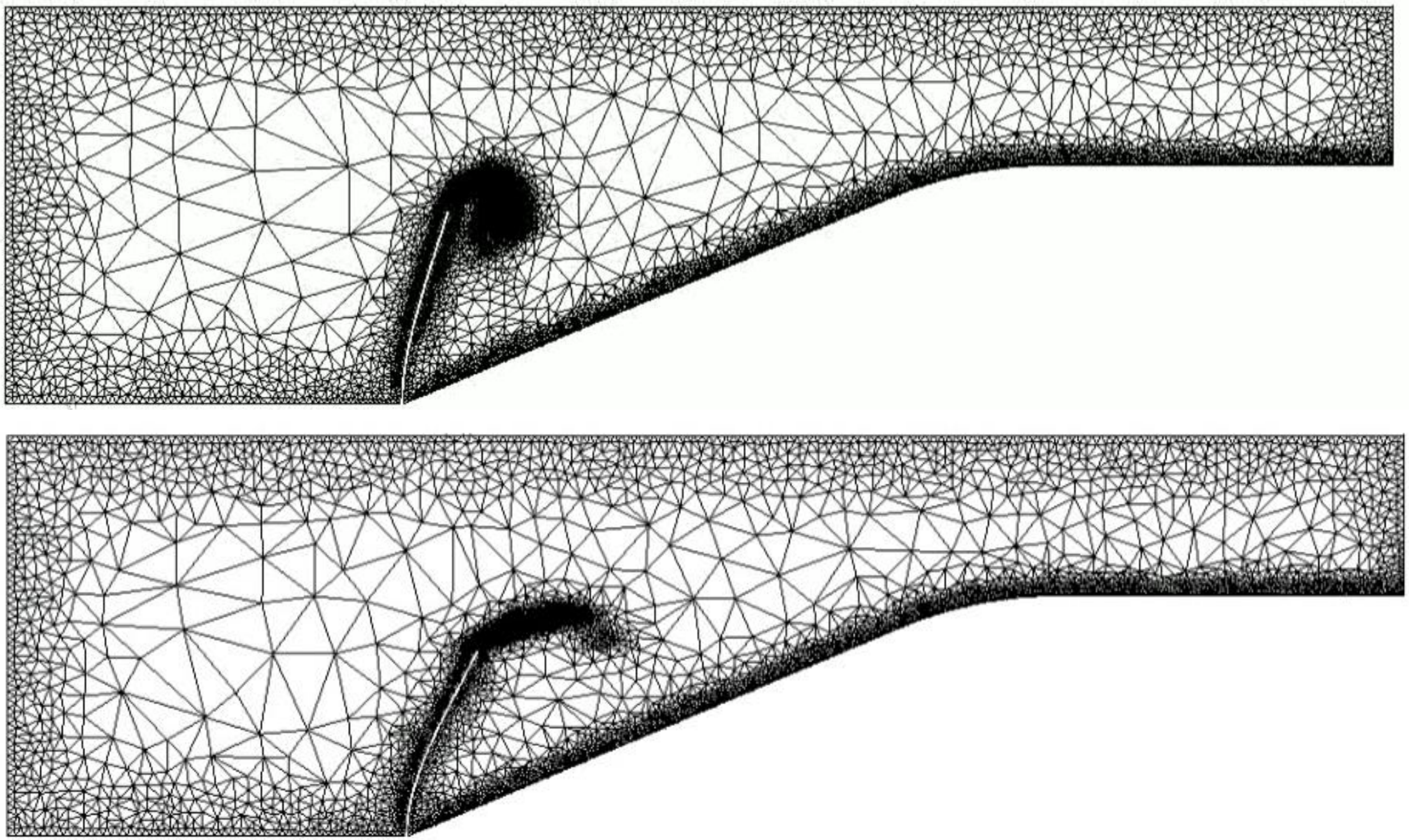
  - **Robust : Tets allow to better represent complex geometries with sharp angles than Hexes.**

  - **Generic : The same automatic volume mesher can be used for any surface geometry provided by the user.**

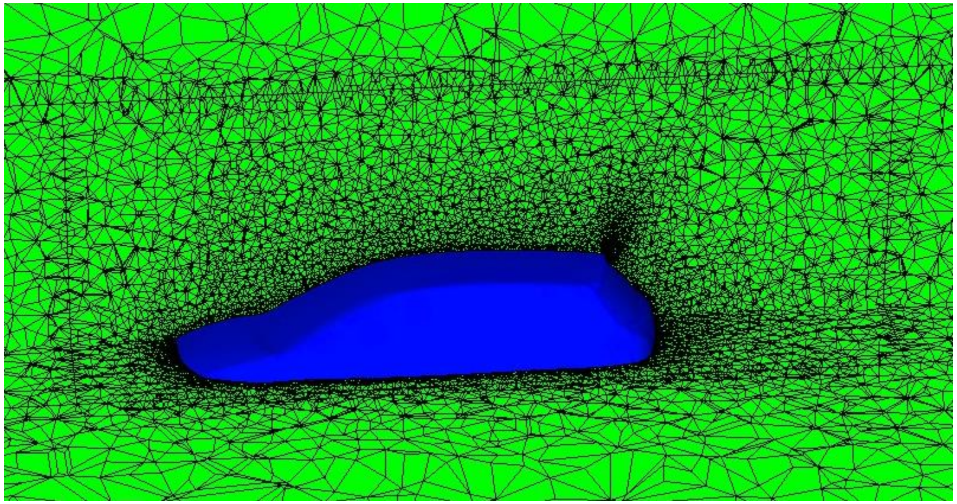
- The ICFD solver also uses an ALE approach for mesh movement :

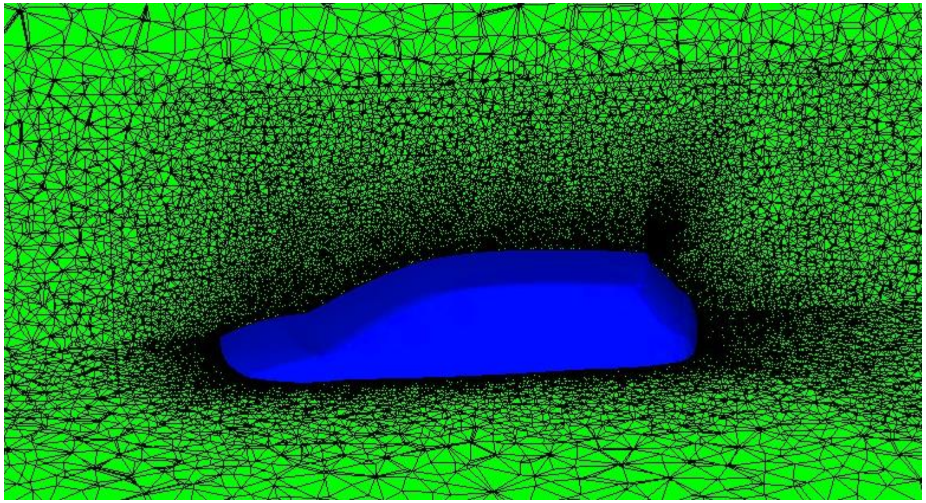


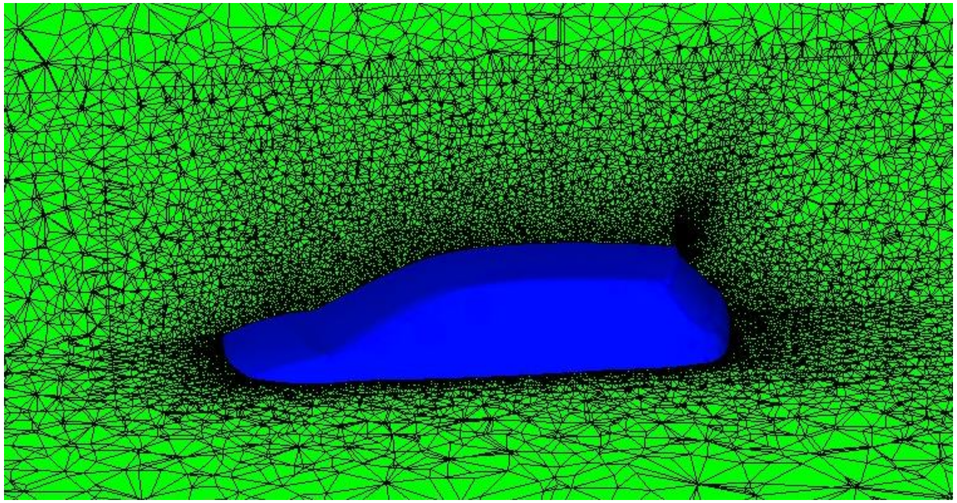
- And also has mesh adaptivity capabilities



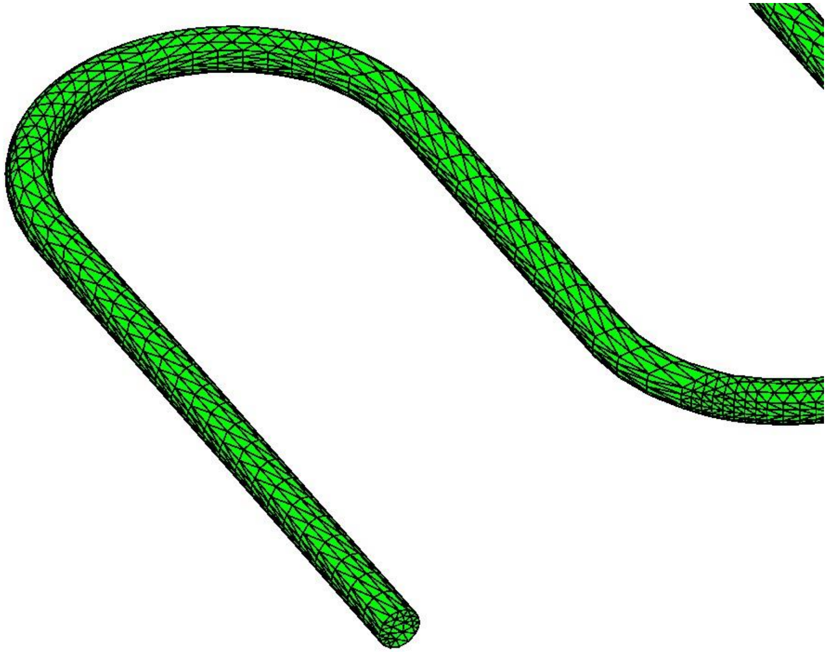
- Option to control the mesh size interpolation :

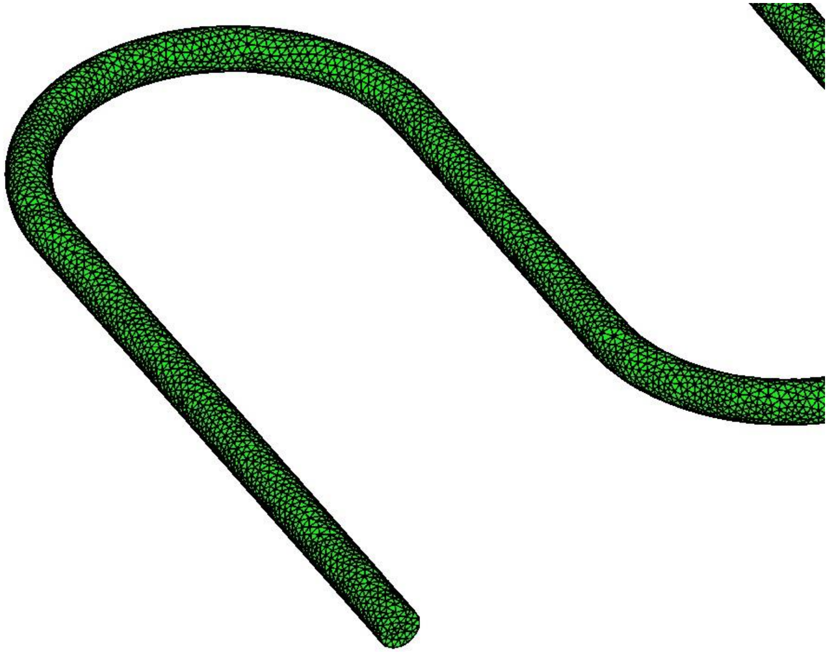






- Option to remesh surface with initial “bad” aspect ratio for better mesh quality :



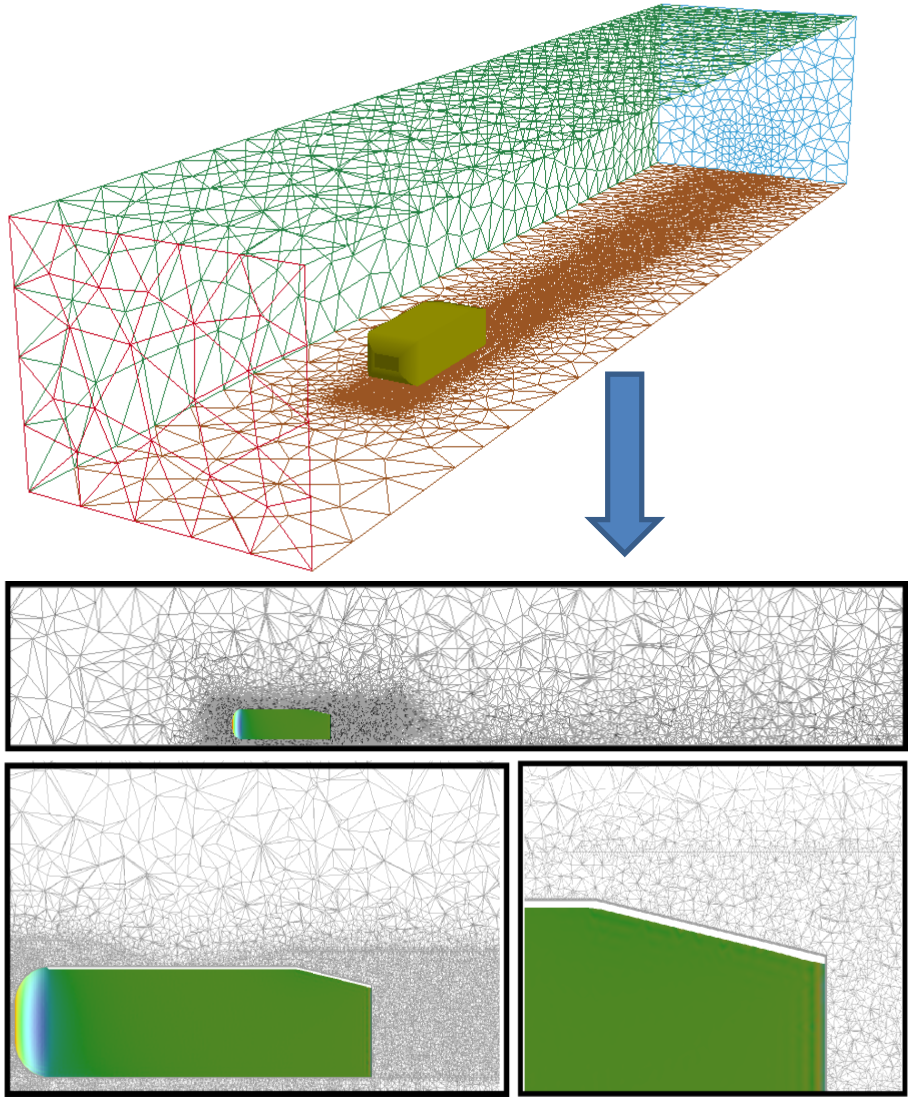


- Only the surfaces meshes have to be provided to define the geometry (No input volume mesh needed).

- 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 own volume mesh (Tets only).



Solver features

2.1 Focus on the incompressible hypothesis

2.2 Focus on the fluid volume mesh

2.3 Focus on the FSI coupling and thermal coupling

2.4 What's new ?

- **Reminder** : The **scope** of the new multi physics solvers is not only to solve their particular equations linked to their respective domains but to fully make use of LS-DYNA® capabilities by **coupling** them with the existing **structural** and/or **thermal** solvers.

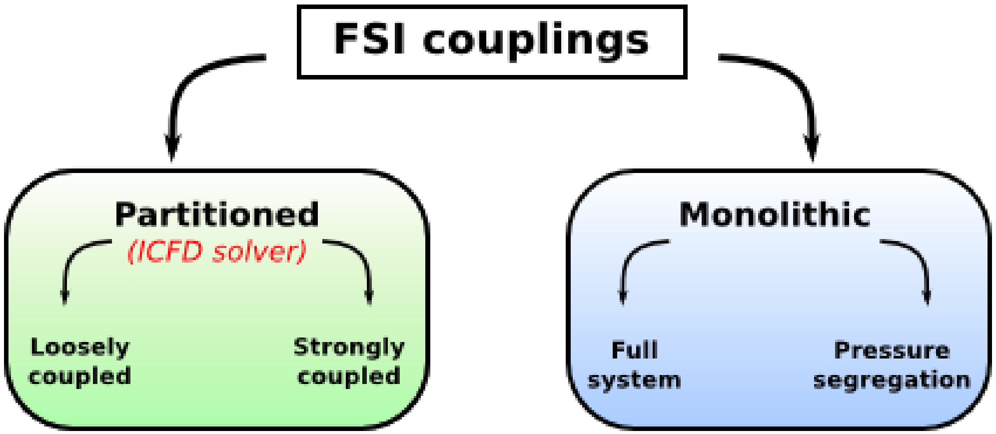
- LS-DYNA has immense solid mechanics capabilities as well as a huge material library.

- LS-DYNA can both run in explicit or implicit.

- LS-DYNA already has a thermal solver for solids.

- LS-DYNA offers the perfect environment in order to develop a state of the art solver allowing complex fluid structure interactions as well as the solving of conjugate heat transfer problems

- Different kind of FSI couplings exist :



- The uncoupling of the fluid and solid equations (**partitioned approach**) offers significant benefits in terms of **efficiency** : smaller and better conditioned subsystems are solved instead of a single problem.

- In addition to the more frequently encountered loosely (or weakly) coupled capabilities, strong coupling capabilities are also available.

- **Strong coupling opens new ranges of applications** but it is important to keep in mind that for FSI problems, instabilities and convergence problems can occur regardless of the type of FSI coupling used and need special treatments to ensure stability and convergence (**Artificial Added mass effect** problems).

- Since the flow is incompressible, the **temperature is uncoupled** and independent from the velocity or pressure terms.

- The solver solves the **heat equation** in the fluid with an **advection** and a **diffusion** term for temperature.

- Monolithic approach used for thermal coupling with the thermal solver for solids.

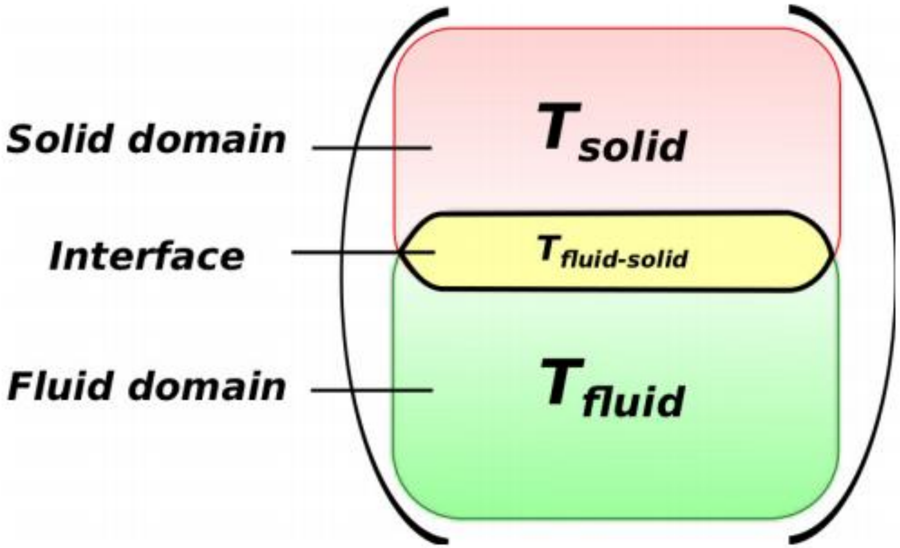
- Very robust but time consuming

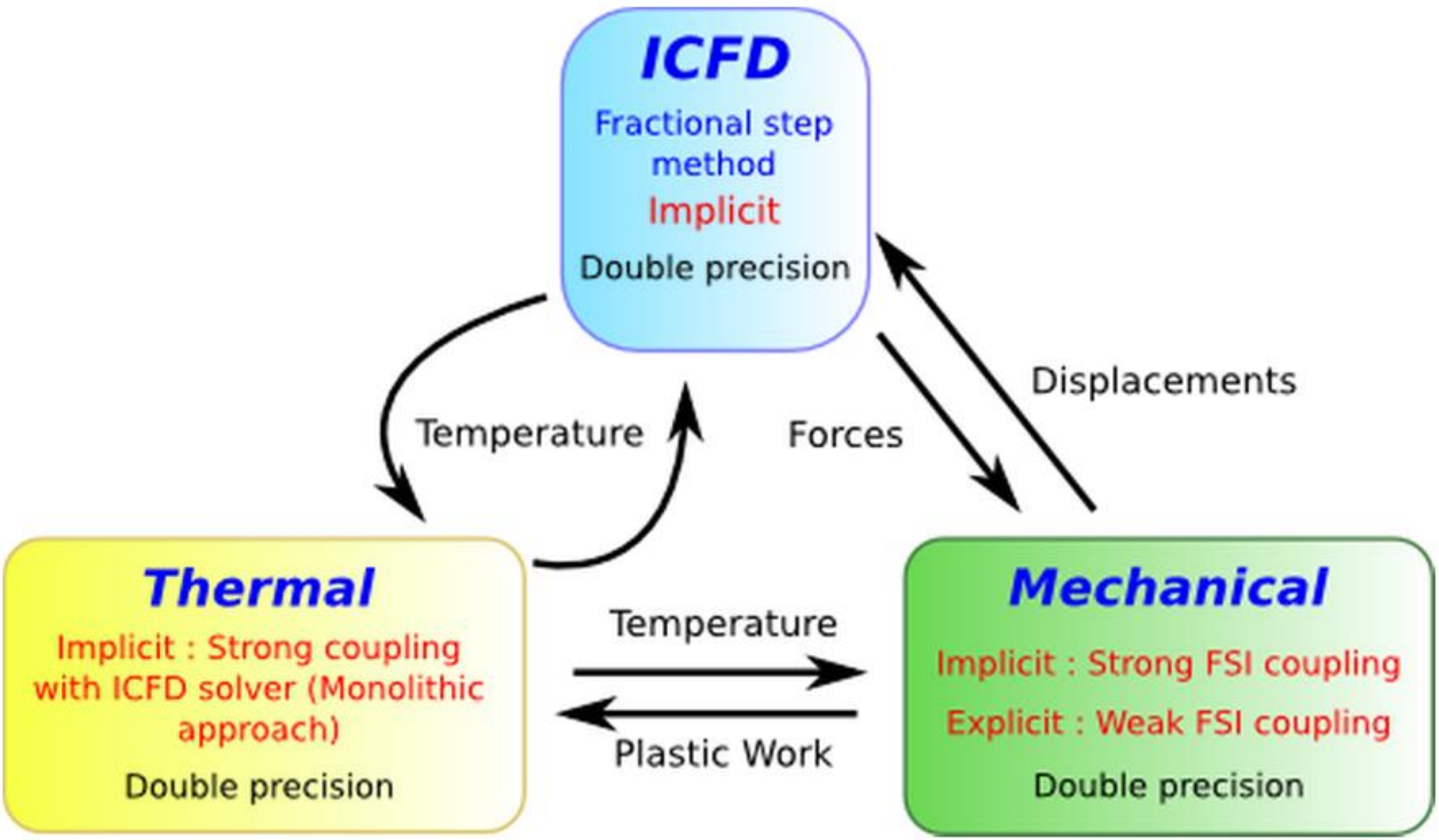
Heat Equation :

Potential source of heat term

$$\frac{\partial T}{\partial t} + u_j \frac{\partial T}{\partial x_j} - \alpha \frac{\partial^2 T}{\partial x_j \partial x_j} = f$$

α, thermal diffusivity coefficient




ICFD
Fractional step method
Implicit
Double precision
Temperature
Forces
Displacements
Thermal
Implicit : Strong coupling with ICFD solver (Monolithic approach)
Double precision
Temperature
Plastic Work
Mechanical
Implicit : Strong FSI coupling
Explicit : Weak FSI coupling
Double precision

Solver features

2.1 Focus on the incompressible hypothesis

2.2 Focus on the fluid volume mesh

2.3 Focus on the FSI coupling and thermal coupling

2.4 What's new ?

Application : External and internal aerodynamics :

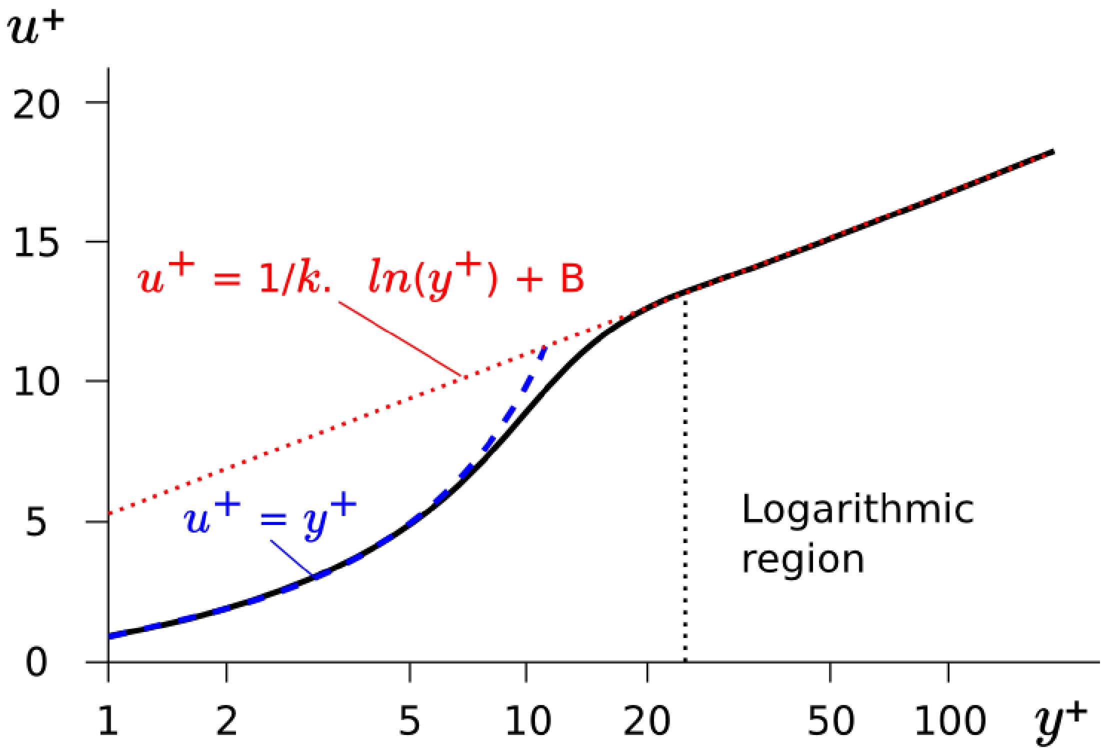
Current development : Adding HRN and LRN laws of the wall for the k-epsilon turbulence model

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

$$y^+ = \frac{yu^*}{\vartheta}$$

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

$$\tau_w = \mu \frac{\partial u}{\partial y}$$



Application : External and internal aerodynamics :

Already available : Space correlated turbulent inlet boundary conditions for incompressible Navier-Stokes equations.

KEYWORD: *ICFD_CONTROL_TURB_SYNTHESIS.

PARAMETERS: PID,Iu; Iv; Iw,Ls.

NOTE: This keyword must be used jointly with VAD=4 of keyword *ICFD_BOUNDARY_PRESCRIBED_VEL

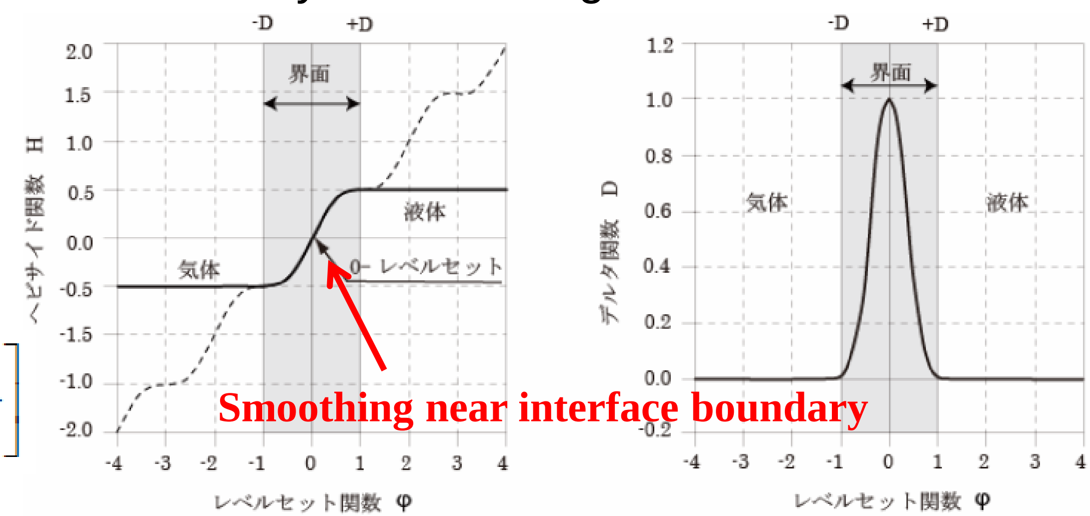
Application : Multiphase flows:

Current development : Implementing first multiphase models for sloshing, impact etc applications (Heaviside parameters). Future developments will investigate more complex models.

In the Level-Set method near the interface boundary, a smoothing method is used with so called Heaviside parameters.

$$\begin{cases}\rho = 0.5\left(\rho_l + \rho_g\right) + H_D\left(\rho_l - \rho_g\right),\\ \mu = 0.5\left(\mu_l + \mu_g\right) + H_D\left(\mu_l - \mu_g\right)\end{cases}$$

$$H_D = 0.5 \cdot \max\left[-1.0, \min\left\{1.0, \frac{\phi}{D} + \frac{1}{\pi}\sin\left(\frac{\pi\phi}{D}\right)\right\}\right]$$



Application : Porous Media

Current development : First generalized flow in porous media model will soon become available for beta testing.

Being ε, κ the porosity and the permeability of the medium respectively :

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

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

$$D_i = -\frac{\mu U_i}{\kappa} + \frac{1{,}75\rho|u|}{\sqrt{150}\sqrt{\kappa}\varepsilon^{1.5}}U_i$$

Application : Thermal problems

Current development : Adding more post treatment tools such as the calculation of the convection coefficient "h" based on a rigorous approach for the bulk temperature.

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

With q the heat flux, Ts the temperature at the surface and Tm the "bulk temperature"

- For solid thermal problems, the h is used as a convection boundary condition in order to approximate the effect of the fluid cooling
- The h can be found in empirical tables
- However, it can sometimes be hard to determine this h especially if geometries are complex.
- The objective of outputting this h is for the user to later apply it to solid thermal problems and to have a good general idea of the fluid's behavior.

Future developments :

- More tools to control the BL mesh size
- Adaptive surface remeshing
- Wave generator for levelset and multiphase applications

Thank you for your attention

Questions ?