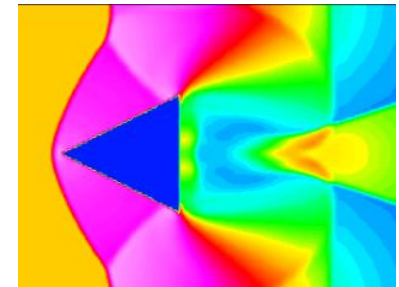
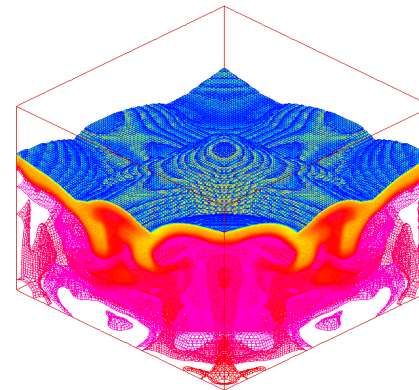
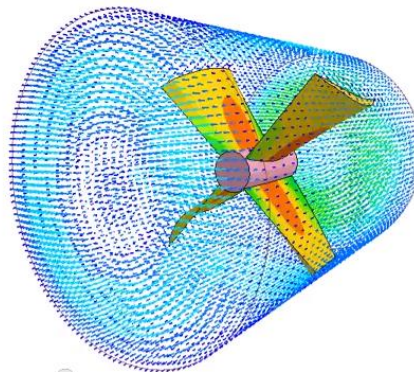
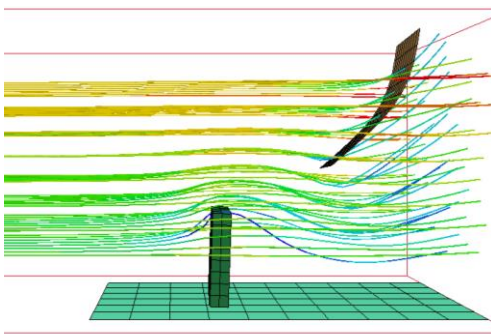


# Compressible CFD (CESE) Module Presentation

***Zeng Chan Zhang, Kyoung-Su Im, Iñaki  
Çaldichoury***



# Introduction and applications

## Characteristics

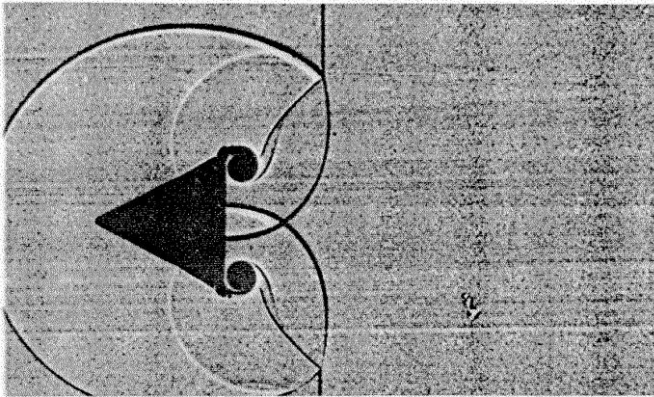
- Double precision.
- Second order Explicit.
- **2D and 2D axisymmetric** solver / **3D** solver.
- FSI available for 3D solver.
- **SMP** and **MPP** versions available.
- Dynamic memory handling.
- New set of keywords starting with **\*CESE** for the solver.
- **Automatically coupled** with LS-DYNA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

## CESE method main advantages

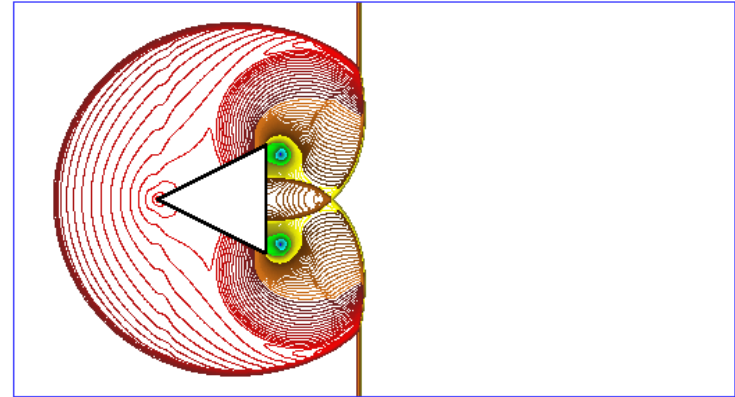
- A **unified treatment of space and time**.  
(By the introduction of **conservation element** (CE) and **solution element** (SE), the conservation of scheme is always maintained in space and time, locally and globally).
- A **novel shock capturing strategy** without using a Riemann solver.
- **High accuracy**.  
(Both flow variables and its spatial derivatives are solved simultaneously).

## Supersonic shock wave capturing:

Experimental picture :

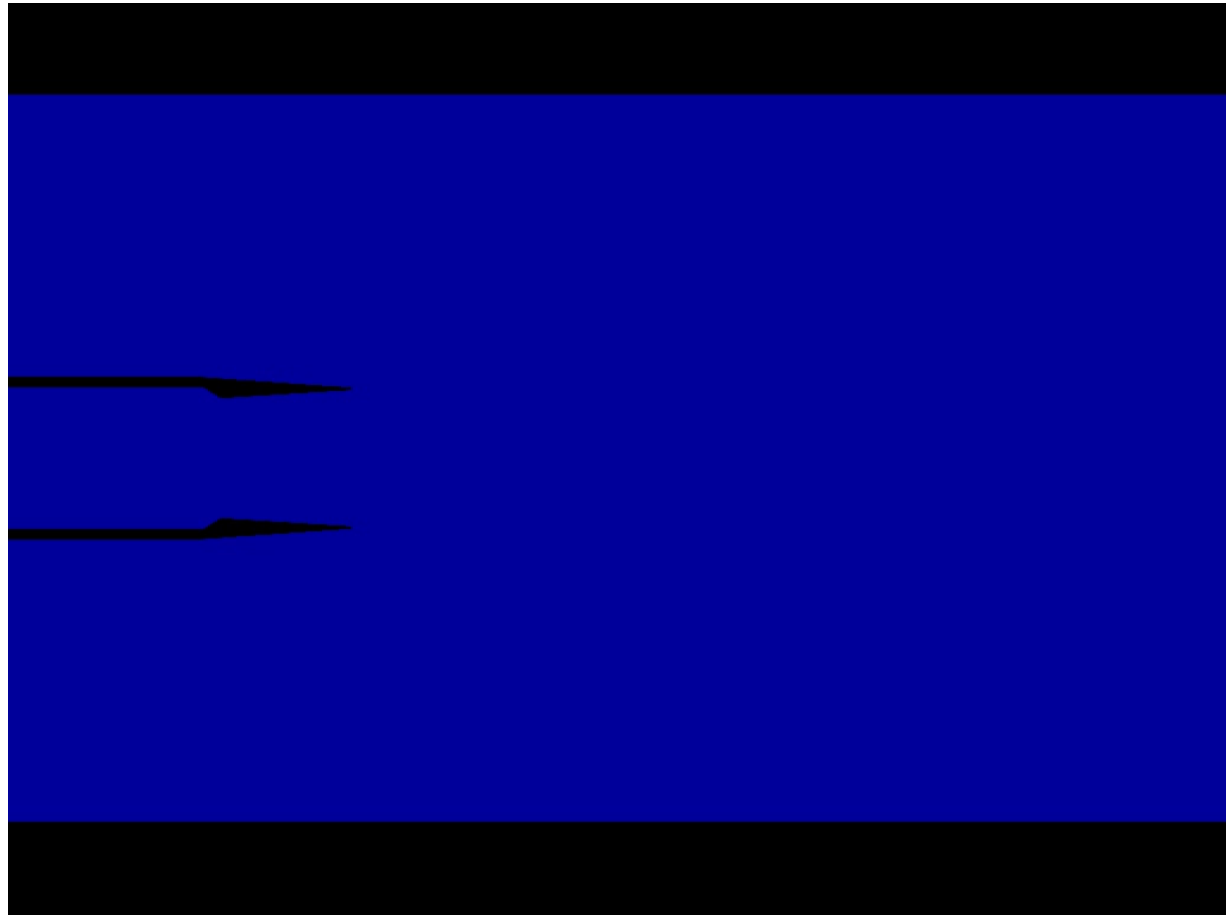


Numerical result:

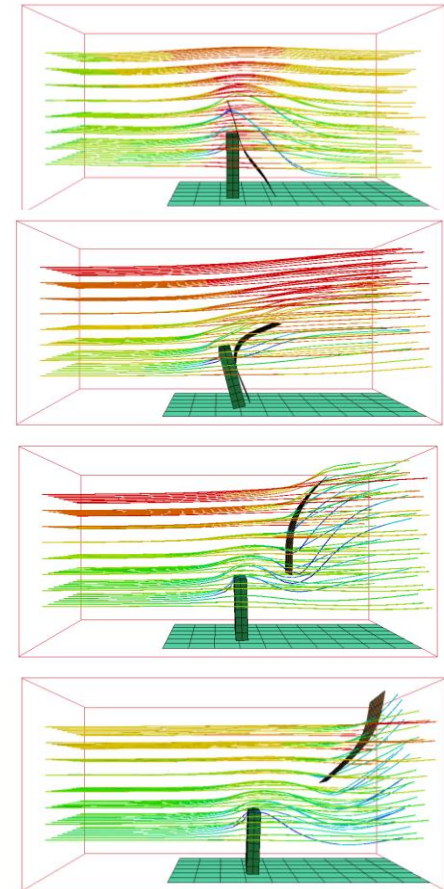
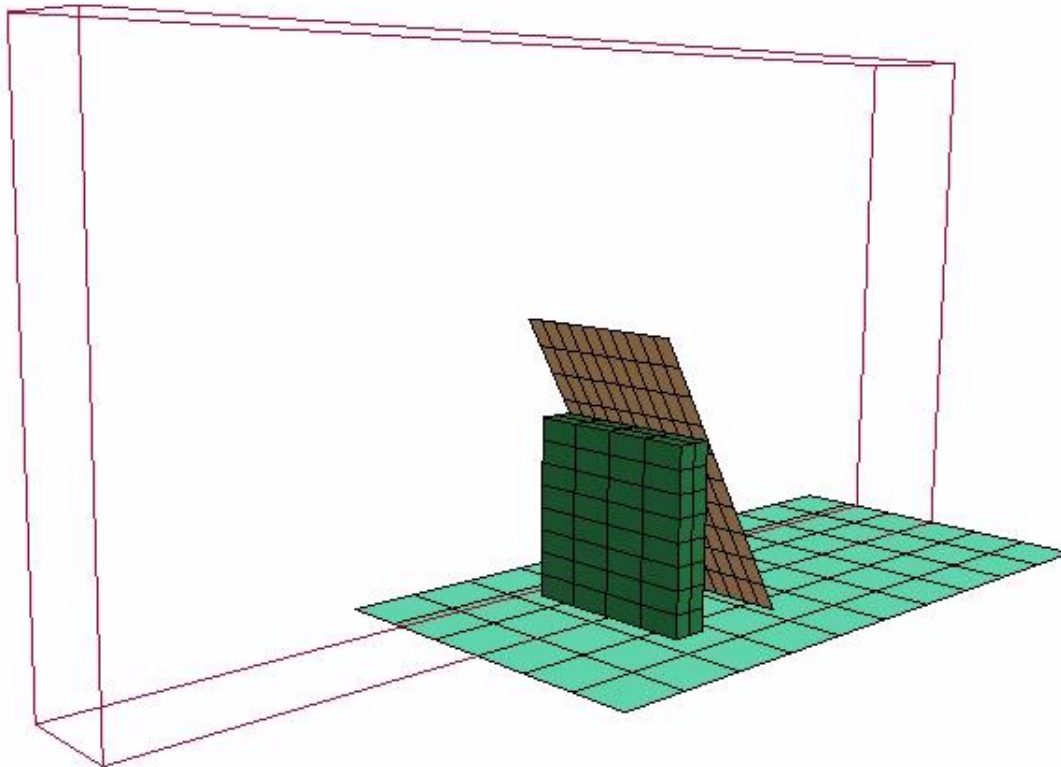


## Flow structure of supersonic jets from conical nozzles (shock diamonds) :

Courtesy of Kazuya  
Yamauchi of  
Lancemore  
Corporation, Japan:



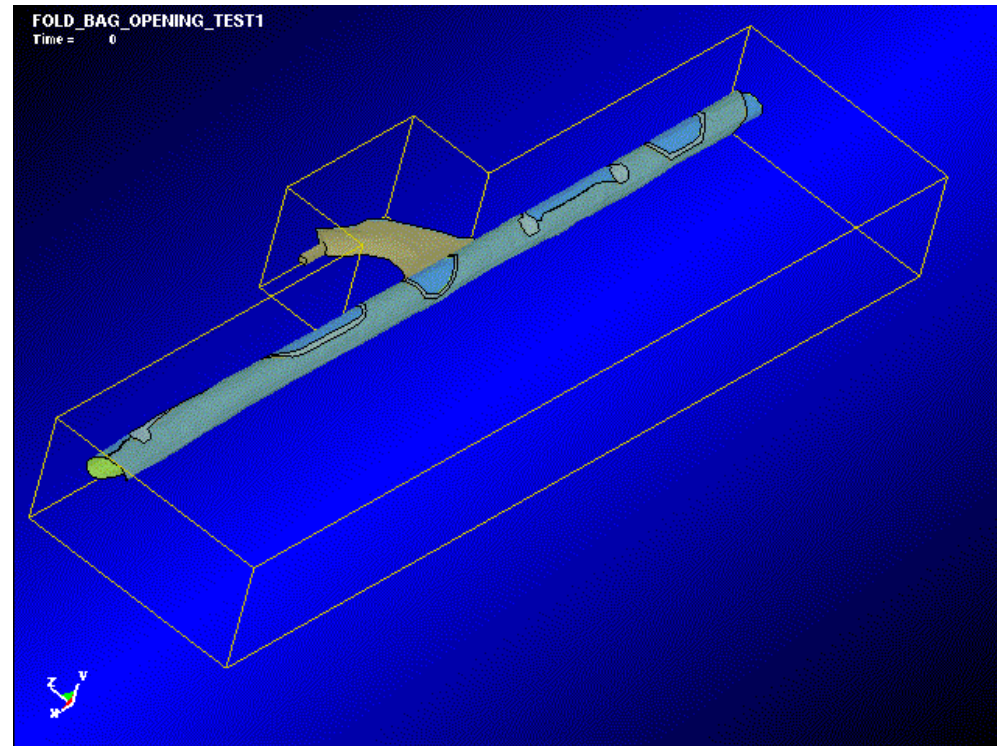
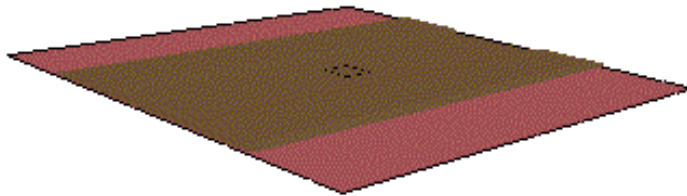
## 3D FSI waving flag problem :





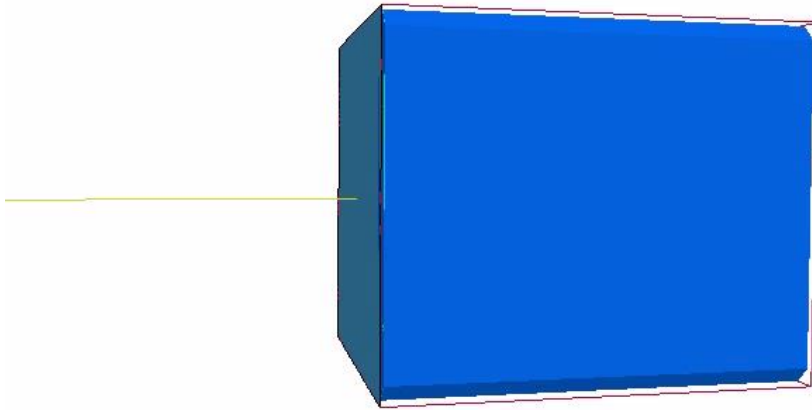
## Airbag applications:

Courtesy of TAKATA corporation, Japan:

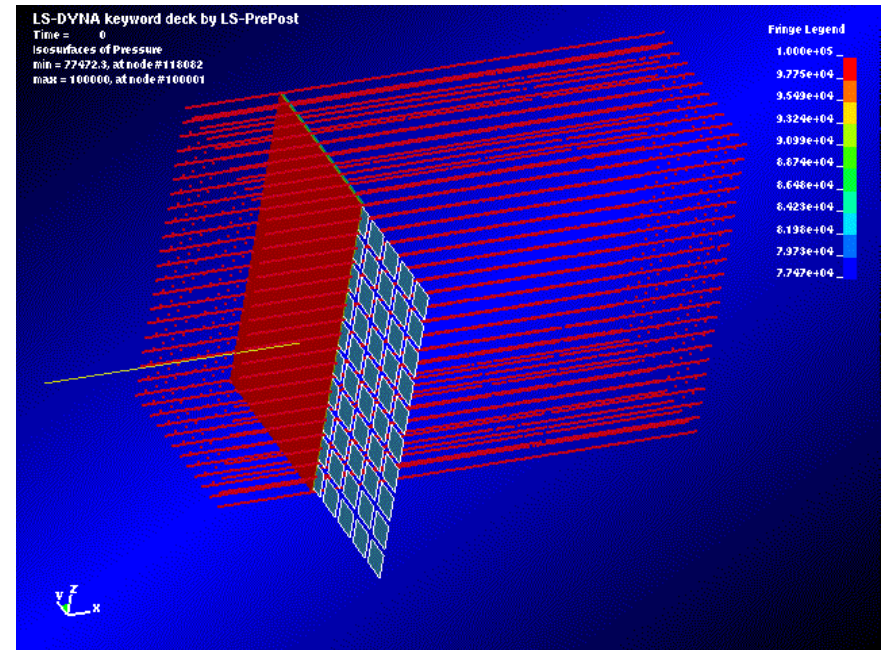




## Piston type applications with or without moving mesh:

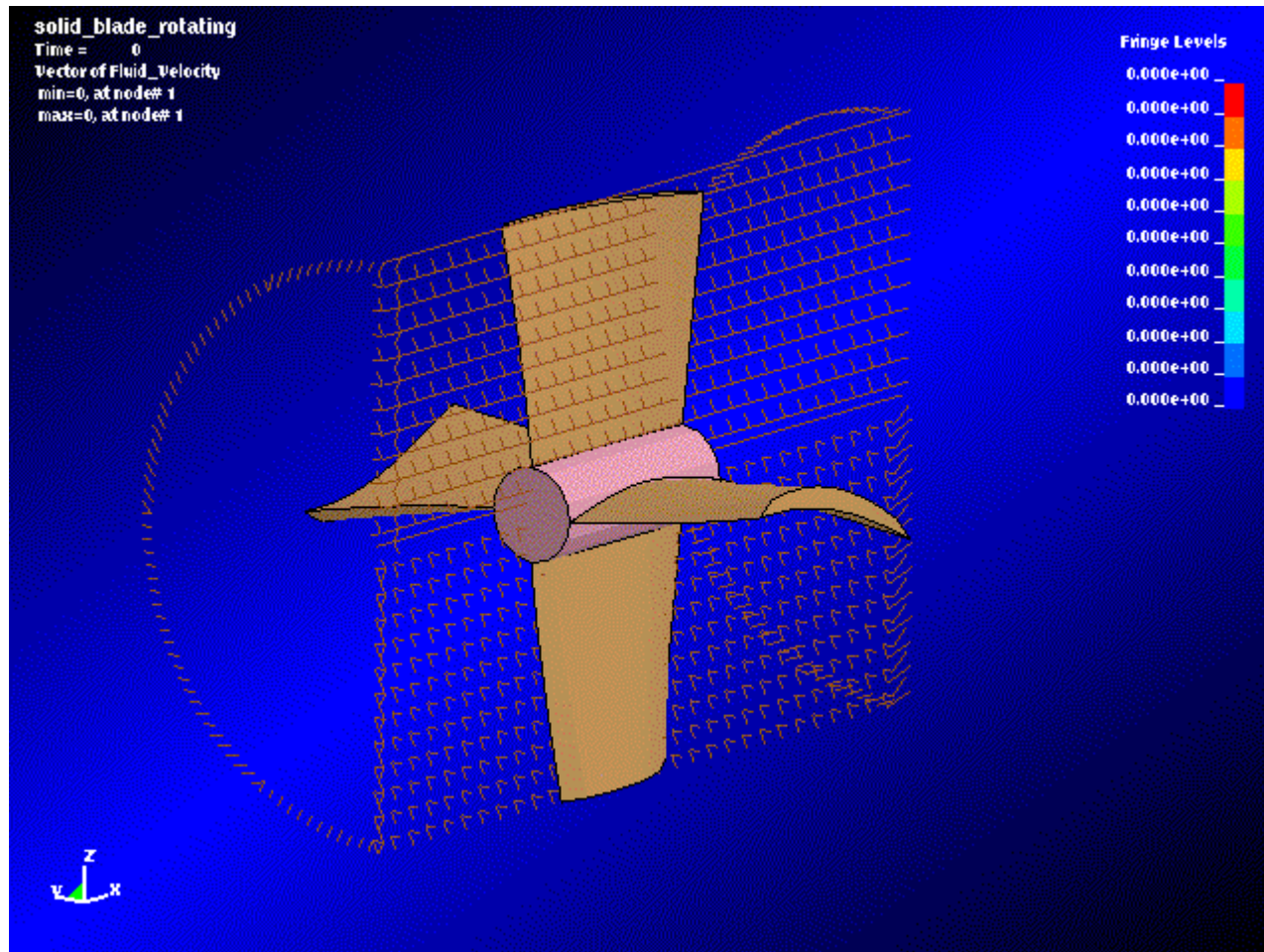


**Moving mesh**



**Embedded mesh**

## Turbomachinery applications:



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

## Detailed reaction model.

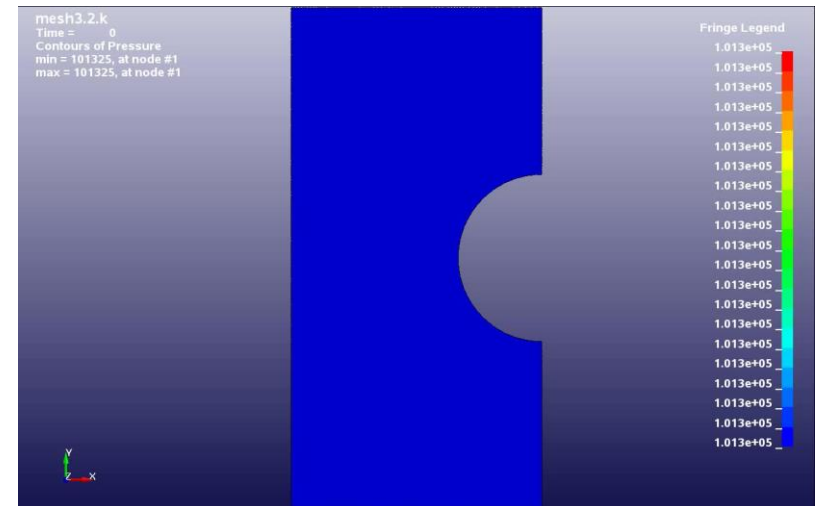
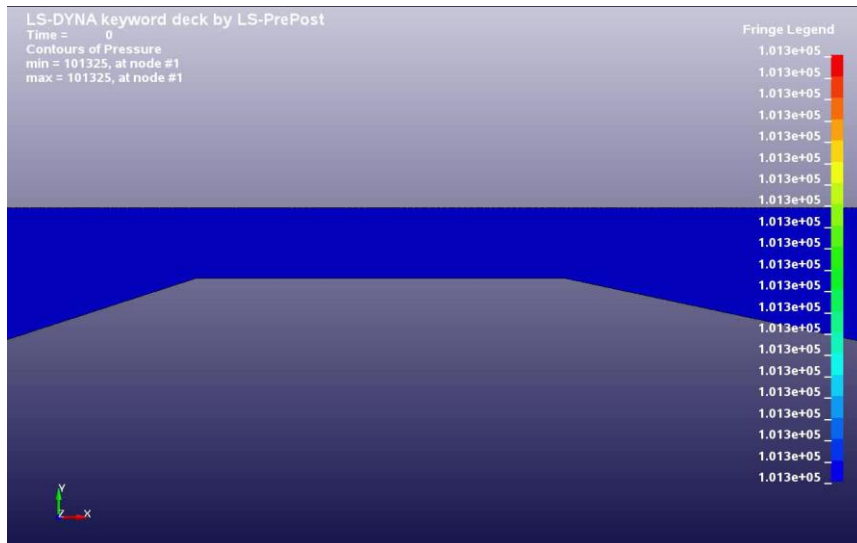
**5 species: O<sub>2</sub>, N<sub>2</sub>, O, N, NO**

## 11 reaction steps

## Initial mixture:

**O<sub>2</sub> + 3.76N<sub>2</sub>**

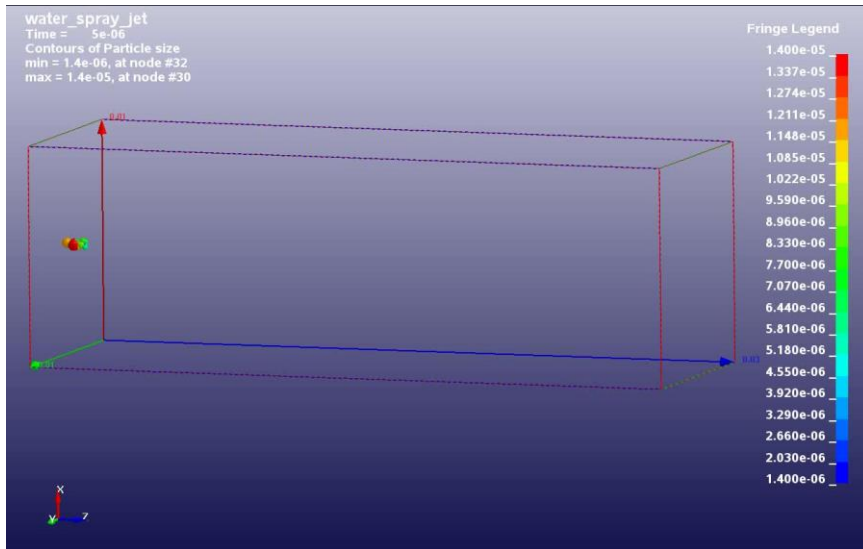
## N-S equation solver:



## Blunt body: hypersonic inflow at $Ma = 7$ & 600 K

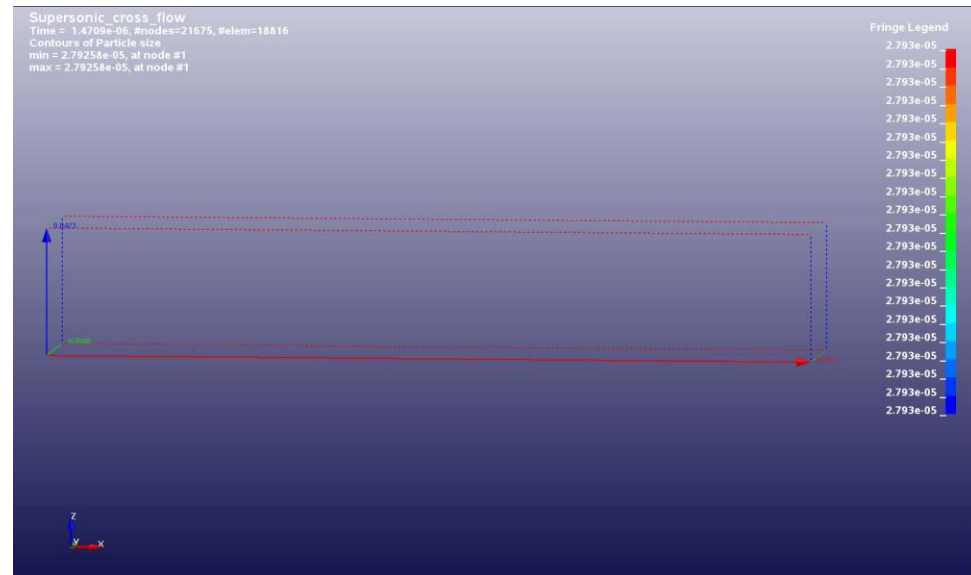
## Ramped duct: hypersonic inflow at $Ma = 4$ & 500 K

## Spray and particle dispersion (beta):



## Water spray jet

## Supersonic cross flow



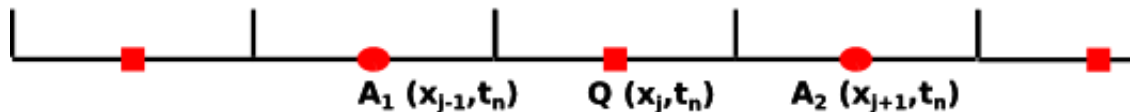
	<b>CESE :</b>	<b>ICFD :</b>	<b>ALE :</b>
<b>Low speed aerodynamics (turbulence)</b>	-	✓	-
<b>High speed aerodynamics (shock waves)</b>	✓	-	-
<b>Explosions using JWL EOS or similar</b>	-	-	✓
<b>Airbags-Pistons</b>	✓	-	✓
<b>Free surface problems (slamming)</b>	-	✓	✓
<b>FSI capabilities</b>	✓	✓	✓
<b>Chemistry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

# The CESE scheme

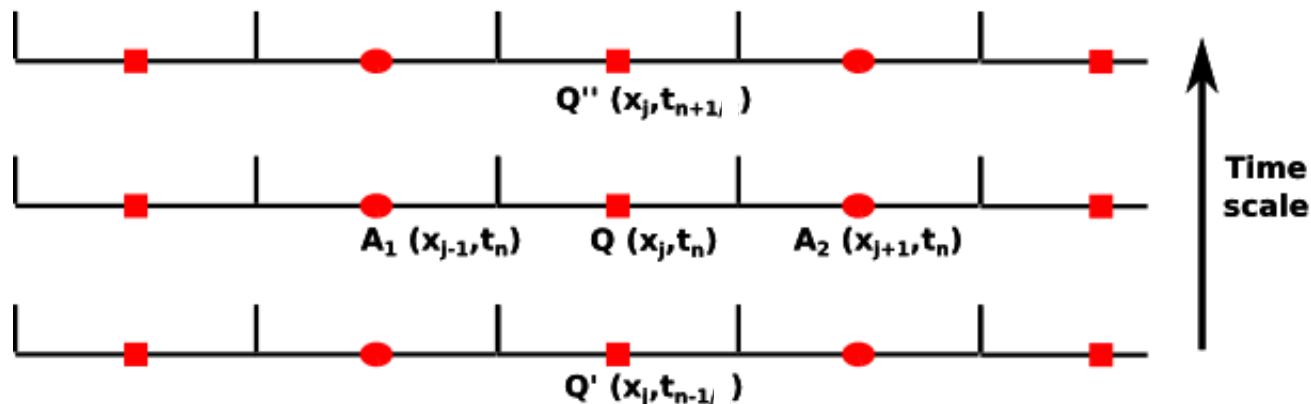


1D Convection equation:  $\frac{\partial u}{\partial t} + a \frac{\partial u}{\partial x} = 0$

- Step 1 : **Element discretization** :



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

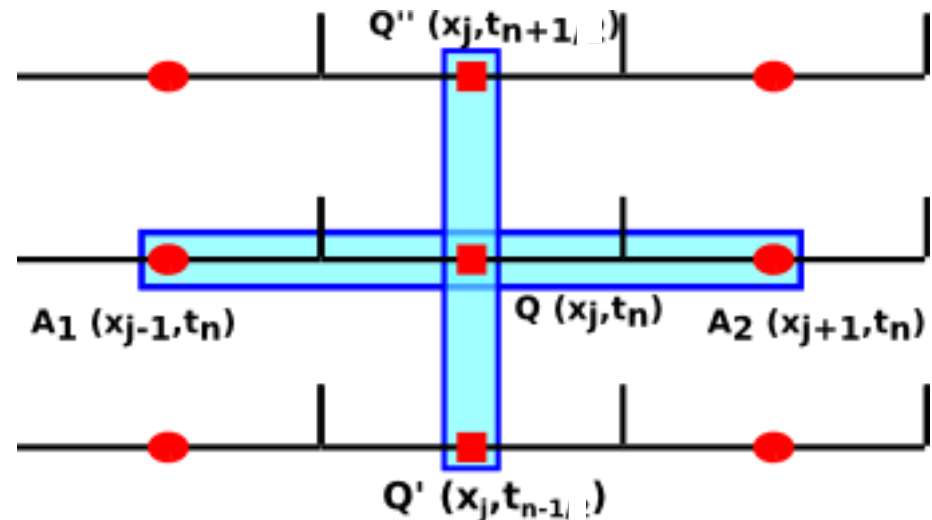


- **Step 3** : Definition of a **SE** :

- 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  $\mathbf{h}$  such as :

$$\iiint_V \nabla h \cdot dV = \iint_{S(V)} \mathbf{h} \cdot d\mathbf{s} = 0$$

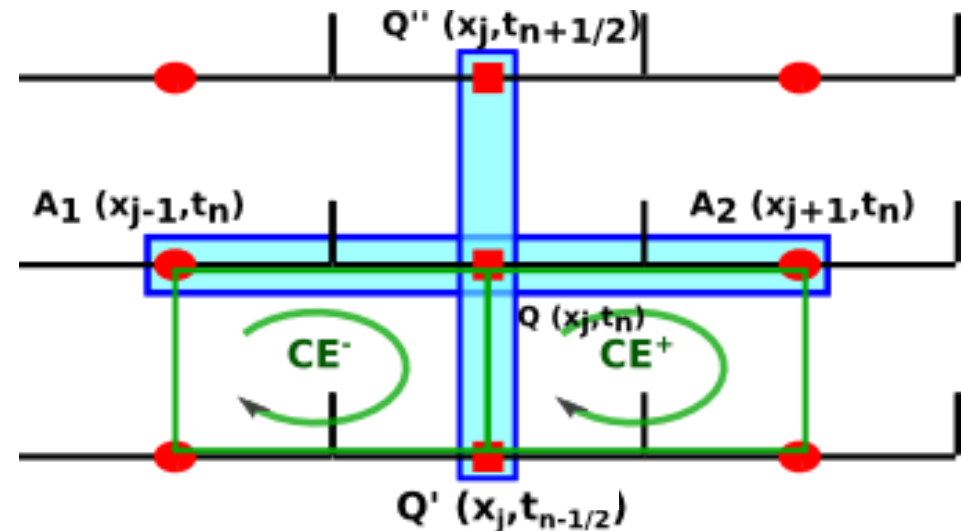
With  $\mathbf{h} \cdot d\mathbf{s}$  now representing the **space time flux** of  $\mathbf{h}$  leaving the Volume  $V$  through the surface element  $ds$ . In the present case  $d\mathbf{s} = (dx, dt)$  and  $\mathbf{h} = (au, u)$  .

- 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.

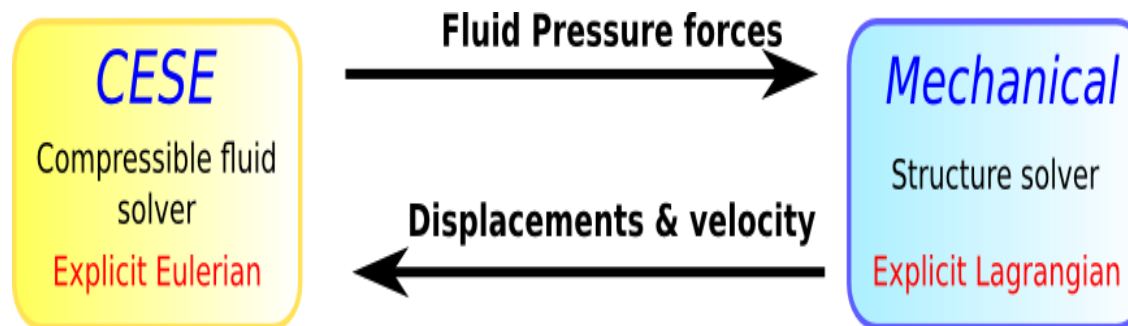
## CESE method summary

- 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

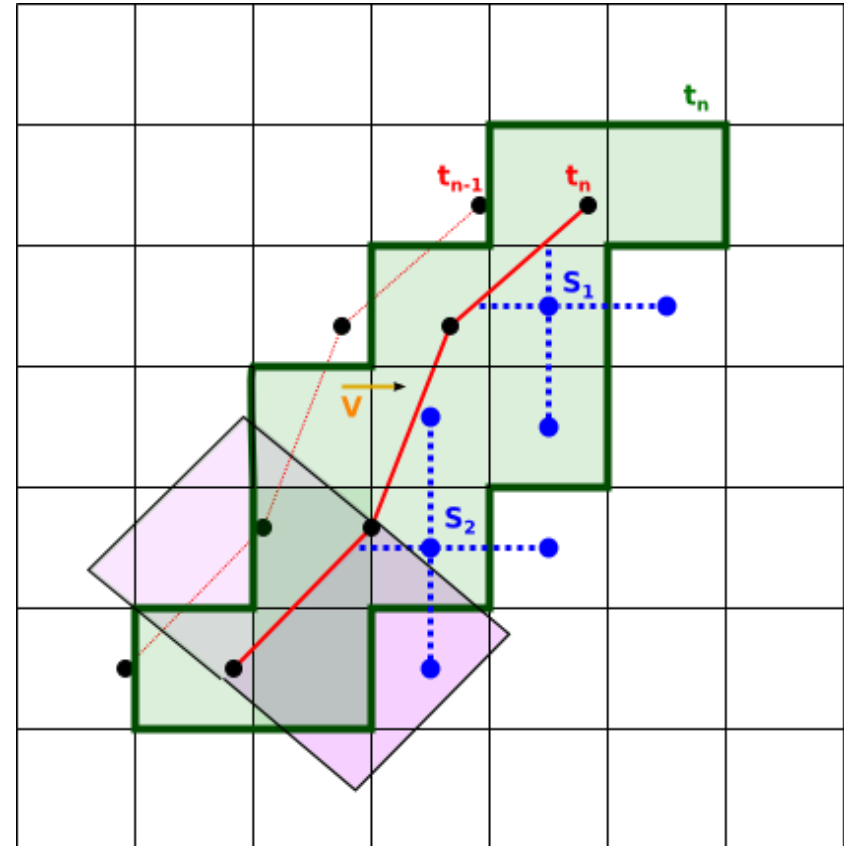


- For the **FSI interaction**, a **loose coupling** method is used where the Lagrangian embedded structure passes **displacement and nodal velocities** information to the fluid solver which in turn, communicates back **pressure forces** that act as exterior loads on the structure.
- For the Fluid-structure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



# **Thank you for your attention**

## **Questions ?**

