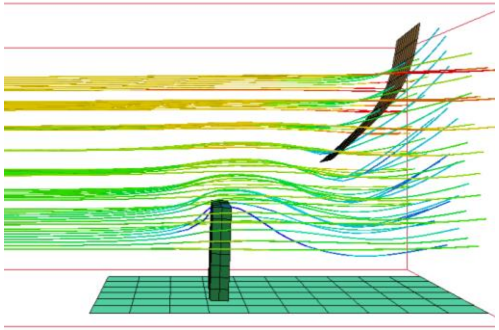
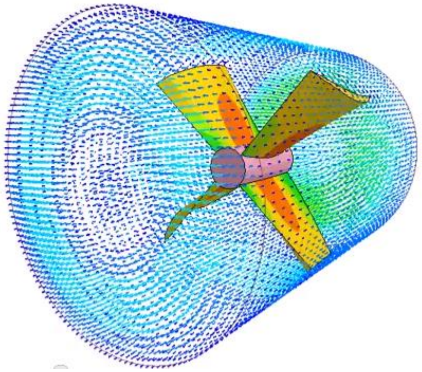
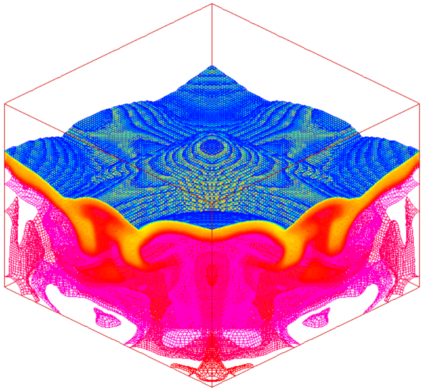
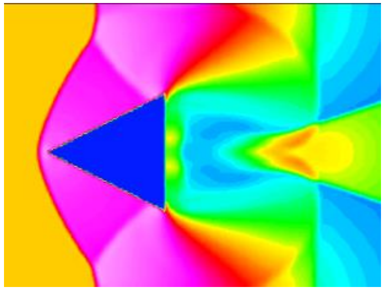
Compressible CFD (CESE) Module Presentation

Zeng Chan Zhang, Kyoung-Su Im, Iñaki Çaldichoury

Introduction and applications

Characteristics

- Double precision.
- Second order Explicit.
- **2D and 2D axisymmetric** solver **/** **3D** solver.
- FSI available for 3D solver.
- **SMP** and **MPP** versions available.
- Dynamic memory handling.
- New set of keywords starting with ***CESE** for the solver.
- **Automatically coupled** with LS-DYNA solid and thermal solvers.
- Coupled with the R7 chemistry and stochastic particle solver (*CHEM and *STOCHASTIC).

CESE method main advantages

- A **unified treatment of space and time**.

  (By the introduction of **conservation element** (CE) and **solution element** (SE), the conservation of scheme is always maintained in space and time, locally and globally).

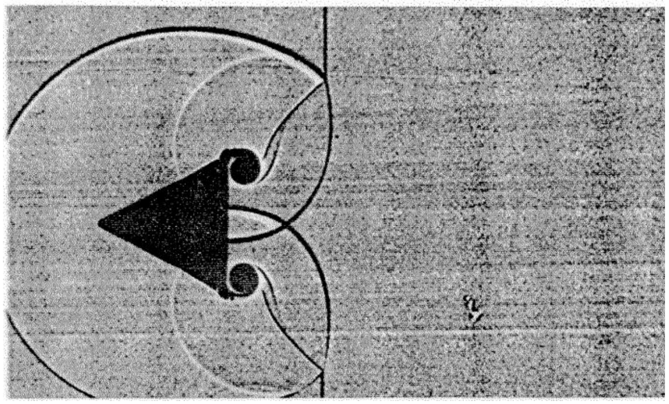
- A **novel shock capturing strategy** without using a Riemann solver.

- **High accuracy**.

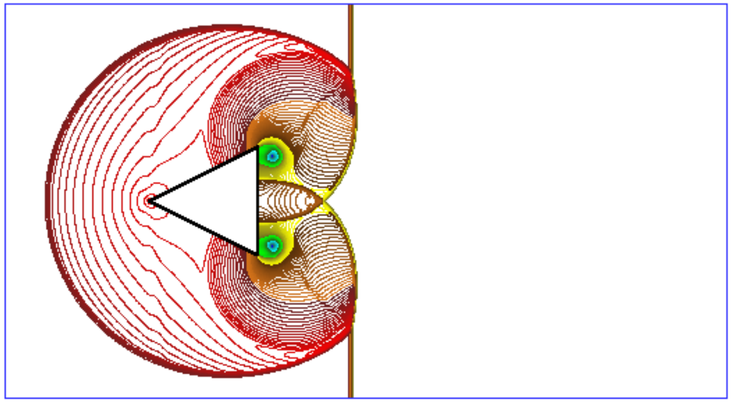
  (Both flow variables and its spatial derivatives are solved simultaneously).

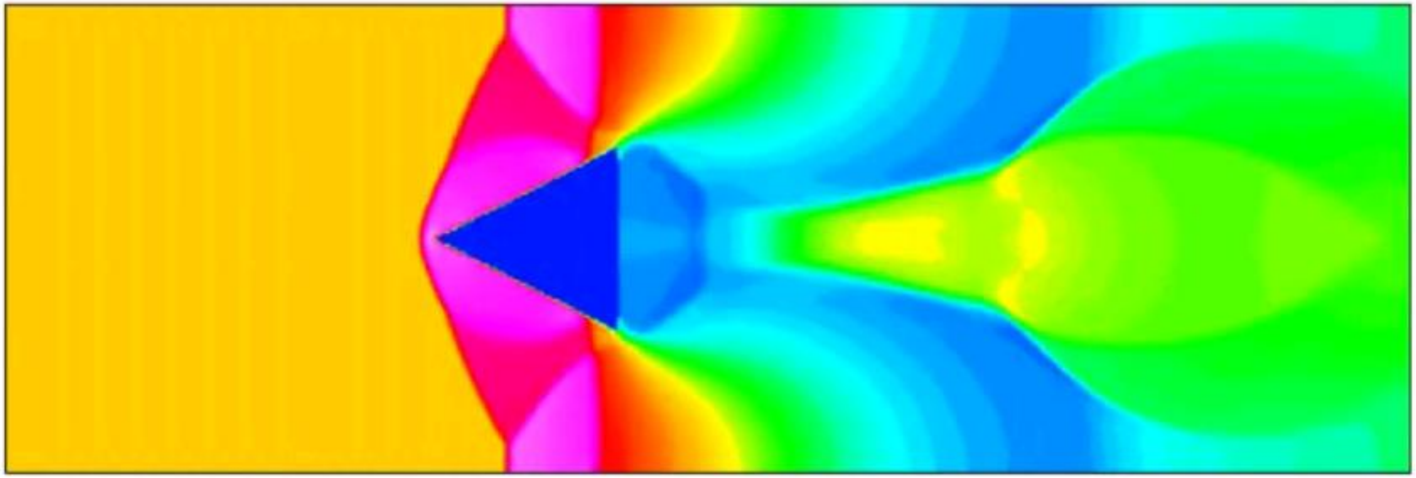
Supersonic shock wave capturing:

Experimental picture :



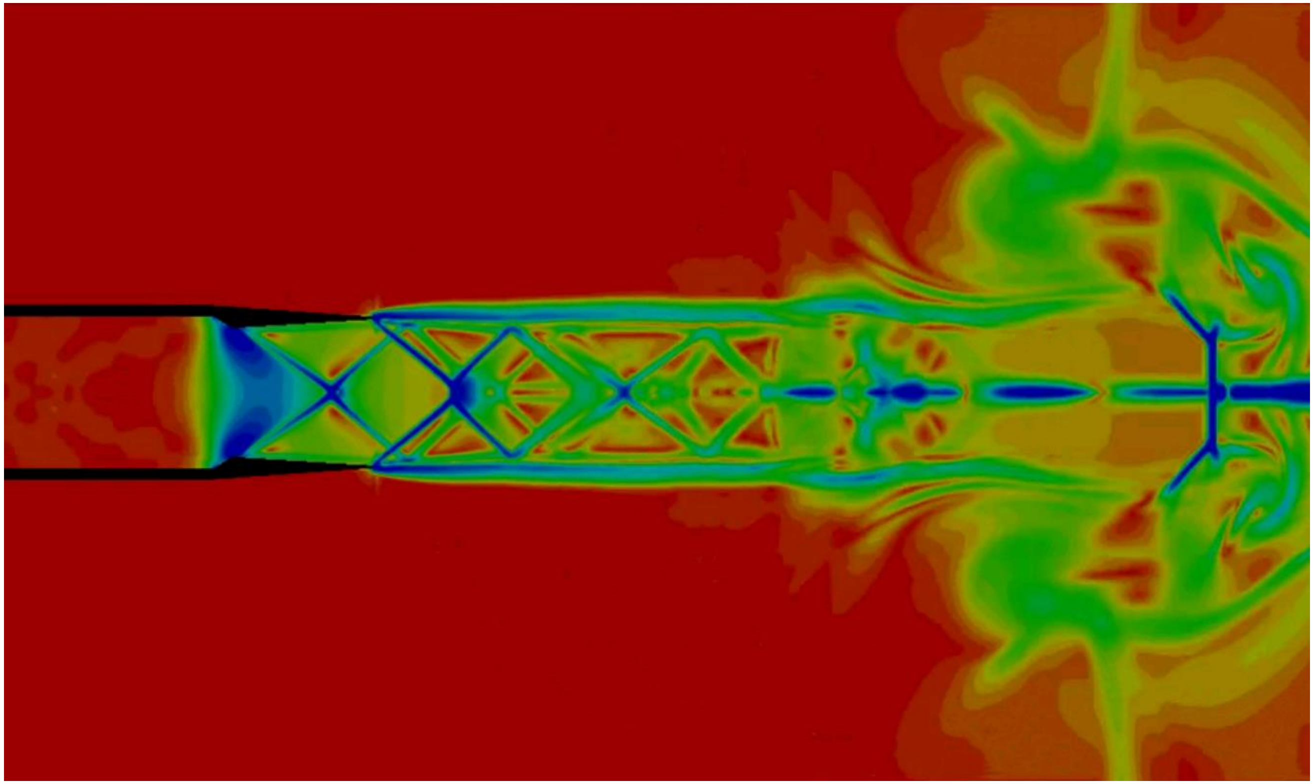
Numerical result:



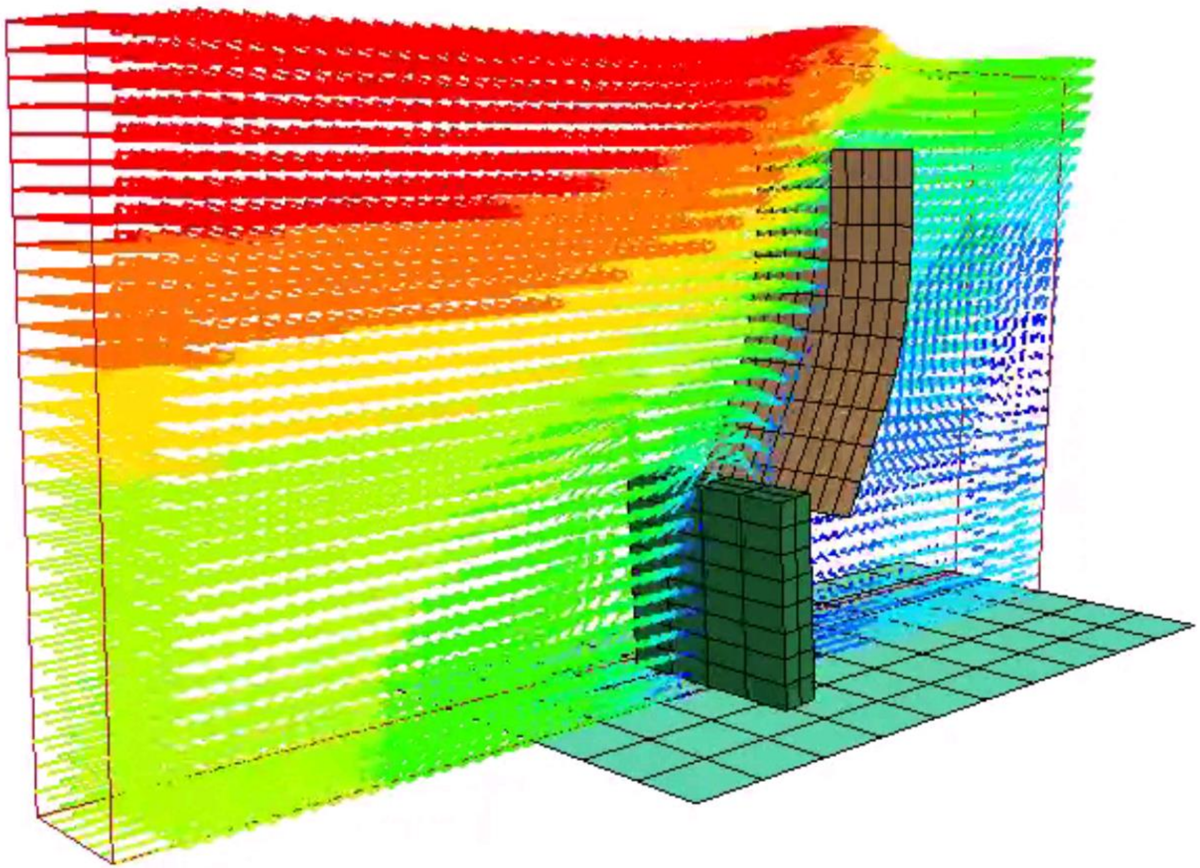


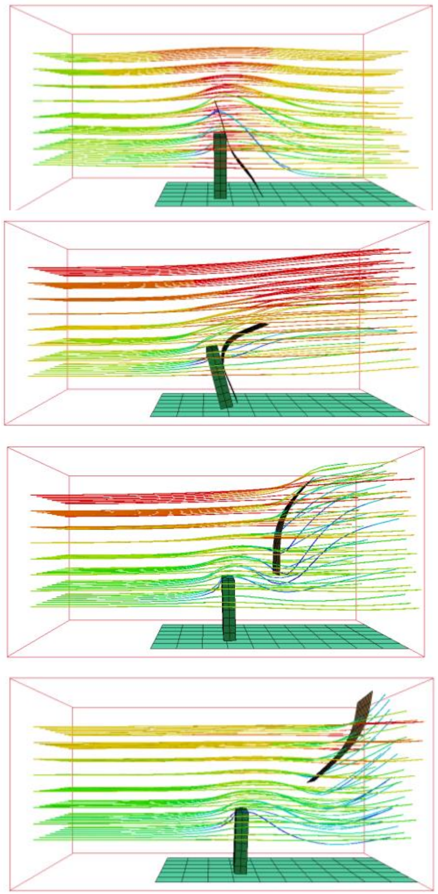
Flow structure of supersonic jets from conical nozzles (shock diamonds) :

Courtesy of Kazuya Yamauchi of Lancemore Corporation, Japan:

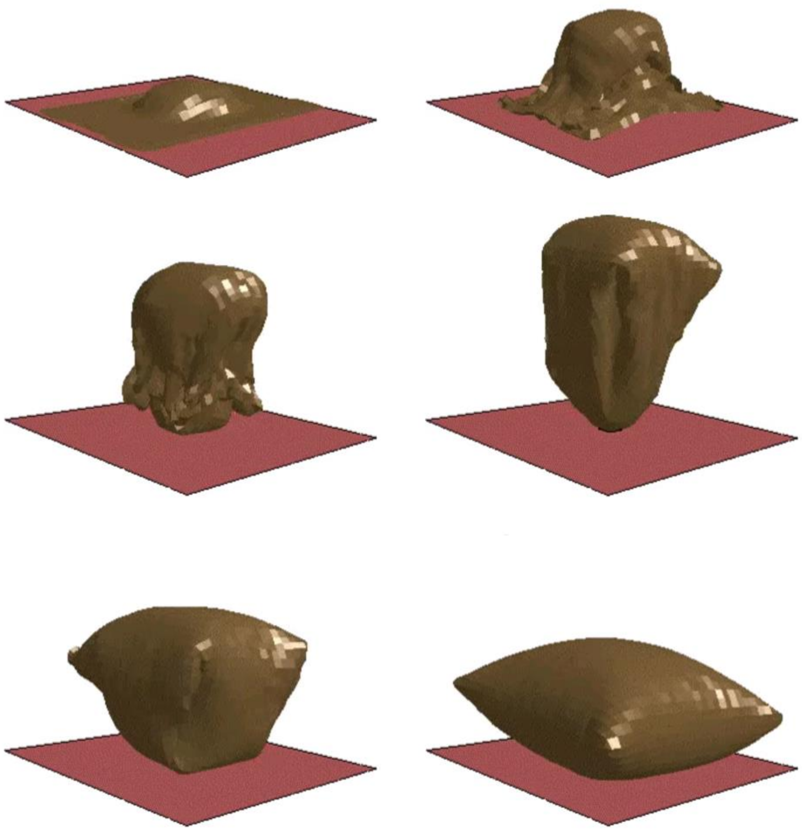


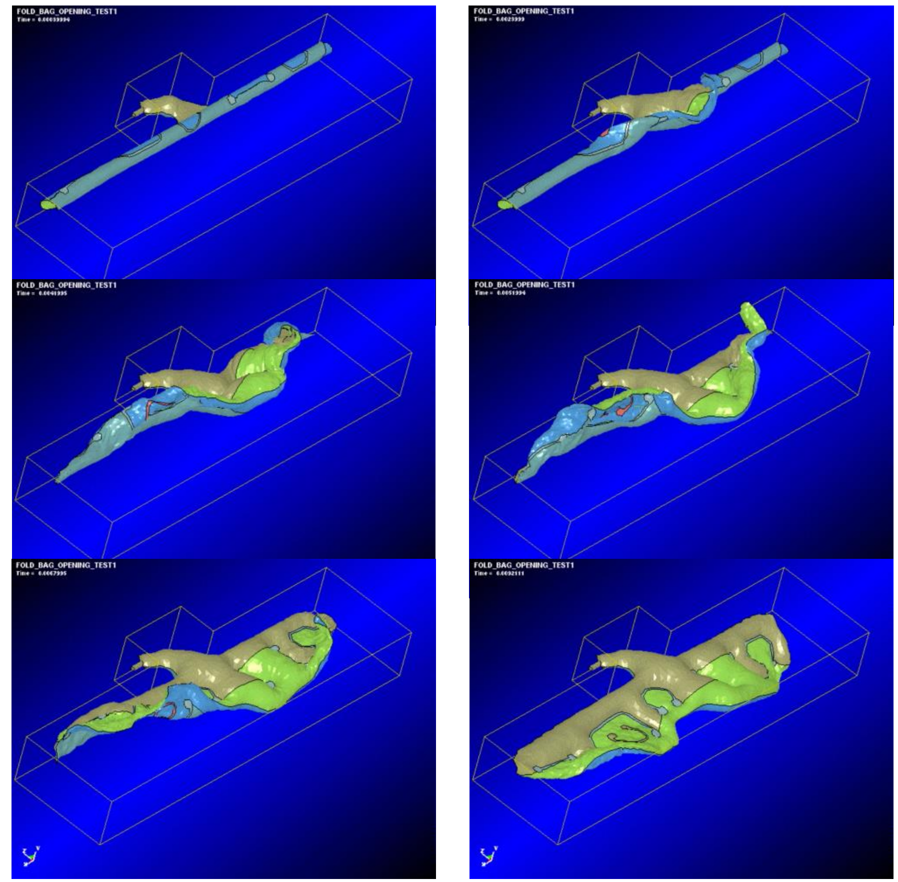
3D FSI waving flag problem :





Airbag applications:

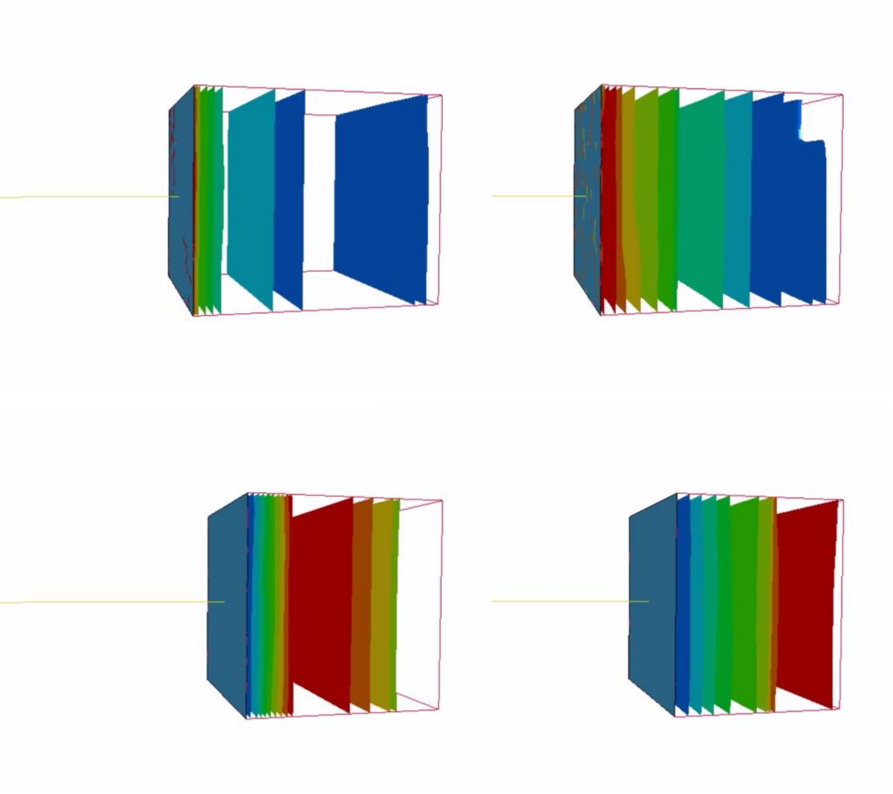




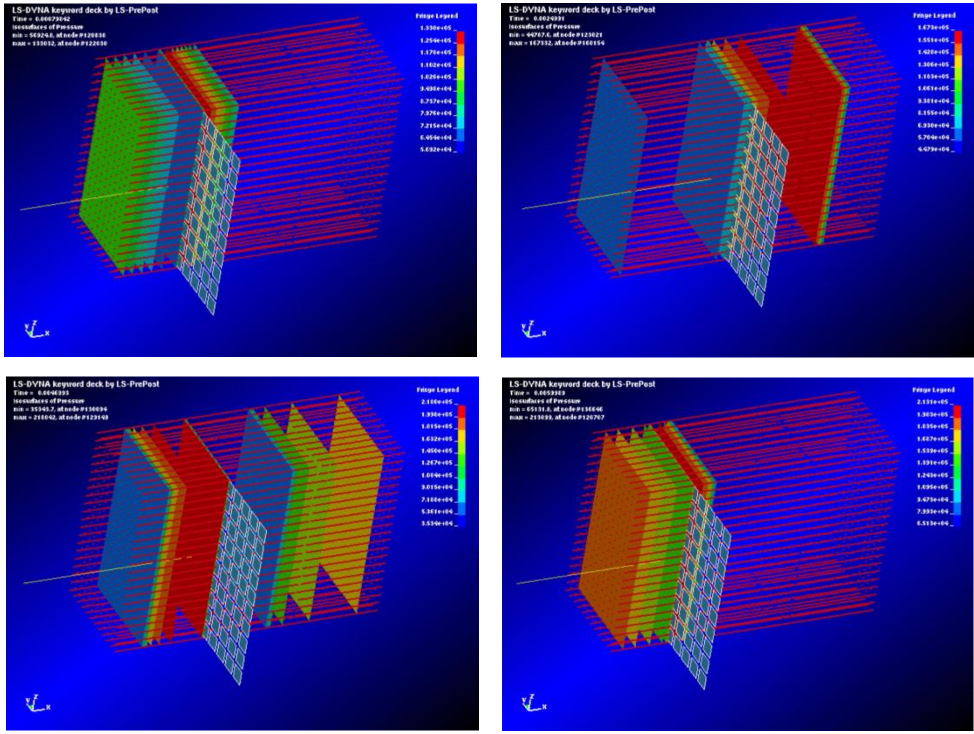
Courtesy of TAKATA Corp., Japan

LSTC
Livermore Software Technology Corp.

Piston type applications with or without moving mesh:

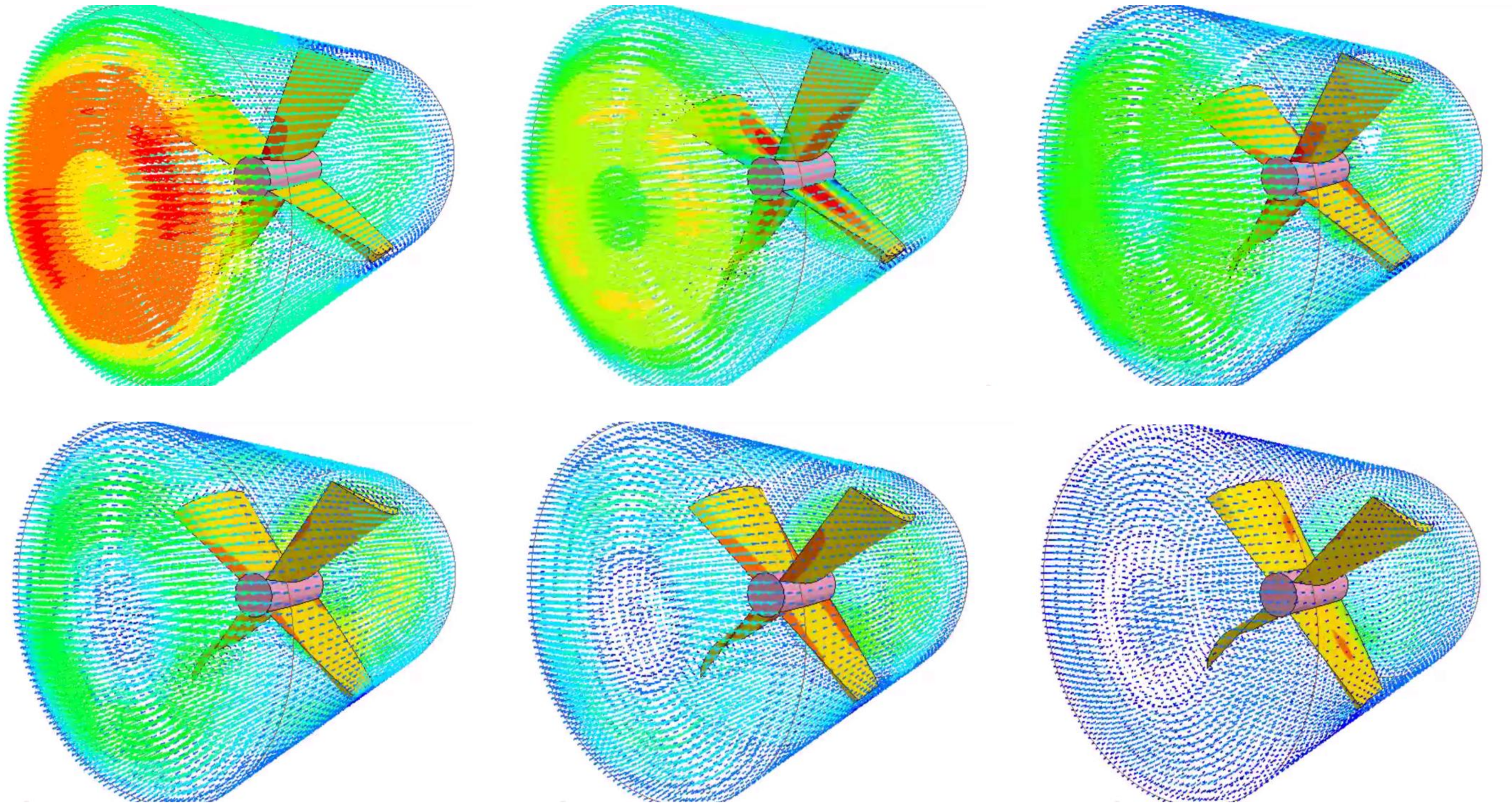


Moving mesh



Embedded mesh

Turbomachinery applications:



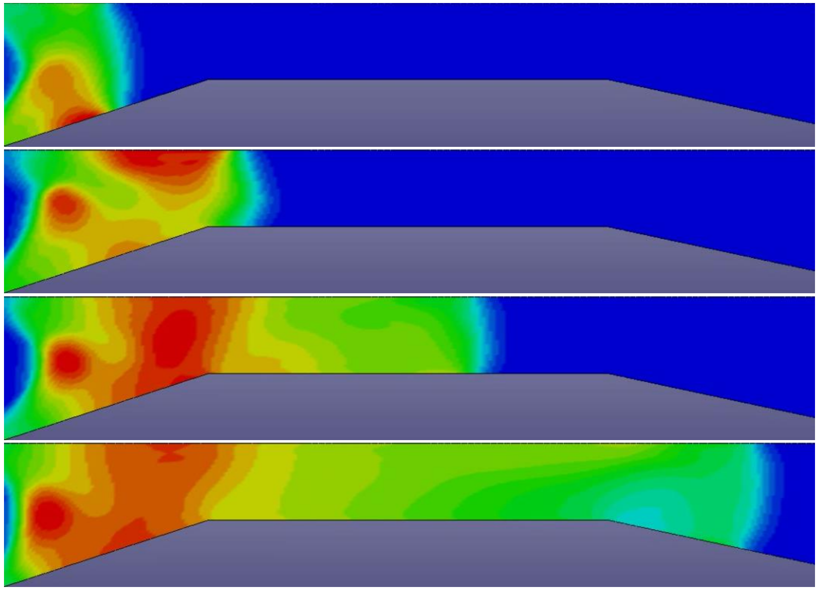
Can be coupled with Chemistry solver (beta) for Chemical reactions at hypersonic speeds:

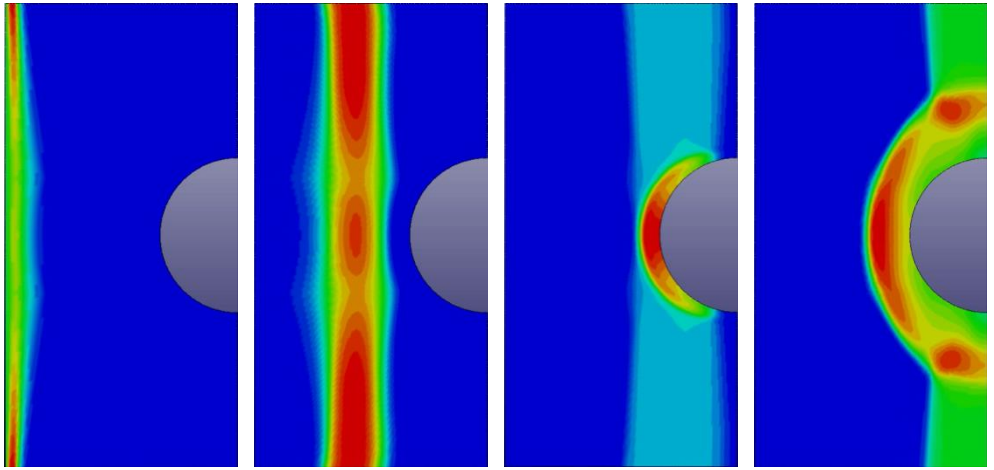
Detailed reaction model

5 species: O_2, N_2, O, N, NO
with 11 reaction steps

Initial mixture: $O_2 + 3.76N_2$

Navier-Stokes solver:

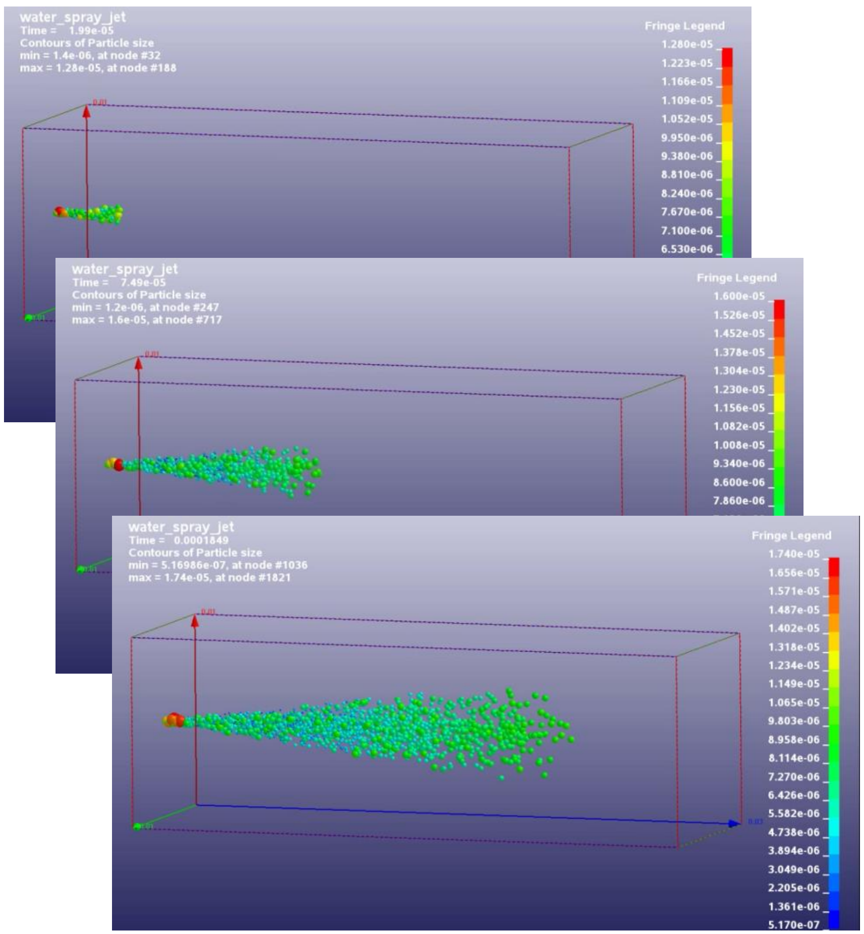


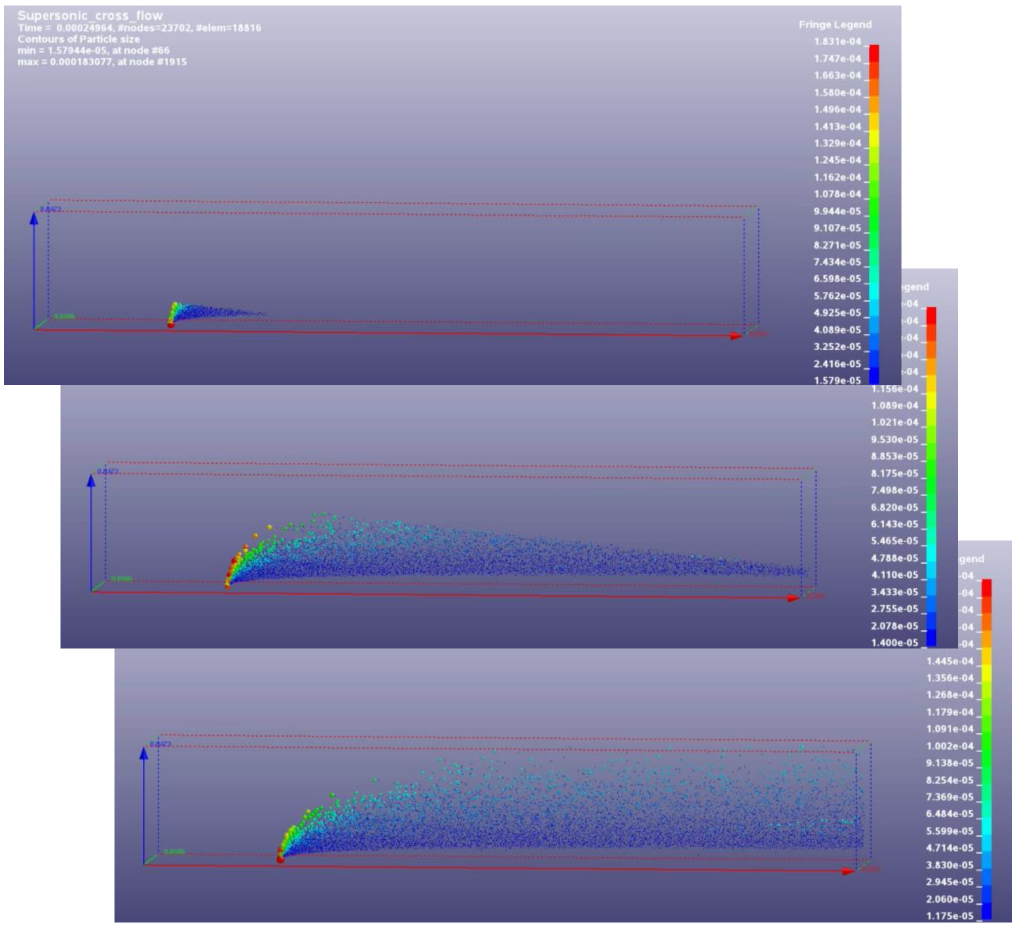


Pressure fringe of a blunt body:
Hypersonic inflow at Ma = 7 & T = 600 K

Pressure fringe of a ramped duct:
Hypersonic inflow at Ma = 4 & T = 500 K

Spray and particle dispersion (beta):



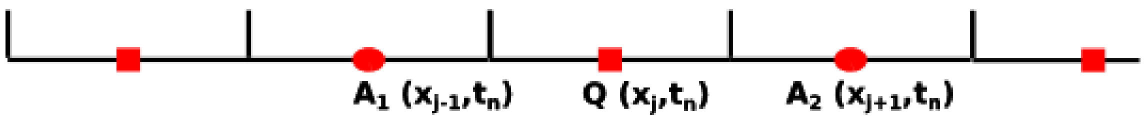


	CESE :	ICFD :	ALE :
Low speed aerodynamics *(turbulence)*	-	✓	-
High speed aerodynamics (shock waves)	✓	-	-
Explosions using JWL EOS or similar	-	-	✓
Airbags-Pistons	✓	-	✓
Free surface problems (slamming)	-	✓	✓
FSI capabilities	✓	✓	✓
Chemistry reactions	✓	-	-
Stochastic particles	✓	-	-

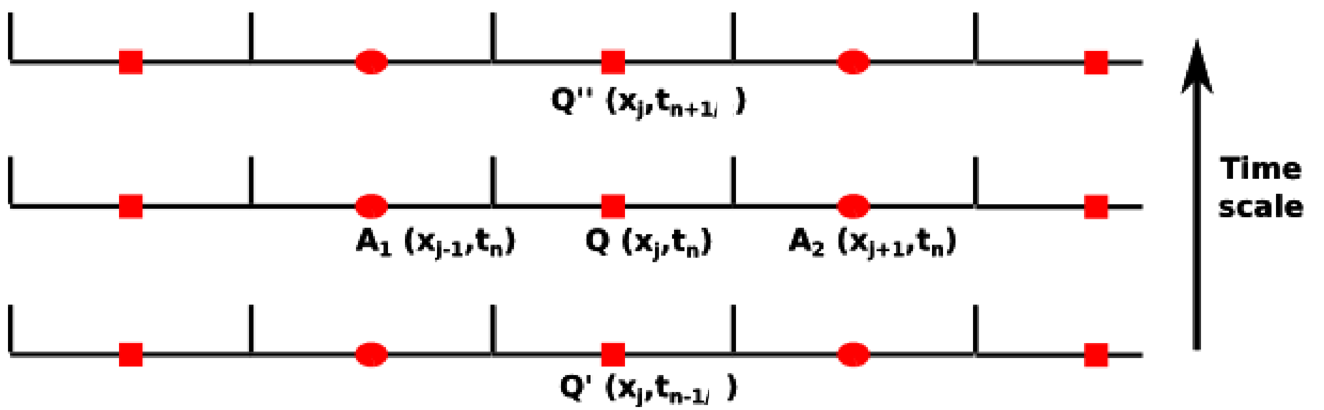
The CESE scheme

1D Convection equation: $\frac{\partial u}{\partial t} + a\frac{\partial u}{\partial x} = 0$

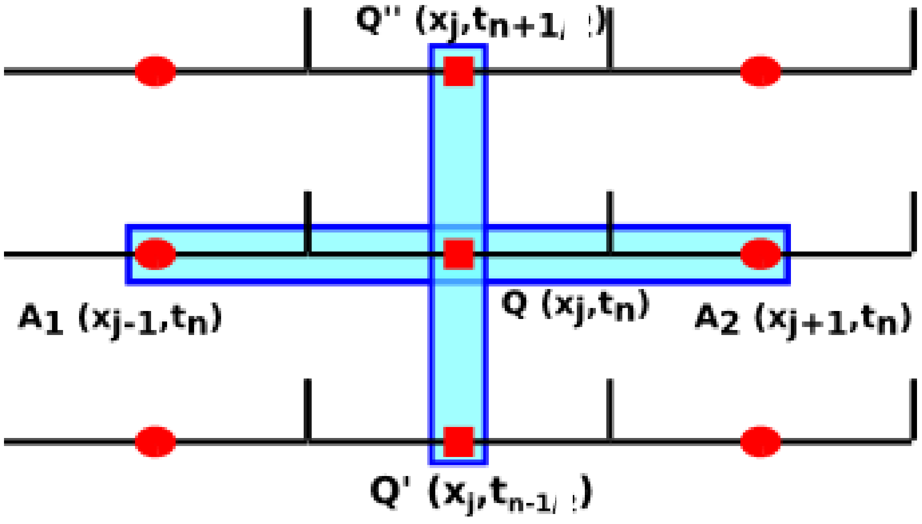
- **Step 1** :**Element discretization** :



- **Step 2** : **Expansion in the time dimension** (time acts as an additional spatial dimension). Euclidian Space $E_2\ (x, t)$:



- **Step 3** : Definition of a **SE** :



  - Along the SE domain, the flow variable will be approximated by a Taylor series :

$$u^*(x,t) = u_q(x,t) + \frac{\partial u_q}{\partial x}(x - x_q) + \frac{\partial u_q}{\partial t}(t - t_q)$$

  - The time and spatial derivatives can be related by using the flow convection-diffusion equation (Euler for perfect flows, N.S for viscous flows, in our case the convection equation : $\frac{\partial u_q}{\partial t} = -a\frac{\partial u_q}{\partial x}$) so that only $u_q(x,t)$ and its spatial derivative $\frac{\partial u_q}{\partial x}$ remain as unknowns to solve.

- **Step 4** : Integral form of convection equation :

  - For the CESE scheme, since the Euclidian space is of dimension n+1 through the introduction of time as a spatial coordinate, it is possible to define a flux **h** such as **:**

$$\iiint_V \nabla h.\, dV = \iint_{S(V)} \boldsymbol{h}.\, d\boldsymbol{s} = 0$$

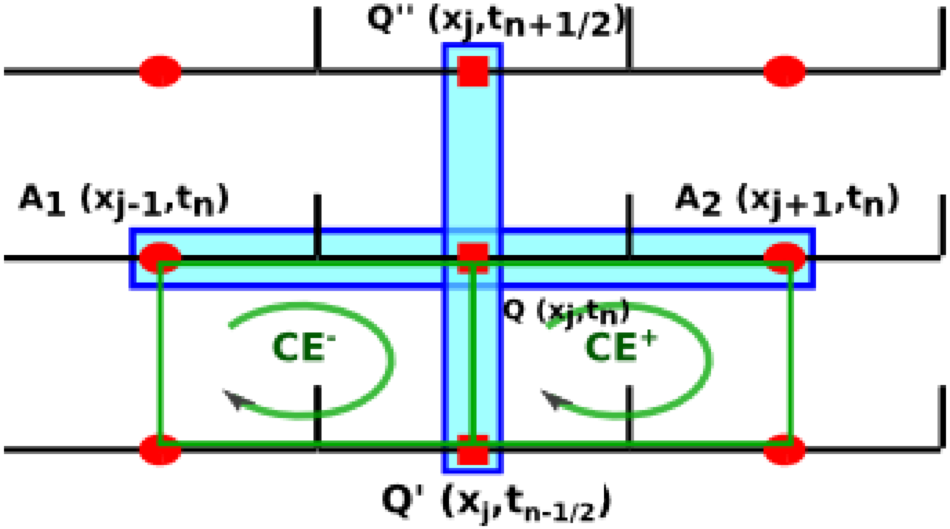
With **h.ds** now representing the **space time flux** of **h** leaving the Volume V though the surface element ds. In the present case $d\boldsymbol{s} = (dx, dt)$ and $\boldsymbol{h} = (au, u)$.

  - By using careful choice for the construction of a conservation element, the **CESE scheme** permits the **conservation** of the solution both **spatially** and **temporarily**.

- Step 4 :**Definition of a CE**:

  - The spate-time integral equation for flow conservation along the lines formed by a CE gives:

$$\oint_{S(CE^{\pm})} \vec{h}_m^{\,*} \cdot d\vec{s} = 0$$



  - CE- and CE+ yield two equations for the two unknowns $u_q(x,t)$, $\frac{\partial u_q}{\partial x}$ function of quantities expressed between *j-1* and *j+1* and *n-1*. This allows for the solving of the complete system.

CESE method summary

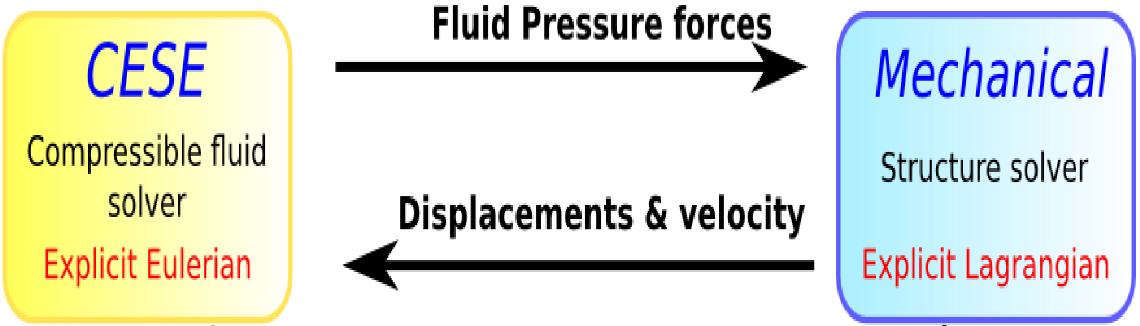
- Solving of integral form of the fluid equations. Integration on **Space-time** domain (CE) : space-time **local and global conservation** of the solution.
- Flow **variables AND** their **derivatives** are **solved simultaneously** : highly accurate second order scheme
- SE used to advance through time.
- No need for further treatment for shock waves (**No Riemann solver**)

FSI resolution steps

- For the **FSI interaction**, a **loose coupling** method is used where the Lagrangian embedded structure passes **displacement and nodal velocities** information to the fluid solver which in turn, communicates back **pressure forces** that act as exterior loads on the structure.

- For the Fluid-structure interface tracking, a **quasi constraint method** is used.

- Since **both meshes are independent**, the interface will be tracked **automatically** and the **FSI problem treated** without any input from the user's perspective.



- It is usually recommended to use a **finer mesh for the fluid** rather than the solid.

- **No leakage** can occur.

- Since both the **CESE solver** and the **solid mechanics LS-DYNA solver** can run **as stand alones**, both have their own timestep bounded by their own CFL condition.

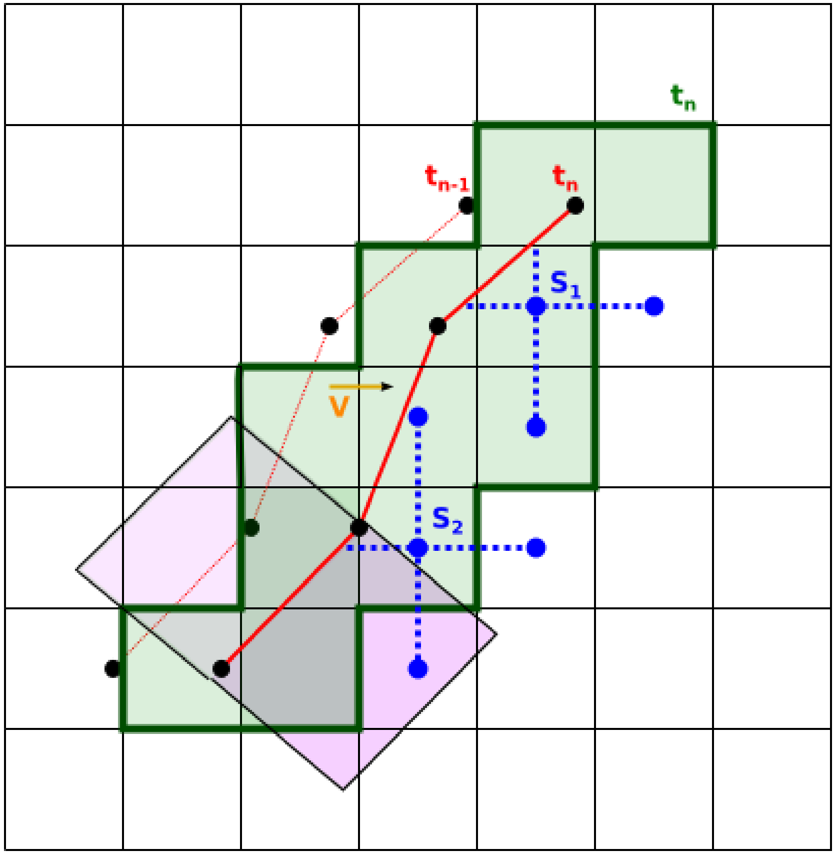
- For FSI problems, the fluid solver will track those two timesteps and use the smallest of the two for both domains.

- **Moving mesh capabilities** exist for special types of FSI problems (**pistons**).

- Step 1 : **Lagrangian embedded structure** moves through fluid mesh and **communicates nodal displacements** to the fluid.

- Step 2 : Eulerian fluid performs a **sorting procedure** in order to determine which elements have at least one **neighbor element that is "blocked" by the solid**. Such neighbors will be treated as **wall boundary conditions**.

- Step 3 : Fluid solver will **track** which **fluid elements** are close to it, compute an **average pressure** and use it as an **exterior pressure load applied to it**.



What’s new ?

Application : External and internal aerodynamics :

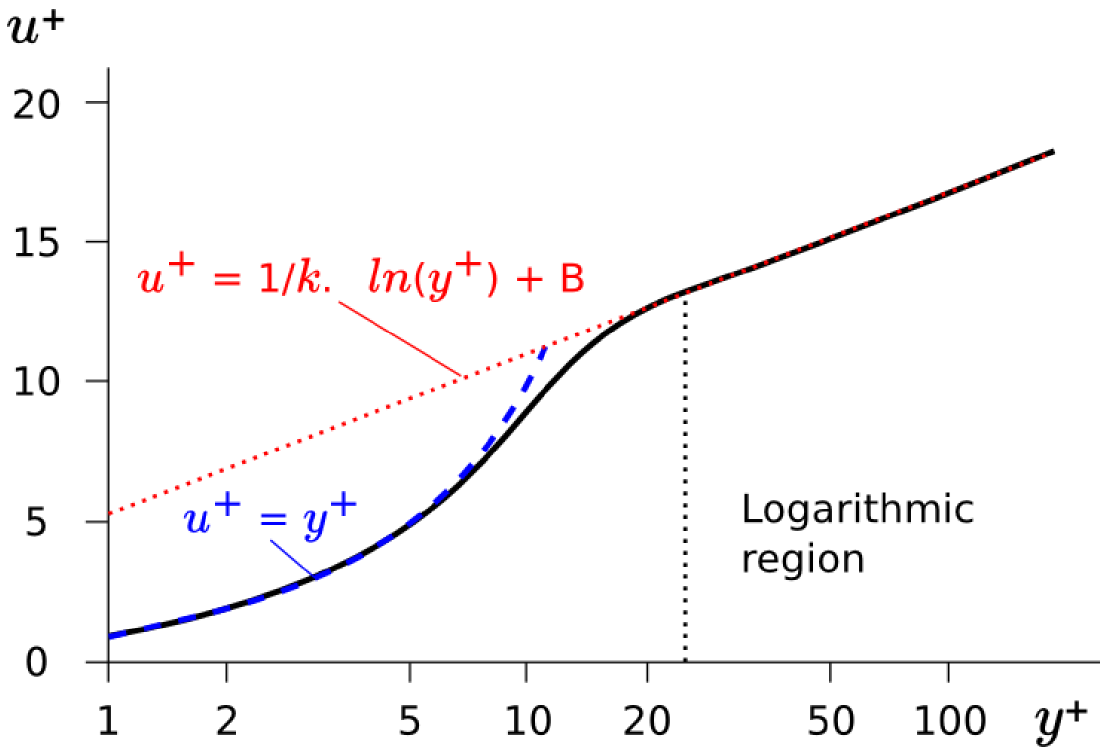
Current development : Adding RANS turbulence models and laws of the wall.

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

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

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

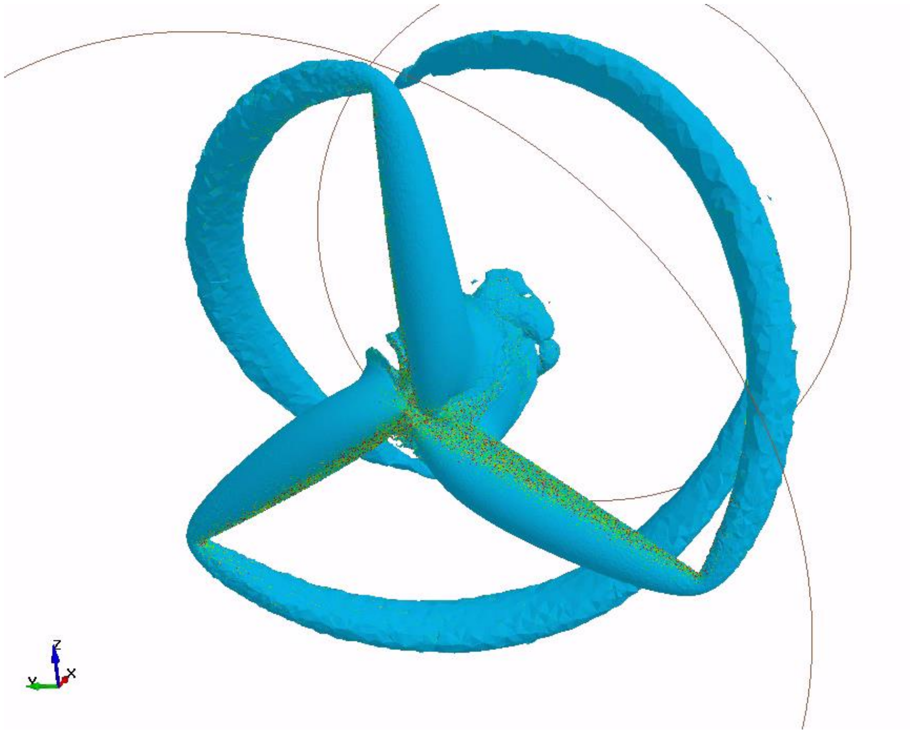
$$\tau_w = \mu \frac{\partial u}{\partial y}$$



Application : External and internal aerodynamics :

Current development : Adding non inertial reference frame. Currently available for beta testing for pure CFD and moving mesh FSI cases.

KEYWORD: *CESE_DEFINE_NON_INERTIAL



Application : Conjugate heat transfer problem:

Current development : Adding coupling with the solid thermal solver for conjugate heat transfer

Future developments :

- **Multi materials for explosions (similar to ALE)**