Euler–Lagrange coupling with damping effects: Application to slamming problems

N. Aquelet a,*, M. Souli a, L. Olovsson b

a Université des Sciences et Technologies de Lille, Laboratoire de Mécanique de Lille, UMR CNRS 8107, Bd. Paul Langevin 59655 Villeneuve d'Ascq, France
b Swedish Defense Research Agency, S-147 25 Tumba, Sweden

Received 17 March 2004; received in revised form 13 January 2005; accepted 13 January 2005

Abstract

During a high velocity impact of a structure on a nearly incompressible fluid, impulse loads with high-pressure peaks occur. This physical phenomenon called ‘slamming’ is a concern in shipbuilding industry because of the possibility of hull damage. Shipbuilding companies have carried out several studies on slamming modeling using FEM software with added mass techniques to represent fluid effects. In the added mass method inertia effects of the fluid are not taken into account and are only valid when the deadrise angle is small. This paper presents the prediction of the local high pressure load on a rigid wedge impacting a free surface, where the fluid is represented by solving Navier–Stokes equations with an Eulerian or ALE formulation. The fluid–structure interaction is simulated using a coupling algorithm; the fluid is treated on a fixed or moving mesh using an ALE formulation and the structure on a deformable mesh using a Lagrangian formulation.

A new coupling algorithm is developed in the paper. The coupling algorithm computes the coupling forces at the fluid–structure interface. These forces are added to the fluid and structure nodal forces, where fluid and structure are solved using an explicit finite element formulation. Predicting the local pressure peak on the structure requires an accurate fluid–structure interaction algorithm. The Euler–Lagrange coupling algorithm presented in this paper uses a penalty based formulation similar to penalty contact in Lagrangian analyses. Both penalty coupling and penalty contact can generate high frequency oscillations due to the nearly incompressible nature of the fluid. In this paper, a damping force based on the relative velocity of the fluid and the structure is introduced to smooth out non-physical high frequency oscillations induced by the penalty springs in the coupling algorithm.

© 2005 Elsevier B.V. All rights reserved.

Keywords: Fluid–structure interaction; Euler–Lagrange coupling; ALE formulation; Hydrodynamic impact; Slamming
1. Introduction

The main concern in fluid–structure interaction problems is the computation of the fluid forces that act on a rigid or deformable structure. In shipbuilding industry, empirical solutions and handbook calculations have been used to estimate forces due to hydrodynamic effects. The hydrodynamic forces are applied as external loads on the structural dynamic model to predict failure, stress fatigue, and creep damage to the structure. Since force calculations rely on an accurate knowledge of the flow field, an accurate fluid–structure analysis requires skilled engineering talent and careful modeling. The application of fluid–structure interaction technology allows us to move beyond handbook design techniques which use generic experimental correlation, to correctly predict the hydrodynamic forces by solving the hydrodynamical equations and using an appropriate coupling algorithm to communicate forces between fluid and structure for dynamic equilibrium.

In the structural design of ship hulls, an estimation of slamming loads is important to avoid substantial damage on the forebody. For instance, the probability of slamming on the fore part of high-speed vessels in rougher seas is significantly high and therefore, wave impacts can damage the bow. Because of the practical importance of the slamming study in shipbuilding engineering, several investigations have been carried out. The former work is due to Von Karman (1929) [1] who developed an asymptotic theory for flat impact problems with linearized free surface and body boundary conditions. The impact load on a sea plane during landing was estimated by the force on a two dimensional wedge upon entry into calm water, neglecting the water surface elevation. But this idealized theory based on momentum conservation underestimates the impact load for wedges with small deadrise angle, which represents the initial incidence angle between the free surface and the structure. The Von Karman formula was modified by Wagner (1932) [2] to account for piled-up water on the wedge. This improvement makes it possible to obtain a theory that is generally well suited to solve the problem of entry of 2D-wedges into water, if the deadrise angle is small. Wagner’s pressure formula is singular on the edge of the expanding plate. To prevent this problem, the effect of the nonlinear jet flow at the interface between the wedge and the free surface was included by Watanabee (1986) [3] by matching the solution in the splash region to the expanding plate solution of Wagner. In 1996, at small deadrise angle, Zhao et al. [4] developed an analytical solution for the pressure distribution on the body by generalizing the work of Wagner [2]. Within the framework of a study carried out by a research laboratory for shipbuilding, this paper compares the results obtained by Zhao’s theory [4] with the numerical results obtained using a fluid–structure coupling algorithm with damping formulation for high frequency modes. This was developed and implemented in LS-DYNA, an explicit finite element code for general fluid–structure interaction problems.

In the fluid–structure coupling algorithm, two superimposed meshes are considered, a fixed Eulerian or ALE mesh for the fluid and a deformable Lagrangian mesh for the structure. Unlike existing algorithms that couple two separate codes, a CFD and a structure code, the fluid–structure interaction algorithm developed in the next section is fully coupled.

The main purpose of this paper is to describe the fluid–structure coupling algorithm, which computes coupling forces at the fluid structure interface. In this paper, both fluid and structure problems are solved using an explicit time integration method, which are suitable for high impact problems. For some problems, the coupling problem generates high frequency oscillations, which disturb the coupling forces. Damping forces computed from relative velocities of fluid and structure are introduced in a coupling algorithm in order to damp out high frequency oscillations. Similar damping forces have already been introduced for contact problems. In contact problems, the slave and master meshes geometrically define the contact interface, whereas in the fluid–structure coupling method developed in this paper, the fluid coupling interface is defined by the material surface.

In the first section of the paper, the governing equations for the fluid and structure are presented together with boundary conditions. In Section 2, a detailed description of the Euler–Lagrange coupling algorithm is
presented together with the description of a regular penalty contact algorithm. The damping formulation implemented in the coupling is described in the last part of this section. In Section 3, the improvements to the Euler–Lagrange coupling are validated in a simple test problem, a column of fluid compressed by an elastic piston. The fluid–structure interaction in this example is modeled using two different methods, an Euler–Lagrange coupling algorithm and a classical Lagrangian formulation using a tied contact at the fluid–structure interface. Since the piston problem is a simple one, the fluid mesh distortion is not severe and so the approach of the problem by a Lagrangian formulation is reliable. The solution of the coupling problem is compared to the Lagrangian solution, which is considered to be a reference solution. In the second part of Section 3, the asymptotical matching of the pressure presented by Zhao et al. [4] is described. The analytical pressure from Zhao et al. is compared to the numerical results obtained by the Euler–Lagrange coupling algorithm with damping. Finally, the slamming modeling is described and the numerical results are compared to existing theoretical results.

2. Description of fluid and structure problems

The fluid is solved by using an Eulerian formulation on a Cartesian grid that overlaps the structure, while the structure is discretised by a Lagrangian approach. For simplicity, the numerical simulations in this paper have been restricted to an Eulerian formulation for the fluid, although the formulation can be extended to an ALE formulation.

The Eulerian formulation is a particular case of the ALE finite element formulation. Thus a general ALE point of view is first described for Navier–Stokes equations. The structure is assumed rigid to allow for comparisons with Zhao’s theoretical solution, however, the method is not restricted to rigid structures.

2.1. General ALE description of Navier–Stokes equations

In the ALE description of motion, an arbitrary referential coordinate is introduced in addition to the Lagrangian and Eulerian coordinates. The total time derivative of a variable \( f \) with respect to a reference coordinate can be described as Eq. (1)

\[
\frac{df(\mathbf{x},t)}{dt} = \frac{\partial f(\mathbf{x},t)}{\partial t} + (\mathbf{v} - \mathbf{w}) \cdot \nabla f(\mathbf{x},t),
\]

where \( \mathbf{X} \) is the Lagrangian coordinate, \( \mathbf{x} \) is the ALE coordinate, \( \mathbf{v} \) is the particle velocity and \( \mathbf{w} \) is the velocity of the reference coordinate, which will represent the grid velocity for the numerical simulation, and the system of reference will be later the ALE grid. Thus substituting the relationship between the total time derivative and the reference configuration time derivative derives the ALE equations.

Let \( \Omega^f \in \mathbb{R}^3 \), represent the domain occupied by the fluid particles, and let \( \partial \Omega^f \) denote its boundary (Fig. 1). The equations of mass, momentum and energy conservation for a Newtonian fluid in ALE formulation in the reference domain, are given by

\[
\frac{\partial \rho}{\partial t} + \rho \text{div}(\mathbf{v}) + (\mathbf{v} - \mathbf{w})\text{grad}(\rho) = 0,
\]

\[
\rho \frac{\partial \mathbf{v}}{\partial t} + \rho(\mathbf{v} - \mathbf{w}) \cdot \text{grad}(\mathbf{v}) = \text{div}(\mathbf{\sigma}) + \mathbf{f},
\]

\[
\rho \frac{\partial e}{\partial t} + \rho(\mathbf{v} - \mathbf{w}) \cdot \text{grad}(e) = \mathbf{\sigma} : \text{grad}(\mathbf{v}) + \mathbf{f} \cdot \mathbf{v},
\]
where \( \rho \) is the density and \( \sigma \) is the total Cauchy stress given by

\[
\bar{\sigma} = -p \cdot \bar{I} d + \mu \text{grad}(\bar{v}) + \text{grad}(\bar{v})^T,
\]

where \( p \) is the pressure and \( \mu \) is the dynamic viscosity. Eqs. (2)–(4) are completed with appropriate boundary conditions. The part of the boundary at which the velocity is assumed to be specified is denoted by \( \partial \Omega_f^l \).

The inflow boundary condition is

\[
\bar{v} = \bar{g}(t) \quad \text{on } \partial \Omega_f^l.
\]

The traction boundary condition associated with Eq. (3) are the conditions on stress components. These conditions are assumed to be imposed on the remaining part of the boundary

\[
\bar{\sigma} \cdot \bar{n} = \bar{h}(t) \quad \text{on } \partial \Omega_f^l.
\]

One of the major difficulties in time integration of the ALE Navier–Stokes equations (2)–(4) is due to the nonlinear term related to the relative velocity (\( \bar{v} - \bar{w} \)). For some ALE formulations, the mesh velocity can be solved using a remeshing and smoothing process.

In the Eulerian formulation, the mesh velocity \( \bar{w} = \bar{0} \), this assumption eliminates the remeshing and smoothing process, but does not simplify the Navier–Stokes equations (2)–(4).

To solve equations (2)–(4), the split approach detailed in [5,6] and implemented in most hydrocodes is adopted in this paper.

Operator splitting is a convenient method for breaking complicated problems into series of less complicated problems. In this approach, first a Lagrangian phase is performed, using an explicit finite element method, in which the mesh moves with the fluid particle. In the CFD community, this phase is referred to as a linear Stokes problem. In this phase, the changes in velocity, pressure and internal energy due to external and internal forces are computed. The equilibrium equations for the Lagrangian phase are

\[
\rho \frac{d\bar{v}}{dt} = \text{div}(\bar{\sigma}) + \bar{f},
\]

\[
\rho \frac{de}{dt} = \bar{\sigma} : \text{grad}(\bar{v}) + \bar{f} \cdot \bar{v}.
\]

The mass conservation equation is used in its integrated form Eq. (10) rather than as a partial differential equation [7]. Although the continuity equation can be used to obtain the density in a Lagrangian formulation, it is simpler and more accurate to use the integrated form Eq. (10) in order to compute the current density \( \rho \)

\[
\rho J = \rho_0,
\]

where \( \rho_0 \) is the initial density and \( J \) is the volumetric strain given by the Jacobian:
In the second phase, called advection or transport phase, the transportation of mass, momentum and energy across element boundaries are computed. This may be thought of as remapping the displaced mesh at the Lagrangian phase back to its initial position. The transport equations for the advection phase are

\[
\frac{\partial \phi}{\partial t} + \vec{c} \cdot \text{grad}(\phi) = 0,
\]

\[
\phi(\vec{x}, 0) = \phi_0(x),
\]

where \( \vec{c} = \vec{v} - \vec{w} \) is the difference between the fluid velocity \( \vec{v} \), and the velocity of the computational domain \( \vec{w} \), which will represent the mesh velocity in the finite element formulation. In some papers \( \vec{c} \) is referred as the convective velocity.

Eq. (12) is solved successively for the conservative variables: mass, momentum and energy with initial condition \( \phi_0(x) \), which is the solution from the Lagrangian calculation of Eqs. (8) and (9) at the current time. In Eq. (12), the time \( t \) is a fictitious time: in this paper, time step is not updated when solving for the transport equation. There are different ways of splitting the Navier–Stokes problems. In some split methods, each of the Stokes problem and transport equation are solved successively for half time step.

The hyperbolic equation system (12) is solved for mass momentum and energy by using a finite volume method. Either a first-order upwind method or second-order Van Leer advection algorithm [8] can be used to solve Eq. (12).

2.2. Governing equations for structure

Let \( \Omega^s \in \mathbb{R}^3 \), the domain occupied by the structure, and let \( \partial \Omega^s \) denote its boundary (Fig. 2). An updated Lagrangian finite element formulation is considered: the movement of the structure \( \Omega^s \) described by \( x_i(t) \) \( (i = 1, 2, 3) \) can be expressed in terms of the reference coordinates \( X_a (a = 1, 2, 3) \) and time \( t \)

\[
x_i = x_i(X_a, t).
\]

The momentum equation is given by Eq. (14) in which \( \vec{\sigma} \) is the Cauchy stress, \( \rho \) is the density, \( f \) is the force density, \( \frac{dv}{dt} \) is acceleration and \( \vec{n} \) is the unit normal oriented outward at the boundary \( \partial \Omega^s \)

\[
\rho \frac{d\vec{v}}{dt} = \text{div}(\vec{\sigma}) + \vec{f}.
\]

The solution of Eq. (14) satisfies the displacement boundary condition Eq. (15) on the boundary \( \partial \Omega^s_1 \) and the traction boundary condition Eq. (16) on the boundary \( \partial \Omega^s_2 \).

\[
\vec{x}(\vec{X}, t) = \vec{D}(t) \quad \text{on} \ \partial \Omega^s_1,
\]

\[
\vec{x}(\vec{X}, t) = \vec{T}(\vec{X}) \quad \text{on} \ \partial \Omega^s_2.
\]
The following section compares two fluid–structure interaction problems: the contact problem where a contact interface is used to separate the fluid mesh from the structure mesh and the Euler–Lagrangian coupling problem where the Eulerian grid overlaps the structure mesh.

3. Fluid–structure interaction

The fluid–structure interaction problem considered in this paper can be treated using two interface algorithms. The first one is the contact formulation, where a contact interface is used to separate the fluid mesh from the structure mesh. For these problems, contact algorithms, described in detail in [9], are used to compute the contact forces applied from the fluid to the structure and conversely. For explicit methods, nodal forces at the contact interface are updated at each time step to account for contact forces. Since the fluid nodes at the contact interface move in order to remain in contact with the Lagrangian structure, an ALE method is required to remesh the fluid domain. For small fluid mesh deformations, classical ALE methods described in [10,11] can be used, but for large mesh distortion, ALE methods cannot be used for the mesh motion. To solve the problem, a rezoning or automatic remesh method is required for the fluid domain, which is CPU time consuming. This method is based on the interpolation algorithm, which is first-order accurate, non-conservative and numerically dissipative. An alternative algorithm to avoid fluid mesh distortion is to use an Euler–Lagrange coupling, an Eulerian formulation for the fluid and a Lagrangian formulation for the structure.

Using Euler–Lagrange coupling allows us to treat the impact problems involving fluids because this coupling treats the interactions between a Lagrangian formulation modeling the structure and an Eulerian formulation modeling the fluid. The Lagrangian finite element formulation uses a computational mesh that follows the material deformation. This approach is efficient and accurate for problems involving moderate deformations like structure motions or flows that are essentially smooth. When this latter departs from this kind of smoothness, the ALE formulation must be used because the finite element mesh is allowed to move independently from the material flow. In fact, ALE codes allow material to flow through the mesh and therefore the remap step in the ALE algorithm needs an advection algorithm like upwind method for first-order advection, or Van Leer or Godunov methods for higher order advection. The convection schemes available in the explicit finite element code used in this paper are the donor cell algorithm, or upwind method, for first-order, and the Van Leer algorithm [8], for second-order. In this paper, we use second-order advection to minimize dissipation and dispersion effects. If the fluid is subjected to large deformations, Lagrangian or ALE meshes are strongly distorted which jeopardizes the simulation because distorted elements have low accuracy and their stable time step sizes are small for explicit time integration algorithms. For single material or multi-material Eulerian formulation, the mesh is fixed in space and materials flow through the mesh using an advection scheme to update fluid velocity and history variables. This takes away all problems associated with distorted mesh that are commonly encountered with a Lagrangian or ALE formulation. Therefore, the Euler Lagrange coupling using Eulerian multi-material formulation for the fluid, is more suitable for solving slamming problems and more generally, fluid–structure interaction problems. First, the multi-material Eulerian formulation is able to simulate large deformations of the free surface and second, the coupling treats fluid–structure impacts. From a mechanical point of view, the coupling algorithm introduced in this paper is similar to penalty contact algorithm because the coupling method is mainly based on force equilibrium, and energy conservation. This method can be described as Eulerian contact. We first present interface conditions for contact and coupling formulations as well as the contact and coupling algorithm description. A classical impact problem is used in order to highlight the limitations.
of contact algorithm when the fluid mesh is highly distorted. The end of this section is devoted to the damping formulation introduced in the coupling algorithm.

3.1. Interface conditions

Let us assume that a fluid $\Omega^f$ and a structure $\Omega^s$ are in contact. If $d$ is the gap normal to the common interface, the following condition must be satisfied:

$$d = 0.$$  

(17)

However, the contacting surfaces in Fig. 3 are depicted separately for clarity. This sketch sums up Eqs. (2)–(7) for the fluid and the Eqs. (14)–(16) for the structure. In a fluid–structure interaction problem, two conditions applied on the interface $\partial \Omega^f_2$ common to $\partial \Omega^s_2$ and $\partial \Omega^f_2$ are added to the previous equations: the fluid–structure traction conditions and the impenetrability condition. The traction condition observes the balance of momentum across the fluid–structure interface. Since this interface has no mass, the sum of traction forces on the fluid and structure must vanish (Newton’s third law). Since a frictionless model is considered, the tangential tractions vanish too. Thus the traction condition is simply the normal traction relation:

$$\bar{\sigma}^s \cdot \bar{n}^s + \bar{\sigma}^f \cdot \bar{n}^f = 0 \quad \text{on } \partial \Omega^c_2$$  

(18)

with

$$\bar{\sigma}^s \cdot \bar{n}^s \leq 0 \quad \text{and} \quad \bar{\sigma}^f \cdot \bar{n}^f \leq 0,$$  

(19)

where $\bar{n}^s$ and $\bar{n}^f$ are normals at the contact point for $\Omega^s$ and $\Omega^f$ respectively, $\bar{\sigma}^s$ and $\bar{\sigma}^f$ are the stress fields of the fluid and structure, respectively.

The impenetrability conditions for $\Omega^f$ and $\Omega^s$ can be stated as

$$\Omega^f \cap \Omega^s = 0 \quad \text{on } \partial \Omega^c_2.$$  

(20)

It is convenient to express the condition Eq. (20) in terms of $\dot{d}$, which is the penetration rate

$$\dot{d} = \bar{v}^s \cdot \bar{n}^s + \bar{v}^f \cdot \bar{n}^f = (\bar{v}^s - \bar{v}^f) \cdot \bar{n}^s \leq 0 \quad \text{on } \partial \Omega^c_2,$$  

(21)

where $\bar{v}^s$ and $\bar{v}^f$ are the contact point velocities of the fluid and structure, respectively.

Fig. 3. Interaction between $\Omega^f$ and $\Omega^s$ (the contacting surfaces are sketched separately for clarity).
The condition Eq. (21) expresses the fact that the fluid and structure must either remain in contact \( (d = 0) \) or separate \( (d < 0) \). It may appear inconsistent to speak of a penetration rate \( \dot{d} \) when impenetrability is an important condition on the solution. However, in many numerical methods, a small amount of interpenetration is allowed: \( d < 0 \). Then, the condition Eq. (21) will not be observed with exactitude. In the penalty method, the impenetrability constraint is imposed as a penalty normal traction along the fluid–structure interface.

### 3.2. Contact algorithms for fluid–structure interaction problems

The detailed description of finite element contact algorithms is not the goal of this paper. However, the coupling method described in this paper is based on the penalty method for contact algorithms. So the contact approach is a good introduction to the coupling method.

In contact algorithms, a contact force is computed proportional to the penetration vector during the timestep. In an explicit FEM method, contact algorithms compute interface forces due to impact of the structure on the fluid (Fig. 5). These forces are applied to the fluid and structure nodes in contact in order to prevent a node from passing through contact interface. An ALE or Lagrangian mesh is used for the fluid. The literature on contact algorithms is extensive, but most of it is devoted to static problems. The literature devoted to contact for dynamic fluid–structure interaction problems is very limited. One of the problems encountered in these applications is the high mesh distortion at the contact interface due to high fluid nodal displacements and velocities. This problem is still unsolved since most of the ALE remeshing algorithms including the equipotential methods, simple and volume average, are not efficient to maintain a regular mesh for the calculation to continue see [6,12]. In contact algorithms, one surface is designated as a slave surface, and the second as a master surface. The nodes lying on both surfaces are also called slave and master nodes respectively. For a fluid–structure coupling problem, for example fluid with an initial velocity impacting a structure, as described in Fig. 5, the fluid nodes at the interface are considered as slave, and the structure elements as master. The first approach for contact is the kinematic contact, constraining fluid and structure nodes to the same velocity. Kinematic contact conserves total momentum, but not total energy. The second approach, the penalty contact, is different from the previous one. The penalty method imposes a resisting force to the slave node, proportional to its penetration through the master element. This force is applied to both the slave node and the nodes of the master element in opposite directions to satisfy equilibrium: the force applied to the nodes of the master element are scaled by the shape functions

\[
F_s = -k \cdot d, \\
F_m^i = N_i \cdot k \cdot d, \\
\text{where } N_i \text{ is the shape function at node } i (i = 1, 2 \text{ in two dimensions, and } i = 1, \ldots, 4 \text{ in three dimensions}) \text{ of the master element surrounding the slave node location, and } d \text{ the penetration vector. In case the slave node coincides exactly with one of the master node (see Fig. 4), node 1 for instance, we will have}
\]

\[
F_s = -k \cdot d, \\
F_m^1 = k \cdot d \quad \text{and} \quad F_m^2 = F_m^3 = F_m^4 = 0. \\
\text{(23)}
\]

The coefficient \( k \) represents the stiffness of a spring. In fact, this method consists in placing springs between all penetrating nodes and the contact surface. The spring stiffness is given by Eq. (24) in terms of the bulk modulus \( K \) of the master material, \( V \) the volume of the master element and \( A \) the area of the master segment

\[
k = \frac{p_t K A}{V}, \\
\text{(24)}
\]
where $p_f$ is scale factor for the interface stiffness, which satisfies $0 \leq p_f \leq 1$. Nevertheless, the contact algorithms have a drawback: for large deformations of materials, the mesh distortion is important. To illustrate the contact algorithm in fluid–structure interaction problem, we consider a volume of fluid with initial velocity $\vec{V}$, impacting an elastic structure, modeled with shell elements, Belytschko type shell, described in detail in [13,14]. Mesh distortion appears on Figs. 6 and 7 at $t = 0.15$ ms and $t = 0.5$ ms. At the earlier time $t = 0.15$ ms, we have small mesh deformations and the time step controlling the explicit calculation does not decrease significantly. But later in time, at 0.5 ms, large mesh distortion causes the time step to decrease, and the accuracy decreases due to highly distorted elements at the contact interface. To prevent high mesh distortion, an explicit Euler–Lagrange coupling method has been developed to compute coupling forces at the fluid–structure interface nodes.
3.3. Euler–Lagrange coupling

For simplicity, we assume the fluid is solved in an Eulerian fixed mesh, although there is no restriction in the coupling formulation to an Eulerian mesh. The method has been extended and used for general ALE problems. In the Euler–Lagrange coupling, the structure is embedded in an Eulerian fixed mesh, as shown in Fig. 9 and Fig. 10. Since the mesh is fixed and the fluid material flows through the mesh, a larger Eulerian mesh than the physical fluid mesh is required. When fluid material leaves the mesh through the element faces, it will occupy adjacent elements that are initially voided or air material elements. During material transport or advection, the interface elements are partially voided, thus the volume fraction $f$ of these elements satisfies $0 < f < 1$.

To track the material interface in elements, the Young method is used. In this method, the material layout is described solely by the volume fraction of the fluid material in the element. Specifically, a straight line using the SLIC technique (simple linear interface calculation) of Woodward and Colella [15] approximates the interface in the cell described on Fig. 8. A detailed multi-material and VOF (volume of fluid) formulation is described in [16], and [17] for sloshing problems. This latter paper describes in detail the split method used for ALE multi-material and single material formulations but coupling with structure is not involved. It is the aim of this paper, however, to describe the coupling and damping algorithms to solve fluid–structure interaction problems.

In an explicit time integration problem, the main part of the procedure in the time step is the calculation of the nodal forces. After computation of fluid and structure nodal forces, we compute the forces due to the coupling, these will only affect nodes that are on the fluid structure interface. For each structure node, a
depth penetration $\bar{d}$ is incrementally updated at each time step, using the relative velocity $(\bar{v}_s - \bar{v}_f)$ at the structure node, which is considered as a slave node, and the master node within the Eulerian element. The location of the master node is computed using the isoparametric coordinates of the fluid element. At time $t = t^n$ (see Fig. 9) $\bar{d}$, the penetration depth represents the amount the interface condition is violated, it is updated incrementally in Eq. (25):

$$\bar{d}^{n+1} = \bar{d}^n + (\bar{v}_s^{n+1/2} - \bar{v}_f^{n+1/2}) \cdot \Delta t.$$ (25)

The fluid velocity $\bar{v}_f$ is the velocity at the master node location, interpolated from the nodes of the fluid element at the current time. For this coupling, the slave node is a structure mesh node, whereas the master node is not a fluid mesh node, it can be viewed as a fluid particle within a fluid element, with mass and velocity interpolated from the fluid element nodes using finite element shape functions. The vector $\bar{d}^n$ represents the penetration depth of the structure inside the fluid during the time step, which is the amount the constraint is violated. The coupling force acts only if penetration occurs, $\bar{n}_s \cdot \bar{d} < 0$, where $\bar{n}_s$ is built up by averaging normals of structure elements connected to the structure node. For clarity, the superscript of the penetration has been omitted, we will use $d$ instead of $\bar{d}^n$, for the penetration vector.

Penalty coupling behaves like a spring system and penalty forces are calculated proportionally to the penetration depth and spring stiffness. The head of the spring is attached to the structure or slave node and the tail of the spring is attached to the master node within a fluid element that is intercepted by the structure, as illustrated in Fig. 9.

Similarly to penalty contact algorithm, the coupling force is described by

$$F = k \cdot d,$$ (26)

where $k$ represents the spring stiffness, and $d$ the penetration. The force $F$ in Eq. (26) is applied to both master and slave nodes in opposite directions to satisfy force equilibrium at the interface coupling, and thus the coupling is consistent with the fluid–structure interface conditions namely the action–reaction principle. At the structure coupling node, we applied a force

$$F_s = -F,$$ (27)

whereas for the fluid, the coupling force is distributed to the fluid element nodes based on the shape functions, at each node $i$ ($i = 1, \ldots, 8$), the fluid force is scaled by the shape function $N_i$. 

Fig. 9. Sketch of the coupling algorithm.
where \( N_i \) is the shape function at node \( i \). Since \( \sum_{i=1}^{h} F^i = F_c \), the action–reaction principle is satisfied at the coupling interface. The coupling force satisfies force equilibrium at the fluid structure interface and reduces fluid penetration into the structure.

In contrast to the constrained method, where the interface condition is imposed and no interpenetration allowed, the penalty method allows some interpenetration. However, coupling forces proportional to the magnitude of the penetrations are applied to keep the penetration small. In the limit, defining a very high penalty stiffness, the penetrations approach zero, satisfying the interface condition. However, the coupling interface adds stiffness to the system affecting its eigenfrequency spectra. Hence, for numerical stability reasons, at a given time step size, there is an upper bound for the maximum possible penalty stiffness coefficient.

The main difficulty in the coupling problem comes from the evaluation of the stiffness coefficient \( k \) in Eq. (26). The stiffness value is problem dependent, a good value for the stiffness should reduce energy interface in order to satisfy total energy conservation, and prevent fluid leakage through the structure. The value of the stiffness \( k \) is still a research topic for explicit contact-impact algorithms in structural mechanics. For fluid structure coupling, the spring stiffness is deduced from explicit contact with penalty method. This value is difficult to obtain unless numerical experiments are run systematically. For some industrial problems, automotive industry for instance, experimental tests are done prior to analysis to evaluate the stiffness value for penalty contact algorithm.

In this paper, the stiffness \( k \) is based on stiffness used in explicit contact algorithms in [9]. \( k \) is given in term of the bulk modulus \( K \) of the fluid element in the coupling containing the slave structure node, the volume \( V \) of the fluid element that contains the master fluid node, and the average area \( A \) of the structure elements connected to the structure node.

\[
k = p_f \frac{K A^2}{V}.
\]

However, to avoid numerical instabilities, a scalar factor \( p_f \), \( 0 \leq p_f \leq 1 \), is introduced as in the contact method. For impact problems, we always have to examine the influence of this parameter on the solution. In impact problems, after contact, the master and slave nodes may separate: this phenomenon is known as release. If the penalty force is very large, the impact and the release may occur in the same time step involving a numerical instability. To avoid this anomaly, the force \( F \) in Eq. (26) can be bounded by the contact force between two spheres, defined by Belytschko and Neal [18], which is given by Eq. (30)

\[
F \leq \frac{M_s M_m (\vec{v}_s - \vec{v}_t)}{\Delta t (M_s + M_m)},
\]

\[
M_m = \sum_{i=1}^{8} N_i \cdot M_m^i,
\]

where \( M_s \) is the mass of the slave or structure node, \( M_m \) is the mass of the master fluid node interpolated from the fluid element nodes (see Eq. (31)), and \( (\vec{v}_s - \vec{v}_t) \) is the relative velocity defined in Eq. (25), \( \Delta t \) is the current time step.

As mentioned in the previous section, the coupling algorithm can be used for problems involving large mesh distortion that contact algorithm cannot handle. To illustrate the ability of the coupling for solving problems where contact algorithms fail because of high mesh distortion, we consider an impact problem described in the previous section (Figs. 5–7) that we run long enough in time for the fluid mesh to get highly distorted. In this case the problem is treated using a multi-material or VOF formulation for the fluid, and
by applying the coupling forces defined by Eq. (26) at the fluid–structure interface. The problem is illustrated in Figs. 10–12. As shown in Figs. 11 and 12, the fluid material flows through a fixed mesh, and the coupling interface between the fluid and structure is no longer a fluid boundary interface defined by the mesh as in a contact algorithm, but is a material surface defined by fluid elements that are partially filled. The impact at time \( t = 0.5 \) ms showed on Figs. 7 and 12 is modeled by two algorithms, coupling

---

**Fig. 10.** Coupling between Lagrangian and Eulerian mesh, time = 0 ms.

**Fig. 11.** Coupling between Lagrangian and Eulerian mesh, time = 0.15 ms.

**Fig. 12.** Coupling between Lagrangian and Eulerian mesh, time = 0.5 ms.
and the contact algorithms, respectively. In the contact algorithm in Fig. 7, the fluid undergoes high mesh distortion, which makes it difficult for the problem to continue, since time step decreases. For the coupling algorithm on Fig. 12, fluid mesh distortion is no longer a problem. However, problems related to fluid leakage through the structure may occur for high velocity impact problems. In such cases the fluid particle penetrates so deep within the structure that the coupling force in Eq. (26), is not large enough to return it to the coupling interface. For general problems, one solution to this problem is to reduce the time step; but for some problems involving highly compressible gas, the expression Eq. (26) needs to be modified: the spring system is nonlinear and the stiffness $K$ should be penetration dependent

$$F = k(d) \cdot d.$$  \hspace{30pt} (32)

Fluid leakage through the structure is a very difficult problem to solve, it is not the aim of this paper to describe the different numerical algorithms to prevent leakage. Most of the research in fluid structure interaction has focused on developing numerical algorithms that prevent leakage and conserve energy.

### 3.4. Damping in the coupling

In general, the numerical dissipation in the numerical scheme is a problem and its limitation is mandatory to respect the physical phenomenon. For instance, Piperno et al. [19] studied the numerical damping errors due to discrete fluid–structure coupling in order to enhance the numerical scheme by compensating these errors. Several methods can be considered to obtain a filtered curve. Some codes of numerical simulation use numerical implicit schemes with integration parameters, which can be adjusted to achieve numerical accuracy and stability. The classical Newmark scheme, second-order and stable scheme can filter out numerical noise. Nevertheless, the numerical damping introduced in the Newmark method brings the order of accuracy to a lower level. The HHT Method developed by Hilber et al. [20] improves the Newmark approach by introducing the $\alpha$-method. In this paper, a viscous coupling is added to the coupling to damp out high frequency oscillations, but a better way could be found to stabilize the numerical scheme. For the penalty coupling algorithm, since the finite element software used in this paper is an explicit code, a damping force is added to the penalty force to have a smoother response and consequently better convergence characteristics. In order to prevent damping force from altering the physics of the problem and give rise to an inconsistent scheme, an optimal damping is used. In fact, the process is a classical one: the spring system described for the contact and coupling force, is completed by a damper as shown on the illustration Fig. 13 for contact and Fig. 14 for coupling.

Let us note $k$, the spring stiffness, $C$, the damping coefficient, $m_{\text{structure}}$ the structure nodal mass and $m_{\text{fluid}}$ the mass of the fluid node or particle. If physical stiffness is small compared to $k$, the coupling and contact

![Fig. 13. Contact algorithm with a dashpot.](image-url)
algorithm with damping enhancement can be represented by the sketch 1 of Fig. 15. This scheme is equivalent to the sketch 2 with $M$ the equivalent mass.

The equilibrium of the inertia force $M \cdot \frac{d^2}{dt^2} d$, the damping force $C \cdot \frac{d}{dt} d$ and the stiffness force $k \cdot d$ satisfy Eq. (33)

$$M \cdot \frac{d^2}{dt^2} d + C \cdot \frac{d}{dt} d + k \cdot d = 0$$

where $M = \frac{m_{\text{structure}} + m_{\text{fluid}}}{m_{\text{structure}}}$ and $C = \ddot{\zeta} \cdot \sqrt{k \cdot M}$ with $\ddot{\zeta}$, the damping factor. Eq. (33) can be written again as

$$\frac{d^2}{dt^2} d + \ddot{\zeta} \omega \cdot \frac{d}{dt} d + \omega^2 \cdot d = 0$$

with the pulsation

$$\omega = \sqrt{\frac{k \cdot m_{\text{structure}} + m_{\text{fluid}}}{m_{\text{structure}} \cdot m_{\text{fluid}}}}.$$ 

Fig. 14. Coupling algorithm with a dashpot.

Fig. 15. Schemes of the penalty method with damping.
The damping is optimal when $\zeta$ is critical. For $\zeta = 2$, the numerical oscillations must be damped completely out. The main drawback of this approach is a possible alteration of the physical aspects if the numerical stiffness $k$ is close to the highest physical stiffness: physical oscillations may be damped out. However, the numerical stiffness is generally large compared to the physical stiffness as shown in the following applications.

4. Numerical applications

4.1. Piston problem

In order to highlight numerical applications of the algorithms evoked in the previous section, we consider the following simple problem: a column of water 0.5 m high is compressed by a piston moving with constant velocity of 10 m/s. Sliding conditions are applied along the walls of the cavity. A Newtonian material model with constant viscosity is used for the water and the equation of state is

$$\Delta P = -\rho c \text{tr}(\dot{\varepsilon}) \Delta t, \quad (36)$$

where $\Delta P$ is the incremental pressure, $\Delta t$ the time step, $\rho$ the density, $c$ the sound velocity and $\dot{\varepsilon}$ the rate of deformation. The bulk modulus, $K = \rho c^2 = 2.25 \text{ GPa}$, is important enough to consider water as slightly compressible. A rigid material model is used for the structure of the piston.

The simple problem can be solved using a classical Lagrangian formulation for the fluid and structure, and shared nodes at the interface with no contact involvement. The fluid mesh remains regular during fluid compression, pressure and forces at the fluid interface can be considered as reference data. The problem is illustrated in Fig. 16, which shows the configuration at initial time. At time $t = 2.5 \text{ ms}$, $5 \text{ ms}$, and $7.5 \text{ ms}$, we keep integrity and regularity of the fluid mesh, the mesh is linearly compressed and mesh regularity is satisfied. In Fig. 17, an Euler–Lagrangian coupling method is used for the same problem. Due to fluid incompressibility, and the nature of the penalty coupling, the coupling method generates high oscillations for interface forces and pressure time history as shown in Fig. 18. The damping force, proportional to relative velocity of the fluid and structure, is added to the coupling force for the evaluation of nodal forces. The damping force developed in the previous section is used to damp out high frequencies. For fluid–structure problems, where the fluid is highly compressible, airbag inflation for instance, pressure or interface force time history are not affected by high oscillations, and no damping force is required.

The equivalent stiffness of the fluid in the piston is determined by Eq. (36):

$$\Delta P = -K \text{tr}(\dot{\varepsilon}) \Delta t = -K \frac{\Delta \omega_f}{\omega_f} = -K \frac{\Delta D}{D}, \quad (37)$$

Fig. 16. Piston problem with shared nodes.
where $\Omega_f$ is the fluid cavity volume, $D = 0.5$ m is the depth, $\Delta D$ is the incremental displacement of the piston. The equivalent stiffness by area unit is

$$\frac{K_D}{D} = \frac{\rho c^2}{D} = 4.5 \text{ GPa m}^{-1}.$$  \hfill (38)

The numerical stiffness by area unit is given by Eq. (29) for $p_f = 1$ and the mesh size $dx = \frac{V}{A} = \frac{D}{x}$ according to Fig. 17

$$\frac{k}{A} = p_f \frac{KA}{V} = \frac{K}{dx} = \frac{8K}{D} = 36 \text{ GPa m}^{-1}.$$  \hfill (39)

Thus the numerical stiffness is sufficiently large compared to the physical stiffness.

4.2. Two-dimensional slamming modeling

In order to highlight the capability of the coupling algorithm described in this paper, we consider the two-dimensional impact problem described in Fig. 19. A rigid wedge impacting water with a deadrise angle of 10 degrees, what is the angle between the water free surface and the wedge. Unlike the piston problem, the contact algorithm fails when solving this problem, due to high mesh distortion for the fluid.
This problem has been selected for the following three reasons:

First there are several papers previously published in literature treating the theoretical approach solving for the pressure applied from the fluid on the structure. For details, we can refer to the work by Zhao et al. [4]. A development of Zhao’s theory is used for a two-dimensional rigid wedge with small deadrise angle. The theoretical approach of the problem is presented next. The numerical results for the slamming problem are compared to the theoretical approach. Second, unlike the piston problem, this problem cannot be solved using a Lagrangian formulation for the fluid. Using a Lagrangian method for the fluid, the fluid mesh will undergo high distortion after few time steps, which causes the time step to decrease. To solve the problem, the coupling method where the fluid is solved with Eulerian formulation and the structure with a Lagrangian formulation, is required. Third, the fluid is incompressible, the penalty coupling using the coupling force based on the spring system, will generate high oscillations, and thus damping forces are required to smooth out high frequencies and reduce the noise in the solution.

4.2.1. Theoretical approach

The two-dimensional slamming problem is approached in the framework of simplifying hypothesis. The incidence between the structure and free surface is small but no air cushioning is created during the impact. Two-dimensional, irrotational, incompressible and inviscid flow is assumed. The wedge is rigid with a deadrise angle of 10° penetrating the calm free surface with a constant velocity. No gravitational acceleration is assumed. The flow is divided into an inner and outer flow field (see Fig. 20). The inner flow domain around \( x = c(t) \) is the splash region and the outer flow region for \( |x| < c(t) \) is the doublet potential flow. Zhao et al. [4] generalized the Wagner’s theory by matching the flow field around a steady planning plate at an attack angle \( \alpha \) (here \( \alpha = 10^\circ \)) for the inner solution and the flow field around the expanding flat plate for the outer solution. Thus, they created a composite solution for the pressure distribution, which is compared with the numerical results from the Euler–Lagrange coupling algorithm.

In the outer region, the hypothesis makes it possible to use the complex potential theory to solve the problem and the complex potential of the outer flow is defined by \( \phi \). An expanding flat plate of half-width, \( c(t) \), represents the wedge. The half-wetted body length \( c(t) \) is given by:

\[
c(t) = \frac{\pi Vt}{2 \cdot \tan(\alpha)}.
\]
The submergence of the wedge, \( V_t \), is assumed much smaller than the expanding half-width \( c(t) \). Thus, an asymptotic approach permits to neglect the quadratic terms compared to the first-order terms. The impact pressure in the outer region is determined from Bernoulli’s equation. In the inner region (see Fig. 20), a jet flow is assumed near the singularity \( c(t) \) by using the Schwarz–Christoffel method. A composite pressure distribution given by Eq. (41) is provided by matching the jet solution in the inner region to the expanding plate solution in the outer region

\[
\frac{p - p_0}{\rho} = \frac{V}{\sqrt{c^2 - x^2}} \frac{dc}{dr} + 2 \left( \frac{dc}{dr} \right)^2 |\tau|^{1/2} \left( 1 + |\tau|^{1/2} \right)^{-2} - V c \left( \frac{dc}{dr} \right) [2c(c - x)]^{-1/2},
\]

where the relationship Eq. (42) between the intermediate variable \(|\tau|\) and the physical variable \(x\) is given by the Schwarz–Christoffel method

\[
x - c = \frac{\delta}{\pi} (- \ln |\tau| - 4 |\tau|^2 - |\tau| + 5)
\]

\(\delta\), the jet thickness obtained by matching the inner and outer solutions, is equal to \( \frac{2\sqrt{y} \left( \frac{dy}{dr} \right)^2}{8} \). When the parameter \(|\tau|\) tends to 0, 1 and \(\infty\), \(x\) tends to \(+\infty\), \(c(t)\) and \(-\infty\), respectively. This composite solution will be used for comparisons with the numerical results given by the following slamming modeling.

4.2.2. Numerical approach of the two-dimensional slamming problem

The model represented by an Eulerian formulation with 8-node brick elements (see Fig. 21) is considered. The grid is sandwiched between both planes of symmetry orthogonal to \(z\)-axis to obtain a two-dimensional
model. A third vertical plane of symmetry orthogonal to x-axis goes through the apex of the wedge to reduce the number of finite elements.

Transmitting boundary conditions are applied to the three other Eulerian boundaries. The wedge is modeled by three jointed strips consisting of Lagrangian shell elements. The position of the three flat plates is presented in Fig. 21. The first one is inclined by an angle of 10 degrees, the second one is vertical and the last one is horizontal. The dimensions of the structure are shown in Fig. 21. The sizes of Eulerian meshes are roughly about $dx = 2.5\, \text{mm}$ and are twice as large as the Lagrangian meshes. It penetrates the calm water with a constant velocity of $V_0 = 5.425\, \text{m/s}$.

4.2.3. Numerical results

The numerical results presented in this section are obtained with the Euler–Lagrange penalty coupling with damping. In the simulation, the value of the local peak pressure on the structure is very sensitive to the scalar factor $pf$ used in Eq. (29) to compute the coupling force. However, the impulse and resultant force on the structure are not sensitive to the variation of the scalar factor, since they are average values on the structure. In order to get a good correlation for the local peak pressure with theoretical results, we set $pf = 0.1$. For this value and the mesh size $dx = 2.5\, \text{mm}$, the numerical stiffness by unit area is computed by Eq. (29)

$$\frac{k}{A} = pf \frac{KA}{V} = pf \frac{K}{dx} = 90 \, \text{GPa m}^{-1}. \quad (43)$$

The maximal local physical stiffness by unit area can be evaluated when the pressure peak on Fig. 22 is reached. The theoretical curve on this graph shows the overpressure is about 1.2 MPa and occurs 24 $\mu$s after the birth time of the peak. Thus, during this time interval, the wedge penetrates 0.13 mm into the free surface. The ratio of the overpressure 1.2 MPa and the penetration 0.13 mm estimates the equivalent local stiffness by unit area: 9.2 GPa m$^{-1}$. The numerical stiffness by unit area is one order of magnitude larger than the maximal physical one. Thus the numerical stiffness is sufficiently large compared to the physical stiffness.

The numerical results are the histories of local pressures at 40 mm from the apex and the vertical force on the wedge. These curves are showed on Figs. 22 and 24, respectively. On these graphs, the numerical pressure curves are superimposed with the corresponding theoretical results. For the local pressure at 40 mm, the penalty coupling is used with and without damping force. The superimposition of both cases is presented on Fig. 23. The curves of Figs. 22 and 24 show good correlations. On Fig. 22, the relative error
on the peak pressure is about 3% because of the calibration of numerical stiffness. Finding the right numerical stiffness for this problem remains an open problem. In this paper, the main purpose is to eliminate the numerical oscillations perturbing local pressure histories. The graph on Fig. 24 shows both curves have similar evolutions. Thus, the numerical data are close to the theoretical results. The slamming modeling is in close agreement with the physical phenomenon results. To highlight the effect of damping on the pressure signal, the graph of Fig. 23 is built by superimposing the pressure at 40 mm from the apex for a penalty coupling using the damping and for a penalty coupling without damping (the former coupling). The comparison of both curves shows an effective damping of oscillations after the pressure peak. The damping contribution introduced in the penalty coupling improves the numerical results by eliminating high frequency modes due to high incompressibility of the fluid.

Fig. 22. Pressure at 40 mm from the apex: (○) theoretical and (∗) numerical.

Fig. 23. Comparisons of numerical pressure at 40 mm from the apex: (○) without damping and (∗) with damping.
5. Conclusion

In the slamming modeling, the model has been restricted to rigid body, since for water impact problem, theoretical solution is available only for rigid structure. In the description of the method, there is no restriction to rigid bodies, the method can be applied to general structures. The slamming modeling with the Euler–Lagrange coupling showed numerical oscillation problems. More specifically, for times close to the instant of pressure peak, “numerical noise” produced by the penalty spring perturbed the investigated curve after the peak instant. To treat this problem, a damping algorithm has been added to the penalty algorithm in order to obtain smooth pressures. In the sample modeling of the piston, the new penalty coupling has given good results by damping high oscillations. The comparison between the results obtained for the Euler–Lagrange coupling with and without damping after penalty stiffness calibration has highlighted the positive effect of damping: the added damping option in the coupling code permits to find the theoretical results given by Wagner’s approach without numerical dissipative errors. The future investigations will focus on the determination of the coupling stiffness. The challenging problem, which remains open, is to predict the value of this numerical parameter for a large range of fluid–structure interaction problems. The coupling method presented in this paper has been successfully used for different applications where classical contact methods fail or cannot be used, including airbag deployment in automotive industry, and bird impact in aerospace industry. There are, however, still class of problems where neither method, contact nor coupling, appears adequate.

References


Fig. 24. The vertical force per unit length on the wedge: (o) theoretical and (x) numerical.


