Overview of the CESE Compressible Fluid and FSI Solvers

Zeng-chan Zhang, Grant Cook, Jr. & Kyoung-su Im

Livermore Software Technology, an ANSYS Company, Livermore, CA 94551, USA

Abstract

The Original CESE solver in LS-DYNA® is a compressible fluid solver. Over time, more and more capabilities and applications have been added, especially coupled with the LS-DYNA structural FEM solver to solve different fluid/structure interaction (FSI) problems. Among the many problem types suited to using the CESE solver are the following applications: flimsy vacuum sucking in tissue processing, airbag deployment, blast wave FSI, cavitation in fuel injection, etc. In this paper, there are three parts: (i) a brief review of the current CESE solver; (ii) introduction of our new dual CESE solver; (iii) our two different FSI solvers and their applications.

1. Current status of the CESE solver

The CESE solver is based on the space-time conservation element and solution element (CESE) method, originally proposed by Chang [1] for solving hyperbolic conservation laws. It has several nontraditional features, including (i) a unified treatment of space and time (thereby ensuring good conservation in both space and time); (ii) unified treatment for arbitrarily shaped polygonal mesh elements; (iii) a simple but efficient discontinuity (shock) treatment. This method is especially useful for high speed compressible flows with complex flows patterns including shock waves and/or detonations. Based on this method, we developed several different CESE solvers, such as:

- CESE fluid solver (2D & 3D), general purpose fluid solvers for inviscid and viscous flow calculations
- CESE FSI solvers, for fluid/structure interaction simulations
- CESE non-inertial solver -------- solving the fluid problem in a non-inertial system (currently only for constant velocity systems)
- CESE Chemical reaction flow solver
- CESE Stochastic particle solver

2. Dual-CESE solvers

In the CESE method, there are two important concepts, i.e., conservation element (CE) and the solution element (SE), that is where the CESE method name come from. In each CE, the flow conservation laws will be enforced, while in the SE, the flow variables will be approximated by some kinds of functions, such as the first order Taylor expressions. Another important definition is the solution point (denoted as SP), it is the place where flow variables are actually solved and stored. For each SP, there is a SE and a CE associated with it.

Fig1 shows portion of a typical two-dimensional mesh with mixed triangle and quadrilateral elements. In the standard CESE method, all the flow variables are solved and stored on the SP of the element center (but not necessarily consistent with it) (see Fig1(a)). For example, for the element $A_1A_2A_3A_0$, its CE is $A_1C_1A_2C_2A_3C_3A_0C_4$ (here we only show it in space, if the current calculation time is at t^n, then the CE should be extended one time step back to t^{n-1}), while its SE can be defined as the element itself extended from $t= t^{n-1}$ to $t= t^{n+1}$, plus the middle plane of $A_1C_1A_2C_2A_3C_3A_0C_4$ at $t= t^n$. In most cases, the SP will not consistent with the element center, for example the element $A_1A_2A_3A_0$, its element center is C_0 , while its SP is S_0 (see Fig.1 (a)).

Recently, we have recoded all of the solvers mentioned above in the new dual-mesh CESE framework in order to increase their accuracy and stability. In this new solver, the flow variables are solved and stored on two different sets of SPs in two successive time steps. One set is associated with the element centers (red squares in Fig.1(b)), the other set is associated with the element vertices (see Fig.1(b)). Here the solution point of the element center will consistent with the element center itself, while the solution point of the element vertices not necessarily consistent with it (e.g., Q_0 is a solution point associated with A_0, see Fig.1(b)). In the dual-mesh CESE, the CE and SE can be defined as the same, for example, for element center C_0, we can define the CE and SE as $A_1A_2A_3A_0$, similar for element vertices, we can define its CE and SE as $C_0m_1C_3m_2C_5m_3C_6m_4C_4m_5$ (of course here we only show it in space, it must be extended in time direction). With this approach, the new CESE solver becomes more accurate than the standard CESE solver if the same mesh size is used. Especially for a two-dimensional triangle or a three-dimensional tetrahedron mesh, the new dual mesh CESE solver will be more stable (and more accurate too), partially because of the number of integration faces increases for each vertex's conservation element.

Fig.2 and Fig.3 shows the comparison results between the standard CESE and the new dual CESE method. The problem is a typical test case [3], i.e., two-dimensional oblique shock reflection problem. In the first case, we use the same number of triangle mesh (very coarse mesh, total 800 triangle elements), Fig.1 shows the mesh (a), pressure contours (c, d) and the pressure distributions at the central line ($y \approx 0.5354$), an analytical solution is also included. We can see the dual mesh CESE solver's results is better than the standard one's. Of course, here a very coarse triangle mesh is used, if the mesh is refined, the difference between these two solvers will also become smaller. Next, we test the coarse quadrilateral element mesh (total 40x10=400 quads) for dual CESE, while for the standard CESE the mesh is almost doubled (i.e. 57x14=798 quads), Fig.3 shows their results (pressure contours and pressure distributions near the middle line). We can see the dual CESE results is almost the same with the standard ones even if only half of mesh number is used.

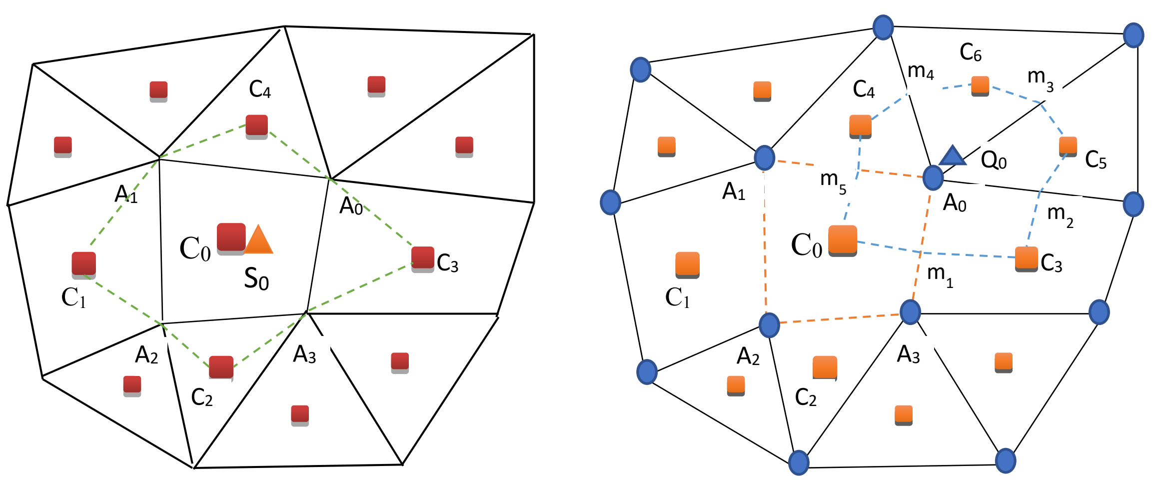
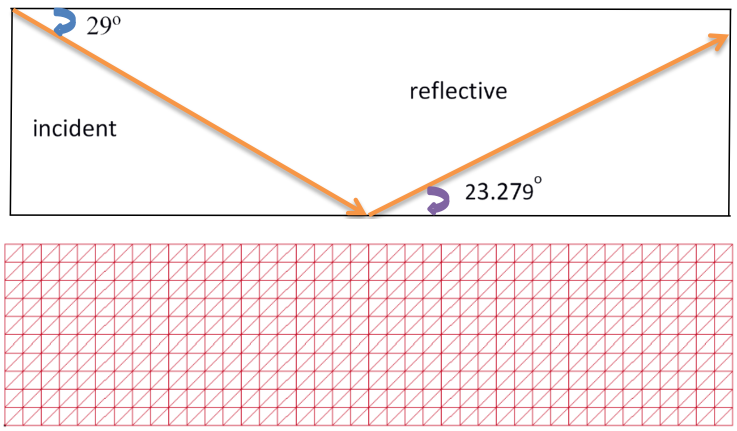
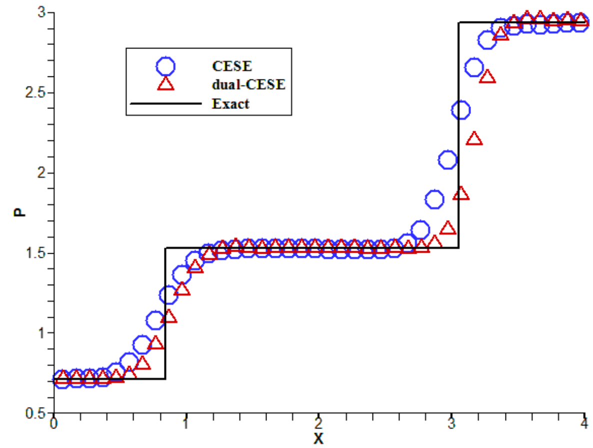


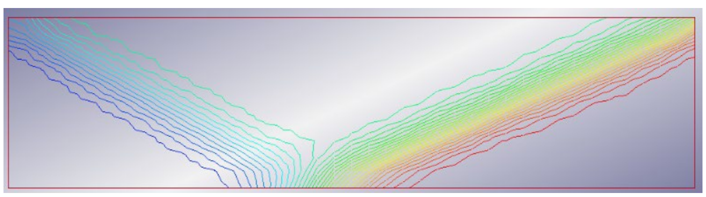
Fig.1 schematic of conservation element (CE) and solution element (SE) under (a) standard CESE and (b) dual CESE solver



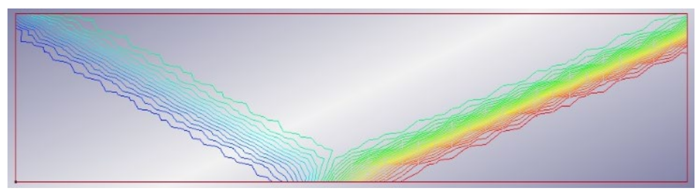
(a) Mesh



(b) pressure distribution at line y=0.5354

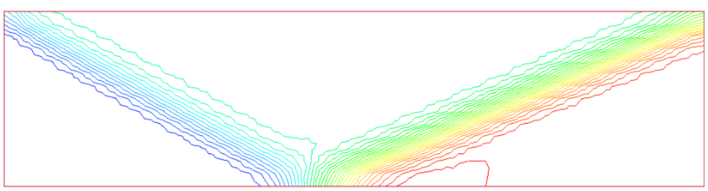


(c) CESE pressure contours

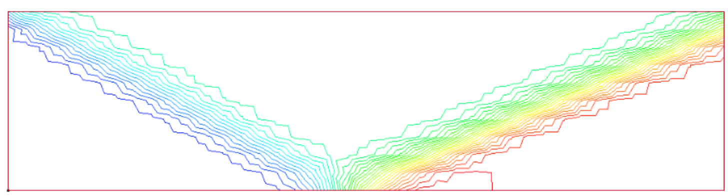


(d) dual CESE pressure contours

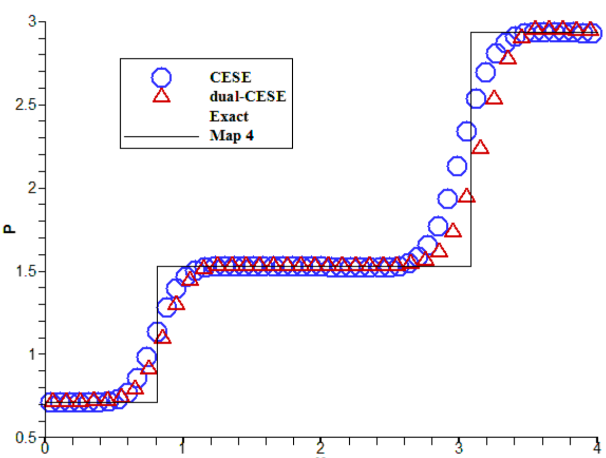
Fig.2 Comparison between the standard CESE and dual CESE solver with same mesh



(a) CESE pressure contours



(b) dual CESE pressure contours



(c) pressure distribution near the middle line

Fig.3 Comparison between the standard CESE and dual CESE solver with different mesh number

3. CESE FSI and Dual-CESE FSI solver

Coupling CESE with the LS-DYNA structural FEM solver, we have developed the following two fluid/structure interaction FSI solvers based on two different strategies, i.e.,

- FSI-ibm solver: It is based on the immersed boundary method (IBM). In this treatment, the fluid mesh is fixed, and the structure can move freely inside the fluid domain (fluid mesh and structural mesh are independent of each other). The fluid solver gets the interface displacements and velocities from the structural solver, and then feeds back the forces and/or heat flux. This solver is very stable and good for large deformation FSI problems.
- FSI-mmm solver: In this treatment, the fluid mesh is adjusted to follow the structure's motion in order to match the fluid-structure interface at every time step. The positions of the interior mesh nodes also need to be recalculated every time step or every several time steps. Because of this, it is more expensive than the FSI-ibm solver. But in our new dual CESE with multi parts capability, the FSI-mmm solver's moving portions mesh can be limited to small regions near the fluid-structure interfaces, this will greatly alleviate the cost of the mesh motion calculation. This solver is more accurate than the FSI-ibm solver, and it is especially useful for small deformation FSI problems.

Fig. 4 shows accelerating bullet flying through a stationary air, an off-body bow shock wave is formed in frond and a high pressure area is created near its head. For more CESE examples, please go to https://www.dynaexamples.com/cese.

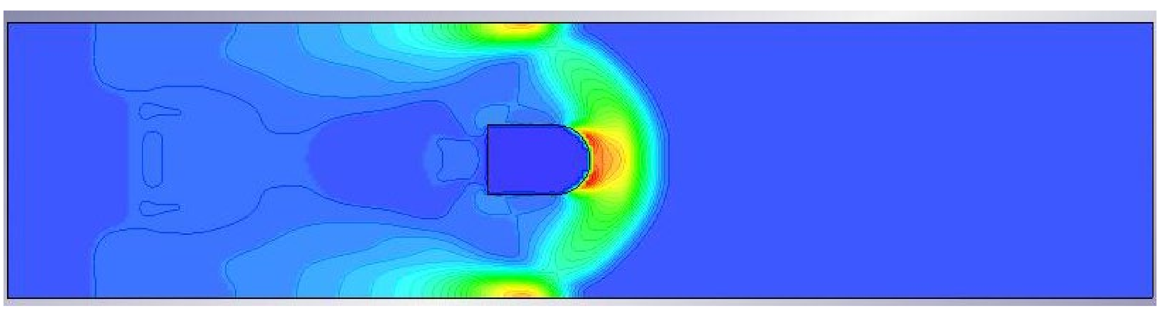


Fig. 4 FSI simulation results (pressure) of bullet flying through a channel

References

[1] S.C. Chang, "The space-time conservation element and solution element – a new approach for solving the Navier-Stokes and Euler equations," J. Comp. Phys. 119 (1995) 295-324

[2] Z.-C. Zhang, S.T.J., Yu and S.-C. Chang, "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. Comp. Phys. 175 (1) (2002) 168-199

[3] H.C. Yee, R.F. Warming and A. Harten, "Implicit Total Variation Diminishing (TVD) Scheme for Steady-State Calculations," AIAA paper 83-1902 (1983).