LS-DYNA® R7: Recent developments, application areas and validation results of the compressible fluid solver (CESE) specialized in high speed flows.

Zeng-Chan Zhang
Iñaki Çaldichoury

Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, CA 94551

Abstract

LS-DYNA version R7 includes CFD solvers for both compressible and incompressible flows. The compressible flow solver is based on the CESE method, a novel numerical method for solving conservation laws. It has many nontraditional features such as space-time conservation, second order accuracy for flow variables and a powerful shock wave capturing strategy.

This paper will focus on some advanced features of the solver namely its FSI capabilities. Several potential industrial applications will be presented such as airbag openings, piston type applications and turbomachines. Some results on high speed supersonic flows will also be presented for illustration and discussion purposes.

1- Introduction

The new compressible fluid solve included in LS-DYNA R7 double precision is based upon the Space-Time Conservation Element and Solution Element Method (CESE).

The CESE method, originally proposed in [1] and further developed in [2] and [3] is a novel integral numerical method for solving conservation laws and includes many nontraditional features such as:

- Local and global space-time conservation of the solution which limits the diffusion of the solution and loss of precision.
- Second order scheme for both flow variables and their spatial derivatives for better solution accuracy.
- Novel supersonic shock capturing strategy which does not involve any Riemann solver resulting in less calculation costs.

Furthermore, the objective of these new solvers included in LS-DYNA R7 is not only to solve for their particular domain of physics but to make full use of LS-DYNA's capabilities and material library in order to solve coupled multiphysics. Consequently, the CESE solver has been extended in LS-DYNA to solve fluid structure interaction problems (FSI). In such cases, the solid can be modeled as a classic Lagrangian solid mechanics part while the fluid flow is based on Eulerian frame.

This current paper will begin by introducing the CESE scheme applied to a one-dimensional case for illustration purposes. Several validation cases will be shown. It will then proceed to the FSI capabilities and will expend on some validation cases and potential industrial applications.

2- The CESE resolution scheme

2.1 1D example

For simplification purposes, we will consider a one dimension form of the 1D convection equation based on the work by [1] :

$$\frac{\partial u}{\partial t} + a\frac{\partial u}{\partial x} = 0$$

With constant advection term a.

This results in the element spatial discretization shown in Figure 1a) with points (A_1, Q, A_2) at various spatial location $x_j$and at a given time n (t_n). The first step of the CESE scheme is to consider time as another additional spatial coordinate thus forming the two dimensional Euclidean space E_2(See Figure 1b)). Let us now define the following current density vector $\boldsymbol{h}$ in E_2 :

$$\boldsymbol{h} = (au, u)$$

It can then be shown that by using Gauss' divergence theorem in the Space-time E_2 , the integral form of the convection-diffusion equation gives:

$$\oiint_{\boldsymbol{S}(\boldsymbol{V})} \boldsymbol{h}.d\boldsymbol{s} = 0$$

Where $\boldsymbol{S}(\boldsymbol{V})$ is the boundary of an arbitrary space-time region in E_2 , $d\boldsymbol{s}$ is the normal area of a surface element on $\boldsymbol{S}(\boldsymbol{V})$ and $\boldsymbol{h}.d\boldsymbol{s}$ is the space-time flux of $\boldsymbol{h}$ leaving the region V through the surface element ds.

It is therefore possible to build in E_2 an elemental volume where space and time are conserved locally and treated in a unified way. This will be the tenet for the construction of CE elements.

For the moment, let us build a solution element as in Figure 1c) that represents the interior of the space time region center on a given point of coordinates (x_q, t_q) in E_2 .

The variations within that SE will be considered small enough so that the solution can be expressed by the following Taylor series expansion:

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

The time and spatial derivatives can be related by using the flow convection-diffusion convection :

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

Consequently, two unknowns, $u_q(x,t)$ and its spatial derivative $\frac{\partial u_q}{\partial x}$ are left to be able to compute the solution anywhere within the space time region centered around (x_q, t_q).

In order to provide the two equations mandatory to close the system, Figure 1d) shows the two CEs that will be defined. The integral form of the conservation is then applied on those two CEs resulting in two equations which allow to solve the system and advance through time, resulting in the previously described nontraditional CESE features.

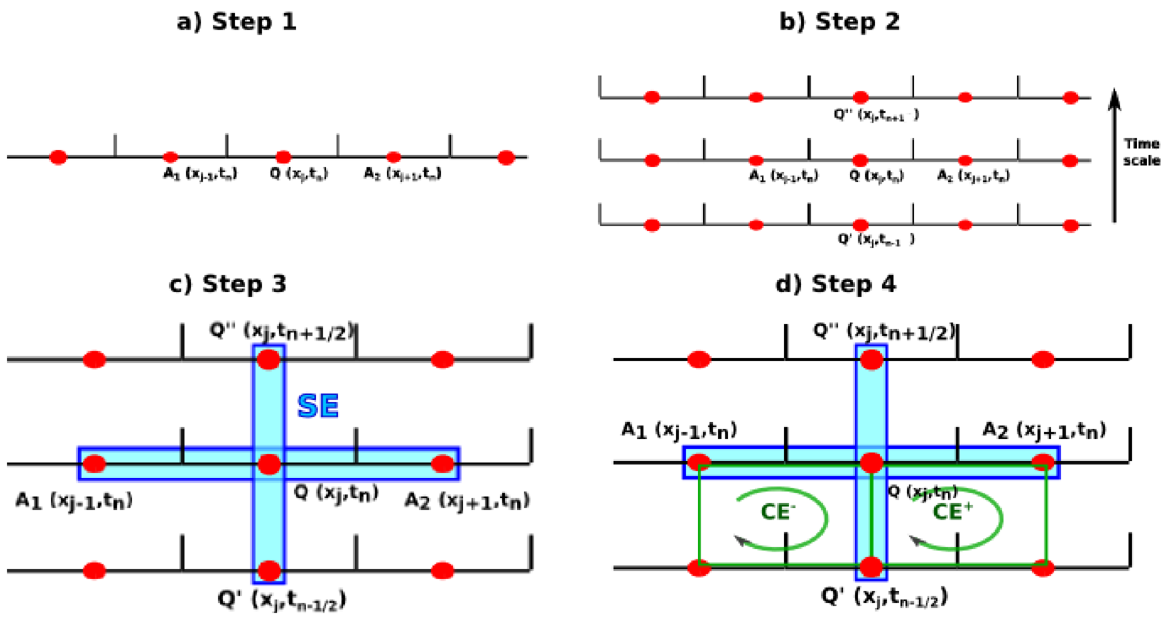

Figure 1 CESE method : 1D resolution steps

2.2 Stabilization methods

The previously described scheme is stable for inviscid flows with no discontinuities. However, for viscous flows and in order to solve shock wave flows, it is necessary to introduce some numerical diffusion for stabilization. Instead of using the two CE- and CE+ in order to solve the system for $u_q(x,t)$ (or simply written u) and its spatial derivative $\frac{\partial u_q}{\partial x}$ (or u_x), the spatial derivative will be estimated by a weighing technique using u_x^- and u_x^+ determined by the previous timestep solution (See SE- and SE+ in Figure 2). Compared to the exact resolution of u_x a diffusive and thus stable solution is obtained. Only one unknown is left therefore only one CE is needed to solve u. Figure 2 sums up the modified stabilized scheme.

The user can choose between a combination of the central difference and another weighted expression in order to express u_x function of u_x^- and u_x^+ or a simple relaxing procedure. More detail can be found in the short theory manual available on the LSTC website.

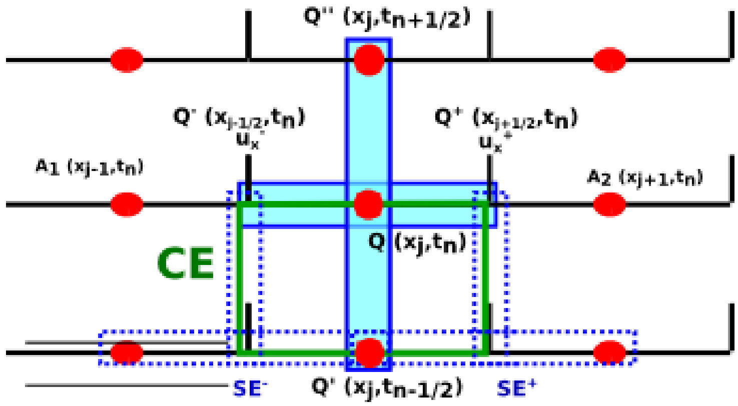

Figure 2 Weighting technique for stable calculation

2.3 Boundary conditions

Several boundary conditions are available to the user. It is possible to impose pressure, density temperature and velocity or to define non-reflective boundaries, reflective boundaries or solid walls. At the domain boundaries, the solver will "extend" the mesh domain by one layer and use the conditions defined by the user as input in this new element layer. This will then be used by the neighboring elements for solving. Figure 3 shows an example for different boundary conditions. Non reflective boundary conditions are used in order to define far field boundary conditions. For the solid wall and reflective boundaries, the normal velocity component is defined in opposite direction to the incoming velocity such as to be exactly zero at the interface (free slip condition). On top of that, for the solid wall condition, the tangent component is defined in the opposite direction such as to be null at the interface (non-slip condition). For inviscid flows, the solid wall boundary condition acts similarly to the reflective boundary condition.

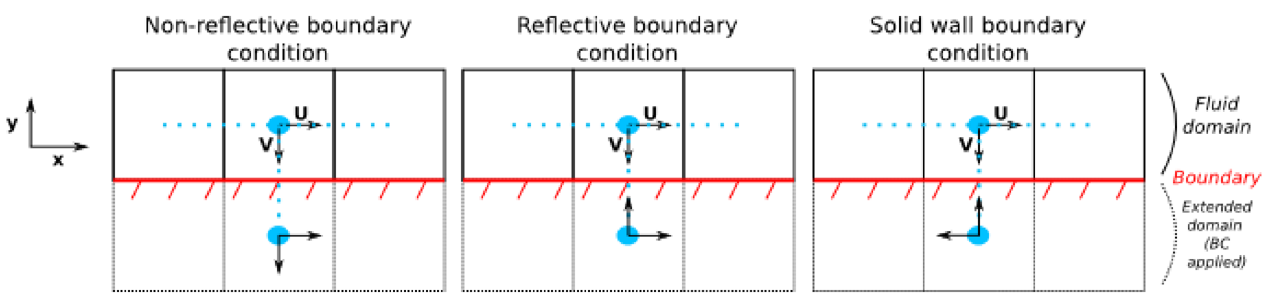

Figure 3 Boundary condition implementation

2.4 Validation problems

Several validation cases for the CESE numerical scheme can be found on the LSTC website. Figure 4, Figure 5 and Figure 6 show some results obtained for a shock wave diffraction around a corner, an incoming supersonic flow around a step (M>3) and the shock wave diffraction patterns forming behind a supersonic wedge (M>1.3).

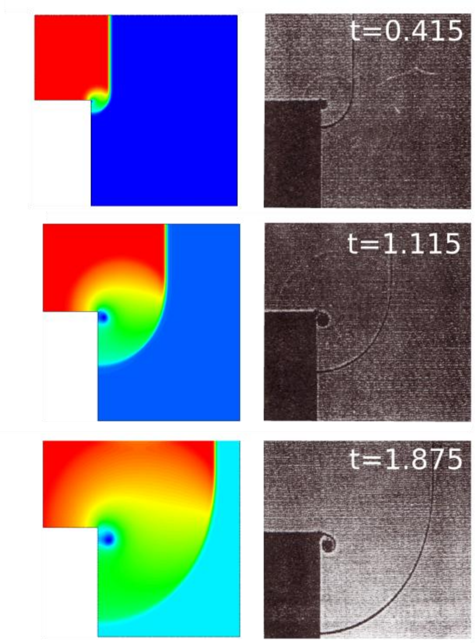

Figure 4 Diffraction of a shock wave on a sharp corner. Comparison with experimental results [4].

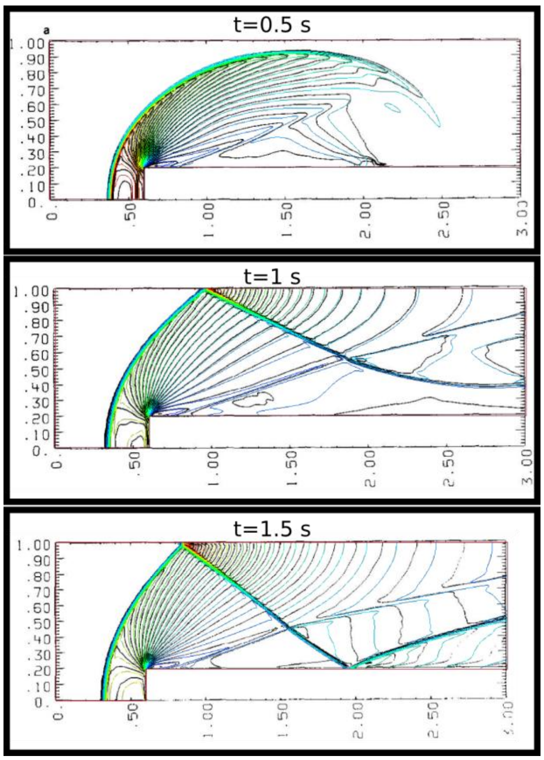

Figure 5 Incoming supersonic flow against a step. Superposition of shock wave patterns with results by [5]

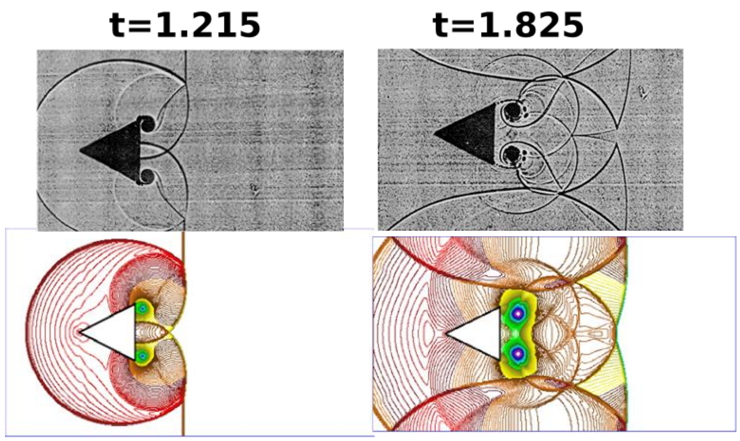

Figure 6 Density isocontours forming around a supersonic wedge. Comparison with experimental pictures.

3- Fluid Structure Coupling (FSI)

3.1 Resolution method

The CESE solver can be coupled with the solid mechanics LS-DYNA solver in order to solve fluid structure interaction problems. Since both solvers admit their own CFL condition on the timestep, the most constraining one from both domains will be used in FSI problems. This way, the fluid and structural solvers advance simultaneously in time. At each timestep, the fluid solver will communicate pressure forces on the structure that will act as exterior loads while the structure will give back its displacements and updated nodal velocities (See Figure 7).

The structure is immersed in the fluid domain. Therefore both meshes are independent and the interface will be automatically tracked by the solver. Figure 8 features a 2D example with a structural beam moving through a 2D fluid mesh with a velocity V. After the structural solver has communicated the nodal positions of the solid at time t_n, the first step would be for the CESE solver to track which fluid elements are closest to the structure and perform a sorting procedure (shown in light green in Figure 8). Then, in order to calculate the solution of those elements, the neighbors that are “blocked” by the solid will be treated as solid wall boundary conditions. For example, in Figure 8, the fluid element S1 sees two solid wall neighbors while S2 only sees one. Finally, a searching procedure based on the fluid mesh size is used in order to determine which fluid elements are close to the solid element and an average of the pressure values will then be computed and transferred to the structural solver acting as an exterior load (in light pink in Figure 8). The solver will automatically know on which side of the solid face those fluid elements are located by using the solid element normal. This way, no leakage can occur. In order for this searching procedure to be able to correctly capture neighboring fluid elements, it is advised to use a finer mesh for the fluid than for the solid.

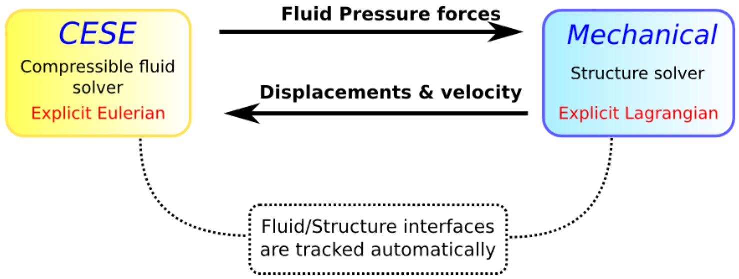

Figure 7 FSI resolution

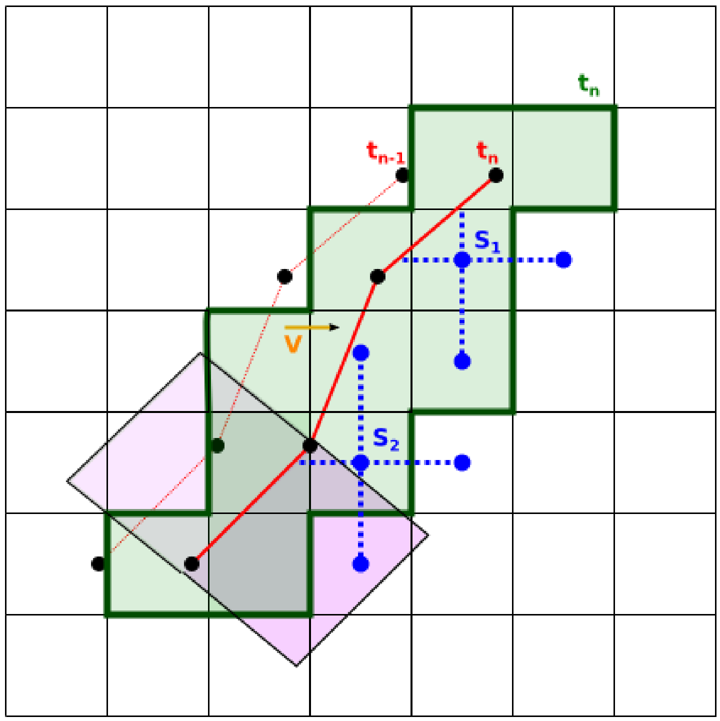

Figure 8 Solid-fluid Interface tracking

3.2 Validation problem

In order to validate the FSI algorithm, the piston problem described by [6] will be considered. This case features a gas contained in a 1D chamber closed on its right hand side by a moving piston and on its left by a fixed wall (spring-back system). For the purpose of this test case, the 1D problem will be moved to the equivalent 3D problem. The piston is of mass m_p, rigidity k_p, unstreched length L_{so}, at rest under pressure length L_{se} and the piston displacement is $u(t)$. The piston is initially loaded so that at initial time it will compress the gas chamber. The reflective pressure wave will then push back the piston thus triggering the spring-back FSI problem. The objective of this test case is to study the piston's response and interaction with the fluid by looking at its displacements function of time.

The gas chamber is at initial pressure and density P_g and ρ_g. Since the structure needs to be fully immersed in the fluid, the fluid properties outside the chamber need also to be considered. Atmospheric conditions will be used here.

The parameters chosen will be taken from [6] and the case of $m_p = 10kg$ will be studied. The mesh size will also be chosen in order to match the reference simulation by [6] i.e 0.1 m in the X-direction for the reference length $L_{so} = 1m$ (See Figure 10).

Figure 11a) shows the different pressure isocontours at a given time t during the gas compression. Figure 11b) shows the oscillation response of the piston function of time. The results are in good agreement with [6] regarding oscillation frequency and amplitude. It is to be noted that the present simulation offers slightly more damping effects. This is due to the fact that the reference simulation did not consider the gas outside the chamber and its interaction with the piston.

The CESE solver also includes moving mesh capabilities which can be applied to this piston problem. While this feature will be detailed during the presentation, it will not be further described in this paper.

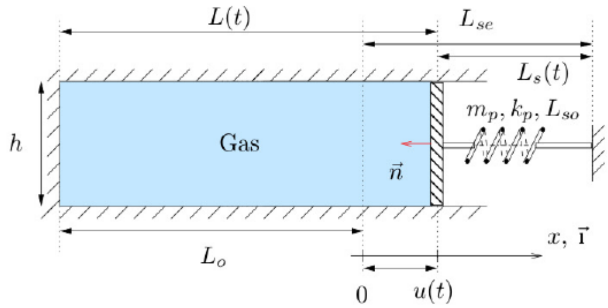

Figure 9 Piston case sketch

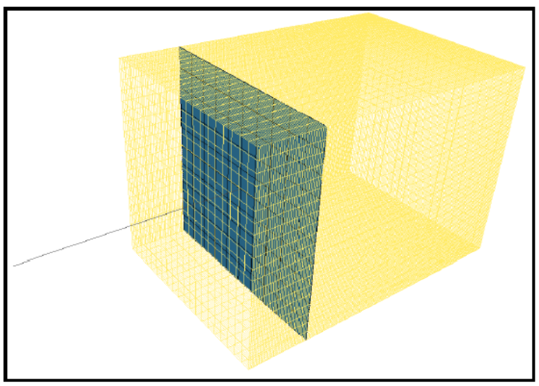

Figure 10 Test case mesh

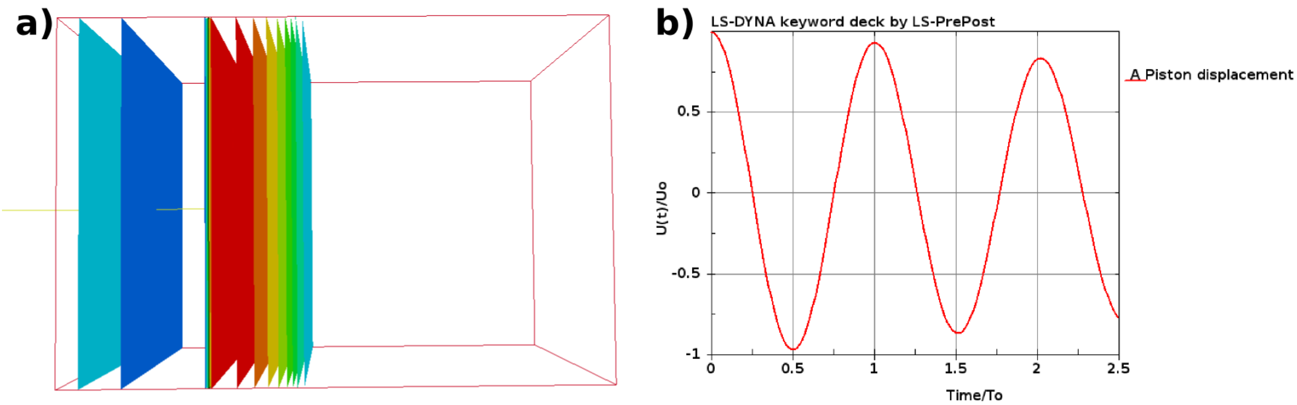

Figure 11 a) Pressure isosurface during piston compression. b) Piston displacements function of time

3.3 Further industrial applications

A few examples involving the CESE scheme and FSI include supersonic inflows or strong pressure waves causing structure deformation of displacement (See Figure 12), Airbag openings (See Figure 13) and transonic flows around turbomachines (See Figure 14).

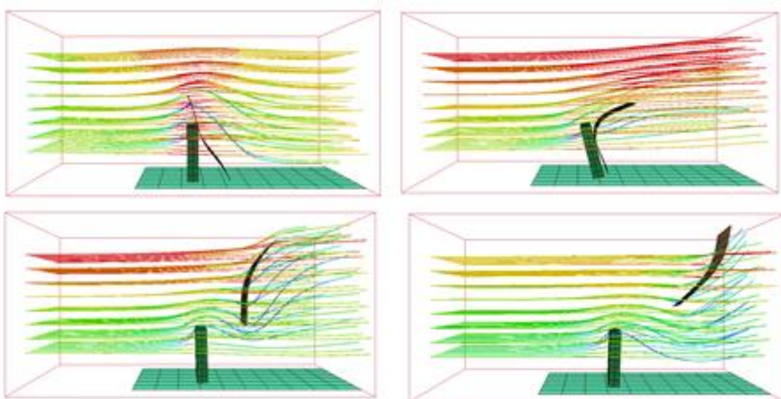

Figure 12 FSI case of a shell being blown away by supersonic flow

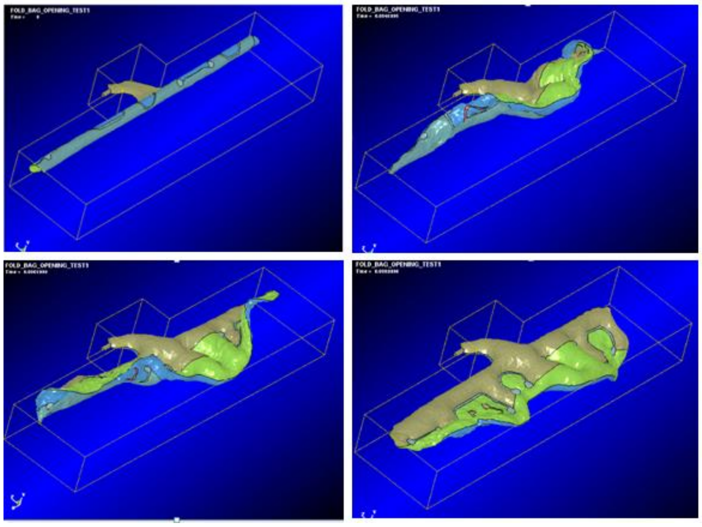

Figure 13 Airbag opening. Courtesy of TAKATA Corporation

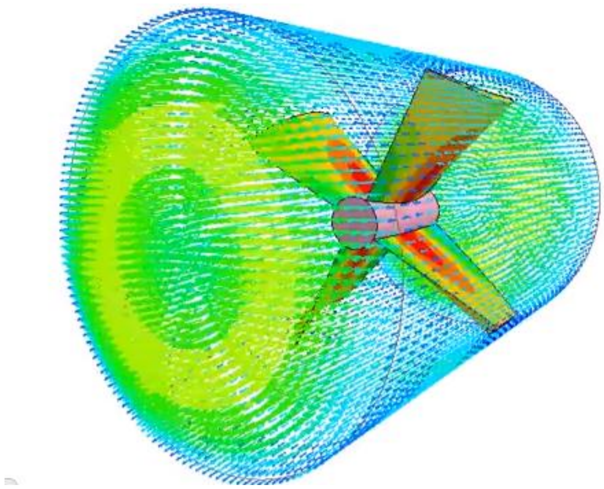

Figure 14 FSI case involving transonic flow around turbomachine

References

[1] S. Chang, «The Method of Space Time Conservation Element and Solution Element-A new approach for solving the Navier Stokes and Euler equations,» *Journal of Computational Physics,* vol. 119, p. 195, 1995.

[2] Z. Zhang, S. Chang et S. Yu, «A Space time Conservation Element and Solution Element for Solving the two and Three Dimensional Unsteady Euler equations using Quadrilateral and Haxagonal Meshes,» *Journal of Computational Physics,* vol. 175, pp. 168-199, 2001.

[3] S. Chang, «Courant Number insensitive CESE Schemes,» *AIAA paper,* 2002.

[4] T. O.I.K, «Shock wave diffraction around a 90 degree sharp corner,» chez *18th ISSW*, 1991.

[5] W. P et C. P, «The numerical simulation of two dimensional fluid flow with strong shocks,» *Journal of Computational Physics,* vol. 54, n° %1115-173, 1984.

[6] E. Lefrançois et J.-P. Boufflet, «An Introduction to Fluid Structure Interaction : Application to the piston problem,» *Society for industrial and applied mechanics,* vol. 52, n° %14, pp. 747-767, 2010.

[7] I. Çaldichoury et P. L'Eplattenier, «Update On The Electromagnetism Module In LS-DYNA,» chez *12th LS-DYNA Users Conference* , Detroit, 2012.