ALE Incompressible Fluid in LS-DYNA

Nicolas Aquelet, Mhamed Souli

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

1. INTRODUCTION

In this paper, we present the algorithm which allows to compute fluid velocity and pressure using explicit time integration for velocity and an implicit method for pressure computation to ensure and enforce fluid incompressibility. For the structure an explicit method is performed. For this, we use a finite element method to solve the governing equations for the structure and the Navier-Stokes equations using a Lagrangian formulation. In order to solve the governing equations for the fluid in Eulerian or Arbitrary Langrangian Eulerian (ALE) formulation, we use the 'split' operator described. In order to solve fluid-structure interaction problems, we have to compute the pressure acting on the structure. The projection method, introduced initially by Chorin and Temam (1968) and proposed by Gresho (1990) is implemented to meet this requirement. The numerical example studied in this paper shows the interest of an implicit pressure for this type of problems. This paper is organized as follows : in the section 2 the fluid governing equations are presented, using a split method, a Lagrangian phase where the nonlinear term in the Navier Stokes equations is not taken into account, followed by an advection phase. For validation, we analyze slamming problem, a rigid structure impacting water at rest; the new material is compared to the classical MAT_NULL in LSDYNA, Hallquist (1998), for sloshing problems Souli et al (2011)

2. Eulerian Formulation for Fluid solver

2.1 Lagrangian phase

In the ALE description, an arbitrary referential coordinate is introduced in addition to the Lagrangian and Eulerian coordinates. The material derivative with respect to the reference coordinate can be described in equation (2.1). Thus substituting the relationship between material time derivative and the reference configuration time derivative leads to the ALE equations in (2.1)

$$\frac{\partial f(X_i,t)}{\partial t} = \frac{\partial f(x_i,t)}{\partial t} + w_i \frac{\partial f(x_i,t)}{\partial x_i} \tag{2.1}$$

where X_i is the Lagrangian coordinate, x_i the Eulerian coordinate, w_i is the relative velocity. Let denote by *v* the velocity of the material and by *u* the velocity of the mesh. In order to simplify the equations we introduce the relative velocity $w = v - u$. Thus the governing equations for the ALE formulation are given by the following conservation equations, mass equation (2,2) and momentum equation (2,3)

$$div(u)=0 \tag{2,2}$$

$$\rho\left(\frac{\partial vi}{\partial t}+wi\frac{\partial vi}{\partial xj}\right)=div(\sigma) \tag{2,3}$$

where σ is the stress tensor defined by $\sigma=-p.I_d+\tau$ where τ is the shear stress from the constitutive model, and p the dynamic pressure.

Note that the Eulerian equations commonly used in fluid mechanics are derived by assuming that the velocity of the reference configuration is zero, $u=0,$ and that the relative velocity between the material and the reference configuration is therefore the material velocity, $w=v$. The term in the relative velocity in (2.3) is usually referred to as the advective term, and accounts for the transport of material past the mesh. It is the additional term in the equations that makes solving the ALE equations much more difficult numerically than the Lagrangian equations, where the relative velocity is zero.

There are two ways to implement the ALE equations, and they correspond to the two approaches taken in implementing the Eulerian viewpoint in fluid mechanics. The first way solves the fully coupled equations for computational fluid mechanics; this approach used by different authors in CFD leads to large linear system to be solved. The alternative approach is referred to as an operator split in the literature, where the calculation, for each time step is divided into two phases. First a Lagrangian phase is performed, in which the mesh moves with the material, in this phase the changes in velocity due to the internal and external forces are calculated. In the Lagrangian formulation the equilibrium equations can be describes by equations (2.4) and (2.5)

$$div(u)=0 \tag{2.4}$$

$$\rho\frac{\partial v_i}{\partial t}=div(\sigma)+f_{ext} \tag{2.5}$$

2.2 Advection phase

In the second phase, the transport of mass, momentum and internal energy across the element boundaries is computed. This phase may be considered as a 're-mapping' phase. The displaced mesh from the Lagrangian phase is remapped into the initial mesh for an Eulerian formulation, or an arbitrary distorted mesh for an ALE formulation.

In this advection phase, we solve a hyperbolic problem, or a transport problem, where the variables are density, momentum per unit volume and internal energy per unit volume. Details of the numerical method used to solve the equations are described in detail in Aquelet et al (2005), where the Donor Cell algorithm, a first order advection method and the Van Leer algorithm, a second order advection method are used. As an example, the equation for mass conservation is:

$$\frac{\partial \rho}{\partial t}+\nabla.(\rho u)=0 \tag{2.6}$$

It is not the goal of this paper to describe the different algorithms used to solve the equation (2.6); these algorithms have already been described in detail in Aquelet et al (2005).

The data necessary for the advection algorithm are the cell volume before and after the Lagrangian phase, nodal velocities, nodal masses and volume fluxes across cells. A finite volume method is used to solve equation (2.6), the method is described in detail by the authors in Aquelet et al (2005).

2.3 Pressure Velocity formulation

The velocity solved in the Lagrangian and advection phase is not divergence free. During the resolution of the two phases, no condition on the velocity divergence has been enforced. To enforce fluid incompressibility, a projection type method described in detail in Chorin (1968) and Gresho (1998) is performed at each time step for pressure correction used as a Lagrange multiplier to enforce incompressibility condition. The projection method consists in deriving a Poisson equation for pressure correction δP. In fact, by taking the divergence of the equation (2,5) at each time step Δt , after time discretisation, and using the incompressibility condition (2.4), we obtain :

$$div(grad(\delta P))=\frac{\rho}{\Delta t}div(v)$$

– **Projection**. As the velocity $\tilde{v}$ does not yet satisfy the incompressibility condition (2,4), it is projected on a divergence free space to get an adequate approximation of the velocity. This is obtained from :

$$\bar{V}=v+\frac{\Delta t}{\rho}.\nabla(\delta P)$$

– ***Pressure update***. Since $\bar{V}$ is the fluid velocity, the pressure *P* can be given from δP .

$$P^{n+1}=P^{n}+\delta P$$

3. 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.

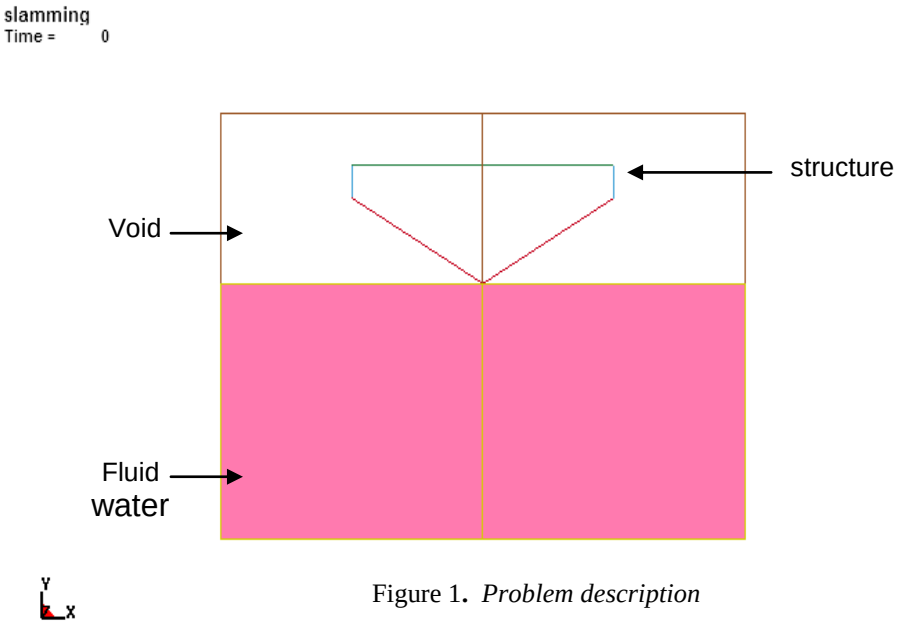

Figure 1. *Problem description*

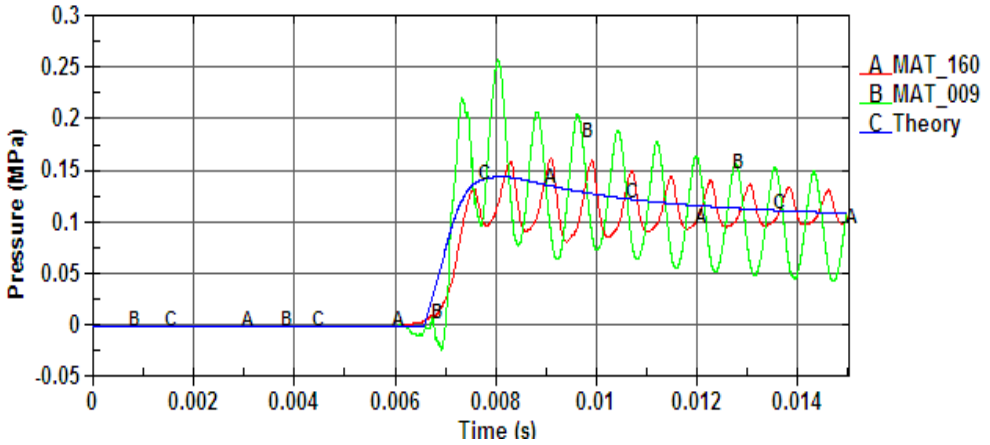

Figure 2: Pressure time history for different formulations
Comparison with theory

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 problems. 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

Souli M., Aquelet N.(2011)
Fluid Structure Interaction for Hydraulic Problems
Houilles Blanches, International Water Journal , n°6, 2011, p. 5-10, 2011.

Aquelet N, Souli M., L. Olovson L., (2005)
Euler Lagrange coupling with damping effects: Application to slamming problems**.**
Computer Methods in Applied Mechanics and Engineering, Vol 195, p 110-132

Chorin A.J., (1968)
Numerical solution of the Navier-Stokes equations", Math. Comp. 22 (1968), pp 745-768

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

Hallquist J.O., (1998)
Theorical manual for LS-DYNA, Livermore Software Technology Corporation, Livermore, 1998.