The New CE/SE Fluid Solver and Fluid/Structure Coupling

Zeng-Chan Zhang and Grant O. Cook, Jr.
Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, CA 94551

Abstract

In this paper, we will discuss a new compressible fluid solver (it will be included with the release of LS980) and its fluid/structure coupling strategy. This fluid solver is based upon the Space-Time Conservation Element and Solution Element Method (or the CE/SE method for short), while a quasi-constraint method is used in the fluid/structure couplings.

Introduction

The space-time conservation element and solution element (CE/SE) method, originally proposed by Chang, is a new numerical framework for solving conservation laws. This method is not an incremental improvement of a previously existing CFD method, and it differs from other well-established CFD methods. It has many nontraditional features. The design principles of the CE/SE method have been extensively illustrated in the references [1, 2, 4]. In the following, a brief review of the CE/SE method is provided.

CESE Method:

In the CE/SE scheme, the integral form of governing equations of fluid mechanics are solved, instead of differential ones. These equations are obtained by using the Gauss divergence theorem in the space-time region. Also, two important element concepts are introduced in the CE/SE scheme formulation, i.e., conservation element (CE) and solution element (SE). The CEs are non-overlapping space-time subdomains introduced such that the computational domain is the union of these subdomains and the flux conservation equations are enforced over each of them. On the other hand, each SE is a space-time subdomain over which any physical flux vector is approximated using simple smooth functions. Here the CE and SE can be any polyhedrons. By the enforcements of the fluid conservation equations over each CE, we obtain the discrete equations for solving the flow variables and its spatial derivatives. The details about the constitution of this scheme can be found in the cited references. Such schemes have many nontraditional features compared with the conventional CFD method. Followings are three of them:

- Both local and global flux conservation are enforced in space and time. This is critical for accurate flow simulations, particularly if they involve long marching times and/or regions of rapid change.
- Both the flow variables and their spatial derivatives are treated as the independent unknowns, and solved simultaneously. Thus, it is more accurate than other schemes with the same accurate order and the same mesh stencils.
- No Riemann solvers and no special limiters are needed to capture shocks, so its computational logic is considerably simpler and the calculations are very efficient.

Currently, this method is being used by more and more people in different fields. To date, numerous highly accurate CE/SE steady and unsteady solutions have been obtained, including: high speed flows, especially with complicated shocks patterns; chemical reaction flows (combustions, detonation waves, etc.); and cavitating flows. Because of its accuracy and space-time conservation features, this method can resolve both strong shocks and small disturbances (e.g., acoustic waves) simultaneously, and this rather unique capability has been verified through several accurate predictions of experimental data. Note that, while numerical dissipation is required for shock resolution, it may also result in annihilation of small disturbances. Thus a solver that can handle both strong shocks and small disturbances simultaneously must be able to overcome this difficulty.

Fluid/Structure Coupling:

In our fluid/structure coupling, a quasi-constraint method is used. For fluid solver, the above CE/SE method is used based on an Eulerian mesh, while FEM is used in the structure solver that is based on a Lagrangian frame. The interface boundary locations and velocities are dictated by the Lagrangian structure. This boundary information is used by the fluid solver as the interface boundary conditions at each time step, and it will feed back the fluid pressures (forces) on each structure segment as exterior forces for the structure solver.

Current Status:

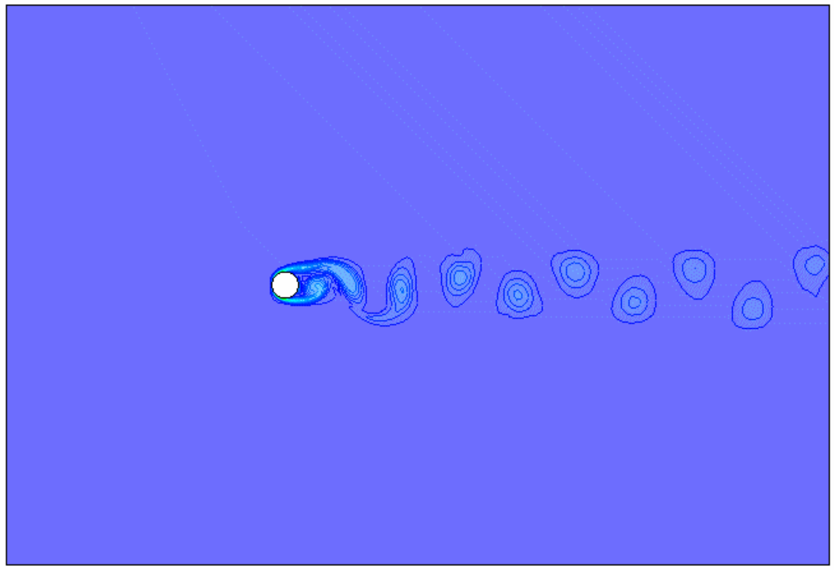
For the fluid solver, both serial and MPP modes are available for the inviscid and viscous flows. The supporting meshes can consist of tetrahedra, wedges, hexahedra or a mixture of these elements.

For the fluid/structure coupling, the shell, solid volume element or both coupling with the fluid all are done. Testing is underway.

Numerical Examples

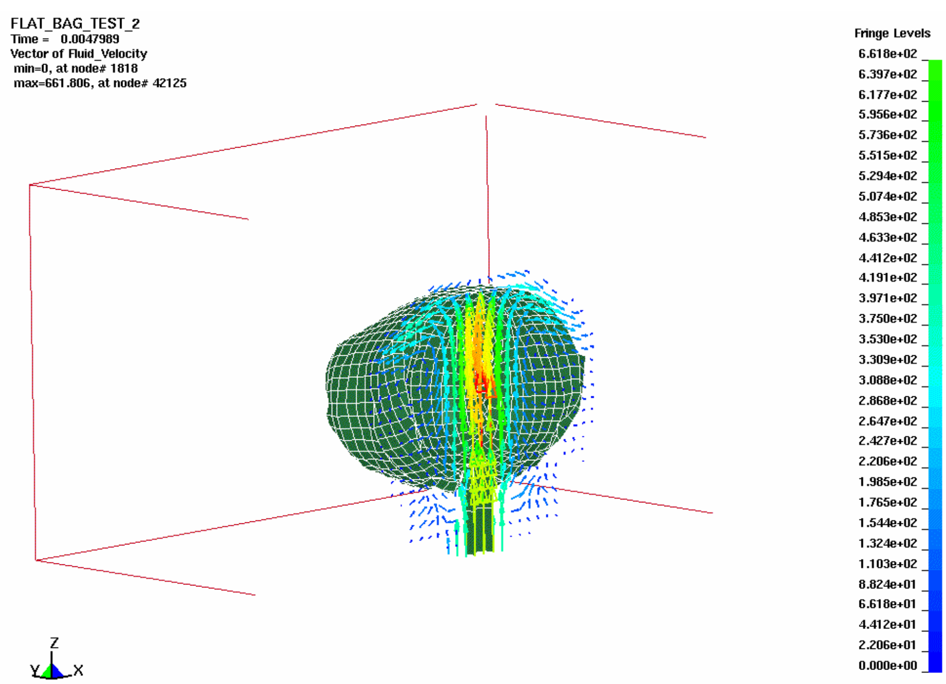
1. Flow Over a Circular Cylinder:

The CE/SE method is most used in high speed flows, but because of its conservation and accurate features, it can also be used in low speed flows even if a small time step should be used and huge number iterations (>100,000) must be run to get to steady state flow. This problem is usually treated as an incompressible flow because of its low speed. But this new fluid solver can also do this well. The figure shows the vortex street sheet in one snapshot after it reaches the steady state.



2. Flat Bag Opening:

This is to test the fluid/structure interactions. High pressure air enters the bag from the bottom hole to open a flat bag. The figure shows the velocity vectors in one snapshot during the bag opening process. The vortex flows are formed on both sides inside the bag.



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.

3. RONALD P. FEDKIW. (2002). “Coupling an Eulerian Fluid Calculation to a Lagrangian Solid Calculation with the Ghost Fluid Method,” J. Comput. Phys., Vol. 175, pp.200-224.

4. CHANG S.C. (1995). “Courant Number Insensitive CE/SE Schemes,” AIAA paper 2002-3890.