

## **GID PROBLEM TYPE FOR THE LAGRANGIAN ANALYSIS OF FLUID FLOWS ENIEF 2004**

**Julio Marti<sup>\*</sup>, Facundo Del Pin<sup>†</sup>, Gerardo Frank<sup>\*</sup> and Sergio Idelsohn<sup>\*†</sup>**

<sup>\*</sup> Centro Internacional de Mecánica Computacional (CIMEC), Instituto de Desarrollo Tecnológico para la Industria Química (INTEC),  
Guemes 3450, (3000) Santa Fe, Argentina  
e-mail: jmarti@ceride.gov.ar

<sup>†</sup> International Center for Numerical Methods in Engineering (CIMNE)  
Universidad Politécnica de Cataluña, Barcelona, Spain.

**Key words:** Lagrangian formulation, fluid-structure interaction, problem type.

**Abstract.** *The objective of this work is the programming of a problem type which configures GID for the pre-process of data that will be used in a fluid mechanic program. This program differs from the traditional Eulerian programs in that a particle fully Lagrangian formulation is implemented. Thus, in writing the problem type special considerations have to be taken regarded to boundary conditions and to the fact that no mesh is needed as input in the calculus program. The nodes generated by GID will be taken as the particles in the initial state of the problem. The Lagrangian program will store the states at different time steps such that they can be read by the GID post-process for a posteriori analysis of the results.*

## 1 INTRODUCTION

Fluidsuite is an object orientated code thus written in C++ to solve the Navier-Stokes equations for incompressible flows. The particular aspect of this code is that the equations of motion are written in a Lagrangian way. Thus, convection is obtained by directly moving the nodes of the discrete problem.

A problem type denominated MFEM was developed to constitute a link between the program and the pre/post processor GID<sup>1</sup>. Some two-dimensional examples are analyzed.

## 2 THEORY

A Lagrangian particle formulation is presented to approximate the continuous fluid mechanics equations. Particle methods are those in which the problem is represented by a discrete number of particles, where each particle moves accordingly to its own mass and the internal/external forces applied to it. A Lagrangian frame of reference has been chosen to represent the convection of the particles. In this way, a set of axis is placed over each particle and the convection is directly obtained by moving the particle. This fashionable way of describing convection simplifies the Navies-Stokes equations by avoiding the advection terms. To complement the Lagrangian formulation the Meshless Finite Element Method (MFEM)<sup>2,3</sup> is introduced. This method does not require a mesh to discretize the domain since the points are connected by means of the Extended Delaunay Tessellation (EDT)<sup>2</sup> and the domain is partitioned into polyhedrons. Special shape functions, called non-Sibsonian<sup>4</sup>, are used to interpolate the unknowns over the polyhedrons, whereas the time integration is evaluated by an semi-implicit way by a fractional step method<sup>5,6</sup>. Then, particles are not only part of the fluid but are also represented by the nodes of the discrete problem.

The mass and momentum conservation equations may be written in a semi Lagrangian formulation as:

Mass conservation:

$$\frac{D\rho}{Dt} + \rho \frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

Momentum conservation:

$$\rho \frac{Du_i}{Dt} = -\frac{\partial}{\partial x_i} p + \frac{\partial}{\partial x_j} \tau_{ij} + \rho f_i \quad (2)$$

where  $\rho$  is the density  $u_i$  are the Cartesian components of the velocity field,  $p$  the pressure,

$\tau_{ij}$  the deviator stress tensor,  $f_i$  the source term (normally the gravity) and  $\frac{D\phi}{Dt}$  represents the total or material time derivative of a function  $\phi$ .

For Newtonian fluids the stress tensor  $\tau_{ij}$  may be expressed as a function of the velocity field through the viscosity  $\mu$  by

$$\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_l}{\partial x_l} \delta_{ij} \right) \quad (3)$$

For near incompressible flows ( $\frac{\partial u_i}{\partial x_i} \ll \frac{\partial u_k}{\partial x_i}$ ) the term :

$$\frac{2\mu}{3} \frac{\partial u_i}{\partial x_i} \approx 0 \quad (4)$$

and it may be neglected in eq.(3). Then:

$$\tau_{ij} \approx \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (5)$$

In the same way, the term  $\frac{\partial}{\partial x_j} \tau_{ij}$  in the momentum equations may be simplified for slow incompressible flows as:

$$\begin{aligned} \frac{\partial}{\partial x_j} \tau_{ij} &= \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right) = \mu \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} \right) + \mu \frac{\partial}{\partial x_j} \left( \frac{\partial u_j}{\partial x_i} \right) = \\ &\mu \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} \right) + \mu \frac{\partial}{\partial x_i} \left( \frac{\partial u_j}{\partial x_j} \right) \approx \mu \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} \right) \end{aligned} \quad (6)$$

Then, the momentum equations can be finally written as:

$$\rho \frac{Du_i}{Dt} = -\frac{\partial}{\partial x_i} p + \frac{\partial}{\partial x_j} \tau_{ij} + \rho f_i \approx -\frac{\partial}{\partial x_i} p + \mu \frac{\partial}{\partial x_j} \left( \frac{\partial u_i}{\partial x_j} \right) + \rho f_i \quad (7)$$

## Boundary conditions

On the boundaries, the standard boundary conditions for the Navier-Stokes equations are:

$$\tau_{ij} \nu_j - p \nu_i = \bar{\sigma}_{ni} \text{ on } \Gamma_\sigma$$

$$u_i \nu_i = \bar{u}_n \text{ on } \Gamma_n$$

$$u_i \zeta_i = \bar{u}_t \text{ on } \Gamma_t$$

where  $\nu_i$  and  $\zeta_i$  are the components of the normal and tangent vector to the boundary.

### 3 DEFINING A PROBLEM TYPE

A problem type is a collection of files used to configure GID for a particular type of analysis with a graphical user friendly interface (GUI). All the files are inside of a directory with the problem type name and extension .gid which is located in the main GID executable directory. For an extended description of the GID's problem types and its definition the reader is referred to<sup>8,9</sup>.

Our problem type called MFEM is obtained using classical files, and an extra programming is needed to create a menu in the post-process with Tcl/Tk<sup>7</sup> extension. The configuration files are:

- MFEM.cnd: definition of node conditions (fluid, solid and mobile solid);
- MFEM.prb: definition of data no associated to geometric figures (physical properties, boundary and initial conditions);
- MFEM.bas: file that indicates what and how the GID should write the input data file to the code (files with extension .dat and .cond) ;
- MFEM.tcl: file to create a menu for viewing the results in the post-processing (MFEM);
- MFEM.bat: file that use commands of the operating system and execute the calculation from GID.

This provides an easy definition of the geometry and meshing while the different conditions are assigned in a natural way for the user. Subsequently all the information is written into the files that are given to the solver dedicated to the analysis of fluid mechanic problems and later to allow the visualizing and analysis of the results.

### 4 USAGE AND IMPLEMENTATION DETAILS

To make the definition of the problem as easy as possible in this interface, a screen of

presentation, appear when the user first specifies the use of the developed interface for the problem (Figure 1).

First, there is a preprocessing stage where the geometry of the problem to be solved is defined and subsequently saved with the project name; the FEM mesh is generated and the nodes conditions (Figure 2) and specific data are defined (Figure 3 and 4). Note that, the definition of the boundary conditions will depend on the kind of analysis performed.

Next stage is to run Fluidsuite (calculate option is selected). The input to this module are two files, one with an extension .dat and the other with an extension .cond, both generated by the MFEM.bas. The first file defines the initial geometry with the initial position of nodes and the second file indicates physical properties, boundary and initial conditions.

The last stage of the analysis is the visualization of the results using the menu mentioned before (Figure 5).

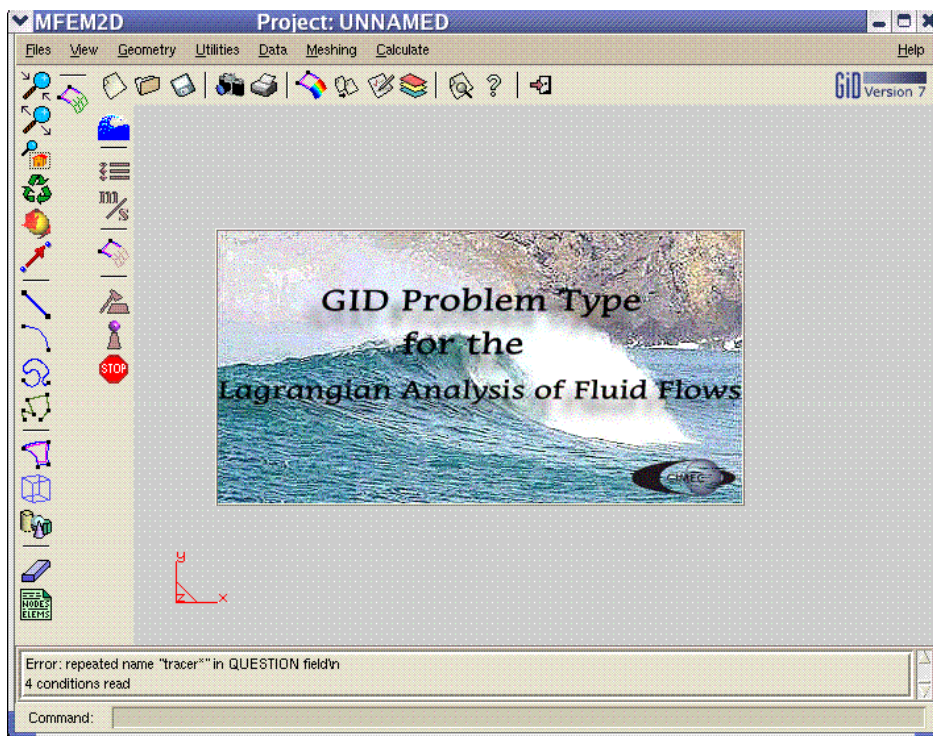


Figure 1: Problem type presentation.

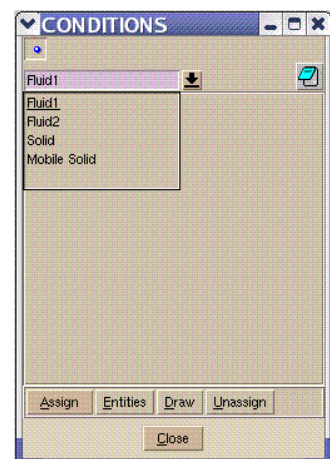


Figure 2: Condition menu

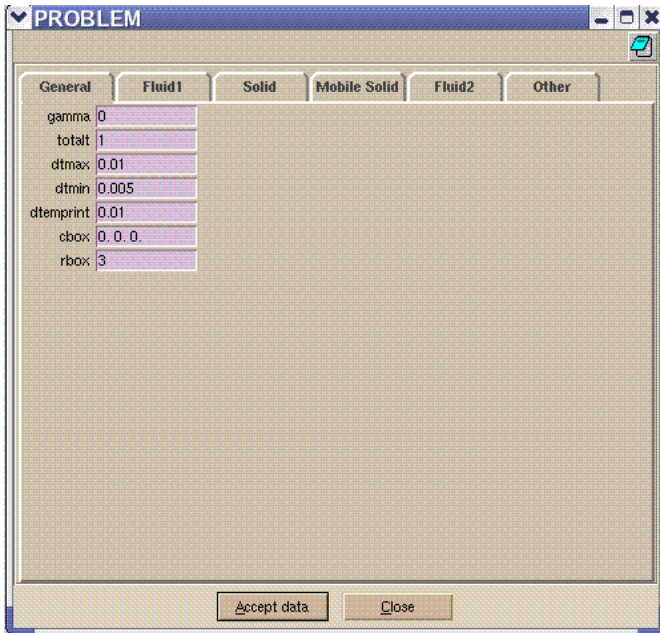


Figure 3 : Problem data menu

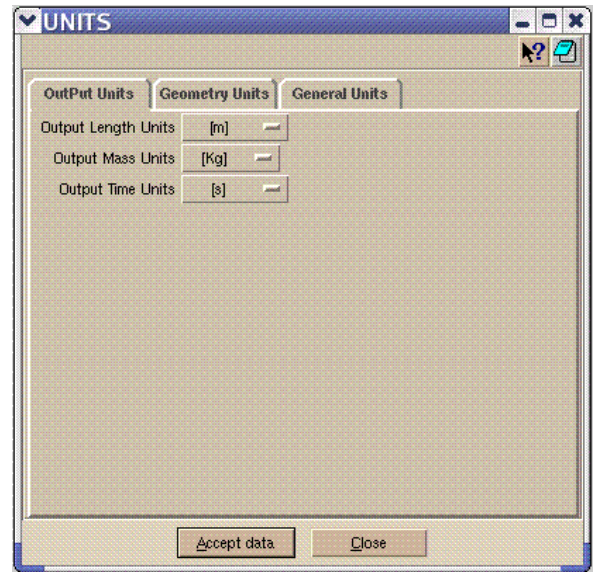


Figure 4 : Units menu

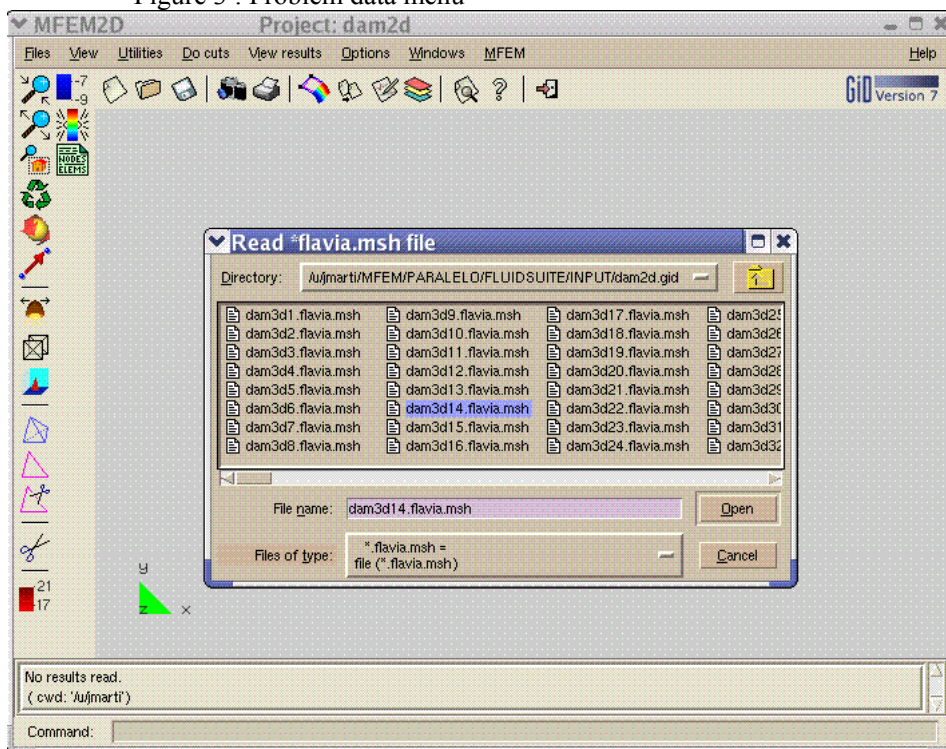


Figure 5: Post- processing menu.

## 5 GRAPHICAL AND NUMERICAL IMPLEMENTATION

A number of free-surface flow and fluid-structure interaction problems will be presented. In a first group of examples the interacting solid will be considered infinitely rigid and fixed. In a second group of examples moving rigid solid motion will be considered. In all cases, the

elastic strains will be neglected. The following examples are considered:

### 5.1 Water column collapse

Figure 6 shows the initial geometry and mesh. The water is initially supported by a removable board and the collapse starts at time  $t=0$ , when the removable board is slip-up.

The point positions at different time steps are shown in Figure 7.



Figure 6: Water column collapse. Geometry and initial mesh.

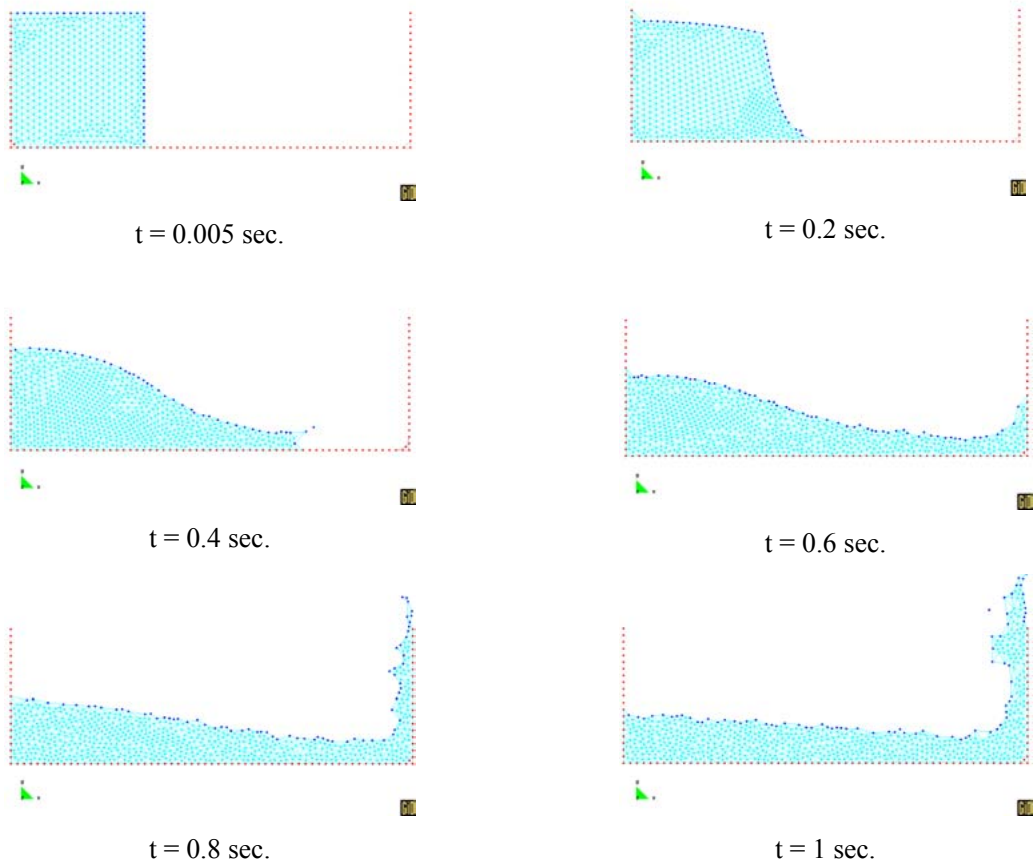


Figure 7: Water column collapse. Point positions at different time steps.

### 5.2 Mixing two different fluids

Figure 8 shows two recipients with fluid. The upper fluid has a density smaller than the bottom fluid. In the Figure 9 the velocity vectors are shown at different time steps.

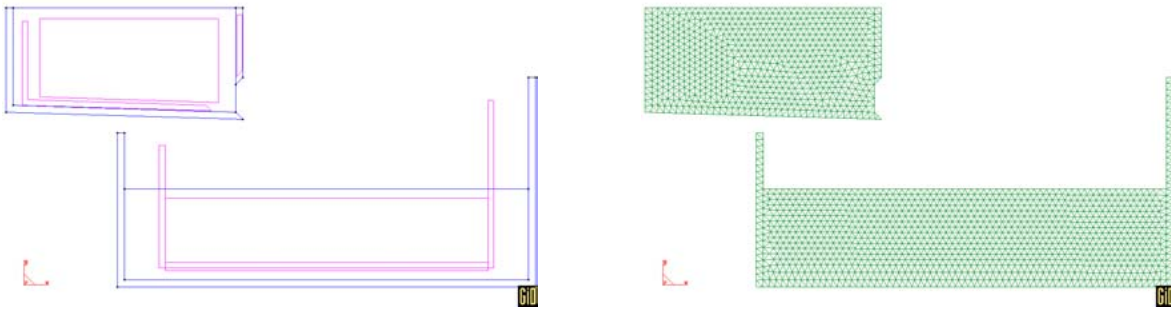
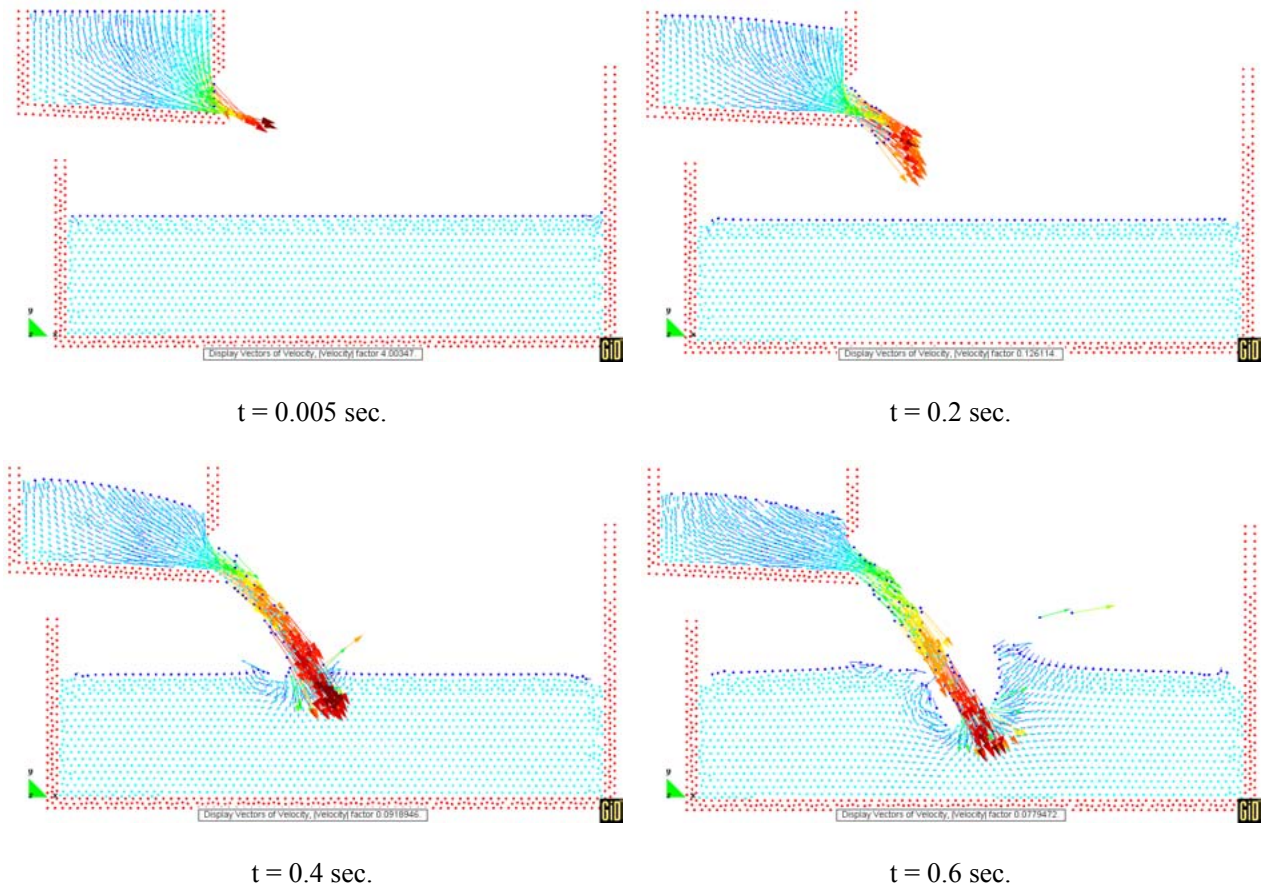


Figure 8: Mixing two different fluids. Geometry and initial mesh





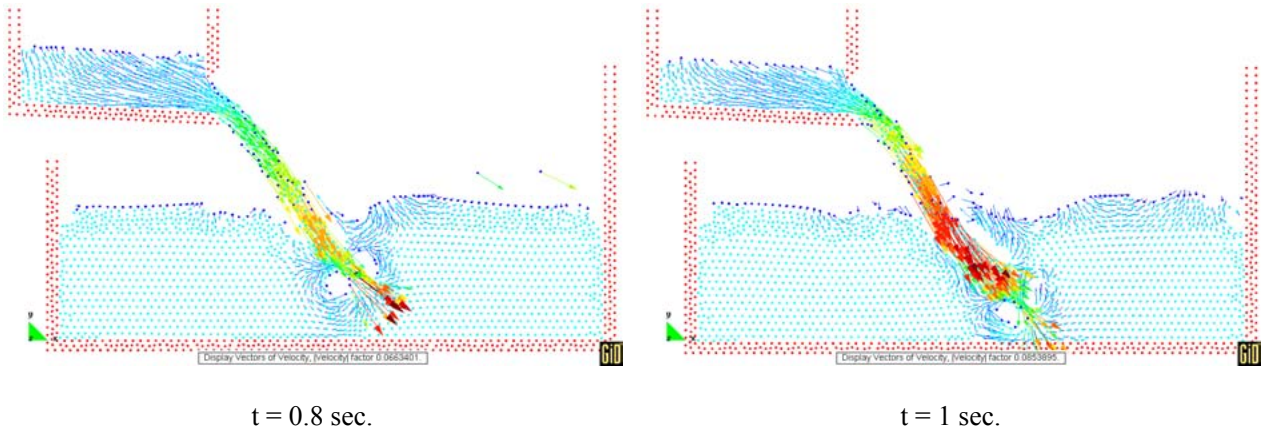


Figure 9: Mixing two different fluids. Velocity vectors at different time steps.

### 5.3 Generation of waves in a channel

Figure 10 shows the generation of waves in a channel, where the left extreme oscillates about its center of mass with a given period  $T$ . The oscillation takes place for 5 seconds. A variable distance  $h$  between particles has been introduced in order to improve the representation of the large gradients around the rocks protection. Figure 11 shows the pressure contours at different time steps.

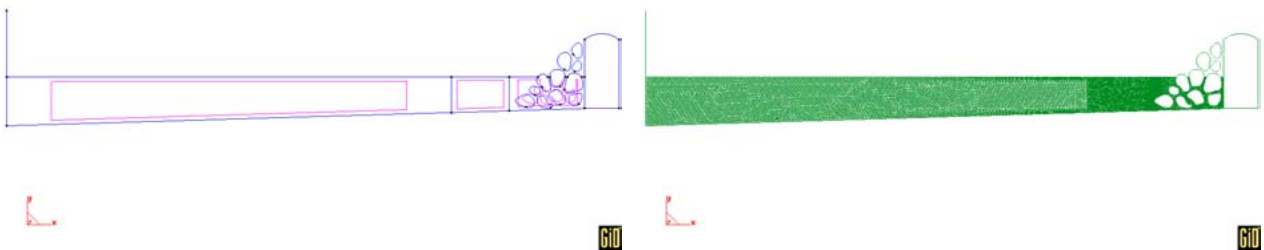


Figure 10: Generation of waves in a channel. Geometry and initial mesh

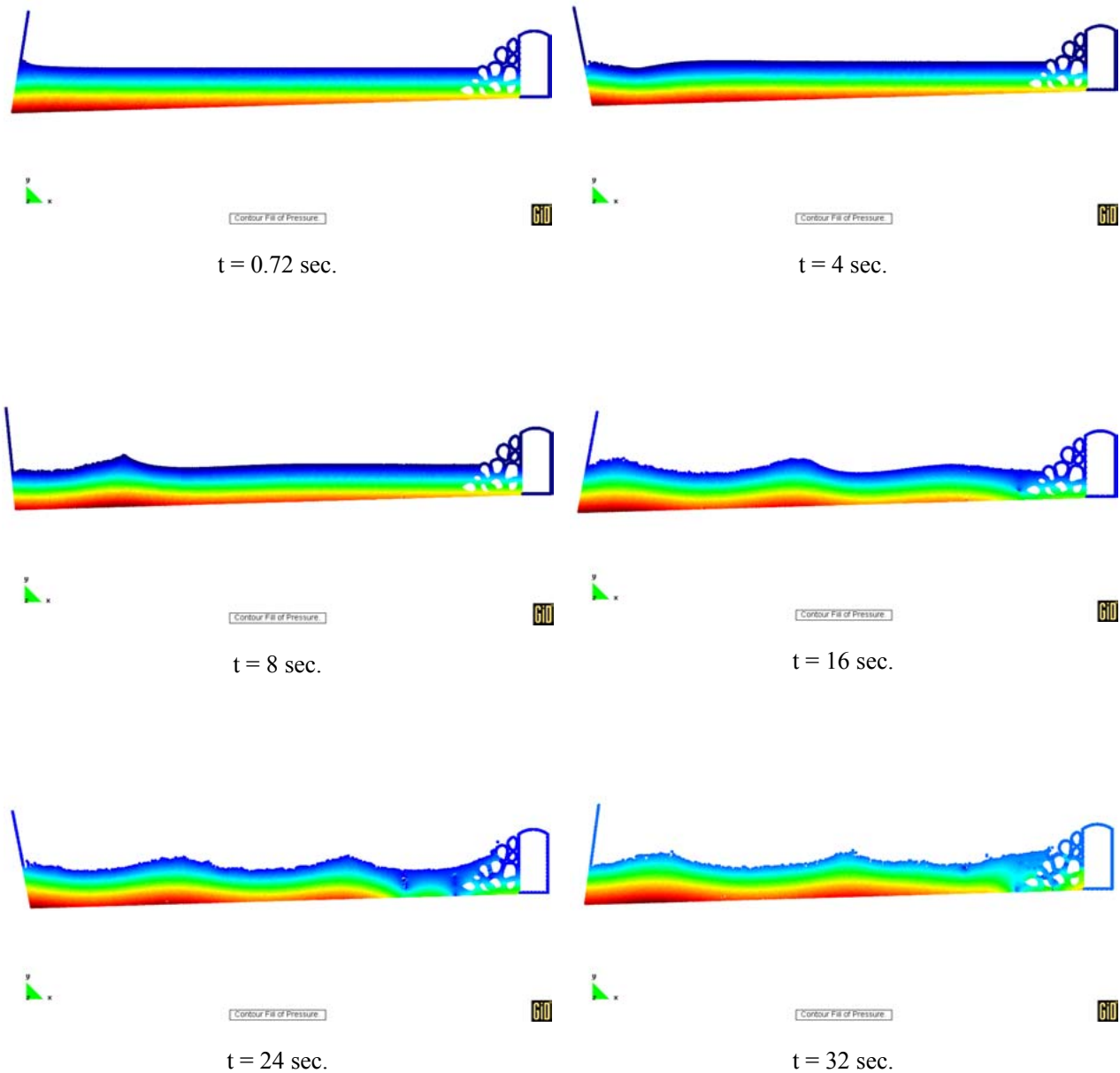


Figure 11: Generation of waves in a channel. Pressure contours at different time steps.

#### 5.4 Cube falling into water.

This example is also a case of fluid-structure interaction. The solid is initially totally free and is falling down into a recipient with fluid (Figure 13). Figure 12 shows the initial position and initial mesh. The solid is considered to be heavy compared to the liquid weight.

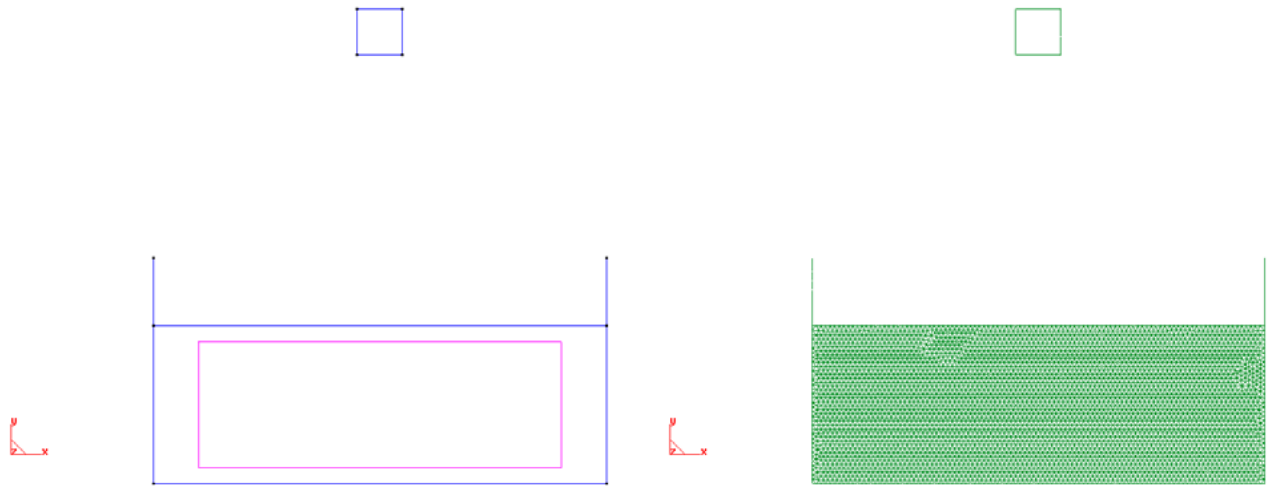
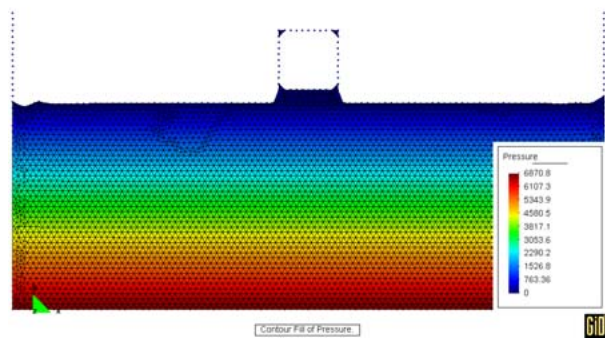
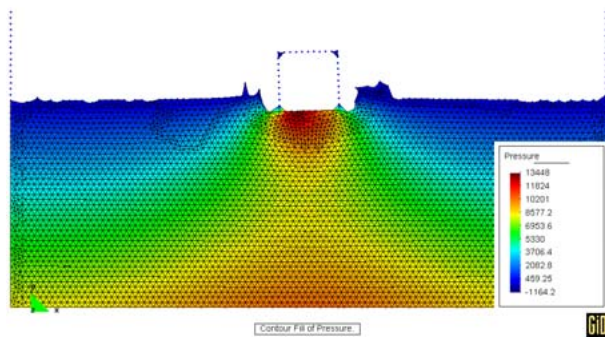


Figure 12: Cube falling into water. Geometry and initial mesh



t = 0.5 sec.



t = 0.51 sec.

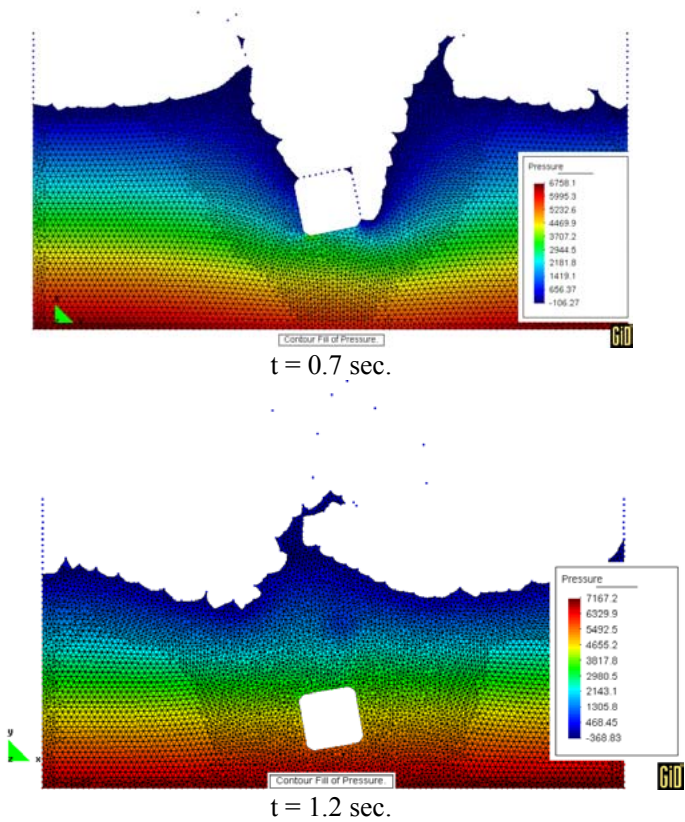


Figure 13: Cube falling into water. Contour fill of pressure for different time steps.

### 5.5 Opening of a radial gate of a weir.

This example shows the maneuver of opening of a radial gate of a weir. The radial (Taintor) gate rotates around its center of mass. Figure 14 shows the geometry and initial mesh and Figure 15 shows the point positions of the fluid at different time steps.

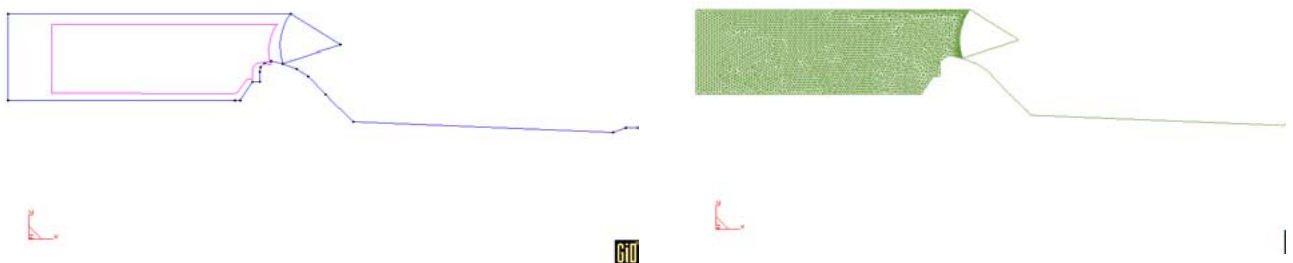
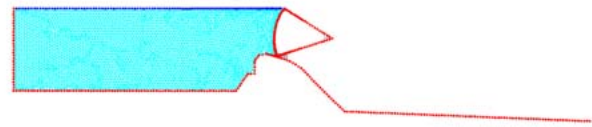


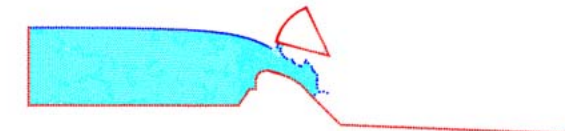
Figure 14: Opening of a radial gate of a weir. Geometry and initial mesh



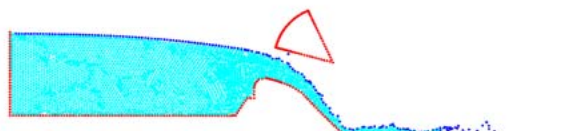
t = 0.005 sec.



t = 1 sec.



t = 2 sec.



t = 4 sec.

Figure 15: Opening of a radial gate of a weir. Positions of the fluid for different time steps.

## 6 CONCLUSIONS

In this work, a Lagrangian formulation was considered in GID to simulate 2D flow problems. Particle methods are interesting for large surface flows, large changes in the geometry (e.g. wave breaking). The problem type presented here permitted a comfortable definition of the geometry and the generation of the initial particles, while the conditions and general parameters of the problem to analyze are assigned more natural to the code. The use of the Tcl/Tk language permitted to create a menu in the post-process of GID that facilitated the selection of the output for visualization. Future work will be to continue adapting the problem type defined in this work to extend our analysis to 3D flow problems.

## 7 ACKNOWLEDGEMENTS

This work has received financial support from Consejo Nacional de Investigaciones Científicas y Tecnológicas (CONICET, Argentina, grants PIP 0198/98, PIP 0266/98), Universidad Nacional del Litoral (Argentina, grants CAI+D 96-004-024, CAI+D 2000/43) and ANPCyT (Argentina, grants PICT 12-0051, PICT 12-06973, PID-99/74, PID 99/76). We made extensive use of freely distributed software as GNU/Linux OS and the commercial GID processor (CIMNE, Barcelona).

## 8 REFERENCES

- [1] GID: The Personal Pre and Postprocessor, International, International Center for Numerical Methods in Engineering (CIMNE), URL: <http://gid.cimne.upc.es>
- [2] IDELSOHN, S., E.OÑATE, CALVO, N., and DEL PIN, F., “The meshless finite element method”, International Journal for Numerical Methods in Engineering, vol. 58-4, 2003.
- [3] DEL PIN, F, “The Meshless Finite Element Method Applied to a Lagrangian Particle Formulation of Fluid Flows”, Ph.D. Thesis, Facultad de Ingeniería y Ciencias Hídricas-Universidad Nacional del Litoral, Santa Fe, Argentina: 2003.
- [4] BELIKOV, V., V.D.IVANOV, V.K.KOMTOROVICH, S.A.KORYTNIK, and A.Y.SEMENOV, “The non-sibsonian interpolation: a new method of interpolation of the values of a function on an arbitrary set of points”, Computational Mathematics and Mathematical Physics, vol. 37, pp. 9-15, 1997.
- [5] CHORIN, A., “Numerical solution of the Navier-Stokes equations”, Math. Comp., vol. 22, pp. 745-762, 1968.
- [6] CHORIN, A., “Numerical study of slightly viscous flow”, Journal of Fluid Mechanics, vol. 57, p. 785, 1973.
- [7] TCL-TK Web page <http://www.tcl.tk/software/tcltk/>
- [8] CIMNE.GID online support. <http://www.gidhome.com/support>
- [9] CIMNE.GID Reference Manual Versión 7, 2003