New LS-DYNA Fluids Solvers

Grant O. Cook, Jr..
Zeng-Chan Zhang

Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, CA 94550

ABSTRACT

This paper discusses two new fluid solvers that will be included with releases of LS-DYNA after LS960. The first one is a new compressible solver based upon the Space-Time Conservation Element and Solution Element Method (or the CE/SE method for short), and the second one is an incompressible fluid FEM solver.

INTRODUCTION

The CE/SE method, originally proposed by Chang from the NASA Glenn Research Center, is a novel numerical framework for conservation laws. It has many non-traditional features, including a unified treatment of space and time, the introduction of conservation element (CE) and solution element (SE), and a novel shock capturing strategy without a Riemann solver. To date, this method has been used in solving a lot of different flow problems, such as detonation waves, shock/acoustic wave interactions, cavitating flows, and chemical reaction flows. This method is being added to LS-DYNA to solve these same kinds of problems, as well as flow/structure interaction problems.

The initial release of a new incompressible flow solver is already included in the LS960 release. However, this version of the incompressible solver does not support coupling of the incompressible fluid with the solids modeling part of LS-DYNA, so for example, no heat conduction in solids coupled to an incompressible fluid is possible in LS960. Of course, coupling between fluids and solids is available in LS960 through the compressible ALE fluids solver (a separate compressible fluid solver). Coupling of the incompressible flow solver to heat conduction in the solids will be supported in the next release of this solver.

Features of future releases are discussed at the ends of next two sections.

NEW COMPRESSIBLE FLOW SOLVER

Space-Time CE/SE Method

The space-time conservation element and solution element (CE/SE) method, original proposed by Chang, is a new numerical framework for solving conservation laws. The CE/SE method is not an incremental improvement of previously existing CFD method, and it differs from other well-established methods. The CE/SE method has many nontraditional features. The design principles of the CE/SE method have been extensively illustrated in the references [1-3, 8-11]. In the following, a brief review of the CE/SE method is provided.

For the convenience of description, consider the following three-dimensional compressible Navier-Stokes equations:

$$\frac{\partial u_m}{\partial t} + \frac{\partial f_m}{\partial x} + \frac{\partial g_m}{\partial y} + \frac{\partial q_m}{\partial z} = 0, \qquad m=1, 2, 3, 4, 5 \tag{1}$$

Let $x_1=x$, $x_2=y$, $x_3=z$ and $x_4=t$ be the coordinates of a four-dimensional Euclidean space E_4. Using the Gauss divergence theorem, we can get the following integral counterpart:

$$\oint_{S(V)} h_m \cdot ds = 0, \quad m=1, 2, 3, 4, 5 \tag{2}$$

Where S(V) is the boundary of an arbitrary space-time region in E_4 and ds=dσ **n** with dσ and **n**, respectively, being the area and the unit outward normal of a surface element on S(V). The CE/SE method is based on the above integral equation (2).

In the CE/SE method, there are two important definitions, i.e., conservation element (CE) and solution element (SE). The CEs are such non-overlapping space-time subdomains introduced such that the computational domain is the union of these subdomains and the flux conservation equations (2) are enforced over each of them. While 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 equation (2) over each CE, we can get 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.

The scheme obtained using the above method has many nontraditional features compared with the conventional CFD method. Followings are some of them:

- Space and time be unified and treated as a single entity. And both local and global flux conservation in space and time be enforced. This is critical for accurate flow simulations, particularly if they involve long marching times and/or regions of rapid change.
- Multidimensional scheme can be constructed without using the dimensional-splitting approach, such that multidimensional effects and source terms can be modeled more realistically.
- 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.
- The more general form of the integral conservation laws is cast in a form in which space and time are treated on an equal footing. This gives flexibility in the shapes of CE and SE, and this make it suitable for the calculations of moving mesh and/or boundary problems and fluid/structure interaction problems.
- Both the flow variables and their spatial derivatives are treated as the independent unknowns, and solved simultaneously. Therefore, there is no need for reconstruction of the flow gradient, and the solution will be more accurate than other schemes with the same mesh stencils.
- Easy implementation of non-reflective boundary conditions, and unified solid boundary treatment for inviscid and viscous flows.
- It 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.

To date, numerous highly accurate CE/SE steady and unsteady solutions have been obtained, including (but not limited to):

— High speed flows (to Mach number 10), especially with complicated shocks patterns.
— Aero-acoustics (acoustic waves/vortices/shocks interactions)
— Chemical reaction flows (combustions, detonation waves).
— Cavitating flows.

Numerical Examples
Example 1: Shock/Vortex Interactions.
In this problem, a steady oblique shock along the diagonal of the computational domain is pre-calculated to form part of the initial condition for the calculation. The computational domain is (0, 1.6)×(0, 1), and 26,000 quads are used in the present calculation. The flow conditions of upstream and downstream of the shock are respectively

$$u_0=2.9,\ v_0=0.0,\ \rho_0=1.0,\ p_0=0.7142857$$

and

$$u_0=2.6193,\ v_0=-0.50632,\ \rho_0=1.7,\ p_0=1.5282$$

A vortex, initially centered at (0.2, 0.6), with the following circumferential velocity (r_0=0.06)

$$u_\theta = \begin{cases} 5r, & r \le r_0 \\ 2-5r, & r_0 < r \le 2r_0 \\ 0, & 2r_0 < r \end{cases}$$

is superimposed on the pre-calculated flow. Figure 1 shows the process of the vortex propagating across a strong shock.

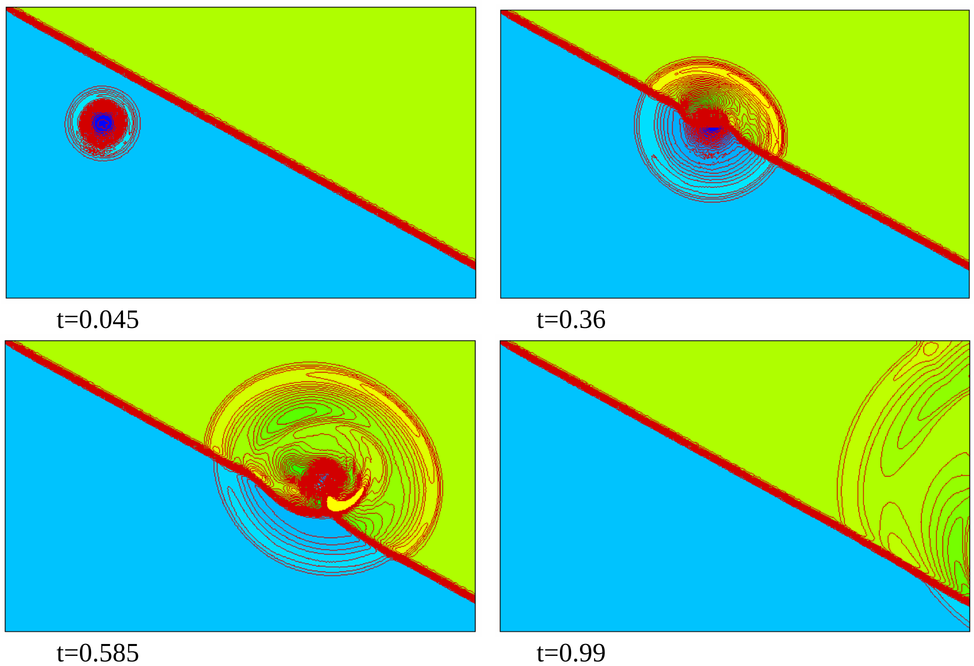

Figure 1. Isobars for the shock/vortex interactions at four different time steps.

Example 2: Three-Dimensional Detonation Waves
In the second example, the three-dimensional simulation of a detonation wave is performed by solving the reacting fluid equations. The chemical reactions are modeled by single-step, irreversible, and finite-rate kinetics. Two chemical species are considered, i.e., the reactant and the product. With proper non-dimensionalization, it can be shown that the defining parameters for this problem are the overdriven factor *f*, the specific heat ratio γ, the activation energy E^+, and the heat release rate q_0. In the present calculation, $f = 1.6$, $\gamma=1.2$, $E^+= 50$, and $q_0 = 50$.

The computation domain is [8×8×6], which is decomposed into 3.2 millions hexahedrons. Reflective wall condition is imposed on the four lateral boundaries. The fresh reactant travels from top to bottom, and is consumed by the flame front. On the top surface, the incoming flow condition is specified. On the bottom surface, a non-reflective boundary condition is imposed. The coordinate system is chosen such that the flame front stays around the *z*=0.5 section of the computational domain. Figure 2 shows one snapshot of product species contours. The flow field is composed of the quiescent state of the reactant before the shock, a flame zone with finite rate reaction, and the equilibrium state after the reaction zone. Due to cellular structure of the detonation, the flow field is very complex. The shock front is characterized by mushroom-shaped incident shocks interacting with a Mach stem. The width of the Mach stem changes in a periodic fashion and many strong vortices are created during the process.

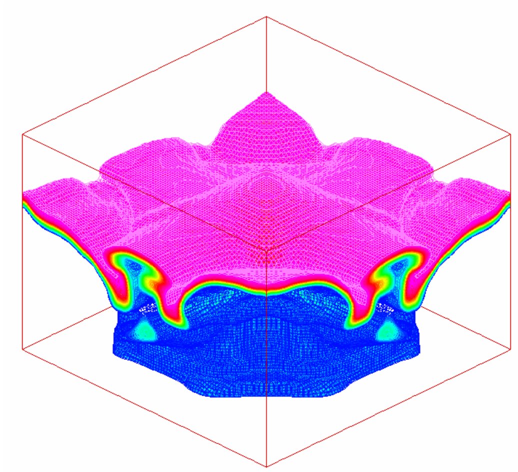

Figure 2. Product species contours in 3D detonation wave inside a square duct.

Numerical results show that the CE/SE method is efficient and robust, and that it has high accuracy and high shock resolution.

Future Features

By the end of 2002, this new CE/SE solver with basic functions will be ready for fluid flow simulations. Then, this fluid solver will be coupled with other physics solvers in LS-DYNA.

INCOMPRESSIBLE FLOW SOLVER

Galerkin FEM Incompressible Navier-Stokes Solver

The LS-DYNA incompressible flow solver is new in version 960 of LS-DYNA. This facility solves the incompressible Navier-Stokes equations with a low-order (Q1Q0) Galerkin finite element formulation. The Q1Q0 element provides a multi-linear approximation to the velocity and temperature fields, and a piecewise constant pressure in each element. While explicit time integration is available, the semi-implicit and implicit time integration methods are more economical and accurate. These implicit algorithms are based upon the second-order projection method of Gresho and Chan[6]. In each of the solution methods, a pressure-Poisson equation (PPE) must be solved at each time step; solving this equation dominates the overall cost of this solver. The current Navier-Stokes model assumes a constant density. In the area of turbulence modeling, a scalar Smagorinsky model is implemented.

One of the main uses for this solver at the time of this writing is for thermal convection problems. A two-dimensional example of natural convection flows in enclosures was studied as part of a comparison of different solvers in [5]. It was demonstrated that the implicit solvers were efficient and reliable.

For a more complete discussion of the incompressible flow solver, the manual is available as `ls-dyna_incompfluid_manual.pdf` in the "ls-dyna/manual" directory on the LSTC `ftp` site. The input files for the examples in this manual can be found in the "ls-dyna/example" directory (`ins_manual_examples-tar.gz`, also on the LSTC `ftp` site).

Recent Feature Enhancements

Output of time history data in the incompressible flow solver is now accomplished through a new database mechanism (*d3thins*). *d3thins* is supported in the latest beta versions of LS-POST, starting in November, 2001, and with a significant performance upgrade in March, 2002. *d3thins* is activated through the usual time history keyword cards, but has different controls in LS-POST. To look at the incompressible flow solver time histories, select the **CFD** button in the panel of buttons labeled "**2**." Bring the CFD time history data into LS-POST by choosing the **LOAD** button and entering the path to the *d3thins* file. After selecting the nodes of interest, time histories of state variables at those nodes can be plotted by choosing the **PLOT** button.

Another new database is the time-averaged output database (*d3mean*). *d3mean* is useful for collecting statistics about turbulent flows, as well as facilitating the identification of average values of state variables in oscillatory phenomena. ***DATABASE_BINARY_D3MEAN** is the keyword card that sets up the output of time-averaged data. The time-averaged data can be visualized in LS-POST by choosing the **D3Mean** button in the **Fcomp** menu on the default panel of buttons. The user then selects the *d3mean* file to load into LS-POST. Viewing controls for these variables operate in very similar fashion to those for snapshot variables that are output through *d3plot*.

Starting with a beta release of LS960 in March 2002, massively parallel processing (MPP) of the incompressible flow solver is working reliably. This involves a parallel conjugate-gradient (CG) linear systems solver that is particularly suited to solving very large problems. All of the output mechanisms are available in the MPP version of the solver. That is, parallel I/O has been implemented for the new *d3thins* and *d3mean* databases, and can be used in the March, 2002 beta release of LS960.

Numerical Example

The example problem is a flow around the Ahmed's body[7] of Re=4.2×10^6. The PPE was solved with a parallel SSOR preconditioned CG method, and each momentum equation was solved with a parallel Jacobi preconditioned CG method. Turbulence was modeled using a Smagorinsky LES subgrid scale turbulence model with C_s=0.1. The mesh was an unstructured mesh consisting of 166,816 hexahedron elements. During this MPP computation, 8 processors on an SGI Origin were used.

Figure 3 shows the isosurfaces of pressure at about halfway through the simulation, while Figure 4 shows the isosurfaces of pressure at the end of the simulation.

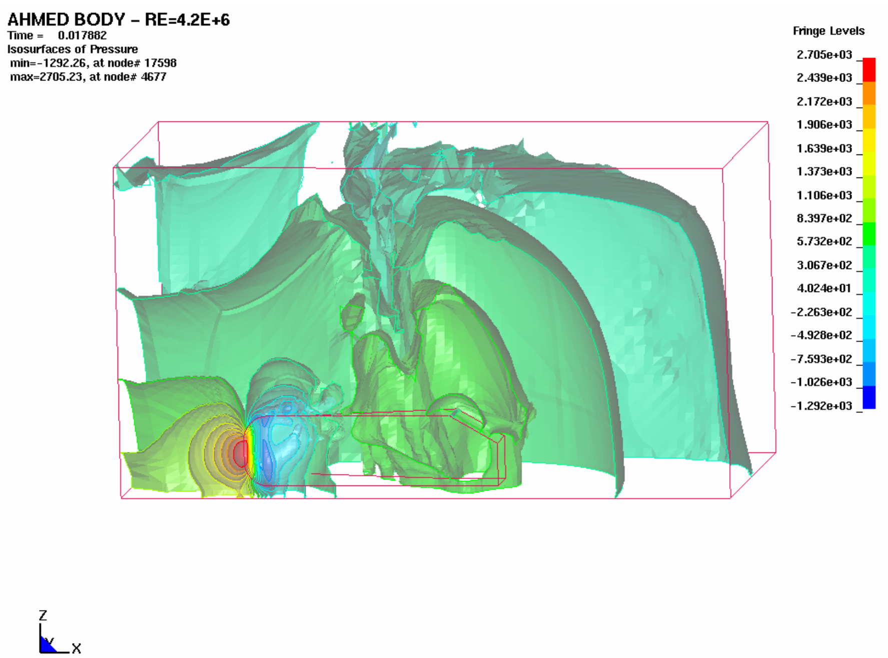

Figure 3. Isosurfaces of pressure at t=0.017

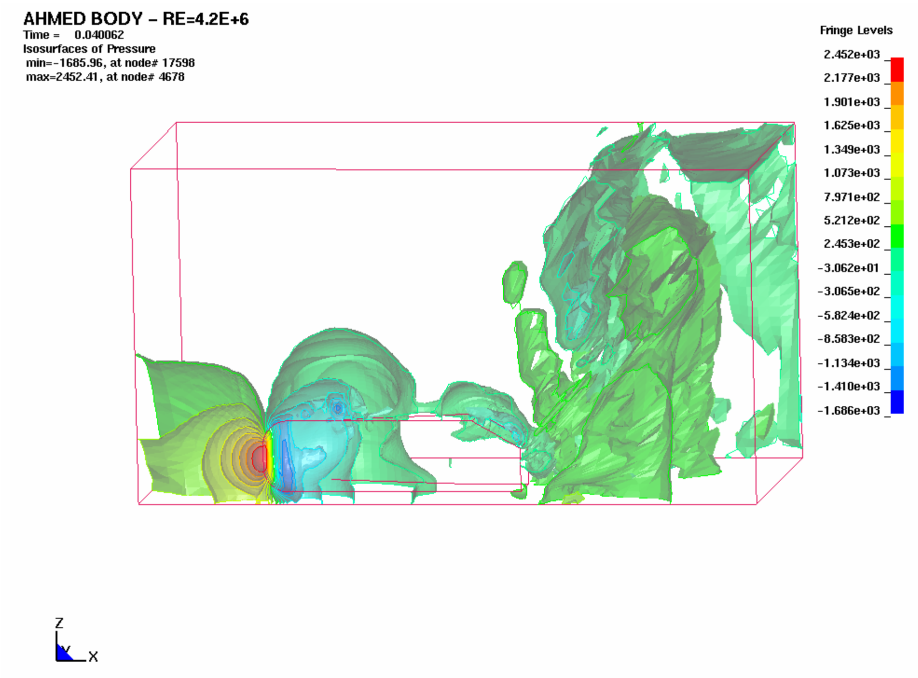

Figure 4. Isosurfaces of pressure at t=0.040

Future Features

In addition to coupling the incompressible flow solver to heat conduction in adjacent solid materials, a number of other features are being added to the next release of this flow solver. First, support for mixed meshes comprised of tetrahedron, wedge, and hexahedron elements is being implemented. Second, an equal-order (Q1Q1) FEM solver is under development for meshes of mixed elements. Third, a coupled radiative heat transfer solver will enable the simulation of radiative and convective heat transfer processes for vehicle engine environments, furnaces, etc. Fourth, improved turbulence modeling will be supported through a k-ε Reynolds-averaged Navier-Stokes (RANS) turbulence model that is appropriate for wall-bounded turbulence.

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. CHANG S.C., WANG X.Y. and CHOW C.Y. (1999). “The Space-Time Conservation Element and Solution Element Method: A New High-Resolution and Genuinely Multidimensional Paradigm for Solving Conservation Laws,” J. Comput. Phys., Vol. 156, p.89.
3. CHANG S.C., WANG X.Y. and TO W.M. (2000). “Application of the Space-Time Conservation Element and Solution Element Method to One-Dimensional Convection-diffusion Problems,” J. Comput. Phys., Vol. 165, p.189.
4. LOH C.Y., HULTGREN L.S. and CHANG S.C. (2001). “Wave computation in compressible flows using the space-time conservation element and solution element method,” AIAA J., Vol. 39, p.794.
5. CHRISTON M.A. (2001). “LS-DYNA and the 8:1 differentially-heated cavity,” First MIT Conference on Computational Fluid and Solid Mechanics, June 12-15, 2001, p. 1460.
6. GRESHO P.M., CHAN S.T. (1990). “On the theory of semi-implicit projection methods for viscous incompressible flow and its implementation via a finite element method that also introduces a nearly consistent mass matrix. Part 2: Implementation,” Int. J. Numer. Methods Fluids, Vol. 11, p. 621.
7. AHMED S.R., RAMM G. and FALTIN G. (1984). “Some Salient features of the time-averaged ground vehicle wake,” SAE paper 840300.
8. ZHANG Z.C. and YU S.T. (1999). “Shock Capturing without Riemann Solver---A Modified Space-Time CE/SE Method for Conservation Laws,” AIAA Paper 99-0904.
9. ZHANG Z.C., YU S.T., CHNAG S.C., HIMANSU A. and JORGENSON P. (1999). “A Modified Space-Time CE/SE Method for Euler and Navier-Stokes Equations,” AIAA Paper 99-3277.
10. ZHANG Z.C., YU S.T., HE H., JORGENSON, P.C.E. (2001). “Direct Calculations of Plume Dynamics of a Pulse Detonation Engine by the Space-Time CE/SE Method,” AIAA Paper 2001-3614, 37th AIAA/ASME/SAE/ASEE JPC Conference, July 8-11, 2001, Salt Lake City, Utah.
11. 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.