# ALE Incompressible Fluid in LS-DYNA®

Nicolas Aquelet
*LSTC Livermore*
*&*
Mhamed Souli
*University of 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 $\phi$ 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 $\Omega_S$ (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.

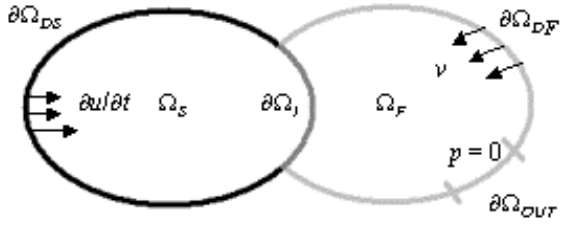

**Figure 1.** *Fluid and structure domains*

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 $\rho_f$ 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 $\tau_{ij}$ is commonly defined by:

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

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

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

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

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

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

And $p = 0$, on the outflow boundary

## Fluid Analysis 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 $\Delta p$ 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 $\Delta p^{n+1}$ has been determined, the final velocity field is obtained from the intermediate velocity $v_i^*$ and $\Delta p^{n+1}$:

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

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

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

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. For each cell (see Fig.2), we calculate the volume swept out by each of faces relative to their Lagrangian positions $x_L$. According to the sign of these volumes, we add or remove momentum to the cell.

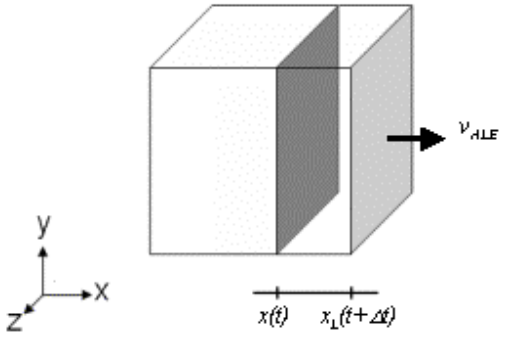


**Figure 2.** *Advected volume*

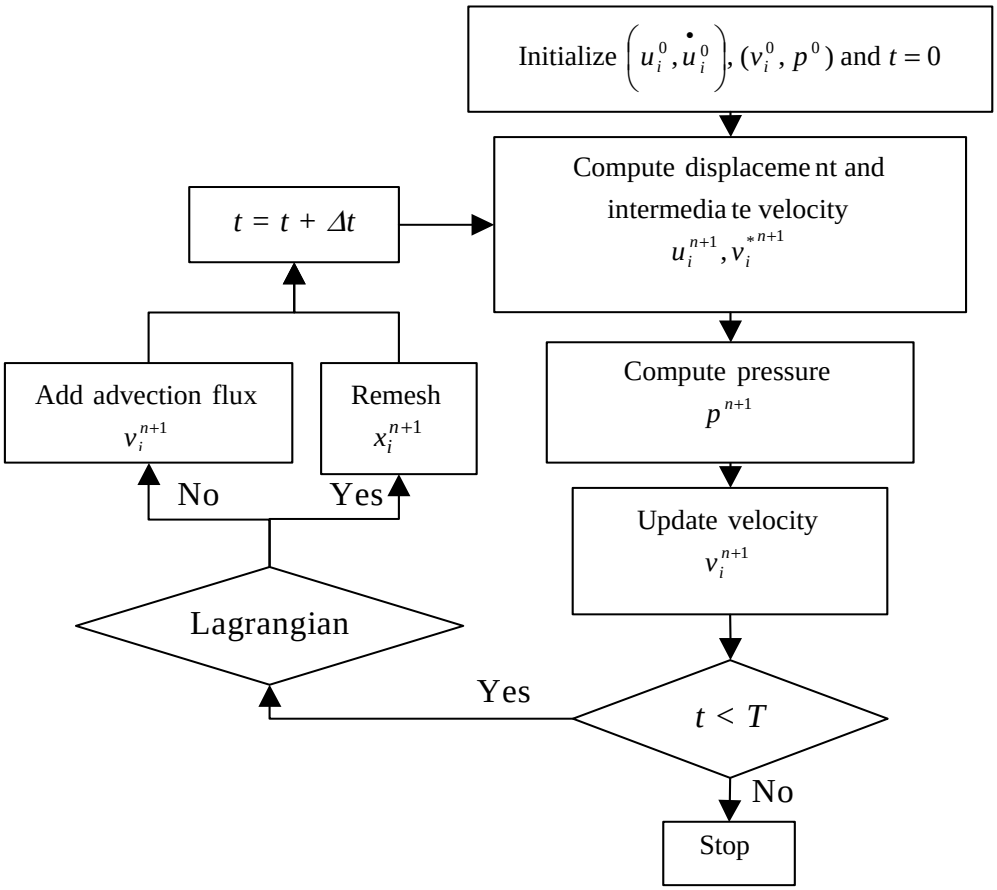


**Figure 3.** *Flowchart for the time-incremental fluid-structure numerical analysis*

## Numerical implementation

The numerical algorithm is sketched on Fig.3 .
In the course of the Lagrangian phase, we compute structural displacements and intermediate velocities necessary to the projection method. To obtain displacements and velocities, we compute nodal forces from respectively Eq.(11) and Eq.(12). This allows us to use the same method to solve the structural behaviour and the liquid dynamic response. The difference between the structural algorithm and the fluid algorithm is the computation of stress tensors. Then, we solve the pressure from Eq.(14) and Eq.(16). So, we obtain the velocity which is solenoidal (div($v$) = 0). For the Lagrangian nodes, we move the domain to update the coordinates of the nodes. And for the other nodes, we compute the momentum flux of the cell in order to update the velocity.

## Numerical results

To illustrate this numerical method, we study the case of a confined flow between two cylinders which are considered as infinite (see Fig.4). The diameter of the outer cylinder is $D = 5.5cm$ and that of the inner cylinder is $d = 2.2cm$. The thickness of the walls is $e = 1mm$ e = 10-3 m. The structural density, the Young modulus, the Poisson number, the fluid density and kinematic viscosity are respectively: $\rho_S = 2700kg.m^{-3}$, $E = 69000MPa$, $\eta = 0.3$, $\rho_F = 1000kg.m^{-3}$ and $\nu = 0.017545m^2s^{-1}$. Only the inner cylinder is excited, the other is fixed. Its velocity has the form :

$$v(x, y, z, t) = v_x(t) = A\sin(\pi f t) \tag{17}$$

with an amplitude $A = 1mm$ and a frequency $f = 38.7Hz$.

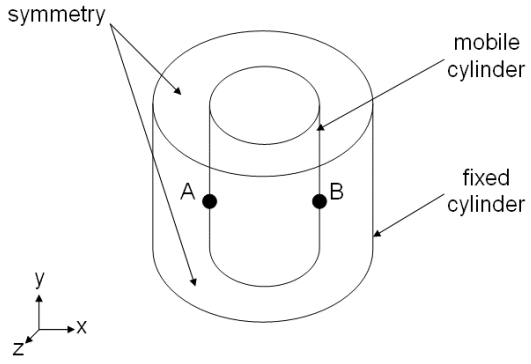


**Figure 4.** *Problem description*

We compare our results obtained by the method described in this paragraph 5 with those obtained by ASTER-SATURN and provided by Electricité De France (EDF). In this paper, we examine the evolution of the pressure at the points A and B which are diametrically opposite (see Fig.4 and Fig.5). To not deform the fluid mesh and create great distortions with the structural displacement, we use an ALE mesh for the elements near the inner mobile cylinder and an Eulerian mesh for the others elements (see Fig.5). This enables us to obtain a good approximation of the pressure of the element since in this case the mesh is not crushed.

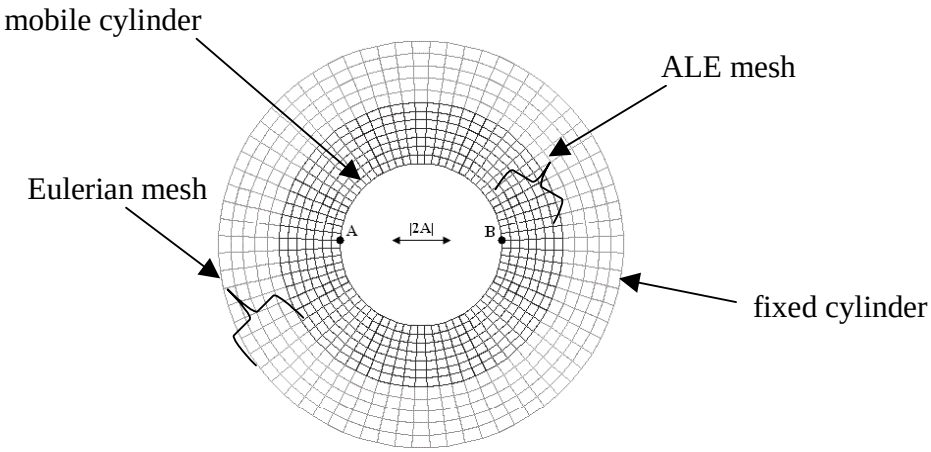


**Figure 5.** *Computational mesh*

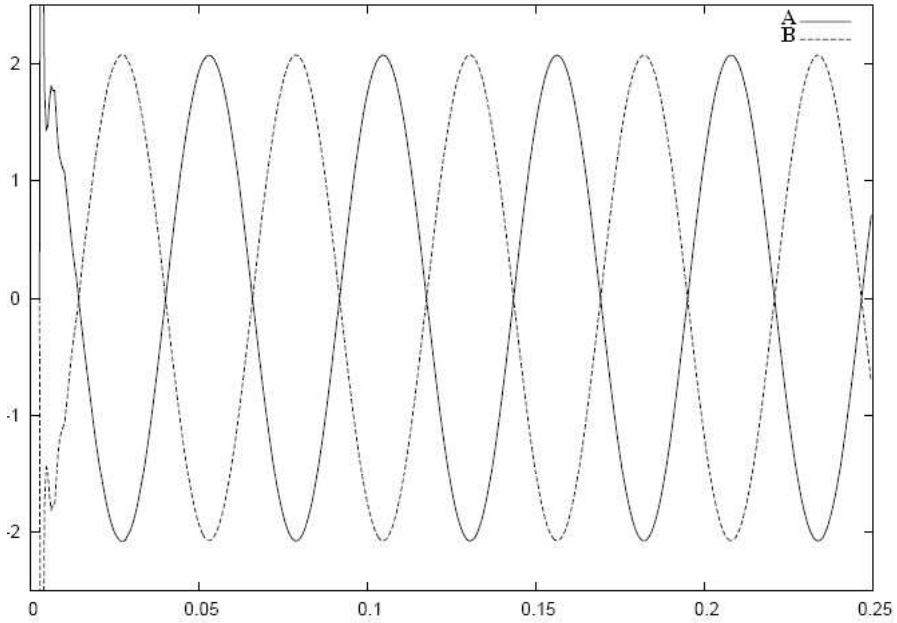


**Figure 6**. *Pressure evolution at the points A and B*

Figures 6 and 7 show the pressure evolution at two points obtained by the method adopted here and ASTER-SATURNE code respectively. We can observe that our results agree well with those reported by EDF. Moreover, the frequency of the response is the same frequency as that the imposed velocity defined by Eq.(17). This study seems to enable us to validate the present numerical method of resolution for the equations described in the paragraph 1 for the rigid body.

## Conclusion

In this paper, an ALE formulation for viscous incompressible flow has been presented. The finite element spatial discretization is used to solve the problem. However, for the advection term of the Navier-Stokes equation, a first order Godunov method is used. For the computation of the liquid dynamic response, the projection method defined by Gresho is implemented in order to handle the pressure. Numerical test shows that the projection method is an appropriate one for predicting fluid-structure interaction problem. The extension of this work will be the computation of fluid forces that act on a deformable structure in order to take into account the real structural behaviour.

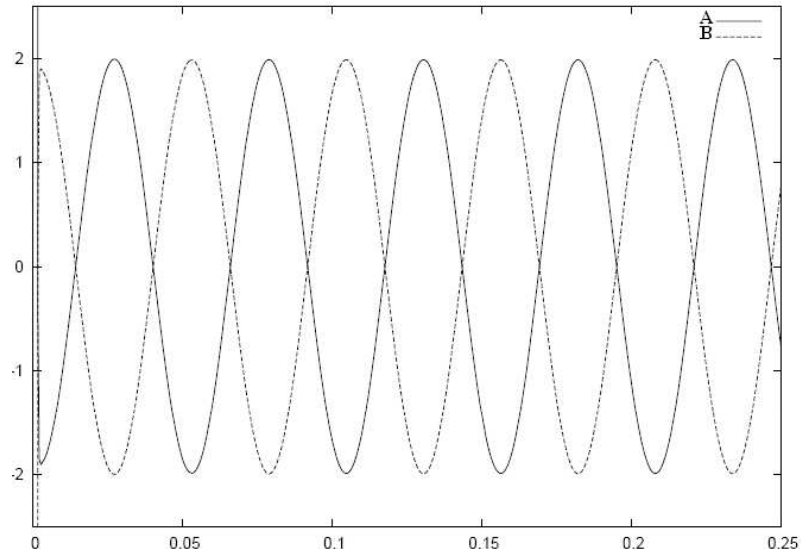


***Figure 7.*** Pressure evolution at the points A and B (ASTER-SATURNE)

## 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.