ALE Incompressible Fluid in LS-DYNA®

Aquelet Nicolas

Livermore Software Technology Corporation
7374 Las Positas Road
Livermore, CA 94551

Souli Mhamed
Université de Lille, Laboratoire Mécanique de Lille, France

Abstract

The computation of fluid forces acting on a rigid or deformable structure constitutes a major problem in fluid-structure interaction. However, the majority of numerical tests consists in using two different codes to separately solve pressure of the fluid and structural displacements. In this paper, a monolithic with an ALE formulation approach is used to implicitly calculate the pressure of an incompressible fluid applied to the structure. The projection method proposed by Gresho is used to decouple the velocity and pressure

Introduction

A computational procedure is developed to solve problems of viscous incompressible flows interacting with rigid or deformable structure. The arbitrary Lagrangian Eulerian method (ALE) is used to move the internal fluid nodes whereas the boundary fluid nodes move with the structure. The coupling of the mesh motion equations and the fluid equations is essentially done through contact surface boundary conditions. In continuum Mechanics, two descriptions are considered for the motion in a continuum media

ALE Description

The ALE description for incompressible viscous flows has been developed by Hughes at al [1], to solve free surface flows and fluid-structure interaction problems. The general kinematics theory developed in [1] serves as the basis of the Lagrangian-Eulerian description. For this purpose, the authors define three domains in space, and mappings from one domain to the other. The first one, called the spatial domain, is considered as the domain on which the fluid problem is posed. The spatial domain is generally in motion, because of moving boundaries. The second domain, called the material domain, is to be thought of as the domain occupied at time t=0 by the material particles which occupy the spatial domain at time t. The third domain, called the reference domain, is defined as a fixed domain throughout. From these domain descriptions, we can see that the Eulerian description is obtained when the spatial domain coincides with the reference domain, whereas the Lagrangian reference is obtained when the material domain coincides with the reference domain.
Both the material and spatial domains are generally in motion with respect to the reference domain; it is convenient to express the material time derivative of a physical property ϕ in the reference configuration.

$$\dot{\phi} = \phi_{,t} + c.\nabla\phi \tag{1}$$

where $\dot{\phi}$ is the material time derivative, and $\phi_{,t}$ is the time derivative when freezing coordinates in the reference domain, c is the convective velocity.

$$c = v - v^{mesh} \tag{2}$$

v is the fluid velocity, and v^{mesh} is the mesh velocity. In the Eulerian description, the mesh velocity is zero, $v^{mesh} = 0$, whereas in the Lagrangian description $v^{mesh} = v$, and $c = 0$.
In the ALE formulation, the mesh nodes move with an arbitrary velocity. The choice of the mesh velocity constitutes one of the major problems with the ALE description. Different techniques have been developed for updating the mesh in a fluid motion, depending on the fluid domain. For problems defined in simple domains, the mesh velocity can be deduced through a uniform or non uniform distribution of the nodes along straight lines ending at the moving boundaries.

Governing equations

The Lagrangian formulations are frequently used to solve the structural behaviour. Indeed, displacements of the nodes and the elements on a Lagrangian mesh correspond to the movements of material. The material edges always coincide with the edges of the elements. Thus, if the material sharply becomes deformed, the mesh is subjected to distortions. In general, the structural deformations are weak so that the Lagrangian mesh remains regular and is not subjected to distortions. The boundary conditions are easily imposed because the edges of the mesh represent the limits of the physical domain during calculation. For these reasons, the Lagrangian formulations are much appreciated. In the Cartesian coordinate system, the displacement of the structure u in a domain Ω_S (see Fig.1) is governed by:

$$\rho_S \frac{\partial^2 u_i}{\partial t^2} = \sigma_{ij}(u)_{,j} + \rho_S g_i \tag{3}$$

with initial and boundary conditions:

$$u_i = \hat{u}_i \quad \text{on } \partial\Omega_{DS} \times [0, T] \tag{4}$$

Two points of view are generally considered to describe the movement of a fluid. The first is Lagrangian where the speed of the mesh follows that of the fluid. The disadvantage of this description is to generate great distortions of mesh. The second is Eulerian and consists in studying the movement of the fluid in fixed positions. The domain of study is fixed and the fluid is updated constantly in this one. This method introduces a term of convection into the equations to be solved. It avoids the great distortions of mesh. However, the difficulty is deferred to the interface where it is difficult to represent the boundary conditions for a problem of interaction fluid-structure.

So, we made recourse to a mixed formulation. This later is the ALE method which combines at the same time Eulerian and Lagrangian descriptions to describe the movement of the fluid particles. In this framework, the velocity of the incompressible viscous fluid in a domain is characterized by the mass and momentum conservation laws such that:

$$v_{i,i} = 0 \quad \text{in } \Omega_F \times [0,T] \tag{5}$$

$$\frac{\partial v_i}{\partial t} + (v_j - v_j^m) v_{i,j} - \frac{1}{\rho_F} \tau_{ij,j} = g_i \quad \text{in } \Omega_F \times [0,T] \tag{6}$$

where v_i and ρ_f indicate, respectively, the flow velocity components and the fluid density. The term v_j^m represents the velocity of the mesh. If $v_j^m = 0$, we obtain the Eulerian formulation because the convective velocity of the mesh is null. If $v_j^m = v_j$, we obtain the Lagrangian formulation for which the convective velocity is the fluid velocity. The quantity $v_j - v_j^m$ is the relative velocity and the stress tensor τ_{ij} is commonly defined by:

$$\tau_{ij} = \mu_F (v_{i,j} + v_{j,i}) - p\delta_{ij} \tag{7}$$

where μ_F is the fluid dynamic viscosity.
The momentum equation is to be solved with the initial condition and the boundary conditions:

$$v_i(0) = 0 \quad \text{in } \Omega_F \tag{8}$$

$$v_i = \hat{v}_i \quad \text{on } \partial\Omega_{DF} \times [0,T] \tag{9}$$

where $\hat{v}_i$ are the imposed velocity components on $\partial\Omega_{DF}$.
The boundary conditions on the fluid-structure interface $\partial\Omega_I$ are given by :

$$v_i = \frac{\partial u_i}{\partial t} \text{ on } \partial\Omega_I \times [0,T] \tag{10}$$

And $p = 0$, on the outflow boundary

Numerical Algorithm

It is well known that the main difficulties arising in the numerical solution of the convection-diffusion equations are due to their no-self-adjoint character. The standard Galerkin method leads to no physical spatial oscillations when applied to the high convective case. To preclude such anomalies, the most popular method being the use of upwind differencing on the convective term via Petrov-Galerkin methods (see, for example, Heinrich & al [2]; Heinrich and Zienkiewicz [3], Belytscho & al. [4]). Although theses methods are precise and stable, we will use a ‘split’ method which is a simple mean to obtain a robust and effective formulation. This time-split method decomposes the time step into two phases :

– Phase 1 is a solution of the Lagrangian equations of motion (advection terms are nil) updating the velocity field by the effects of all forces. For the fluid, the velocity-pressure formulation of the discretized problem is decoupled by the projection method (for more details, see Cho and Lee [5]).

– Phase 2 adds advection contributions, and is required for runs that are Eulerian or contain some relative motion of mesh and fluid.

In order to effectively solve the pressure and velocities satisfying the continuity constraint Eq.(5) for the phase 1, we adopt the fractional method proposed by Gresho [6]. The idea of these methods is to decouple the velocity v and the pressure p. These are based on a resolution in three steps of the Navier-Stokes equations.
Hereafter, we describe briefly the above method in Lagrangian formulation:

– Intermediate velocity. The first step consists in calculating an intermediate velocity v_i^* , solution of the Naviers-Stokes equation without taking into account the continuity constraint.

$$v_i^{*n+1} = v_i^n + \Delta t\left(\frac{\mu_F}{\rho_F} v_{i,jj}^n - \frac{1}{\rho_F} p_{,i}^n + g_i^n\right) \text{ in } \Omega_F \tag{11}$$

$$v_i^{*n+1} = \frac{\partial u_i^n}{\partial t} \text{ on } \partial\Omega_I \tag{12}$$

– Projection. As the velocity v_i^* does not yet satisfy the incompressibility condition Eq.(5), it is projected on a divergence free space to get an adequate approximation of the velocity. This is obtained from :

$$v_i^* = v_i + \frac{\Delta t}{\rho_F} \Delta p_{,i} \tag{13}$$

with $v_{i,i}^{n+1} = 0$. The term Δp is a pressure increment.
The second step consists in deriving a Poisson equation for the pressure *p*. In fact, by taking the divergence of Eq.(13) and using the incompressibility condition Eq.(5), we obtain :

$$\frac{1}{\rho_F} \Delta p_{,ii}^{n+1} = \frac{1}{\Delta t} v_{i,i}^{*\,n+1} \quad \text{in } \Omega_F \tag{14}$$

Once the corrective pressure Δp^{n+1} has been determined, the final velocity field is obtained from the intermediate velocity v_i^* and Δp^{n+1}:

$$v_i^{n+1} = v_i^{*n+1} - \frac{\Delta t}{\rho_F} \Delta p_{,i}^{n+1} \quad \text{in } \Omega_F \tag{15}$$

– Pressure update. Since v is the physical velocity, the pressure p can be given from Δp^{n+1}.

$$p^{n+1} = p^n + \Delta p^{n+1} \tag{16}$$

For the phase 2, we used a first order Godunov method : the Donor Cell (see Benson [7] and Amsden & al. [8]). This step is bypassed for a purely Lagrangian calculation. In all other cases

(Eulerian and ALE calculation) the relative velocity $v_{ALE} = v_j - v_j^m$ is not null, and we must calculate the flux of momentum between cells.

Numerical results

To illustrate this numerical method, we study the case of a rigid structure impacting a fluid at a velocity of 1650mm/sec. This problem is very common in Naval industry is called slamming. For a rigid structure, theoretical results are available in the literature. Time history pressure is plotted for both ALE formulations, using classical MAT_NULL with an equation of state, and MAT_ALE_INCOMPRESSIBLE with no equation of state. The problem set-up is described in figure1.

We can show from figure 2, that new incompressible material generates less oscillations for pressure history that the classical material MAT_NULL. It has been also observed that time step is higher when running the incompressible material than the class MAT_NULL material, since the time step only depends on the element size and the fluid velocity and not on the material speed of sound.

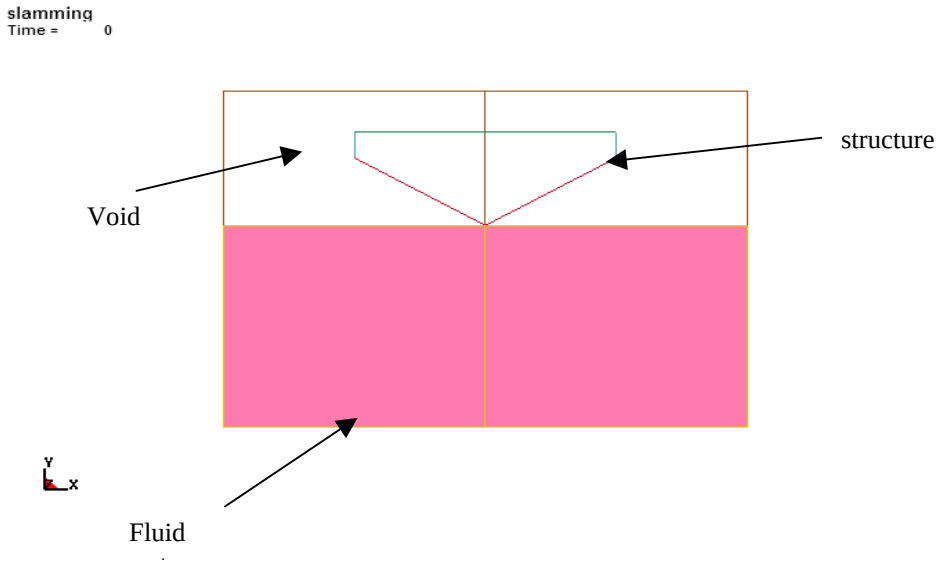

Figure 1. *Problem description*

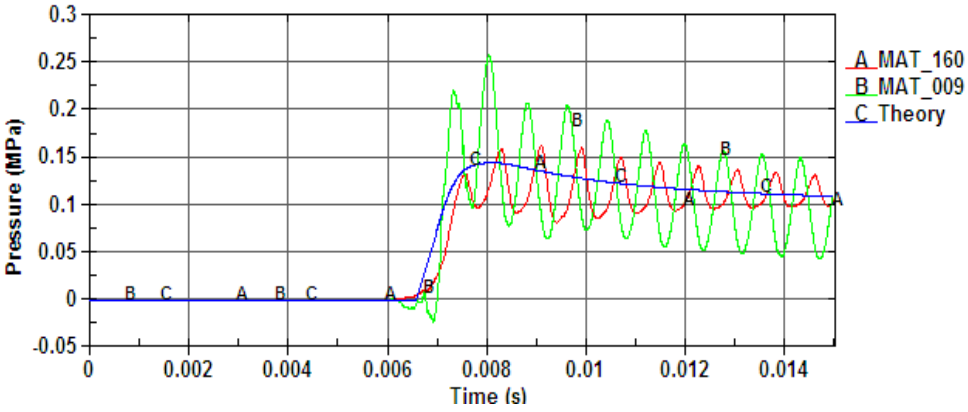

Figure 2: Pressure time history for different formulations
Comparison with theory

Conclusion

This paper describes the new incompressible material that has been developed in LSDYNA code. This material can be used will all LSDYNA capabilities, including contact algorithms, coupling using the CONSTRAINED_LAGRANGE_IN_SOLID for fluid structure interaction probems. Users can use this material for most CFD applications for Newtonian viscous fluid for laminar flow. It is our goal to extend this material for flow turbulence modelling using Large eddy simulation (LES), a mathematical model for turbulence used in computational fluid dynamics.

References

[1] T.J.R.Hughes, W.K.Liu and T.Zimmerman Lagrangian Eulerian finite element formulation for viscous flows. Comp Meth in Applied Mech and Eng 29 (1981) 329-349

[2] Heinrich J.C., Huyakorn P.S., Zienkiewicz O.C. and Mitchel A.R. "An upwind finite element for two-dimensional convective transport equations", Int. Journ. Num. Meth. Eng., 11, pp 131-143, (1977).

[3] Heinrich J.C. and Zienkiewicz O.C. "Quadratic finite element schemes for two-dimensional convective transport problems", Int. Journ. Num. Meth. Eng., 11, pp 1831-1844, (1977).

[4] Belytscho T., Liu W. K., Moran B., Nonlinear finite elements for continua and structures, Wiley, Chichester, 2000.

[5] Cho J.R., Lee S.Y., "Dynamic analysis of baffled fule-storage tanks using the ALE finite element method", Int. Journ. Num. Meth. Fluids 41 (2003) pp 185-208.

[6] Gresho P.M., "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 1 : theory ", Int. Journ. Num. Meth. Fluids 11 (1990) pp 587-620.

[7] Benson D.J., "Lagrangian and Eulerian hydrocodes", Comput. Methods Appl. Mech. Eng,. 99 pp 235-394, (1992).

[8] Amsden A. A., Ruppel H. M., Hirt C. W., SALE – a simplified ALE computer program for fluid flow at all speeds, Report, 1980, Los Alamos National Laboratory.