Anexo 1

Desarrollos del Método de Elementos Finitos y Partículas (PFEM) para estudio del movimiento de objetos bajo acción de las olas. Aplicación al estudio de la estabilidad de diques
POSSIBILITIES OF THE PARTICLE FINITE ELEMENT METHOD FOR COMPLEX COUPLED PROBLEMS IN FLUID AND SOLID MECHANICS

Eugenio Oñate, Sergio R. Idelsohn, Miguel A. Celigueta, Riccardo Rossi and Salvador Latorre

International Center for Numerical Methods in Engineering (CIMNE)
Universidad Politecnica de Cataluna
Campus Norte UPC, 08034 Barcelona, Spain
Email: onate@cimne.upc.edu
Web page: http://www.cimne.com

Abstract. We present some developments in the formulation of the Particle Finite Element Method (PFEM) for analysis of complex coupled problems in fluid and solid mechanics accounting for fluid-structure interaction and coupled thermal effects. The PFEM uses an updated Lagrangian description to model the motion of nodes (particles) in both the fluid and the structure domains. Nodes are viewed as material points which can freely move and even separate from the main analysis domain representing, for instance, the effect of water drops. A mesh connects the nodes defining the discretized domain where the governing equations are solved as in the standard FEM. The necessary stabilization for dealing with the incompressibility of the fluid is introduced via the finite calculus (FIC) method. An incremental iterative scheme for the solution of the non linear transient coupled fluid-structure problem is described. Extensions of the PFEM to allow for frictional contact conditions at fluid-solid and solid-solid interfaces via mesh generation are described. A simple algorithm to treat erosion in the fluid bed is presented. Examples of application of the PFEM to solve a number of coupled problems such as the effect of large wave on structures, the large motions of floating and submerged bodies, bed erosion situations and melting and dripping of polymers under the effect of fire are given.

Key words: Lagrangian formulation, fluid-structure, particle finite element method

1 INTRODUCTION

The analysis of problems involving the interaction of fluids and structures accounting for large motions of the fluid free surface and the existence of fully or partially submerged bodies which interact among themselves is of big relevance in many areas of engineering. Examples are common in ship hydrodynamics, off-shore and harbour structures, spill-ways in dams, free surface channel flows, environmental flows, liquid containers, stirring reactors, mould filling processes, etc.
Typical difficulties of fluid-multibody interaction analysis in free surface flows using the FEM with both the Eulerian and ALE formulation include the treatment of the convective terms and the incompressibility constraint in the fluid equations, the modelling and tracking of the free surface in the fluid, the transfer of information between the fluid and the moving solid domains via the contact interfaces, the modeling of wave splashing, the possibility to deal with large motions of the bodies within the fluid domain, the efficient updating of the finite element meshes for both the structure and the fluid, etc. For a comprehensive list of references in FEM for fluid flow problems see [7, 37] and the references there included. A survey of recent works in fluid-structure interaction analysis can be found in [18], [27] and [35].

Most of the above problems disappear if a Lagrangian description is used to formulate the governing equations of both the solid and the fluid domains. In the Lagrangian formulation the motion of the individual particles are followed and, consequently, nodes in a finite element mesh can be viewed as moving material points (hereforth called “particles”). Hence, the motion of the mesh discretizing the total domain (including both the fluid and solid parts) is followed during the transient solution.

The authors have successfully developed in previous works a particular class of Lagrangian formulation for solving problems involving complex interaction between fluids and solids. The method, called the particle finite element method (PFEM, www.cimne.com/pfem), treats the mesh nodes in the fluid and solid domains as particles which can freely move and even separate from the main fluid domain representing, for instance, the effect of water drops. A mesh connects the nodes discretizing the domain where the governing equations are solved using a stabilized FEM.

The FEM solution of the variables in the (incompressible) fluid domain implies solving the momentum and incompressibility equations. This is not such as simple problem as the incompressibility condition limits the choice of the FE approximations for the velocity and pressure to overcome the well known div-stability condition [7, 37]. In our work we use a stabilized mixed FEM based on the Finite Calculus (FIC) approach which allows for a linear approximation for the velocity and pressure variables.

An advantage of the Lagrangian formulation is that the convective terms disappear from the fluid equations. The difficulty is however transferred to the problem of adequately (and efficiently) moving the mesh nodes. We use a mesh regeneration procedure blending elements of different shapes using an extended Delaunay tesselation with special shape functions [11, 13]. The theory and applications of the PFEM are reported in [2, 6, 11, 12, 14, 15, 26, 27, 28, 30, 31, 32].

The aim of this paper is to describe recent advances of the PFEM for a) the analysis of the interaction between a collection of bodies which are floating and/or submerged in the fluid, b) the modeling of bed erosion in open channel flows and c) the analysis of melting and dripping of polymer objects in fire situations. These problems are of great relevance in many areas of civil, marine and naval engineering, among others. It is shown that the PFEM provides a general analysis methodology for treat such a complex problems in a simple and efficient manner.
The layout of the paper is the following. In the next section the key ideas of the PFEM are outlined. Next the basic equations for an incompressible thermal flow using a Lagrangian description and the FIC formulation are presented. Then an algorithm for the transient solution is briefly described. The treatment of the coupled FSI problem and the methods for mesh generation and for identification of the free surface nodes are outlined. The procedure for treating at mesh generation level the contact conditions at fluid-wall interfaces and the frictional contact interaction between moving solids is explained. A methodology for modeling bed erosion due to fluid forces is described. Finally, the potential of the PFEM is shown in its application to problems involving large flow motions, surface waves, moving bodies in water, bed erosion and melting and dripping of polymers in fire situations.

2 THE BASIS OF THE PARTICLE FINITE ELEMENT METHOD

Let us consider a domain containing both fluid and solid subdomains. The moving fluid particles interact with the solid boundaries thereby inducing the deformation of the solid which in turn affects the flow motion and, therefore, the problem is fully coupled.

In the PFEM both the fluid and the solid domains are modelled using an updated Lagrangian formulation. That is, all variables in the fluid and solid domains are assumed to be known in the current configuration at time \( t \). The new set of variables in both domains are sought for in the next or updated configuration at time \( t + \Delta t \) (Figure 1). The finite element method (FEM) is used to solve the continuum equations in both domains. Hence a mesh discretizing these domains must be generated in order to solve the governing equations for both the fluid and solid problems in the standard FEM fashion. Recall that the nodes discretizing the fluid and solid domains are treated as material particles which motion is tracked during the transient solution. This is useful to model the separation of fluid particles from the main fluid domain in a splashing wave, or soil particles in a bed erosion problem, and to follow their subsequent motion as individual particles with a known density, an initial acceleration and velocity and subject to gravity forces. The mass of a given domain is obtained by integrating the density at the different material points over the domain.

The quality of the numerical solution depends on the discretization chosen as in the standard FEM. Adaptive mesh refinement techniques can be used to improve the solution in zones where large motions of the fluid or the structure occur.

2.1 Basic steps of the PFEM

For clarity purposes we will define the collection or cloud of nodes (C) pertaining to the fluid and solid domains, the volume (V) defining the analysis domain for the fluid and the solid and the mesh (M) discretizing both domains.

A typical solution with the PFEM involves the following steps.

1. The starting point at each time step is the cloud of points in the fluid and solid domains. For instance \(^nC\) denotes the cloud at time \( t = t_n \) (Figure 2).
2. Identify the boundaries for both the fluid and solid domains defining the analysis domain \( V \) in the fluid and the solid. This is an essential step as some boundaries (such as the free surface in fluids) may be severely distorted during the solution, including separation and re-entering of nodes. The Alpha Shape method [8] is used for the boundary definition (Section 5).

3. Discretize the fluid and solid domains with a finite element mesh \( M \). In our work we use an innovative mesh generation scheme based on the extended Delaunay tesselation (Section 4) [11, 12, 14].

4. Solve the coupled Lagrangian equations of motion for the fluid and the solid domains. Compute the state variables in both domains at the next (updated) configuration for \( t + \Delta t \): velocities, pressure, viscous stresses and temperature in the fluid and displacements, stresses, strains and temperature in the solid.

5. Move the mesh nodes to a new position \( n + 1 \) where \( n + 1 \) denotes the time \( t_n + \Delta t \), in terms of the time increment size. This step is typically a consequence of the solution process of step 4.

6. Go back to step 1 and repeat the solution process for the next time step to obtain \( n + 2 \). The process is shown in Figure 2.

Figure 3 shows another conceptual example of application of the PFEM to model the melting and dripping of a polymer object under a heat source acting at a boundary.
3 FIC/FEM FORMULATION FOR A LAGRANGIAN INCOMPRESSIBLE THERMAL FLUID

3.1 Governing equations

The key equations to be solved in the incompressible thermal flow problem, written in the Lagrangian frame of reference, are the following:

Momentum

\[ \rho \frac{\partial v_i}{\partial t} = \frac{\partial \sigma_{ij}}{\partial x_j} + b_i \quad \text{in } \Omega \]  

(1)

Mass balance

\[ \frac{\partial v_i}{\partial x_i} = 0 \quad \text{in } \Omega \]  

(2)
Figure 3: Sequence of steps to update in time a “cloud” of nodes representing a polymer object under thermal loads represented by a prescribed boundary heat flux $q$ using the PFEM. Crossed circles denote prescribed nodes at the boundary.

Heat transport

$$\rho c \frac{\partial T}{\partial t} = \frac{\partial}{\partial x_i} \left( k_i \frac{\partial T}{\partial x_i} \right) + Q \quad \text{in } \Omega \quad (3)$$

In above equations $v_i$ is the velocity along the $i$th global (cartesian) axis, $T$ is the temperature, $\rho$, $c$, and $k_i$ are the density (assumed constant), the specific heat and the conductivity of the material along the $i$th coordinate direction, respectively, $b_i$ and $Q$ are the body forces and the heat source per unit mass, respectively and $\sigma_{ij}$ are the (Cauchy) stresses related to the velocities by the standard constitutive equation (for incompressible Newtonian material)

$$\sigma_{ij} = s_{ij} - p\delta_{ij} \quad (4a)$$
Figure 4: Breakage of a water column. (a) Discretization of the fluid domain and the solid walls. Boundary nodes are marked with circles. (b) and (c) Mesh in the fluid domain at two different times.

\[ s_{ij} = 2\mu \left( \dot{\varepsilon}_{ij} - \frac{1}{3} \delta_{ij} \dot{\varepsilon}_{ii} \right) , \quad \dot{\varepsilon}_{ij} = \frac{1}{2} \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) \] (4b)

In Eqs.(4), \( s_{ij} \) is the deviatoric stresses, \( p \) is the pressure (assumed to be positive in compression), \( \dot{\varepsilon}_{ij} \) is the rate of deformation, \( \mu \) is the viscosity and \( \delta_{ij} \) is the Kronecker delta. In the following we will assume the viscosity \( \mu \) to be a known function of temperature, i.e \( \mu = \mu(T) \).

Indexes in Eqs.(1)–(4) range from \( i, j = 1, n_d \), where \( n_d \) is the number of space dimensions of the problem (i.e. \( n_d = 2 \) for two-dimensional problems).

Eqs.(1)–(4) are completed with the standard boundary conditions of prescribed velocities and surface tractions in the mechanical problem and prescribed temperature and prescribed normal heat flux in the thermal problem [2, 7].

We note that Eqs.(1)–(3) are the standard ones for modeling the deformation of viscoplastic materials using the so called “flow approach” [38, 39]. In our work the dependence of the viscosity with the strain typical of viscoplastic flows has been simplified to the Newtonian form of Eq.(4b).

3.2 Discretization of the equations

A key problem in the numerical solution of Eqs.(1)–(4) is the satisfaction of the incompressibility condition (Eq.(2)). A number of procedures to solve his problem exist in the finite element literature [7, 37]. In our approach we use a stabilized formulation based in the so-called finite calculus procedure [19]–[21],[28, 30, 32]. The essence of this method is the solution of a modified mass balance equation
which is written as

\[ \frac{\partial v_i}{\partial x_i} + \sum_{i=1}^{3} \tau_i \left[ \frac{\partial p}{\partial x_i} + \pi_i \right] = 0 \]  

(5)

where \( \tau \) is a stabilization parameter given by [10]

\[ \tau = \frac{3h^2}{8\mu} \]  

(6)

In above \( h \) is a characteristic length of each finite element (such as \([A^{(e)}]^{1/2}\) for 2D elements). In Eq.(5) \( \pi_i \) are auxiliary pressure projection variables chosen so as to ensure that the second term in Eq.(5) can be interpreted as weighted sum of the residuals of the momentum equations and therefore it vanishes for the exact solution. The set of governing equations for the velocities, the pressure and the \( \pi_i \) variables is completed by adding the following constraint equation to the set of governing equations [28, 32]

\[ \int_V \tau w_i \left( \frac{\partial p}{\partial x_i} + \pi_i \right) dV = 0 \quad i = 1, n_d \]  

(7)

where \( w_i \) are arbitrary weighting functions.

The rest of the integral equations are obtained by applying the standard Galerkin technique to the governing equations (1), (2), (3) and (5) and the corresponding boundary conditions [28, 32].

We interpolate next in the standard finite element fashion the set of problem variables. For 3D problems these are the three velocities \( v_i \), the pressure \( p \), the temperature \( T \) and the three pressure gradient projections \( \pi_i \). In our work we use equal order \textit{linear interpolation} for all variables over meshes of 3-noded triangles (in 2D) and 4-noded tetrahedra (in 3D) [28, 32, 40]. The resulting set of discretized equations has the following form

**Momentum**

\[ M\ddot{\bar{v}} + K(\mu)\bar{v} - G\bar{p} = f \]  

(8)

**Mass balance**

\[ G\bar{v} + L\bar{p} + Q^T\pi = 0 \]  

(9)

**Pressure gradient projection**

\[ \tilde{M}\pi + Q^T\bar{p} = 0 \]  

(10)

**Heat transport**

\[ C\dot{T} + H\dot{T} = q \]  

(11)

In Eqs.(8)–(11) \( \bar{\cdot} \) denotes nodal variables, \( \dot{\bar{\cdot}} = \frac{\partial}{\partial t}(\bar{\cdot}) \). The different matrices and vectors are given in the Appendix.

The solution in time of Eqs.(8)–(11) can be performed using any time integration schemes typical of the updated Lagrangian finite element method. A basic
1. LOOP OVER TIME STEPS, \( t = 1 \), NTIME

Known values
\( \{ x_t, v_t, p_t, T_t, \mu, d_t, q_t, C_t, V_t, M_t \} \)

2. LOOP OVER NUMBER OF ITERATIONS, \( i = 1 \), NITER

- Compute the nodal velocities by solving Eq.(8)
  \[
  \left[ \frac{1}{\Delta t} (M + K) \right]^{i+1} \hat{v} = \frac{1}{\Delta t} f + G^{i+1} \hat{p} + \frac{1}{\Delta t} M \hat{v}
  \]

- Compute nodal pressures from Eq.(9)
  \[
  L^{i+1} \hat{p} = -G^{i+1} \hat{v} - Q^{i+1} \Pi
  \]

- Compute nodal pressure gradient projections from Eq.(10)
  \[
  \Pi^{i+1} = -\hat{\Pi}_D \left[ Q^{i+1} \right] \hat{v} \quad \hat{\Pi}_D = \text{diag}\left[ \hat{M}_D \right]
  \]

- Compute nodal temperature from Eq.(11)
  \[
  \left[ \frac{1}{\Delta t} (C + H) \right]^{i+1} \hat{T} = \frac{1}{\Delta t} q - \frac{1}{\Delta t} C \hat{T}
  \]

- Update position of analysis domain nodes:
  \[
  \hat{x}_{i+1} = \hat{x}_{i} + \Delta t \hat{v}
  \]

Define new “cloud” of nodes \( \hat{x}_{i+1} \)

- Update viscosity values in terms of temperature
  \[
  \mu_t = \mu \left( \frac{\hat{T}}{T_0} \right)
  \]

Check convergence → NO → Next iteration \( i \rightarrow i + 1 \)
↓ YES

Next time step \( t \rightarrow t + 1 \)

- Identify new analysis domain boundary: \( \hat{\Gamma}_V \)
- Generate mesh: \( \hat{\Gamma}_M \)

\text{Go to 1}

**Box I.** Flow chart of basic PFEM algorithm for the fluid domain

Algorithm following the conceptual process described in Section 2.1 is presented in Box I. \( n+1(\hat{a})^{j+1} \) denotes the values of the nodal variables \( \hat{a} \) at time \( n + 1 \) and the \( j + 1 \) iterations. We note the coupling of the flow and thermal equations via the dependence of the viscosity \( \mu \) with the temperature.

## 4 OVERVIEW OF THE COUPLED FSI ALGORITHM

Figure 5 shows a typical domain \( V \) with external boundaries \( \Gamma_V \) and \( \Gamma_t \) where the velocity and the surface tractions are prescribed, respectively. The domain \( V \) is formed by fluid \( V_F \) and solid \( V_S \) subdomains (i.e. \( V = V_F \cup V_S \)). Both subdomains interact at a common boundary \( \Gamma_{FS} \) where the surface tractions and the kinematic variables (displacements, velocities and accelerations) are the same for both subdomains. Note that both set of variables (the surface tractions and the kinematic variables) are equivalent in the equilibrium configuration.

Let us define \( ^tS \) and \( ^tF \) the set of variables defining the kinematics and the stress-strain fields at the solid and fluid domains at time \( t \), respectively, i.e.

\[
^tS := [^t x_s, ^t u_s, ^t v_s, ^t a_s, ^t \varepsilon_s, ^t \sigma_s, ^t T_s]^T
\]

\[
^tF := [^t x_F, ^t u_F, ^t v_F, ^t a_F, ^t \varepsilon_F, ^t \sigma_F, ^t T_F]^T
\]

where \( x \) is the nodal coordinate vector, \( u, v \) and \( a \) are the vector of displacements,
Note: $t_{FS}$ and $V_{FS}$ are equivalent

The coupled fluid-structure interaction (FSI) problem of Figure 4 is solved, in this work, using the following *strongly coupled staggered scheme*:

0. We assume that the variables in the solid and fluid domains at time $t$ ($t^F$ and $t^S$) are known.

1. Solve for the variables at the solid domain at time $t + \Delta t$ ($t^{+\Delta t}S$) under prescribed surface tractions at the fluid-solid boundary $\Gamma_{FS}$. The boundary conditions at the part of the external boundary intersecting the domain are the standard ones in solid mechanics.

The variables at the solid domain $t^{+\Delta t}S$ are found via the integration of the equations of dynamic motion in the solid written as $[40]$

$$ M_s \ddot{a}_s + \mathbf{g}_s - \mathbf{f}_s = 0 $$

(14)
BOX II. Staggered scheme for the FSI problem (see also Figure 3)

\[
\begin{array}{c}
\text{LOOP OVER TIME STEPS } n = 1, \ldots, n_{\text{time}} \\
\text{LOOP OVER STAGGERED SOLUTION } j = 1, \ldots, n_{\text{stag}} \\
\text{Solve for solid variables (prescribed tractions at } n_{\text{stag}} \Gamma_{r} \text{)} \\
\text{LOOP OVER ITERATIONS } i = 1, \ldots, n_{\text{iter}} \\
\text{Solve for } n_{\text{iter}} S'_{j} \\
\text{Integrate Eq.(3) using a Newmark scheme} \\
\text{Check convergence. Yes: solve for fluid variables} \\
\text{NO: Next iteration } i \leftarrow i + 1 \\
\text{Solve for fluid variables (prescribed velocities at } n_{\text{stag}} \Gamma_{FS} \text{)} \\
\text{LOOP OVER ITERATIONS } i = 1, \ldots, n_{\text{iter}} \\
\text{Solve for } n_{\text{iter}} F'_{j} \text{ using the scheme of Section 3.4} \\
\text{Check convergence. Yes: go to C} \\
\text{Next iteration } i \leftarrow i + 1 \\
\text{C Check convergence of surface tractions at } n_{\text{stag}} \Gamma_{FS} \\
\text{Yes: Next time step} \\
\text{Next staggered solution } j \leftarrow j + 1, \ i \leftarrow i + 1 \\
\text{Next time step } n_{\text{stag}} S \leftarrow n_{\text{stag}} S'_{j}, \ n_{\text{stag}} F \leftarrow n_{\text{stag}} F'_{j} \\
\end{array}
\]

where $\mathbf{a}$ is the vector of nodal accelerations and $\mathbf{M}_s$, $\mathbf{g}_s$ and $\mathbf{f}_s$ are the mass matrix, the internal node force vector and the external nodal force vector in the solid domain. The time integration of Eq.(14) is performed using a standard Newmark method.

Solve for the variables at the fluid domain at time $t + \Delta t$ ($t + \Delta t F'$) under prescribed surface tractions at the external boundary $\Gamma_r$ and prescribed velocities at the external and internal boundaries $\Gamma_V$ and $\Gamma_{FS}$, respectively. An incremental iterative scheme is implemented within each time step to account for non linear geometrical and material effects.

Iterate between 1 and 2 until convergence.

The above FSI solution algorithm is shown schematically in Box II.

5 GENERATION OF A NEW MESH

One of the key points for the success of the PFEM is the fast regeneration of a mesh at every time step on the basis of the position of the nodes in the space domain. Indeed, any fast meshing algorithm can be used for this purpose. In our work the mesh is generated at each time step using the so called extended Delaunay tesselation (EDT) presented in [11, 13, 14]. The EDT allows one to generate non standard meshes combining elements of arbitrary polyhedral shapes (triangles,
Figure 6: Generation of non standard meshes combining different polygons (in 2D) and polyhedra (in 3D) using the extended Delaunay technique.

quadrilaterals and other polygons in 2D and tetrahedra, hexahedra and arbitrary polyhedra in 3D) in a computing time of order $n$, where $n$ is the total number of nodes in the mesh (Figure 6). The $C^0$ continuous shape functions of the elements can be simply obtained using the so called meshless finite element interpolation (MFEM). In our work the simpler linear $C^0$ interpolation has been chosen [11, 13, 14].

Figure 7 shows the evolution of the CPU time required for generating the mesh, for solving the system of equations and for assembling such a system in terms of the number of nodes. The numbers correspond to the solution of a 3D flow in an open channel with the PFEM [32]. The figure shows the CPU time in seconds for each time step of the algorithm of Section 3.2. We see that the CPU time required for meshing grows linearly with the number of nodes, as expected. Note also that the CPU time for solving the equations exceeds that required for meshing as the number of nodes increases. This situation has been found in all the problems solved with the PFEM. As a general rule for large 3D problems meshing consumes around 30% of the total CPU time for each time step, while the solution of the equations and the assembling of the system consume approximately 40% and 20% of the CPU time for each time step, respectively. These figures prove that the generation of the mesh has an acceptable cost in the PFEM.

6 IDENTIFICATION OF BOUNDARY SURFACES

One of the main tasks in the PFEM is the correct definition of the boundary domain. Boundary nodes are sometimes explicitly identified. In other cases, the total set of nodes is the only information available and the algorithm must recognize the boundary nodes.

In our work we use an extended Delaunay partition for recognizing boundary nodes. Considering that the nodes follow a variable $h(x)$ distribution, where $h(x)$ is typically the minimum distance between two nodes, the following criterion has been used. All nodes on an empty sphere with a radius greater than $\alpha h$, are considered as boundary nodes. In practice $\alpha$ is a parameter close to, but greater than one. Values of $\alpha$ ranging between 1.3 and 1.5 have been found to be optimal in all examples analyzed. This criterion is coincident with the Alpha Shape concept [8]. Figure 8 shows an example of the boundary recognition using the Alpha Shape technique.

Once a decision has been made concerning which nodes are on the boundaries,
the boundary surface is defined by all the polyhedral surfaces (or polygons in 2D) having all their nodes on the boundary and belonging to just one polyhedron.

The method described also allows one to identify isolated fluid particles outside the main fluid domain. These particles are treated as part of the external boundary where the pressure is fixed to the atmospheric value. We recall that each particle is a material point characterized by the density of the solid or fluid domain to which it belongs. The mass which is lost when a boundary element is eliminated due to departure of a node (a particle) from the main analysis domain is again regained when the “flying” node falls down and a new boundary element is created by the Alpha Shape algorithm (Figures 2 and 8).

The boundary recognition method above described is also useful for detecting contact conditions between the fluid domain and a fixed boundary, as well as between different solids interacting with each other. The contact detection procedure is detailed in the next section.

We note that the main difference between the PFEM and the classical FEM is just the remeshing technique and the identification of the domain boundary at each time step. The rest of the steps in the computation are coincident with those of the classical FEM.
7 TREATMENT OF CONTACT CONDITIONS IN THE PFEM

7.1 Contact between the fluid and a fixed boundary

The motion of the solid is governed by the action of the fluid flow forces induced by the pressure and the viscous stresses acting at the common boundary $\Gamma_{FS}$, as mentioned above.

The condition of prescribed velocities at the fixed boundaries in the PFEM are applied in strong form to the boundary nodes. These nodes might belong to fixed external boundaries or to moving boundaries linked to the interacting solids. Contact between the fluid particles and the fixed boundaries is accounted for by the incompressibility condition which naturally prevents the fluid nodes to penetrate into the solid boundaries (Figure 9). This simple way to treat the fluid-wall contact at mesh generation level is a distinct and attractive feature of the PFEM formulation.

7.2 Contact between solid-solid interfaces

The contact between two solid interfaces is simply treated by introducing a layer of contact elements between the two interacting solid interfaces. This layer is automatically created during the mesh generation step by prescribing a minimum distance ($h_c$) between two solid boundaries. If the distance exceeds the minimum value ($h_c$) then the generated elements are treated as fluid elements. Otherwise the elements are treated as contact elements where a relationship between the tangential and normal forces and the corresponding displacement is introduced so as to model elastic and frictional contact effects in the normal and tangential directions, respectively (Figure 10).

This algorithm has proven to be very effective and it allows to identifying and modeling complex frictional contact conditions between two or more interacting bodies moving in water in an extremely simple manner. Of course the accuracy of this contact model depends on the critical distance above mentioned.

This contact algorithm can also be used effectively to model frictional contact conditions between rigid or elastic solids in standard structural mechanics applications. Figures 11–14 show examples of application of the contact algorithm to the bumping of a ball falling in a container, the failure of an arch formed by a collection
Contact between fluid and fixed boundary

Contact is detected during mesh generation
There is no need for a contact search algorithm

Figure 9: Automatic treatment of contact conditions at the fluid-wall interface

of stone blocks under a seismic loading and the motion of five tetrapods as they fall and slip over an inclined plane, respectively. The images in Figures 11 and 14 show explicitly the layer of contact elements which controls the accuracy of the contact algorithm.

8 MODELING OF BED EROSION

Prediction of bed erosion and sediment transport in open channel flows are important tasks in many areas of river and environmental engineering. Bed erosion can lead to instabilities of the river basin slopes. It can also undermine the foundation of bridge piles thereby favouring structural failure. Modeling of bed erosion is also relevant for predicting the evolution of surface material dragged in earth dams in
Contact between solid boundaries

overspill situations. Bed erosion is one of the main causes of environmental damage in floods.

Bed erosion models are traditionally based on a relationship between the rate of erosion and the shear stress level [16, 36]. The effect of water velocity on soil erosion was studied in [34]. In a recent work we have proposed an extension of the PFEM to model bed erosion [31]. The erosion model is based on the frictional work at the bed surface originated by the shear stresses in the fluid. The resulting erosion model resembles Archard law typically used for modeling abrasive wear in surfaces under frictional contact conditions [1, 24].

The algorithm for modeling the erosion of soil/rock particles at the fluid bed is the following:

1. Compute at every point of the bed surface the resultant tangential stress \( \tau \) induced by the fluid motion. In 3D problems \( \tau = (\tau_s^2 + \tau_t^2)^{1/2} \) where \( \tau_s \) and \( \tau_t \) are the tangential stresses in the plane defined by the normal direction \( n \) at the bed node. The value of \( \tau \) for 2D problems can be estimated as follows:

\[
\tau_t = \mu \gamma_t \quad (15a)
\]

with

\[
\gamma_t = \frac{1}{2} \frac{\partial v_t}{\partial n} = \frac{v_t^k}{2h_k} \quad (15b)
\]

where \( v_t^k \) is the modulus of the tangential velocity at the node \( k \) and \( h_k \) is a prescribed distance along the normal of the bed node \( k \). Typically \( h_k \) is of the order of magnitude of the smallest fluid element adjacent to node \( k \) (Figure 15).

Figure 10: Contact conditions at a solid-solid interface
2. Compute the frictional work originated by the tangential stresses at the bed surface as

\[ W_f = \int_0^t \tau_l \gamma_l \, dt = \int_0^t \mu \left( \frac{v_k}{h_k} \right)^2 \, dt \quad (16) \]

Eq.(16) is integrated in time using a simple scheme as

\[ ^n W_f = ^{n-1} W_f + \tau_l \gamma_l \Delta t \quad (17) \]

3. The onset of erosion at a bed point occurs when \(^n W_f\) exceeds a critical threshold value \(W_c\) defined empirically according to the specific properties of the bed material.
4. If \( w_f > w_c \) at a bed node, then the node is detached from the bed region and it is allowed to move with the fluid flow, i.e. it becomes a fluid node. As a consequence, the mass of the patch of bed elements surrounding the bed node vanishes in the bed domain and it is transferred to the new fluid node. This mass is subsequently transported with the fluid. Conservation of mass of the bed particles within the fluid is guaranteed by changing the density of the new fluid node so that the mass of the suspended sediment traveling with the fluid equals the mass originally assigned to the bed node. Recall that the
mass assigned to a node is computed by multiplying the node density by the tributary domain of the node.

5. Sediment deposition can be modeled by an inverse process to that described in the previous step. Hence, a suspended node adjacent to the bed surface with a velocity below a threshold value is assigned to the bed surface. This automatically leads to the generation of new bed elements adjacent to the boundary of the bed region. The original mass of the bed region is recovered by adjusting the density of the newly generated bed elements.
Figure 15 shows an schematic view of the bed erosion algorithm proposed.

9 EXAMPLES

The examples presented below show the applicability of the PFEM to solve problems involving large motions of the free surface, fluid-multibody interactions, bed erosion and melting and dripping of polymers in fire situations.

9.1 Rigid objects falling into water

The analysis of the motion of submerged or floating objects in water is of great interest in many areas of harbour and coastal engineering and naval architecture among others.

Figure 16 shows the penetration and evolution of a cube and a cylinder of rigid shape in a container with water. The colours denote the different sizes of the elements at several times. In order to increase the accuracy of the FSI problem smaller size elements have been generated in the vicinity of the moving bodies during their motion (Figure 17).

9.2 Impact of water streams on rigid structures

Figure 18 shows an example of a wave breaking within a prismatic container including a vertical cylinder. Figure 19 shows the impact of a wave on a vertical column sustained by four pillars. The objective of this example was to model the impact of a water stream on a bridge pier accounting for the foundation effects.
9.3 Dragging of objects by water streams

Figure 20 shows the effect of a wave impacting on a rigid cube representing a vehicle. This situation is typical in flooding and Tsunami situations. Note the layer of contact elements modeling the frictional contact conditions between the cube and the bottom surface.
9.4 Impact of sea waves on piers and breakwaters

Figure 21 shows the 3D simulation of the interaction of a wave with a vertical pier formed by a collection of reinforced concrete cylinders.

Figure 22 shows the simulation of the falling of two tetrapods in a water container. Figure 23 shows the motion of a collection of ten tetrapods placed in the slope of a breakwaters under an incident wave.

Figure 24 shows a detail of the complex three-dimensional interactions between water particles and tetrapods and between the tetrapods themselves.

Figures 25 and 26 show the analysis of the effect of breaking waves on two different sites of a breakwater containing reinforced concrete blocks (each one of 4 × 4 mts). The figures correspond to the study of Langosteira harbour in La Coruna, Spain using PFEM.

Figure 27 displays the effect of an overtopping wave on a truck circulating by the perimetral road of the harbour adjacent to the breakwater.

9.5 Erosion of a 3D earth dam due to an overspill stream

We present a simple, schematic, but very illustrative example showing the potential of the PFEM to model bed erosion in free surface flows.

The example represents the erosion of an earth dam under a water stream running over the dam top. A schematic geometry of the dam has been chosen to simplify the computations. Sediment deposition is not considered in the solution. The images of Figure 28 show the progressive erosion of the dam until the whole dam is dragged
out by the fluid flow.

Other applications of the PFEM to bed erosion problems can be found in [31].

9.6 Melting and spread of polymer objects in fire

In the next example shown the PFEM is used to simulate an experiment performed at the National Institute for Stanford and Technology (NIST) in which a slab of polymeric material is mounted vertically and exposed to uniform radiant heating on one face. It is assumed that the polymer melt flow is governed by the equations of an incompressible fluid with a temperature dependent viscosity. A quasi-rigid behaviour of the polymer object at room temperature is reproduced by using a very high value of the viscosity parameter. As temperature increases in the thermoplastic object due to heat exposure, the viscosity decreases in several orders of magnitude as a function of temperature and this induces the melt and flow of the particles in the heated zone. Polymer melt is captured by a pan below the sample.

A schematic of the apparatus used in the experiments is shown in Figure 29.
rectangular polymeric sample of dimensions 10 cm high by 10 cm wide by 2.5 cm thick is mounted upright and exposed to uniform heating on one face from a radiant cone heater placed on its side. The sample is insulated on its lateral and rear faces. The melt flows down the heated face of the sample and drips onto a surface below. A load cell monitors the mass of polymer remaining in the sample, and a laboratory balance measures the mass of polymer falling onto the catch surface. Details of the experimental setup are given in [4].

Figure 29 shows all three curves of viscosity vs. temperature for the polypropylene type PP702N, a low viscosity commercial injection molding resin formulation. The relationship used in the model, as shown by the black line, connects the curve for the undegraded polymer to points A and B extrapolated from the viscosity curve for each melt sample to the temperature at which the sample was formed. The result is an empirical viscosity-temperature curve that implicitly accounts for molecular weight changes.

The finite element mesh used for the analysis has 3098 nodes and 5832 triangular elements. No nodes are added during the course of the run. The addition of a catch
Figure 20: Dragging of a cubic object by a water stream.

pan to capture the dripping polymer melt tests the ability of the PFEM model to recover mass when a particle or set of particles reaches the catch surface. For this problem, heat flux is only applied to free surfaces above the midpoint between the catch pan and the base of the sample. However, every free surface is subject to radiative and convective heat losses. To keep the melt fluid, the catch pan is set to a temperature of 600 K. Figure 30 shows four snapshots of the time evolution of the melt flow into the catch pan.

To test the ability of the PFEM to solve this type of problem in three dimensions, a 3D problem for flow from a heated sample was run. The same boundary conditions are used as in the 2D problem illustrated in Figure 29, but the initial dimensions of the sample are reduced to $10 \times 2.5 \times 2.5$ cm. The initial size of the model is 22475 nodes and 97600 four-noded tetrahedra. The shape of the surface and temperature field at different times after heating begins are shown in Figure 31.

Although the resolution for this problem is not fine enough to achieve high accuracy, the qualitative agreement of the 3D model with 2D flow and the ability to carry out this problem in a reasonable amount of time suggest that the PFEM can be used to model melt flow and spread of complex 3D polymer geometry.

Figure 32 shows results for the analysis of the melt flow of a triangular thermoplastic object into a catch pan. The material properties for the polymer are the same as for the previous example. The PFEM succeeds to predicting in a very realistic manner the progressive melting and slip of the polymer particles along the vertical
Figure 21: Interaction of a wave with a vertical pier formed by reinforced concrete cylinders.

wall separating the triangular object and the catch pan. The analysis follows until the whole object has fully melt and its mass is transferred to the catch pan.

We note that the total mass was preserved with an accuracy of 0.5% in both these studies. Gasification, in-depth absorption or radiation were not taken into account in these analysis. More examples of application of PFEM to the melting and dripping of polymers are reported in [33].

10 CONCLUSIONS

The particle finite element method (PFEM) is ideal to treat problems involving fluid-structure interaction, large motion of fluid or solid particles, surface waves, water splashing, separation of water drops, frictional contact situations between fluid-solid and solid-solid interfaces, bed erosion, coupled thermal effects, melting and dripping of objects, etc. The success of the PFEM lies in the accurate and efficient solution of the equations of an incompressible fluid and of solid dynamics using an updated Lagrangian formulation and a stabilized finite element method, allowing the use of low order elements with equal order interpolation for all the variables. Other essential solution ingredients are the efficient regeneration of the finite element mesh using an extended Delaunay tessellation, the identification of the boundary nodes using the Alpha-Shape technique and the simple algorithm to treat frictional contact conditions at fluid-solid and solid-solid interfaces via mesh generation. The examples presented have shown the great potential of the PFEM for
solving a wide class of practical FSI problems in engineering. Examples of validation of the PFEM results with data from experimental tests are reported in [17].

Acknowledgements

Thanks are given to Mrs. M. de Mier for many useful suggestions. This research was partially supported by project SEDUREC of the Consolider Programme of the Ministerio de Educación y Ciencia of Spain and Project SAYOM of CDTI Spain. Thanks are also given to the Spanish construction company Dragados for financial support for the study of harbour engineering problems.

REFERENCES


Figure 23: Motion of ten tetrapods on a slope under an incident wave.


Figure 24: Detail of the motion of ten tetrapods on a slope under an incident wave. The figure shows the complex interactions between the water particles and the tetrapods.


Figure 25: Effect of breaking waves on a breakwater slope containing reinforced concrete blocks. Detail of the mesh of 4-noded tetrahedra near the slope at two different times.

Figure 26: Study of breaking waves on the edge of a breakwater structure formed by reinforced concrete blocks


Figure 27: Effect of an overtopping wave on a truck passing by the perimetral road of a harbour adjacent to the breakwater
Figure 28: Erosion of a 3D earth dam due to an overspill stream.


Figure 29: Polymer melt experiment. Viscosity vs. temperature for PP702N polypropylene in its initial undegraded form and after exposure to 30 kW/m$^2$ and 40 kW/m$^2$ heat fluxes. The black curve follows the extrapolation of viscosity to high temperatures.


Figure 30: Evolution of the melt flow into the catch pan at $t = 400\text{s}$, $550\text{s}$, $700\text{s}$ and $1000\text{s}$


[29] E. Oñate, A. Valls, J. García, FIC/FEM formulation with matrix stabilizing terms for incompressible flows at low and high Reynold’s numbers, Computational Mechanics 38 (4-5) (2006a) 440-455.


Figure 31: Solution of a 3D polymer melt problem with the PFEM. Melt flow from a heated prismatic sample at different times.


Figure 32: Melt flow of a heated triangular object into a catch pan.


APPENDIX

The matrices and vectors in Eqs.(8)-(11) for a 4-noded tetrahedron are:

\[ M_{ij} = \int_{V_e} \rho N_i^T N_j dV, \quad K_{ij} = \int_{V_e} B_i^T D B_j dV \]

\[ G_{ij} = \int_{V_e} B_i^T m N_j dV, \quad f_i = \int_{V_e} N_i^T b dV + \int_{\Gamma_e} N_i^T t d\Gamma \]

\[ L_{ij} = \int_{V_e} \nabla^T N_i \nabla N_j dV, \quad \nabla = \left[ \frac{\partial}{\partial x_1}, \frac{\partial}{\partial x_2}, \frac{\partial}{\partial x_3} \right]^T \]

\[ Q = [Q_1, Q_2, Q_3] \quad [Q_k]_{ij} = \int_{V_e} \tau_k \frac{\partial N_i}{\partial x_k} N_j dV, \quad \text{no sum in} \ k \]

\[ \dot{M} = [\dot{M}_1, \dot{M}_2, \dot{M}_3], \quad [\dot{M}]_{kl} = \left( \int_{V_e} \tau_k N_i N_j dV \right) \delta_{kl}, \quad k, l = 1, 2, 3 \]

\[ B = [B_1, B_2, B_3, B_4]; \quad B_i = \begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N_i}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N_i}{\partial z} \\ \frac{\partial N_i}{\partial y} & 0 & \frac{\partial N_i}{\partial x} \\ 0 & \frac{\partial N_i}{\partial z} & \frac{\partial N_i}{\partial y} \end{bmatrix}; \quad D = \mu \begin{bmatrix} 2 & 0 & 0 & 0 & 0 \\ 0 & 2 & 0 & 0 & 0 \\ 0 & 0 & 2 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 1 \end{bmatrix} \]

\[ N = [N_1, N_2, N_3, N_4]; \quad N_i = N_i I_3 \quad I_3 \text{ is the unit matrix} \]

\[ C_{ij} = \int_{V_e} \rho c N_i N_j dV, \quad H_{ij} = \int_{V_e} \nabla^T N_i [k] \nabla N_j dV \]

\[ [k] = \begin{bmatrix} k_1 & 0 & 0 \\ 0 & k_2 & 0 \\ 0 & 0 & k_3 \end{bmatrix}, \quad q_i = \int_{V_e} N_i Q dV - \int_{\Gamma_q} N_i q_n d\Gamma \]

In above equations indexes \( i, j \) run from 1 to the number of element nodes (4 for a tetrahedron), \( q_n \) is the heat flow prescribed at the external boundary \( \Gamma_q \), \( t \) is the surface traction vector \( t = [t_x, t_y, t_z]^T \) and \( V_e \) and \( \Gamma_e \) are the element volume and the element boundary, respectively.
Anexo 2

Resultados de los experimentos realizados en el Laboratorio de Hidráulica de la E.T.S. de Ingenieros de Caminos, Canales y Puertos de Barcelona sobre estudios de los frentes de onda generados por la apertura rápida de una compuerta
ANÁLISIS EXPERIMENTAL DE FRENTE DE ONDA GENERADOS POR LA APERTURA RÁPIDA DE UNA COMPUERTA

Autores:

José Luis Aragón Hernández
Belén Martí Cardona
Martí Sánchez-Juny

Diciembre de 2008.
Contenido
1 Descripción del trabajo ................................................................................................................ 1
  1.1 Descripción de la instalación.................................................................................................... 1
  1.2 Sistema de apertura de la compuerta .................................................................................. 2
  1.3 Equipo de medición .............................................................................................................. 3
  1.4 Procesado de las imágenes .................................................................................................. 3
2 Campaña Experimental .............................................................................................................. 5
  2.1 Descripción de los ensayos realizados .................................................................................. 5
  2.2 Leyes de apertura de la compuerta ..................................................................................... 5
3 Resultados .................................................................................................................................... 7
  3.1 Perfil de la superficie libre del agua ..................................................................................... 7
    3.1.1 Ensayos T1 ..................................................................................................................... 8
    3.1.2 Ensayos T2 ................................................................................................................... 12
4 Conclusiones ................................................................................................................................ 18
1 Descripción del trabajo

En el Laboratorio de Hidráulica y Mecánica de Fluidos, que el grupo de Investigación FLUMEN dispone, se ha llevado a cabo el diseño de una instalación que permite la reproducción experimental de un frente de onda provocado por la apertura rápida de una compuerta.

1.1 Descripción de la instalación

Los ensayos se realizaron en el interior del Canal de Pendiente Variable del grupo FLUMEN. El canal dispone de cajeros de vidrio que permiten la visualización del flujo en su interior, es de sección rectangular de ancho 0.60 m y los ensayos se realizaron con pendiente horizontal (Figura 1).

La instalación consiste en un recinto cerrado de 1.44 m de longitud, con una compuerta deslizante de 2 cm de espesor, cuyo eje está situado a 0.47 m del contorno de cierre del extremo aguas arriba.

Figura 1. Caracterización de la instalación en el Canal de Pendiente Variable del grupo FLUMEN.
La compuerta está ligada a un sistema de contrapesos y poleas que permite una apertura rápida, controlada y repetitiva para iguales cargas de agua, aguas arriba de la misma. Dichos niveles establecen la condición inicial de cada ensayo. El recinto aguas abajo de la compuerta tiene una longitud mayor que el de aguas arriba, en concreto 0.96 m.

Figura 2. Vista de perfil del recinto de ensayo. Puede apreciarse igualmente el sistema de poleas que sujeta la compuerta.

1.2 Sistema de apertura de la compuerta

La compuerta consiste en una plancha de metacrilato rectangular que dispone de su extremo inferior biselado y, como ya se ha comentado, está sujeta a un sistema de contrapeso y poleas que, a igualdad de carga aguas arriba de la compuerta, permite su izado de manera repetitiva.

Este sistema de izado fue diseñado para asegurar:

1. Una apertura lo más rápida posible para la formación de un frente onda lo más rápidamente variable posible, y
2. Igualdad de la fuerza de izado en todos los ensayos a fin de poder realizar ensayos comparables entre sí.

Así, el sistema que consiste en un cable de acero de 5 mm de diámetro sujeto a la compuerta por un extremo y por el otro unido a dos bloques cilíndricos de hormigón de aproximadamente 35 kg, mediante un sistema de tres poleas. Antes de iniciar cada ensayo los bloques de hormigón se encuentran apoyados en un soporte a una altura suficiente que permite mantener la compuerta cerrada. Un mecanismo permite hacer caer el soporte y con él los bloques de hormigón, provocándose el izado súbito de la compuerta.
1.3 **Equipo de medición**

Para la medición de las variables hidráulicas en laboratorio y, tratándose en especial de un fenómeno que tiene un comportamiento rápidamente variable, se ha usado una cámara de alta velocidad MotionPro RedLake Midas, cedida por el Laboratorio de Tecnología de Estructuras del Departamento de Ingeniería de la Construcción de la UPC. Se trata de un sistema de captación de imágenes no intrusivo que, por tanto no alterará el flujo en el interior del recinto de ensayo.

La cámara se configuró para una captura de 100 instantáneas por segundo que se comprobó suficiente para captar con detalle el fenómeno. Ello requirió de un laborioso post-proceso para transformar las imágenes capturadas en secuencias temporales de perfiles de la lámina de agua y de calados.

1.4 **Procesado de las imágenes**

Como ya se ha descrito, los ensayos se filmaron con la cámara de alta velocidad MotionPro RedLake Midas con frecuencia de adquisición de 100 imágenes por segundo. Las filmaciones de los ensayos constan del número de imágenes indicadas en la Tabla 1 adjunta. Estas imágenes se procesaron para determinar en cada una de ellas la altura de la lámina de agua adyacente al cristal anterior del recinto de ensayo en una serie de secciones transversales del recinto referidas como...
posiciones de los sensores. La Tabla 1 indica la posición de los sensores en función de su distancia al extremo aguas arriba del recinto, así como el número de imágenes que se procesó en cada caso.

<table>
<thead>
<tr>
<th>Tipo</th>
<th>Fondo</th>
<th>$H_a$ (m)</th>
<th>$H_b$ (m)</th>
<th>Objeto Aguas Abajo</th>
<th>N° Imágenes</th>
<th>Posición Sensores (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>T1</td>
<td>Seco</td>
<td>0.00</td>
<td>0.30</td>
<td>Ninguno</td>
<td>1.545</td>
<td>0.127 0.310 0.663 0.861 1.062 1.161 1.262</td>
</tr>
<tr>
<td></td>
<td>Mojado</td>
<td>0.05</td>
<td>0.30</td>
<td>Ninguno</td>
<td>2.030</td>
<td></td>
</tr>
<tr>
<td>T2</td>
<td>Seco</td>
<td>0.00</td>
<td>0.30</td>
<td>Ninguno</td>
<td>410</td>
<td>0.127 0.310 0.709 0.961 1.210 - -</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Cilindro circular</td>
<td>395</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Prisma cuadrada</td>
<td>400</td>
<td></td>
</tr>
</tbody>
</table>

Todas las imágenes fueron filtradas mediante un cierre y apertura morfológicos consecutivos, con un elemento estructural de $3 \times 3$ píxeles. Estos filtros eliminan de la imagen puntos oscuros, como los de la malla milimetrada en el cristal anterior del recinto del ensayo, así como pequeños brillos debidos en general a la reflexión de la luz sobre gotas de agua adheridas al cristal del recinto.

En la imagen inicial de cada ensayo se determinó mediante observación directa la altura de agua en píxeles en la columna correspondiente a la posición los distintos sensores. Un código programado en Matlab permitió obtener la altura de agua en píxeles en la posición de cada sensor para el resto de las imágenes de los ensayos. Si la altura de agua en la columna del sensor $S$ en la imagen $I$ es $H$, el código evalúa en las ocho direcciones posibles el gradiente de intensidad de la imagen $I+1$ en la columna $S$ en un entorno de la altura $H$. La altura de agua en la columna $S$ de la imagen $I+1$ viene determinada por el máximo gradiente de intensidad.

Una malla milimetrada situada sobre el cristal anterior del recinto del ensayo permitió calcular la función de transformación de calado en píxeles a calado en metros, en las columnas de la imagen correspondientes a la posición de los sensores. Mediante estas funciones los calados obtenidos en número de píxeles por el código Matlab se transformaron en medidas de longitud.
2 Campaña Experimental

2.1 Descripción de los ensayos realizados

En la Tabla 2 se resumen las principales características de los ensayos que se llevaron a cabo. Se realizaron dos tipologías de ensayos:

- **Ensayos T1.** La instalación funciona como un recinto cerrado, es decir, existe una pared aguas abajo, por lo que se producirá el rebote de la onda generada por la apertura de la compuerta dentro del depósito.

- **Ensayos T2.** Se retiró la pared del extremo aguas abajo por lo que no se producirán los rebotes del frente hacia aguas arriba.

Los ensayos se realizaron imponiendo como condición inicial, un nivel aguas arriba de la compuerta aproximado de 0.30 m (carga hidráulica) y aguas abajo con fondo seco (0.00 m) en los ensayos T1 y T2 y mojado (aproximadamente 0.05 m) para los ensayos T1. Hay que tener en cuenta que los ensayos llamados con fondo seco aguas abajo de la compuerta, en la práctica tenían unos niveles de agua, en cualquier caso del orden de unos pocos milímetros, debido a pequeñas fugas de agua existentes en la compuerta. Se dispone de las medidas de dichos niveles por lo que es posible tenerlos en cuenta para una simulación más precisa.

En cada caso, y también para la condición de, aproximadamente, 0.30 m aguas arriba de la compuerta, se colocaron unos sólidos rígidos de dos diferentes geometrías en el centro del espacio aguas abajo de la compuerta a fin de analizar su posible arrastre por el flujo. En la Tabla 2 se muestran las principales características de los cinco ensayos. En la Tabla 3 se indican las características los objetos expuestos al flujo.

<table>
<thead>
<tr>
<th>Tabla 2. Características de los ensayos realizados.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tipo</td>
</tr>
<tr>
<td>Recinto Cerrado</td>
</tr>
<tr>
<td>Recinto Abierto</td>
</tr>
<tr>
<td>Recinto Cerrado</td>
</tr>
<tr>
<td>Recinto Abierto</td>
</tr>
</tbody>
</table>

$h_a$, calado aguas arriba de la compuerta; $h_d$, calado aguas abajo de la compuerta

<table>
<thead>
<tr>
<th>Tabla 3. Caracterización de los objetos expuestos al flujo.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Objeto</td>
</tr>
<tr>
<td>Prisma cuadrado</td>
</tr>
<tr>
<td>Cilindro circular</td>
</tr>
</tbody>
</table>

2.2 Leyes de apertura de la compuerta

Las condiciones de frontera impuestas son: aguas arriba la ley de apertura de la compuerta, la cual puede variar de un ensayo a otro y aguas abajo el ensayo puede funcionar como recinto cerrado o abierto según el tipo de ensayo.
En la Figura 4 se presentan las leyes de apertura de la compuerta para los diferentes ensayos. En la Figura 5 se muestran las mismas leyes desplazadas al mismo origen de tiempo. Puede observarse que de los cinco ensayos mostrados, cuatro corresponden al caso en que la carga hidráulica aguas arriba de la compuerta es aproximadamente 0.30 m y aguas abajo el recinto se encuentra seco (línea azul continua, puntos rojos, cuadrados verdes y línea roja discontinua), y el quinto corresponde a una carga de 0.30 m aguas arriba pero con un nivel de 0.05 m aguas abajo. Se aprecia que dos de ellos presentan un comportamiento algo distinto:

- Las diferencias más destacadas aparecen en el caso de carga 0.30 m y fondo seco aguas abajo de la compuerta (línea roja discontinua). En este caso el izado es significativamente más lento que en los demás ensayos. Ello es así debido a que en este ensayo particular la compuerta se encontraba sellada, mediante el uso de cinta adhesiva, en su contacto con la ranura de guiado del canal.

- En la Figura 5 puede observarse que el ensayo con 0.05 m de agua, abajo de la compuerta (línea azul discontinua) presenta una ley algo distinta de las obtenidas con fondo seco. En tal caso, se aprecia que la compuerta sube de manera algo más lenta.

---

**Figura 4.** Leyes de apertura de la compuerta en cada ensayo.

**Figura 5.** Leyes de apertura de la compuerta con el mismo origen de tiempo.
3 Resultados

Mediante análisis y tratamiento de la secuencia de imágenes digitalizadas se determinaron la ley de apertura de la compuerta, el perfil de la superficie libre del agua y posición de los objetos, para cada ensayo y para cada instante de tiempo. Como ya se ha dicho la frecuencia de adquisición de imágenes fue de 100 Hz.

3.1 Perfil de la superficie libre del agua.

En la Figura 6 se presentan algunos fotogramas del ensayo T1, para el caso de carga hidráulica aguas arriba de la compuerta 0.30 m aproximadamente, y fondo seco aguas abajo. En dichos fotogramas se aprecia el perfil de la lámina de agua durante los primeros instantes del ensayo en el que el frente sale de la compuerta y se desplaza hacia aguas abajo, hasta impactar contra el contorno del extremo del recinto, produciéndose en dicho impacto el rebote y por tanto el retorno del frente hacia aguas arriba.

![Figura 6. Ensayo T1, hu=0.30 m y hd= 0.05 m. Imágenes capturadas de los primeros instantes del ensayo.](image)

Igualmente en la Figura 7 se muestran, a modo de ejemplo, algunos fotogramas del ensayo T2, para el caso de carga aguas arriba 0.30 m aproximadamente, fondo seco aguas abajo y acción sobre un objeto cilíndrico situado a 0.939 m del extremo aguas arriba del recinto. Puede apreciarse en dichas imágenes el primer impacto del flujo sobre el cilindro y como éste es arrastrado hacia aguas abajo, aunque sin producirse su volcado.

Los resultados que se muestran a continuación consisten en la evolución temporal de los perfiles de la lámina de agua en diferentes verticales de medida que se ubican en la Tabla 4. En cualquier caso las verticales S1 y S2 se encuentran aguas arriba de la compuerta y las restantes aguas abajo de la misma.
3.1.1 Ensayos T1

**Carga 0.30 m, fondo seco aguas abajo**

En la Figura 8 y la Figura 9 se presenta la evolución temporal de los niveles de agua registrados en las siete verticales de medida, indicadas en la Tabla 4.

En dichas figuras puede apreciarse el comportamiento ondulante del perfil de la lámina de agua en cada vertical. Así, por ejemplo, las dos primeras verticales (S1 y S2) se aprecia desde el momento en que se empieza a abrir la compuerta, a los 0.14 segundos del inicio del ensayo, como el calado va disminuyendo desde la condición inicial (0.296 m) hasta transcurridos 1.82 segundos del inicio del ensayo cuando se alcanza un valor mínimo en ambas verticales (0.037 m y 0.017 m respectivamente), momento en el que el nivel en dichos puntos empieza a subir debido al retorno del frente de onda. A partir de dicho momento se produce un movimiento ondulatorio provocado por la superposición de los rebotes de los frentes de ondas desde los extremos aguas arriba y aguas abajo.

Las verticales de medida aguas abajo de la compuerta S3 a S7, en el primer paso del frente, muestran un comportamiento análogo a las dos primeras, aunque partiendo de niveles de agua prácticamente en seco y aumentando el calado en cada una a medida que el frente de onda las alcanza.
De la ley de apertura de la compuerta (Figura 4) se observa que desde el inicio del ensayo la compuerta empezó a abrirse a los 0.14 segundos y en llegar a la primera vertical de medida aguas abajo de la compuerta (S3) a los 0.28 segundos, lo que implica una celeridad de propagación del frente de unos 0.76 m/s. En la figura se indican los tiempos de viaje estimados, del primer paso del frente de onda entre las diversas verticales de medida situadas abajo de la compuerta.

El frente tarda en llegar a la vertical S7, la más alejada de la compuerta, 0.58 segundos desde el inicio del ensayo, lo que indica una celeridad media del frente de 0.56 m/s.

Figura 8. Ensayo T1, \( h_u = 0.30 \) m y \( h_d = 0 \) m. Sensores situados a 0.127 m (S1), 0.310 m (S2), 0.663 m (S3) y 0.861 m (S4) del extremo aguas arriba del recinto.
Figura 9. Ensayo T1, $h_r=0.30$ m y $h_d=0$ m. Sensores situados a 1.062 m (S5), 1.161 m (S6) y 1.262 m (S7) del extremo aguas arriba del recinto.

**Carga 0.30 m, nivel 0.05 m aguas abajo**

En la Figura 10 y la Figura 11 se presentan los cambios en el tiempo de la lámina de agua obtenidos en las siete verticales de medida, indicadas en la Tabla 4.

Puede observarse que los perfiles de la lámina de agua tienen un comportamiento análogo al descrito en los diferentes puntos de control en el caso anterior de fondo seco aguas abajo de la compuerta. A pesar de ello se pueden destacar algunas diferencias esenciales:
El calado en los puntos S1 y S2 aguas arriba de la compuerta no bajan del nivel dado de 0.05 m, mientras que en el caso de fondo seco llegó a 0.017 m.

En cualquier caso, una vez pasado el primer frente, la fluctuación de los niveles de agua en el resto de puntos de medida se establece por encima del nivel inicial de 0.05 m prefijado.

**Figura 10.** Ensayo T1, h₀=0.30 m y hₐ= 0.05 m. Sensores situados a 0.127 m (S1), 0.310 m (S2), 0.663 m (S3) y 0.861 m(S4) del extremo aguas arriba del recinto.
3.1.2 Ensayos T2

**Carga 0.30 m, fondo seco aguas abajo**
En la Figura 12 se muestra la variación en el tiempo de los calados obtenidos en las cinco verticales de medida, indicadas en la Tabla 4.

En estos ensayos al no encontrarse cerrados en su extremo aguas abajo, se registrará el vaciado del canal. Es decir, para los puntos de medida aguas arriba de la compuerta (S1 y S2), los niveles de
agua medidos en un punto deben resultar monótonamente decrecientes a lo largo del tiempo. En cambio, los puntos de medida aguas abajo (S3, S4 y S5) se encontrarán prácticamente en seco hasta que sean cruzados por el frente, momento en el que se captará un ascenso brusco de niveles que, de nuevo, se irá amortiguando monótonamente en el tiempo.

Figura 12. Ensayo T2, \( h_c = 0.30 \) m y \( h_d = 0 \) m. Sensores situados a 0.127 m (S1), 0.310 m (S2), 0.709 m (S3), 0.961 m (S4) y 1.210 m (S5) del extremo aguas arriba del recinto.
Carga 0.30 m, fondo seco aguas abajo, acción sobre un prisma de sección cuadrada

En la Figura 13 se presenta la evolución en el tiempo de los niveles de agua obtenidos en las cinco verticales de medida, indicadas en la Tabla 4.

Figura 13. Ensayo T2, \( h_0 = 0.30 \text{ m} \) y \( h_d = 0 \text{ m} \), arrastre del elemento prismatico de sección cuadrada. Sensores situados a 0.127 m (S1), 0.310 m (S2), 0.709 m (S3), 0.961 m (S4) y 1.210 m (S5) del extremo aguas arriba del recinto.
El prisma se encuentra a una distancia de 0.965 m del extremo aguas arriba y sobre el centro de simetría del canal. En concreto se encuentra 0.004 m aguas abajo de la vertical de medida S4 (ver Tabla 4).

Al igual que en el caso anterior, aguas arriba de la compuerta se obtiene una variación monótona decreciente en el tiempo del calado. Los restantes puntos de medida aguas abajo de la compuerta captan el crecimiento súbito del calado al paso del frente y más en concreto el punto S4, que coincide a la práctica con la ubicación del bloque prismático, y el punto S5, captan la interferencia del bloque con el flujo.

En este caso, el elemento permanece inamovible al paso del flujo como se describe en la Figura 14.

**Carga 0.30 m, fondo seco aguas abajo, acción sobre un cilindro**

En la Figura 16 se muestran los perfiles de la lámina de agua obtenidos en las cinco verticales de medida, indicadas en la Tabla 4.

El cilindro se encuentra a una distancia de 0.939 m del extremo aguas arriba y sobre el centro de simetría del canal. En concreto se encuentra 0.022 m, aguas arriba de la vertical de medida S4 (ver Tabla 4).

Los puntos de medida aguas arriba y aguas abajo de la compuerta muestran un comportamiento análogo al que se muestra en el ensayo descrito anteriormente. Comparando las medidas en S4 y S5 de la Figura 13 y la Figura 16, se observa la menor interferencia que el cilindro produce al flujo.

En este caso el objeto es arrastrado en su interacción con el flujo, aunque no volcado. El desplazamiento del objeto se describe como se describe en la Figura 14 y la Figura 15.

**Figura 14.** Ensayo T2, hu=0.30 m y hd= 0 m. Posición (x, y) de los elementos prismático y cilíndrico a lo largo de los respectivos ensayos.

**Figura 15.** Ensayo T2, hu=0.30 m y hd= 0 m. Localización de del elemento cilíndrico en función del tiempo.
Figura 16. Ensayo T2, $h_a=0.30$ m y $h_d=0$ m, arrastre del elemento cilíndrico. Sensores situados a 0.127 m (S1), 0.310 m (S2), 0.709 m (S3), 0.961 m (S4) y 1.210 m (S5) del extremo aguas arriba del recinto.
El coeficiente de arrastre del flujo sobre un cilindro recto circular, cuya relación $h/\phi = 2.66$, con su eje normal al flujo es aproximadamente $C_D = 0.7$, mientras que el mismo coeficiente para un prisma de sección cuadrada con su eje normal al flujo es $C_D = 2.0$ (Lencastre\(^1\), 1957). Ello pone de manifiesto que la sección circular es más hidrodinámica que la cuadrada, puesto que a igualdad de área y energía de velocidad, la fuerza que ejercerá el flujo sobre el elemento será mayor en el caso del prisma cuadrado que del cilindro circular. Ello se traducirá en una mayor alteración de la superficie libre, tal como se aprecia en los niveles registrados en el entorno del bloque prismático en comparación con los del bloque cilíndrico (ver puntos de medida S4 y S5 de la Figura 13 y la Figura 16).

A pesar de ello, en los ensayos realizados el bloque prismático no fue movido por el impacto del flujo debido a su mayor peso, ya que tal como se detalla en la Tabla 3, dicho bloque es de acero (masa de 5.641 kgr) mientras que el cilíndrico es de PVC (masa de 0.406 kgr).

---

4 Conclusiones

A la vista de lo expuesto hasta ahora se pueden destacar las siguientes conclusiones:

- La instalación experimental puesta a punto se muestra útil y adecuada para el análisis de fenómenos de apertura súbita de compuerta. Se han analizado diversas situaciones: con recinto cerrado y fondo seco y mojado aguas abajo de la compuerta, y con recinto abierto y fondo seco hacia aguas abajo. Igualmente, en este caso se han estudiado la incidencia del flujo sobre dos elementos móviles: un prisma cuadrado y un cilindro circular.

- Se ha utilizado como sistema de medida una cámara de alta velocidad. Los ensayos se han registrado con una frecuencia de adquisición de imágenes de 100 Hz. Dicha frecuencia se ha mostrado suficiente para captar estos fenómenos rápidamente variables.

- El sistema de post-proceso de las imágenes ha sido adecuado para reproducir la evolución de los niveles de agua en el recinto de medida, para los diversos ensayos realizados.

- En el apartado 3 se presentan los resultados obtenidos para cada ensayo. En los ensayos correspondientes al recinto cerrado, los calados presentan un comportamiento ondulante amortiguado en el tiempo producido por la superposición de frentes de onda que rebotan en los contornos de los extremos aguas arriba y abajo. Por otro lado los ensayos correspondientes al recinto abierto en su extremo aguas abajo, los niveles de agua tienen un comportamiento monótono decreciente en el tiempo.

- Mientras que el bloque prismático cuadrado no fue arrastrado ni volcado por el impacto del flujo debido a su mayor peso el elemento cilíndrico si fue arrastrado aunque no volcado. La geometría cuadrada, enfrentada al flujo, provoca una mayor variabilidad en los niveles de agua que la geometría circular debido al mayor coeficiente de arrastre que presentan dicha geometría cuadrada (comportamiento menos hidrodinámico).