Using the New Compressible Fluid Solver in LS-DYNA®
— CESE Solver and the Input File Setup

Zeng-Chan Zhang
Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, CA 94551

Abstract

In this new compressible fluid solver, the conservation element and solution element (CESE) method[1, 2] is used. The CESE method is a novel numerical method for solving conservation laws and it has many nontraditional features, such as:

- *Space and time conservation — Flux conservation can be maintained very well both locally and globally in space & time.*
- *Accurate — It is 2nd order for both flow variables and their spatial derivatives. Thus, it is more accurate than other 2nd order schemes.*
- *Novel & simple shock-capturing strategy — only a simple weighted averaging technique is used, no Riemann solver and no special limiters are needed to capture shocks.*
- *Both strong shocks and small disturbances can be handled very well simultaneously.*

Because of these advantages, this CESE solver is a good choice for the following problem simulations:

- *Compressible flow problems, especially for high speed flows with complex shocks.*
- *Acoustic (noise) problems (near field).*

This solver is also combined with the LS-DYNA® structure solver to solve the fluid structure interaction (FSI) problems. There, the fluid solver is based on the Eulerian frame while the structure solver is the Lagrangian one. These meshes are independent of each other, and the interfaces will be tracked by the fluid solver automatically. A quasi-constraint method is used in the interface treatment, i.e., the fluid solver get the displacements and velocity of the interfaces from the structure solver and feeds back the fluid pressures (forces) on the interfaces.

Currently, both serial & MPP models are available for this compressible fluid & FSI solver (in LS-DYNA® 980 β-version). The fluid mesh can be made up of hexahedra, wedges, tetrahedra, or a mixture of these elements, while the structural mesh can be made up of shells (thin) or solid volume elements.

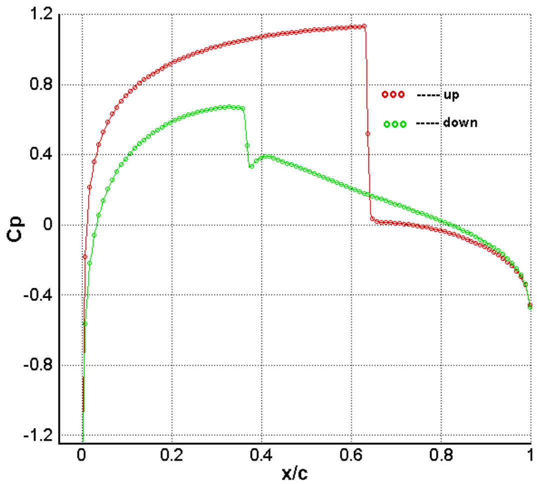
In this talk, a brief review of this new compressible fluid solver will be given first. Then, an introduction of how to use this new solver will be emphasized, including:

- *Computational domain & mesh determinations, especially for FSI problems*
- *Input deck (keywords cards) setup*
  - *— some general control parameters for the method*
  - *— initial flow field setup*
  - *— boundary condition (BC) choice at each boundary*
- *The use of LS-PrePost® for this new solver's output*

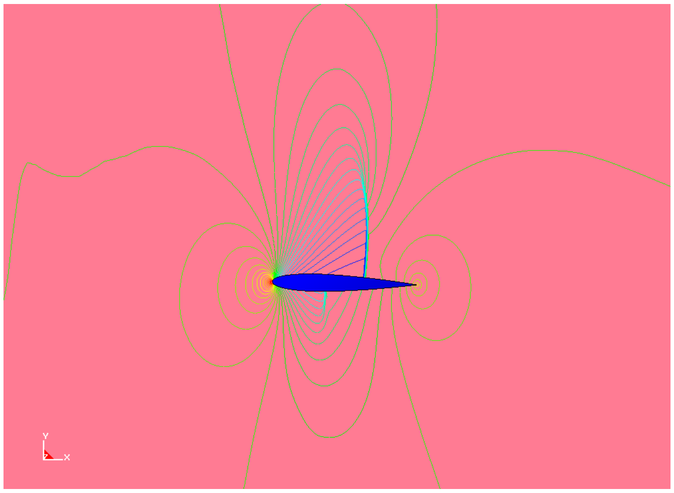
In addition, some new features will be introduced, followed by some remarks. The limitations of this solver will be pointed out too. Finally, some features under development will be mentioned.

References

1. CHANG S.C. (1995). "The Method of Space-Time Conservation Element and Solution Element – A New Approach for Solving the Navier Stokes and Euler Equations." J. Comput. Phys., Vol. 119, p.295.
2. ZHANG Z.C., CHANG S.C. and YU S.T. (2001). "A Space-Time Conservation Element and Solution Element for Solving the Two- and Three-Dimensional Unsteady Euler Equations Using Quadrilateral and Hexagonal Meshes," J. Comput. Phys., Vol. 175, pp.168-199.



(a)



(b)

Fig.1 Inviscid transonic flows around NACA0012 aerofoil:
(a) Pressure coefficients on the up and down faces (b) Pressure contours.

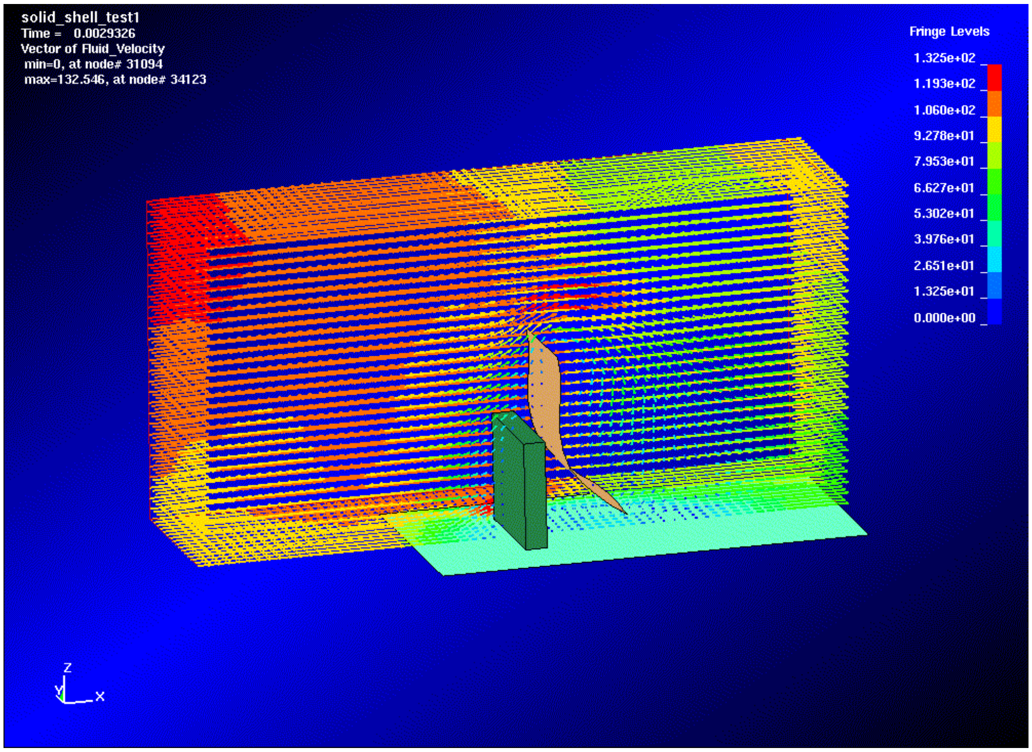


Fig.2 Fluid / shell + solid volume element interaction, velocity vectors