The CE/SE Compressible Fluid Solver

(Theory)

I. CE/SE Method

This new compressible fluid solver in LS-DYNA is based upon the Space-Time Conservation Element and Solution Element Method (or the CE/SE method for short).

The space-time conservation element and solution element (CE/SE) method, originally proposed by Chang[1], is a new numerical framework for solving conservation laws. The design principles and new features of the CE/SE method can be found in the references. In the following, a brief review of the CE/SE method is provided.

Consider the following three-dimensional compressible Navier-Stokes equations:

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

Let (x, y, z, t) be the coordinates of a four-dimensional Euclidean space E_4. Using the Gauss divergence theorem, we can get the following integral counterpart:

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

$$h_m = (f_m, g_m, h_m, u_m) \tag{3}$$

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).

Let the fluid computational domain divide into N volume elements E_i (i = 1, 2, 3, … N). There are two important definitions in the CE/SE method, i.e., conservation element (CE) and solution element (SE). The CEs are non-overlapping space-time sub-domains introduced such that the computational domain is the union of these sub-domains and the flux conservation equations (2) are enforced over each of them. Usually, there are several sub-CEs connected with each mesh element depending on that element shape. The union of those sub-CEs is called the main or general CE. The centroid of each main CE is called the solution point and all of the flow variables will be solved at these points.

On the other hand, a SE is a space-time sub-domain over which the physical flow vector is approximated using some kind of smooth functions. Here, connected with each mesh element, we define a SE in which the flow vector is approximated by the 1st order Taylor expansion, i.e.,

$$u^*(x,y,z,t)_{(i,n)} = u + u_x(x-x_i) + u_y(y-y_i) + u_z(z-z_i) + u_t(t-t^n) \tag{4}$$

Where (x_i, y_i, z_i) is the solution point connected with the mesh element E_i. In the CESE method, flow variables u and their spatial derivatives u_x, u_y, u_z are all treated as the independent unknowns, and solved simultaneously, while the time derivative u_t is calculated by equation (1).

If we substitute (4) into (3) and enforce equation (2) over the main CE and those sub-CEs at each mesh element and then solve these equations, we obtain the flow variables and their respective spatial derivatives (denoted by u and ($u_x^{(a)}$, $u_y^{(a)}$, $u_z^{(a)}$), called the a-scheme). Since there is no dissipation in this scheme, it is not stable for non-linear problems. There are other ways to calculate the spatial derivatives (u_x, u_y, u_z), and the following are some of them.

First, if we use a quasi-central-difference method based on the solutions at neighbor solution points (by Taylor expansion in the time direction), we get another set of spatial derivatives, denoted by ($u_x^{(c)}$, $u_y^{(c)}$, $u_z^{(c)}$).

Furthermore, we can use a weighting technique instead of the above central difference method, e.g.,

$$u_x^w = w_- \cdot u_x^- + w_+ \cdot u_x^+ \qquad , \tag{5}$$

where

$$w_{\pm} = \frac{1+\sigma\eta_{\mp}}{2+\sigma(\eta_{+}+\eta_{-})} \; ; \qquad \sigma = 0.5/|\nu| \tag{6}$$

$$\eta_{\pm} = |u_x^{\pm}|^{\alpha} / \min\left\{ |u_x^{-}|^{\alpha} + |u_x^{+}|^{\alpha} \right\} + 1 \tag{7}$$

It is defined similarly for the other two derivatives, u_y^w and u_z^w. Then we get another weighting set of derivatives, denoted by ($u_x^{(w)}$, $u_y^{(w)}$, $u_z^{(w)}$). In equations (5) and (7), u_x^+ and u_x^- are the forward and backward differences, $|\nu|$ is the local CFL number and $\alpha \geq 0$ is the weighting parameter. If α=0.0, the above weighted average becomes the simple average, and increasing the value of α will add more numerical dissipation and the calculation will be more stable. If the flow is smooth, the above weighted and simple averages become identical. Usually, α=1 (default value) is good enough for most cases. But if the shock is too strong, a larger value is needed.

Now we can choose to use one of the above methods or a combination of them to calculate the spatial derivatives (u_x, u_y, u_z). In this solver, we provide the following two choices:

(1) Due to the instability of the a-scheme for non-linear problems, a relaxation procedure is added in solving ($u_x^{(a)}$, $u_y^{(a)}$, $u_z^{(a)}$). We denote such relaxed derivatives by ($u_x^{(\varepsilon)}$, $u_x^{(\varepsilon)}$, $u_x^{(\varepsilon)}$). The amount of relaxation is controlled by an input parameter ε ($\varepsilon \geq 0.0$) and the solution itself at each time step. If ε =0.0, ($u_x^{(\varepsilon)}$, $u_x^{(\varepsilon)}$, $u_x^{(\varepsilon)}$) become ($u_x^{(a)}$, $u_y^{(a)}$, $u_z^{(a)}$). This is usually not stable because of the non-linearity of equation (1). So typically, ε>0.0, with larger values of ε giving a more stable calculation. Since accuracy goes down as ε increases, very large values should be avoided. Alternatively, if ε=0.0, the following set of equations will be used instead.

(2) A combination of the central-difference and weighted one, i.e.,

$$u_x^{(c)} + \beta(u_x^{(w)} - u_x^{(c)}) \tag{8a}$$

$$u_y^{(c)} + \beta(u_y^{(w)} - u_y^{(c)}) \tag{8b}$$

$$u_z^{(c)} + \beta(u_z^{(w)} - u_z^{(c)}) \tag{8c}$$

Where $0.0 \leq \beta \leq 1.0$. (this set of derivative approximations is denoted by $u_x^{(wc)}$, $u_x^{(wc)}$, $u_x^{(wc)}$)

For a smooth flow, β=0.0 can be used. But if a flow contains large gradients or discontinuities like shocks, β=1.0 is highly recommended.

The users can choose either one of the above two methods. If $\varepsilon \neq 0.0$, the first one will be used and parameters α and β will be ignored, otherwise the second one will be used. For most cases, there will not be much difference between the results of these two methods. The details about the CESE method can be found in the cited references.

II. Fluid/Structure Coupling (FSI)

In fluid/structure coupling, a quasi-constraint method is used. For the fluid part, the CE/SE method just described is used based upon an Eulerian frame, while for the structural parts, the FEM solver in LS-DYNA is employed based upon the Lagrangian frame.

The interface boundary locations and velocities are dictated by the Lagrangian structure. This information will be used by the fluid solver as the interface conditions at each time step, and the CE/SE solver feeds back the fluid pressures (forces) on the structural interface as exterior forces for the structural solver.

III. Cavitating Flow

Cavitation is a very complex vapor-liquid multi-phase flow including phase changes and viscous effects. In spite of significant research, the actual structure of cavitation is not yet fully understood. So, numerical simulation becomes a very important tool. Its importance grows in view of the impact of cavitation on the safety and efficiency of many industrial operations. Actually, cavitation is a common hydrodynamics phenomenon that has received much attention over the past several decades. For example, the presence of cavitation in hydraulic machinery can lead to significant flow blockages and changes in the flow pattern. These in turn lead to efficiency losses, changes in the hydrodynamics loads and corresponding head loss in these machines. Cavitation can also result in critical erosion damage and noise in underwater vehicles.

On the other hand, for high-pressure, high-speed modern automotive diesel injectors, the influence of injector orifice design has long been known to be a key feature influencing the spray breakup characteristics, and thus overall engine performance. However, it is extremely difficult, if not impossible, to directly observe the highly transient internal nozzle cavitating flow because of the small size of the nozzle orifice and high-speed flow in such a nontransparent enclosure. Therefore, numerical simulation is the perfect tool to shed more light on this problem.

Basically, cavitation is a multiphase phenomenon where the gas phase can be generated inside a liquid phase if suitable conditions are reached in the flow properties, such as a large drop in pressure. However, simulation of cavitation is an extremely difficult problem because there always exists a very stiff source term related to the interaction between phases (liquid density/gas density ~ 1000).

There are two alternative ways to resolve this problem. The first one is to construct an EOS valid for the liquid phase, the two-phase mixture, and the gas phase. Such an approach is well-posed if one can provide a valid EOS for the mixture phase. Currently, most of existing models can be categorized as this type of model. The second alternative is to use a non-equilibrium multiphase flow model. In this case, each phase has its own EOS, with the gas phase typically being modeled by an ideal gas EOS.

As a first step, we have implemented the Schmidt's homogeneous equilibrium model[4] (HEM), which is based on the acoustic speed of the mixture of liquid and vapor, in LS-DYNA's CESE fluid solver. This model is simple but contains the important characteristic information of cavitating flows.

In HEM, the ideal gas EOS is used for the gas phase. The EOS for the mixture is given by,

$$p = p_l^{sat} + \frac{\rho_v a_v^2 \rho_l a_l^2 (\rho_v - \rho_l)}{\rho_v^2 a_v^2 - \rho_l^2 a_l^2} \log \frac{\rho_v a_v^2 \rho_l + \alpha(\rho_v + \rho_l)}{\rho_l (\rho_v a_v^2 - \rho_l a_l^2)} \qquad (9)$$

Where, (ρ_v, ρ_l) and (a_v, a_l) are the density and sound speed for vapor and liquid respectively, p_l^{sat} is the saturation pressure, and α is the void fraction, which is defined as,

$$\alpha \overset{def}{=} \frac{\rho - \rho_l}{\rho_v - \rho_l} \tag{10}$$

Since all parameters except the void fraction are given by fluid properties, the EOS of the mixture phase is a function of the mixture density, ρ, only. The EOS of the pure liquid phase can also be obtained by using an isentropic relation combined with saturation pressure as follows,

$$p = p_l^{sat} + a_l^2 \rho \tag{11}$$

Also, the viscosity is assumed to depend linearly on α for simplicity, i.e.,

$$\mu = \alpha \mu_v + (1-\alpha)\mu_l \tag{12}$$

Where μ_v and μ_l are the viscosities for the vapor and liquid respectively.

<u>Summary</u>: (ρ_v, a_v), (ρ_l, a_l), (μ_v, μ_l) and p_l^{sat} are the seven input parameters that need to be specified for this cavitation model in the "***CESE_EOS_CAV_HOMOG_EQUILIB***" EOS card. This cavitation model is suitable for high-speed flows in small geometry, such as the diesel injection systems mentioned above.

IV. Other Aspects

2D code

The default of this solver is 3D. But we also provide an option for 2D problem calculations. If the user wants to use this 2D solver, they must have a mesh that has only a single layer of elements in the z-direction (i.e., a 2D x-y mesh (quadrilateral or triangular mesh) but extended in the z-direction). Also, the user should not define the boundary conditions on the z-direction boundary surfaces (both sides). Then the code will automatically check for this setup, and use the 2D solver if the above requirements are met. This saves a lot of CPU time for some problems.

2D axisymmetric code

A 2D axisymmetric flow option is also available with this solver. In order to make use of this option, the above 2D requirements must be satisfied first. In addition, the x and y coordinates must correspond to the axial- and radial-directions respectively.

Current Status

For the fluid solver, both serial and MPP solvers are available for inviscid and viscous flows (currently laminar flows only). Supporting meshes can consist of tetrahedra, wedges, or hexahedra or a mixture of these elements.

For the fluid/structure coupling, fluid coupling with shell elements (thin), solid volume elements, or both in the same simulation is available.

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. CHANG S.C. (2002). "Courant Number Insensitive CE/SE Schemes," AIAA paper 2002-3890.
4. Schmidt, D.P.,Rutland, C. J., Corradini, M. L. (1999), "A Fully Compressible Two-Dimensional Model of Small, High Speed Cavitating Nozzles," Atomisation Sprays Techology 9: 255-276.