International Journal of Scientific Engineering and Applied Science (IJSEAS) – Volume-6, Issue-7, July 2020
ISSN: 2395-3470
www.ijseas.com

Free Surface Flow study using the Smoothed Particle Hydrodynamics model

José Gustavo Coelho[1], Artur Elias de Morais Bertoldi[2]
[1] Institute of Exact and Technological Sciences: Dept. of Mechanical Engineering
Federal University of Triângulo Mineiro, Uberaba, Brazil
[2] Faculty of Gama, Aerospace Engineering
University of Brasília, Brasília, Brazil

Abstract

Free surface flow is a natural event involving complex effects, such as turbulence, vortex generation, strong interaction between waves and structures, etc. This work simulates a wave that breaks on an inclined surface using the Smoothed Particle Hydrodynamics (SPH) model. SPH is a meshless method and it is based on the idea that flow is considered as a set of parts of the fluid volume in motion, by using the concept of particles. In this method, the pressure field is determined by a state equation. The code is written in C ++ and validated with a reference case. Subsequently the model is applied to study the wave breaking on an inclined surface.

Keywords: *SPH, free surface, meshless, numerical simulation, fluid mechanics*

1. Introduction

The Smoothed Particle Hydrodynamics (SPH) method is relatively recent. It was created by Lucy in 1977 in the context of modeling astrophysical phenomena [1]. Currently, this method can be used to model a wide range of engineering problems, such as elastic materials [2], explosions [3], fluid mechanics [2, 4], multiphase flow [5], large deformations [6] another other.

SPH is a Lagrangian method without mesh that uses the idea of particles to model the flow. It integrates the hydrodynamic equations of motion for each unique particle in Lagrangian formalism, where physical quantities are computed for each individual particle through an interpolation of the values of the nearest neighboring ones. Subsequently, the particles move according to these values. The laws of conservation of fluid mechanics, in differential form, are transformed into their particle forms by integral equations, through the use of an interpolation function that provides the estimate of the probability density function (kernel) of the field variables in a dot. Numerically, the data is known only at discrete points (particles), so that integrals are evaluated as sums in neighboring particles.

The present study uses the SPH method to simulate the flow in the free surface condition where a wave breaks on an inclined surface. The challenges to represent the behavior of waves in coastal problems using the SPH approach have been investigated by some studies such as Barreiro et al. [7] and Altomare et al. [8]. They explored the potentials of SPH in situations of wave breaking problems, pointing out the difficulty of implementing and validating numerical methods in a structure of complex real problems. The present article investigates some of these difficulties, with the objective of obtaining more realistic results by a code written in the C ++ language.

The mathematical model considers the continuity compressible and the equations of the incompressible moment. A flow in a two-dimensional rectangular domain was used to validate the constructed code, comparing the results with experimental data. After, a case of a wave breaking on an inclined surface is analyzed. The free ParaView software was used to post-process the simulation data. The work is organized into five sections: introduction in section 1, the mathematical model of flow in section 2, followed by a description of the SPH numerical methods in section 3. Section 4 provides all the results of the work. Finally, the main conclusions of the work are shown in section 5.

2. Mathematical Formulation

In the Lagrangian form, the principle of conservation of mass and moment can be expressed by eq. 1 and eq. 2

$$\frac{D\rho}{Dt} = -\rho\nabla\cdot\boldsymbol{u} \qquad (1)$$

$$\frac{D\boldsymbol{u}}{Dt} = -\frac{1}{\rho}\nabla P + \frac{1}{\rho}\nabla\cdot\boldsymbol{\tau} + \boldsymbol{F} \qquad (2)$$

where ρ is the specific gravity of the fluid, **u** is the velocity field (which is the velocity of the particle of the material), P is the pressure field, **F** is an external force per unit mass and τ is the deviating part of the stress tensor of the fluid. The nabla operator, ∇, has the conventional meaning, being the derivative of the material, defined by $D / Dt = \partial / \partial t + u \cdot \nabla$.

To determine the pressure field, the model found in Batchelor [9] is used, according to eq. 3

$$\mathrm{P} = \frac{\rho_0 c^2}{\gamma}\left[\left(\frac{\rho}{\rho_0}\right)^{\gamma} - 1\right] \qquad (3)$$

where y = 7 and c is the speed of sound across the medium. For the determination of this constant, we use the formulation of Liu [8], given by eq. 4.

$$c^2 \cong max\left(\frac{U_0}{\delta_*}, \frac{\nu U_0}{L\delta_*}, \frac{FL}{\delta_*}\right) \qquad (4)$$

where $\delta * = \Delta\rho/\rho 0$, ρ0 is the reference density, U0 a characteristic speed, L a characteristic length and ν the viscosity. A condition that needs to be satisfied to guarantee incompressibility is that $\Delta\rho/\rho 0 < 0.03$ [3].

3. Smoothed Particle Hydrodynamics Model

In the SPH, the fluid is represented by a set of discrete particles, where each unique particle has physical properties such as pressure, mass, volume, specific mass and speed. The only parameter that does not vary is the mass of the particle. All others vary according to the evolution of time and these properties represent a spatial average over a given part of the domain. Considering a generic property α (which can be scalar, vector or tensorial), the value of a particle at a given position r is given by eq. 5.

$$\alpha(\boldsymbol{r}) = \int_{\Omega} \alpha(\boldsymbol{r'})W(\boldsymbol{r} - \boldsymbol{r'})d\boldsymbol{r'} \qquad (5)$$

where W is an interpolation function, or kernel, that obeys all the properties of a typical probability density function, as explained in [10] and Ω is the domain. This integration is done only in a part of the domain, which defines the neighborhood of the particle. In this article, we use the cubic spline kernel, defined by eq. 6

$$W(s) = \begin{cases} 1 - \frac{3s^2}{2} + \frac{3s^3}{4} & if\ 0 \leq s < 1 \\ \frac{2 - s^3}{4} & if\ 1 \leq s < 2 \\ 0 & if\ \geq 2 \end{cases} \qquad (6)$$

where s = r / h, h is the length of the filter and $r = \sqrt{(r - r') \cdot (r - r')}$. The integral in eq. 5 is approximated by a summation of the discrete particles in the vicinity of the particle. Thus, eq. 5 is reduced to eq. 7

$$\alpha(\boldsymbol{r}_a) \approx \sum_b \frac{m_b}{\rho_b} \alpha_b W_{(r_{ab})} \qquad (7)$$

where b indicates that the summation is performed over the vicinity of the particle, m is the mass and r_{ab} is the distance between particles *a* and *b*.

The moment equation in the SPH method can be written as eq. 8

$$\frac{D\boldsymbol{u}_a}{Dt} = -\sum_b m_b \left(\frac{P_a}{\rho_a^2} + \frac{P_b}{\rho_b^2}\right) \nabla_a W_{(r_{ab})} + \sum_b \frac{m_b(\mu_a + \mu_b)\boldsymbol{u}_{ab}}{\rho_a \rho_b} \left(\frac{\mathbf{1}}{r_{ab}} \frac{\partial W_{ab}}{\partial r_a}\right) + \boldsymbol{F}_a \qquad (8)$$

where μ is viscosity. F_a is the strength of the body evaluated in particle *a* and the second part of the equation is the viscous contribution. More information on this formulation is found in [11].

The continuity equation is calculated as shown in eq. 9

$$\frac{D\rho_a}{Dt} = \sum_b m_b \boldsymbol{u}_{ab} \cdot \nabla_a W_{ab} \qquad (9)$$

The time step is calculated by eq. 10

$$\Delta t = min \left\{ \begin{array}{c} 0.25 \frac{h}{c}, \\ 0.25 \min_a \sqrt{\frac{h}{f_a}}, 0.125 \min_a \frac{h^2}{\nu} \end{array} \right\} \qquad (10)$$

To avoid particle penetration and regularize the speed field, the speed equation is rewritten according to eq. 11.

$$\frac{D\boldsymbol{u}_a}{Dt} = \boldsymbol{u}_a - \varepsilon \sum_{b=1}^{N} m_b \boldsymbol{u}_{ab} \cdot \nabla_a W_{ab} \qquad (11)$$

where the constant ε varies between 0 and 0.5.

The boundary conditions used are based on the Leonard-Jones models, according to eq. 12

$$FP_{ab} = \begin{cases} D\left[\left(\frac{r_0}{r_{ab}}\right)^{1}2 - \left(\frac{r_0}{r_{ab}}\right)^{4}\right]\frac{x_{ab}}{r_{ab}^2} & if\ \frac{r_0}{r_{ab}} \leq 1 \\ 0 & if\ \frac{r_0}{r_{ab}} > 1 \end{cases} \quad (12)$$

This method uses the idea of repulsive force on particles in the domain wall. In eq. 12, D is the magnitude of the square of the maximum speed in the domain and r_0 is the initial distance between the particles. More information on boundary conditions can be found in [8] and [10].

4. Results

4.1 Dam break case

The case studied initially refers to a dam break. This example is widely used to validate numerical methods, particularly with the free surface condition. In Fig. 1, there is an illustration of the domain, where a = 1 m. The active force in this problem is gravitational and the dam is broken at time t = 0 s, when the dam is removed. In this simulation we use a spatial resolution discretization of H / 50 that results in a fluid with 20,000 particles. For the wall, we use 4 layers of particles. The filter length, h, is 1.3Δx.

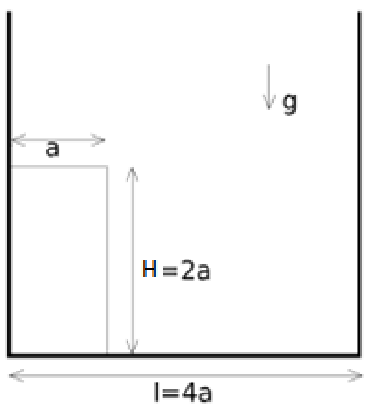

Figure 1: Geometry used in the dam break simulation.

Using negative gravity in the y direction, Monaghan [12] suggests that the initial density is determined by eq. 13

$$\rho = \rho_0 \left(1 + \frac{7\rho_0 g(H - y)}{\rho_0 c^2}\right)^{1/7} \quad (13)$$

where ρ0 = 1000 kg/m3 and H is the height of the water column in meters. When a dam of height H collapses, an approximate upper limit to the speed u of the water is given by u2 = 2gH [12]. Therefore $c = \sqrt{200gH} \approx 63\ m/s$. However, this value results in a specific mass variation greater than 1%. For a specific mass variation below this limit, we adopt c = 100 m/s. The initial flow condition is shown in Fig. 2.

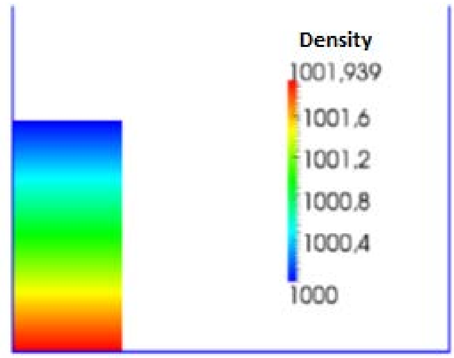

Figure 2: Initial condition of the density.

An important parameter refers to the constant ε. It is directly linked with the dissipation of the method. If it is too small, the model becomes less dissipative. If, on the other hand, it is a large value, the model becomes very dissipative, reaching the point where particle "explosions" may occur, and such behavior is totally numerical and not represent any physical effect. The value used for ε was 0.03, also used by Zheng and Duan [13]. A flow comparison is shown in Fig 3, where the results obtained numerically are compared with experimental research carried out by Koshizuka and Oka [14].

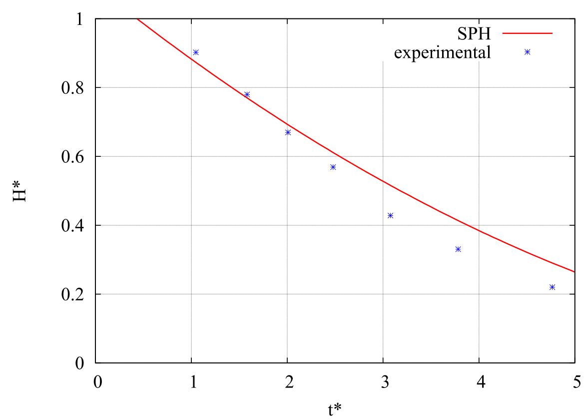

Figure 3: Comparison between numerical and experimental results, where H* = H/(2a) and

$$t^* = t\sqrt{(2\,g\,/\,a)}.$$

Figure 4 shows the flow development at different time points. You can see the wave formation and also the return of water after hitting the wall on the right side of the control volume. Another noticeable factor is the formation of the tube in the wave, which is a difficult phenomenon to be found using a mesh method. As the SPH is meshless, it is possible to capture this physical phenomenon, showing the robustness of the written code.

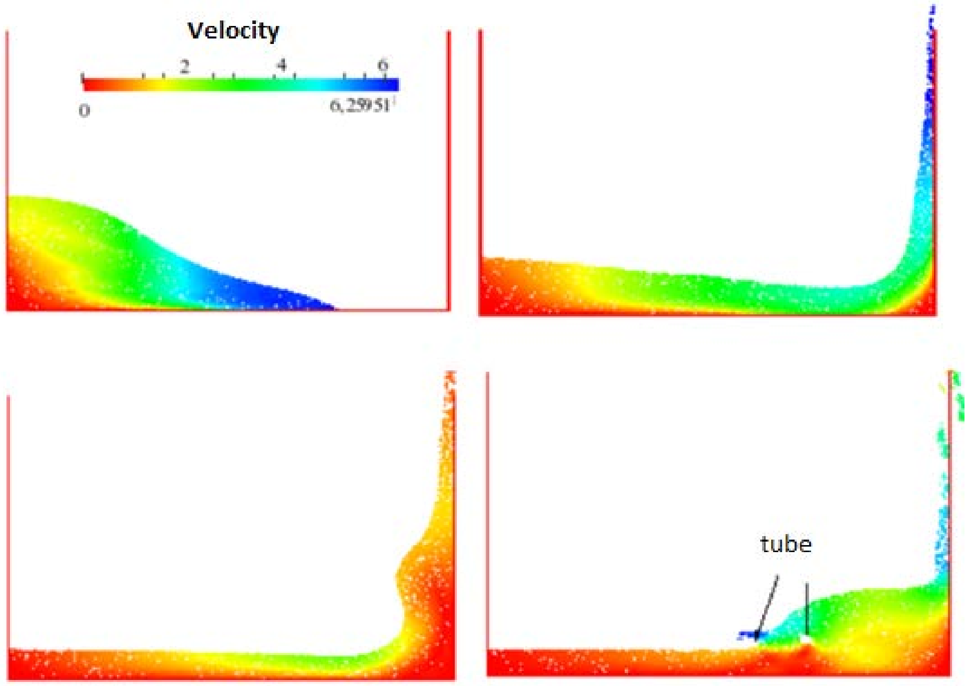

Figure 4: Numerical results for 2, 4, 6 and 8s.

4.2 Impact of waves case

This section shows the impact of waves on a sloping surface. In this simulation, the spatial resolution of the discretization was H / 100, sufficient to capture the desired effects, such as wave breaking, tube formation, etc. This discretization results in 40039 particles (39950 of fluid and 8089 of wall). The domain used is shown in Fig. 5, where one of the vertical walls has an angle of 15 °. The horizontal length is 0.5 m and the height of the water column is 0.4 m. The horizontal length was chosen to prevent smoothing of the wave amplitudes and also to reduce the computational cost. The wave is generated by moving the vertical wall as shown in Fig. 6.

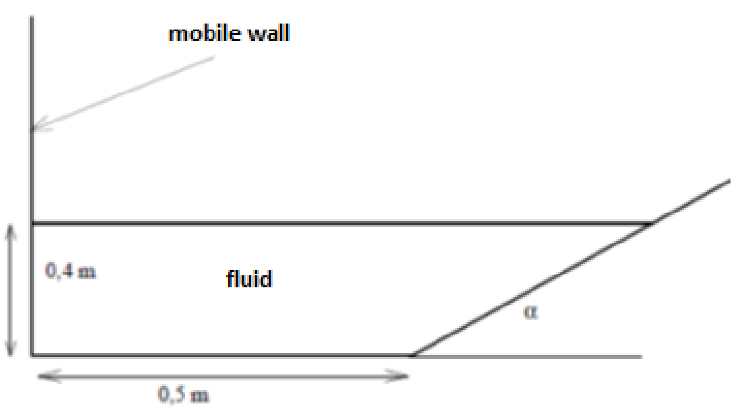

Figure 5: Domain used for simulating the impact of waves.

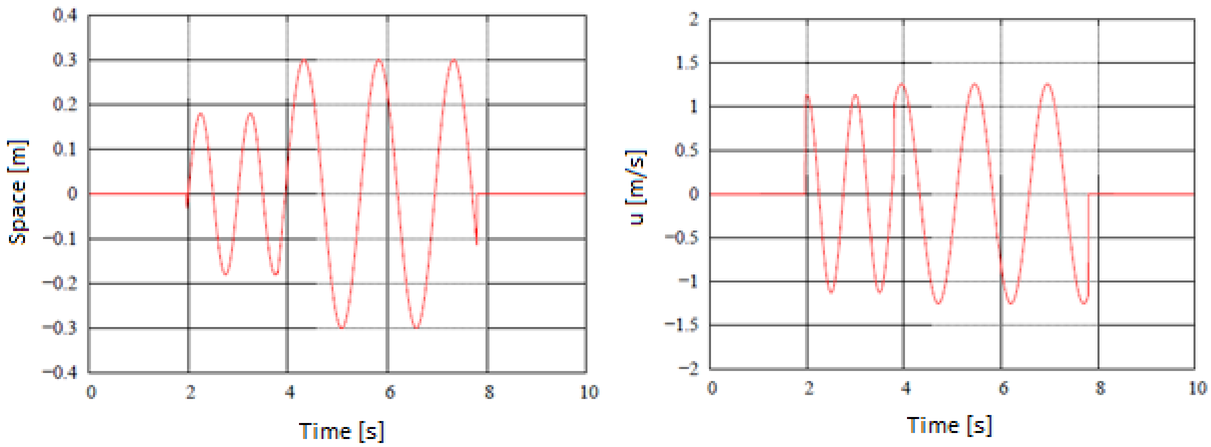

Figure 6: Displacement (up) and speed (down) of the wave generator on the horizontal axis.

Figure 7 shows the flow development at different time points. As expected, it is noticed that the velocity field presents higher values at the front of the flow. In addition, it is also noted that (in Fig. 7) the flow starts to have velocities moving upstream and downstream, while the velocity in the central part presents lower values. Figure 8 shows the moment of impact of the second wave. It is perceived the formation of recirculation that occurs because of the impact of the wave. Also, it is possible to verify the formation of the wave tube.

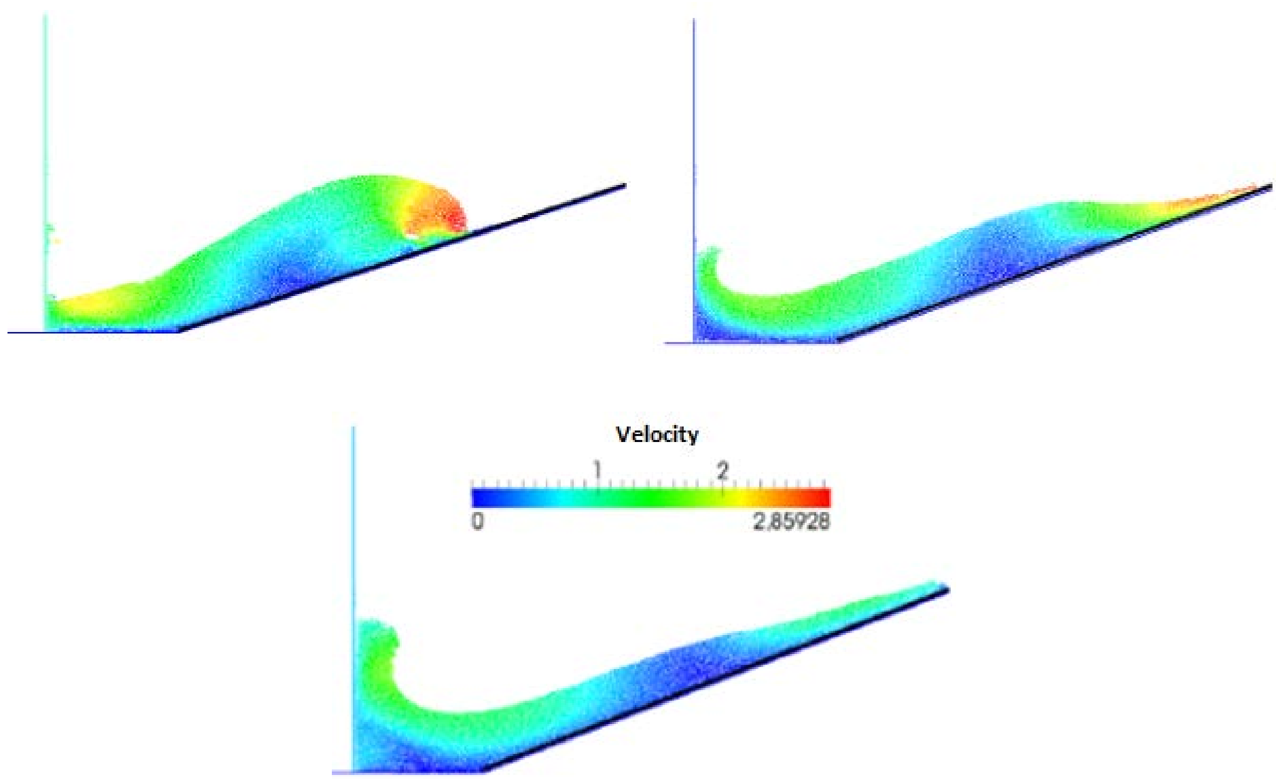

Figure 7: Flow development in 2.12 s (top left), 2.23 s (top right) and 2.38 s (down).

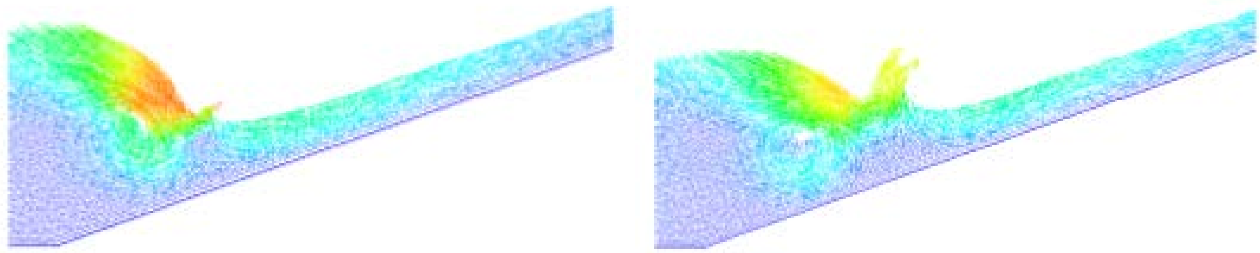

Figure 8: Flow velocity vector in time 2.71 s (left) and 2.81 s (rigth).

5. Conclusion

Using the Smoothed Particle Hydrodynamic (SPH) method, a C ++ code was developed for flow simulations with the free surface condition. The code presented satisfactory results in the analysis of the dam break compared to experimental data. After validation of the model, the wave break on a sloping surface was simulated. The code was able to reproduce important physical phenomena such as wave speed variation, recirculation zones and wave tube formation. This shows the robustness of the written code and the authors believe that the next steps of this work will address the insertion of forces in the wall. This type of analysis can be used in different fields of engineering, ranging from the soil erosion to the analysis of forces of a hydraulic turbine blade.

Acknowledgment

This work is supported by FAPEMIG project APQ-00620-14.

References

[1] Lucy, L.B. (1977). Numerical approach to testing the fission hypothesis. Astronomical Journal, 82, 1013-1024.

[2] Morris, J.P. (2000). Simulating surface tension with smoothed particle hydrodynamics. International Journal for Numerical Methods in Fluids, 33(3), 333-353.

[3] Liu, G.R.; and Liu, M.B. (2003). Smoothed particle hydrodynamics - A meshfree particle method. World Scientific, New Jersey.

[4] Hu, X.Y.; and Adams, N.A. (2007). An incompressible multi-phase SPH method. Journal of Computational Physics, 227(1), 264-278.

[5] Jian C., Ognjen O., Kenneth W., Jingijing M., Chegzhi L. "A coupled DEM-SPH model for moisture migration in unsaturated granular material under oscillation", International Journal of Mechanical Sciences, 169, 105313, 2020.

[6] Serroukh H. K., Mabssout M., Herreros M. I. "Updated Lagrangian Taylor-SPH method for large deformation in dynamic problems", Applied Mathematical Modelling, 80, 238-256, 2020.

[7] Barreiro, A.; Crespo, A.J.C.; Dominguez, J.M.; and Gomez-Gesteira, M. (2013). Smoothed particle hydrodynamics for coastal engineering problems. Computer and Structures, 120, 96-106.

[8] Altomare, C.; Crespo, A.J.C.; Rogers, B.D.; Dominguez, J.M.; Gironella, X.; and Gomez-Gesteira, M. (2014). Numerical modeling of armour block sea breakwater with smoothed particle hydrodynamics. Computer and Structures, 130, 34-45.
[9] Batchelor, G.K. (1967). An introduction to fluid dynamics. Cambridge University, Cambridge.
[10] Dezbo, E.; Zagar, D.; Krzyk, M.; Cetina, M.; and Petkovsek, G. (2014). Different ways of defining wall shear in the smoothed particle hydrodynamics simulations of a dam-break wave. Journal of Hydraulic Research, .52 (4), 453-464.
[11] Violeau, D.; and Issa, R. (2006). Numerical modelling of complex turbulent free-surface flows with the SPH method: an overview. International Journal for Numerical Methods in Fluids, 53(2), 277-304.
[12] Monaghan, J.J. (1994). Simulating free surface flows with SPH. Journal of Computational Physics, 110 (2), 399-406.
[13] Zheng, X.; and Duan, W. (2010). Numerical simulation of dam breaking using smoothed particle hydrodynamics and viscosity behavoir. Journal of Marine Science and Applications, 9(1), 34-41.
[14] Koshizuka, S.; and Oka, Y. (1996). Moving particle semi-implicit method for fragmentation for compressible fluid. Nuclear Science Engineering, 123, 421-434.

José Gustavo Coelho Graduated in Mathematics, Master in Mechanical Engineering and Doctorate (sandwich) in Mechanical Engineering at the University of Brasília (UnB) and Ecole Polytechnique Fédérale de Lausanne (EPFL / Switzerland), at the Laboratory for Hydraulic Machines. He has experience in the area of numerical flow simulation, using commercial software and also using his own program, called XSPH-UnB. Currently has work in progress in the areas of fluid mechanics, flow machines and aerospace simulation, using Eulerian (Finite Volumes and Finite Elements) and Lagrangian models, in particular the Smoothed Particle Hydrodynamics (SPH) and Discrete Element Method (DEM).

.

Artur Elias de Morais Bertoldi Professor Artur E. M. Bertoldi, graduated in Physics at Federal University of São Carlos – UFSCar (São Paulo, Brazil). He has Master and PhD degrees in Mechanical Sciences at University of Brasília - UnB (Brazil) with International Certificate in doctoral research training (CIRD) from the Aero-Thermo-Mechanics Department at Université Libre de Bruxelles (Belgium). He has professional experience in public and private institutions and is currently professor for the Aerospace Engineering undergraduate course at University of Brasília. His work focuses specifically on the following themes: hybrid rockets design, regression rate studies, liquid injection system design and testing, combustion instabilities and experimental techniques for rocket propulsion.