How to Use the New CESE Compressible Fluid Solver in LS-DYNA®

Zeng-Chan Zhang

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

Abstract

This new solver is based on the conservation element and solution element (CESE) method[1, 2]. The CESE method is a novel numerical method for solving conservation laws, and it has many nontraditional features, such as: space-time conservation; high accuracy (2nd order for both flow variables and their spatial derivatives); novel shock-capturing strategy; both strong shocks and small disturbances can be handled very well simultaneously, etc. Because of these advantages, this CESE solver is a good choice for high-speed compressible flows with complex shocks and acoustic (noise) problems (near field).

The solver has also been used to solve fluid/structure interaction (FSI) problems. For these problems, the fluid solver is based in an Eulerian frame while the structure solver is a Lagrangian frame. Their meshes are independent of each other, and the structural boundaries (fluid-structure interfaces) are tracked by the fluid solver automatically. The fluid solver gets the displacements and velocity of the interfaces from the structural solver and feeds back the fluid pressures (forces).

Current status
Currently, both serial & MPP solvers are available for this compressible fluid & FSI solver (in the ls980 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 for CESE FSI problems.

Input deck setup:
In this presentation, we will talk about how to use this new solver, this will include:

- *How to use LS-prepost to create a CESE input deck & display the final results*
- *How to setup an input deck, including*
  - *setting some control parameters in the CESE method*
  - *initial flow field setup*
  - *boundary condition (BC) choice at each boundary*
- *some example input decks*

In addition, a couple of new features will be introduced, followed by some remarks. The limitations of this solver will be pointed out too.

Some examples
With the new release of the ls980g beta version, we will provide ten examples (fluid & FSI). In each example, we will have problem description, input deck, numerical results and comparisons with analytical or experimental results (where available). Here we show two of them.

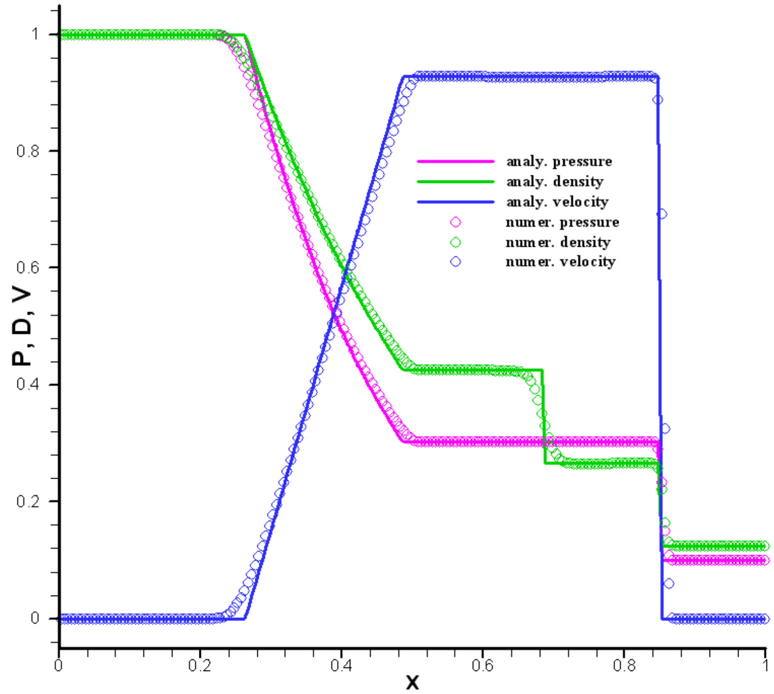
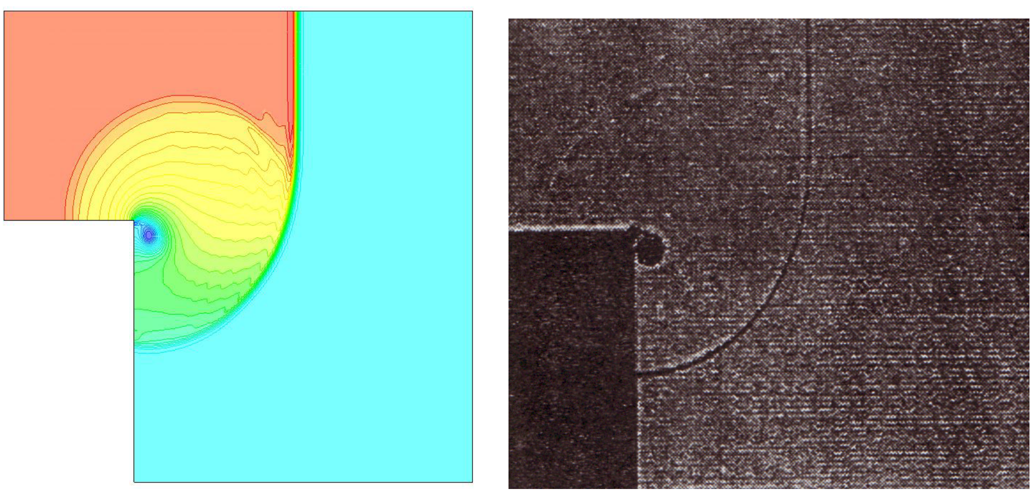


Fig.1 Sod's 1-D shock tube problem: the comparison of numerical results with the analytical solutions for pressure, density and velocity at t=0.2.



(a) *(b)*

Fig.1 Shock wave diffraction around a 90^o corner: (a) numerical results (density contours at t=1.115); (b) experimental results at the same time level.

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.