New Features of CE/SE Compressible Fluid Solver in LS-DYNA®

Zeng-Chan Zhang, Grant O. Cook, Jr. & Kyoung-Su Im

Livermore Software Technology Corporation (LSTC)
7374 Las Positas Road
Livermore, CA 94551

Brief review of the CESE solver

CESE compressible fluid solver is one of the new solvers in LS-DYNA R7.0. This solver is based on the space-time conservation element and solution element (CE/SE) method, originally proposed by Chang [1]. The CE/SE method has many non-traditional features, such as (i) both local and global flux conservation are well maintained in space and time; (ii) shock waves can be captured automatically without using Riemann solvers or special limiters, etc. For more details about the CE/SE method, see the references [1, 2, 3]. This method is suitable for high-speed flows, especially with complex shock waves. In the past, the CESE method has been widely used in many different CFD-related areas, e.g., shock/acoustic wave interaction, detonation waves, cavitation, chemically reacting flows, etc.

Based on the CE/SE method, we developed the CESE compressible fluid solver in LS-DYNA. And furthermore, by coupling with the structural FEM and heat transfer solvers, the CESE-FSI solver was developed to solve fluid/structure interaction (FSI) and conjugate heat transfer problems. Currently, the CESE-fluid and CESE-FSI solvers can be used for:

- 3D and 2D/2D-axisymmetric compressible fluid problems, especially high speed flows with shock waves

- 3D fluid/structure interaction problems. We provide two coupling strategies for the users to choose from, one is the immersed boundary method (FSI-IBM) and the other is the moving mesh method (FSI-MMM).

- 3D and 2D/2D-axisymmetric cavitating flows. Here Schmidt's homogeneous equilibrium model [4] is used. This model is suitable for high-speed flows in small geometries, like diesel engine injection systems.

- 3D and 2D chemically reacting flows both in premixed and diffusion combustions. The multi-component mixing problems are also available. Recently, we added airbag inflator models for a conventional pyrotechnic inflator and heated gas inflator.

New Features

Recently, we have added more features to this solver, including:

- 2D/2D-axisymmetric CESE-FSI solver to solve 2D/2D-axisymmetric fluid/structure interaction problems

- Conjugate heat transfer solver for 3D and 2D/2D-axisymmetric fluid/structure heat transfer problems

- 3D Fluid and FSI problems in a non-inertial frame

- For turbulence models, Chien's κ-ε two-equation RANS turbulence model, with more models to be added soon.

- Chemically reacting flows in both premixed and diffusion combustion, multi-component mixing problems without combustion, and airbag inflator models.

Some Remarks

Following are some observations regarding the CESE fluid solver:

1. As to the computational mesh, different kinds of mesh element shapes can be used in the CESE fluid & FSI solvers, i.e., tetrahedron, wedge, hexahedron or a mixed mesh of these elements for 3D; triangle and/or quadrilaterals for 2D/2D-axisymmetric meshes. But just like other CFD methods, the accuracy of the results will be better when using a hexahedron (3D) and quadrilateral (2D) mesh than a tetrahedron (3D) or triangle (2D) mesh.

2. There are two ways to provide the computational mesh. First, the standard input of volume elements via keyword input. Second, an automatically-generated volume mesh is created by a built-in mesher that works with user-input boundary surfaces meshes that must enclose the volume to be meshed.

3. The CESE-FSI solver provides two coupling strategies for users to choose from, i.e., the immersed boundary method (FSI-IBM) and the moving mesh method (FSI-MMM). The IBM method is simpler and more stable and suitable for FSI problems with large deformations, while the MMM approach is more accurate, especially at the FSI interface, and is good for FSI problems with small deformations. Nevertheless, the MMM method is more time consuming due to the overhead in computing the mesh motion.

4. Several things can affect the stability of the calculation, including (but not limited to):

    - Mesh quality, e.g., element shape, size, angle, aspect ratio, skewness, etc.; usually, the more uniform the mesh the better.

    - Boundary condition (BC) set up; for example, the whole problem setup needs to be well-posed in order to get a reasonable and physical solution. Also, some BC setup needs to be chosen carefully, such as non-reflecting BCs that must be placed as far away from the main flows as possible.

    - CESE limiter control parameter setup, especially for flows with strong shock waves where more numerical dissipation is needed.

Numerical Examples

1. Piston problem

Here we use both CESE FSI-IBM and FSI-MMM to calculate this problem. Fig.1 (a) and (b) display the meshes for IBM and MMM solvers respectively, and (c) and (d) show the corresponding piston displacement. We can see that the dissipation of the MMM treatment is a little less than that of the IBM, i.e., MMM is more accurate than IBM. Both results compare well with the reference [5], as for the total fluid mass conservation, our result is ΔM_f / M_f (0) ≤ 0.049%, better than paper [5]'s 0.08%.

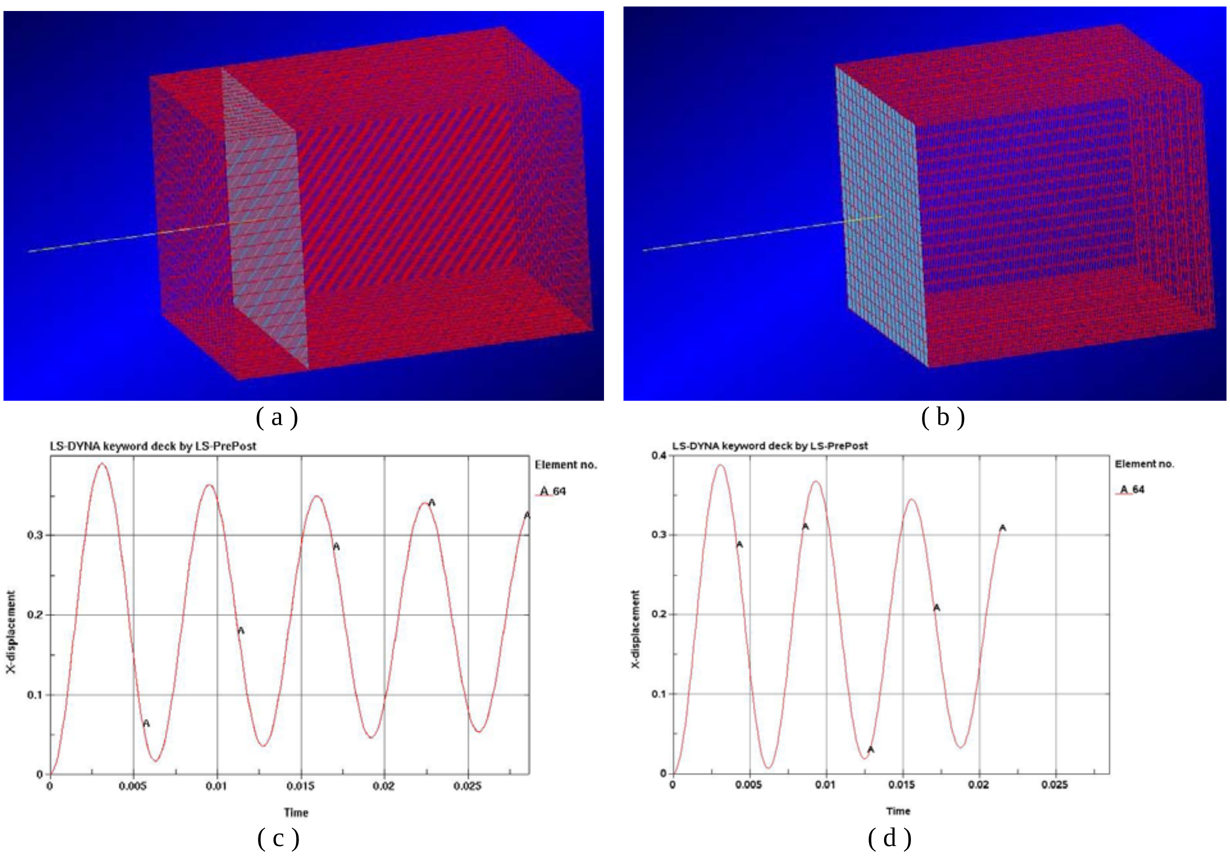

Figure 1. piston problem

2. Traffic information plate swaying in a strong wind

This simulation of a traffic information plate swaying in a strong wind uses the CESE-FSI solver with the IBM option. Figure 2 shows flow field and fluid/structure interaction at four different times.

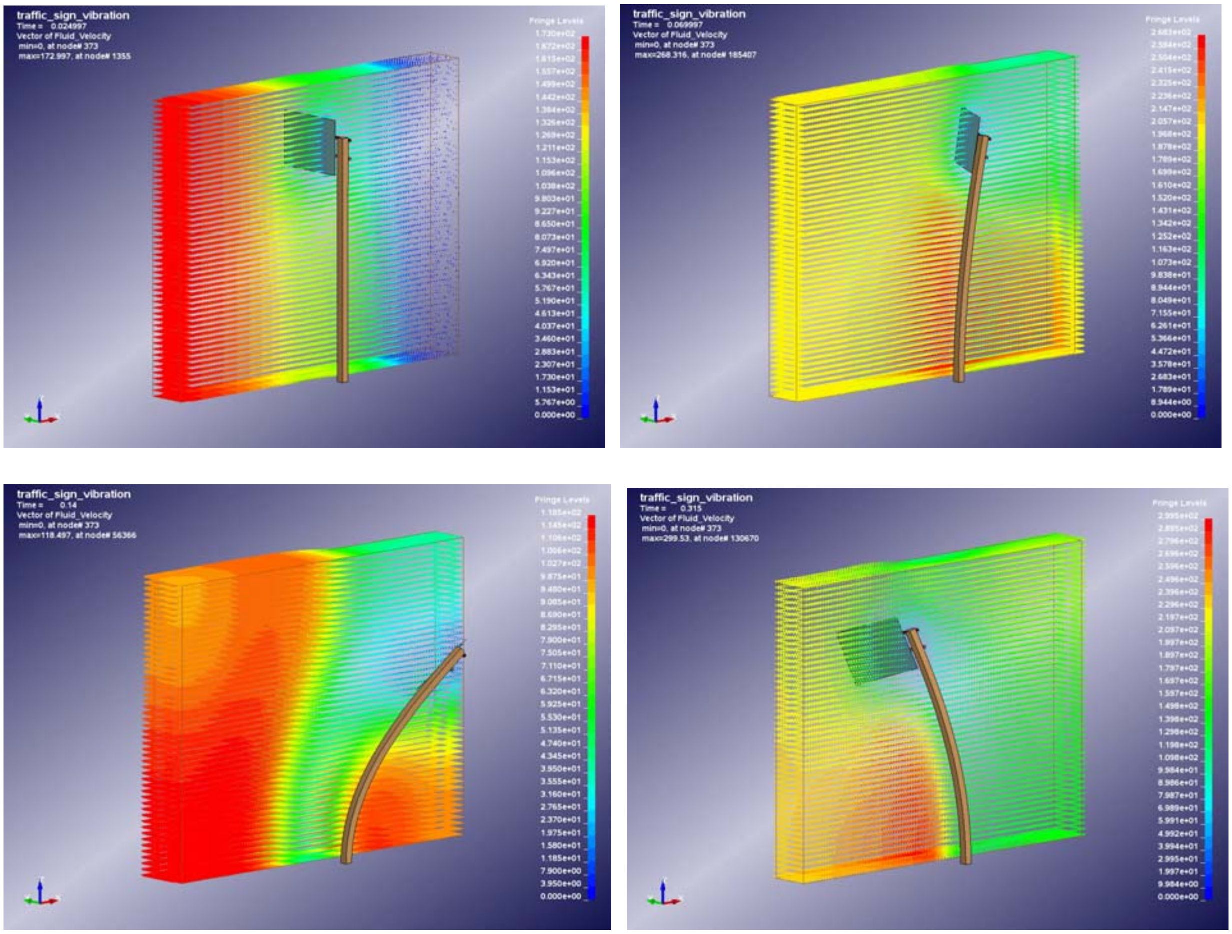

Figure 2. Traffic info plate swaying at four different times

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 Method for Solving the Two- and Three-Dimensional Unsteady Euler Equations Using Quadrilateral and Hexahedral Meshes," J. Comput. Phys., Vol. 175, pp.168-199.

3. Chang S.C., Chang C.Y. and Yen J.C. (2013). "Recent Developments in the CESE Method for the Solution of the Navier-Stokes Equations Using Unstructured Triangular or Tetrahedral Meshes with High Aspect Ratio." 21st AIAA Computational Fluid Dynamics Meeting, 24-27 June 2013, San Diego, California, USA.

4. Schmidt, D.P.,*Rutland, C. J., Corradini, M. L., Atomisation Sprays Techology* 9: 1999, 255-276

5. Emmanuel Lefrancois and Jean-Paul Boufflet, "An Introduction to Fluid-Structure Interaction: Application to the Piston Problem," SIAM Review, Vol. 52, No. 4, pp. 747–767