23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

NUMERICAL STUDY OF THE PROPAGATION OF WAVES ON FLAT BEACHES: AN APPLICATION IN ENGINEERING USING SPHysics AND FUNWAVE MODELS

Carlos Alberto Dutra Fraga Filho
Instituto Federal do Espírito Santo – IFES
Avenida Vitória, 1729 – Jucutuquara – 29040-780 – Vitória – ES – Brazil
cadff1@gmail.com

Fabio Pavan Piccoli
Universidade do Federal do Espírito Santo, Centro Tecnológico - Laboratório de Simulação de Escoamentos com Superfície Livre
Av. Fernando Ferrari, 514 – Goiabeiras – 29075-910 – Vitória – ES – Brazil
fabio_p_p@hotmail.com

Grupo de Pesquisas Costeiras NS Ambiental (NavierSat Meio Ambiente Ltda) – Vitória – ES – Brazil
naviersat@gmail.com

Danilo de Almeida Barbosa
Universidade do Federal do Espírito Santo, Centro Tecnológico - Laboratório de Simulação de Escoamentos com Superfície Livre
Av. Fernando Ferrari, 514, Campus Universitário – Goiabeiras – 29075-910 – Vitória – ES – Brazil
danilofisico@yahoo.com.br

8d (Octopus Dofleini Environmental Engineering)
Octopusengenharia@gmail.com

Julio Tomás Aquije Chacaltana
Universidade do Federal do Espírito Santo, Centro Tecnológico - Laboratório de Simulação de Escoamentos com Superfície Livre
Av. Fernando Ferrari, 514, Campus Universitário – Goiabeiras – 29075-910 – Vitória – ES – Brazil
juliotac@gmail.com

Abstract: *This study presents a comparison between the application of a Lagrangian meshless and a Eulerian model, that employs meshes, to study the generation and propagation of waves in coastal regions. The Eulerian numerical model is based on nonlinear wave equations of Boussinesq type and the Lagrangian model employs the Smoothed Particle Hydrodynamics (SPH) method. The results of numerical simulations show agreement between both models on the behaviour of the free surface. The largest discrepancy occurred in the surf zone, during the breaking, and in the wave overtopping (runup).*

Keywords*: Smoothed Particle Hydrodynamics, nonlinear wave equations of Boussinesq type, numerical method, surf zone, coastal region*

1. INTRODUCTION

Historically, the Eulerian view is that most commonly used in the modelling of physical problems involving solids and fluids. This model requires the use of numerical methods, the finite difference method (FDM), finite volume method (FVM) and finite element method (FEM), which employ grids or meshes for the solution of differential or integral formulations (Liu and Liu, 2010). Numerical methods based on a predefined mesh properly convert the governing equations into a set of algebraic equations. Generally, the unknowns of the field variables are evaluated at nodal points. However, Eulerian methods have limitations in applications that involve problems with complex geometry and in the modelling of physical processes, both related to the spatial resolution scale.

One of the major difficulties of using meshes is ensuring the consistency of the algebraic equation to be used in the solution of the physical problem. The employing of meshes can lead to several difficulties in the treatment of fluid flows presenting free surfaces, mobile interfaces, complex geometries and topological changes. If these features exist, the generation of a mesh (a prerequisite for simulation quality) is a difficult, time-consuming and expensive process.

Recently, Lagrangian modelling has been presented as an alternative for solving problems of small and large spatial scale and improving understanding of physical processes. In the Lagrangian method, the physical domain of the problem is divided into small parcels of matter that interact with each other. In this procedure the volume of the fluid domain is divided into a set of discrete particles with defined masses (Lagrangian elements). The mass of each particle is a material point which receives the coordinates that define its position in space. Thus, the physical properties of each Lagrangian fluid element are mapped in the domain for each moment of time.

The Lagrangian particle model does not employ meshes and is an alternative method used in the search for more effective computational ways of solving more complex problems, where stable and accurate numerical solutions to integral equations and partial differential equations are required.

The advantage of using Lagrangian modelling rather than Eulerian is its simplicity in dealing with complex geometries and physical processes. Also, there is no need to use meshes, there is no production numerical oscillations and the mass conservation at the Lagrangian element is guaranteed. Visualisation of the results obtained by the Lagrangian method allows better understanding of the spatiotemporal evolution of flow.

2. MATHEMATICAL MODELLING

The evolution of the velocities field and the transport of mass and energy across space and time is performed by differential equations that accompany matter and by the physical laws that govern the conservation of mass, momentum and energy.

2.1 Smoothed Particle Hydrodynamics Lagrangian Method

The Lagrangian method known as Smoothed Particle Hydrodynamics (SPH) was developed in the late 1970s for modelling three-dimensional astrophysical phenomena (Lucy, 1977; Gingold and Monaghan, 1977). Over time, the method was extended to other areas of applied science such as solid mechanics and fluid mechanics.

To solve the differential equations of conservation, the SPH method discretises the continuous domain of matter in a finite number of material particles. The value of the physical quantity associated with each fixed particle is obtained by weighted interpolation of physical properties contained in the neighboring particles. The particles used for interpolation are within an influence domain (delimited by the circle of radius kh that is the product of a scale factor and the support radius, h) and contribute to the approximation of the physical quantity in the fixed particle (or particle a), as shown in Fig. 1.

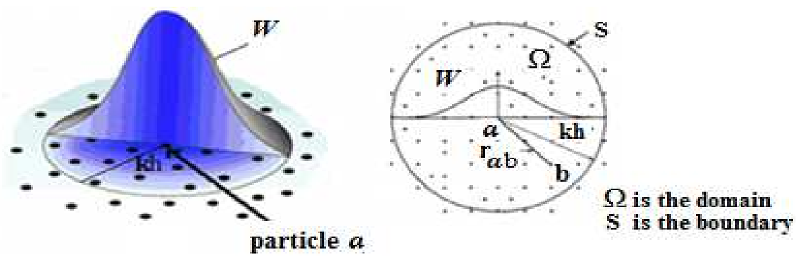

Figure 1. Graphical representation of the domain of influence. The fixed particle a has as neighbors all particles within the influence domain (r_{ab} is the distance between two particles).

The smoothing function, or kernel (W), quantifies the effects of a particle in its neighborhood, ensuring the greatest contribution of neighboring particles closest to the value of the physical quantity in the fixed particle. The choice of an appropriate length for the support radius defines the stability and accuracy of computer simulations. The initial number of neighboring particles should be 5, 21 and 57, in 1D, 2D and 3D, respectively (Liu and Liu, 2003).

In the SPH method, different kernels can be used for the distribution of the weights used in the interpolation. They must satisfy the properties of softness, normalisation, and positivity, be decreasing, symmetric and convergent and have compact supports (Kelager, 2006).

Through the SPH, second-order approximations are obtained for the values of the physical properties for a fluid or a solid. The general equation commonly used in the SPH method to represent the interpolation of a physical property of the neighbors in the particle reference is given by the following expression:

$$A^a = \sum_{b=1}^{n} m_b \frac{A^b}{\rho^b} W(r^a - r^b, h) \tag{1}$$

where:
A^a is the quantity being approximated and concerns the fixed particle a,
n is the number of particles inside the influence domain,
m_b is the mass of the neighboring particle b,
W is the kernel,

h is the support radius,
r^a is the position of the fixed particle,
r^b is the position of the neighboring particle,
ρ_b is the density of the neighboring particle,
A^b is the value of the quantity in each particle b, in the neighboring of fixed particle a.

Different kernels are used in SPH (Gaussian, quadratic, cubic spline, quintic Wendland, among others). In the literature, the cubic spline kernel is one of the most popular, as it provides the mathematical properties previously listed beyond to present a desirable behavior of its first derivatives.

The following cubic spline kernel is used in the method to two or three dimensions (Gomez-Gesteira *et al.*, 2010).

$$W(r,h) = \text{factor}\begin{cases} 1 - \dfrac{3}{2}\left(\dfrac{|r|}{h}\right)^2 + 4\left(\dfrac{|r|}{h}\right)^3, & se\ \ 0 \le |r| \le h \\[2ex] \dfrac{1}{4}\left[2 - \left(\dfrac{|r|}{h}\right)^3\right], & se\ \ 0 \le |r| \le 2h \\[2ex] 0\ , \ \text{in the other case.} \end{cases} \tag{2}$$

where:

$$r = r^a - r^b \tag{3}$$

$$\text{factor} = \begin{cases} \dfrac{15}{7\pi h^2}, & \text{in } 2D, \\ \dfrac{1}{\pi h^3}, & \text{in } 3D. \end{cases} \tag{4}$$

In the present work, this kernel was used for two-dimensional domains.

In the case of fluid flow with free surfaces, the SPH method presents some techniques for their treatment without the need for remeshing at each iteration of the numerical simulation (which complicates the use of traditional methods of meshes for this type of problem). This is one of its clear advantages.

Forces applied to the free surface of the fluid are usually not modelled by the conservation laws; they are considered as boundary conditions. On the free surface, the forces are unbalanced because of the action of surface tension forces. The resultant force on each particle acts in the direction normal to the free surface of the fluid, pointing toward the centre, tending to minimise the curvature and the area, as shown in Fig. 2. For the definition of the free surface, additional field greatness is employed, known as colour field, which allows the calculation of the force acting on each particle belonging to that surface (Muller *et al.*, 2003; Kelager, 2006).

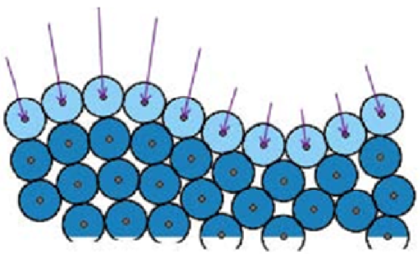

Figure 2. The action of surface tension forces (arrows pointing into the fluid)
particles on the free surface (lighter blue) (Kelager, 2006).

Turbulence in the SPH method can be treated by the application of the large eddy simulation method (Gomez-Gesteira *et al.*, 2010).

One method of treating solid boundaries is based on molecular dynamics and dissipative particle dynamics. Areas with solid boundaries are filled with frozen particles that prevent the penetration of mobile particles. The prevention of penetration by moving particles is performed by a repulsive force applied by virtual particles, simulating the boundary conditions of zero normal velocity. The use of a line of virtual particles (type I) located on the solid contours producing repulsion is commonly applied in SPH to implement the boundary conditions. However, there are situations where the particles in the line can have motions to simulate wavemakers, for example (Monaghan, 1994).Virtual particles of the second type (type II) can also be allocated outside the boundary, as shown in Fig. 3 (Liu and Liu, 2003).

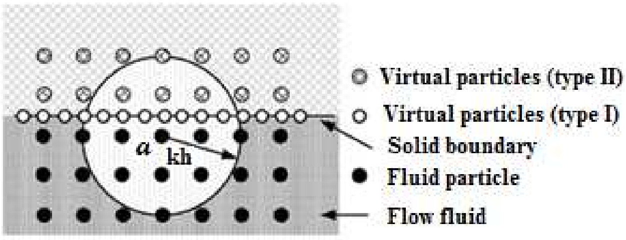

Figure 3. Schematic illustration of the solid boundary region. Configuration of a line of virtual particles (type I). Outside the contour, the virtual particles are type II. Adapted from (Liu and Liu, 2003).

The *SPHysics* Lagrangian numerical model was used to simulate the wavemaker. This is an open computer code developed in *FORTRAN* language and based on Smoothed Particle Hydrodynamics (SPH) for the study of free surface flows. It is the result of collaboration by researchers from Johns Hopkins University (USA), the University of Vigo (Spain) and the University of Manchester (UK). Figure 4 shows the flowchart of the computational algorithm of the SPH method.

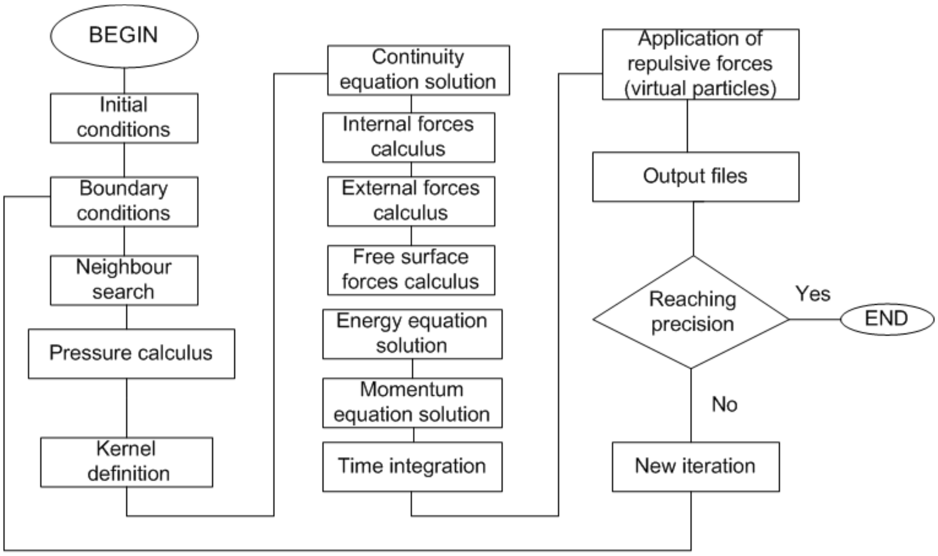

Figure 4. Flowchart of the computational algorithm using the SPH method.

2.2 Eulerian Method: Fully Nonlinear Boussinesq Type for Waves

The Eulerian numerical model *FUNWAVE2D* is a non-commercial software, produced by the Center for Applied Coastal Research (CACR, Delaware, USA), in *FORTRAN* language. It is capable of simulating surface waves in coastal regions including areas of external and internal surfing. The model is based on Boussinesq type equations for second-order dispersion. These equations are obtained after integration of mass and momentum conservation equations in the vertical direction (Wei *et al.*, 1995).

Continuity equation:

$$\frac{\partial \eta}{\partial t} + \frac{\partial M_i}{\partial x_i} = 0 \qquad (5)$$

where the flux M_i is:

$$M_i = (h+\delta\eta)\left\{u_i + \mu^2\left[\frac{1}{2}z^2 - \frac{1}{6}\left(h^2 - h\delta\eta + \delta^2\eta^2\right)\right]\frac{\partial}{\partial x_i}\left(\frac{\partial u_j}{\partial x_j}\right) + \right.$$

$$\left. + \ \mu^2\left[z + \frac{1}{2}(h-\delta\eta)\right]\frac{\partial}{\partial x_i}\left[\frac{\partial\left(hu_j\right)}{\partial x_j}\right]\right\} + O\left(\mu^4\right) \tag{6}$$

Momentum conservation:

$$\frac{\partial u_i}{\partial t} + \delta\left(u_j\frac{\partial u_i}{\partial x_j}\right) + \frac{\partial\eta}{\partial x_i} + \mu^2 J_i + \delta\mu^2 K_i = O\left(\mu^4\right) \tag{7}$$

where non-linear terms J_i and K_i are given as:

$$J_i = \frac{1}{2}z^2\frac{\partial}{\partial x_i}\left[\frac{\partial}{\partial x_j}\left(\frac{\partial u_j}{\partial t}\right)\right] + z\frac{\partial}{\partial x_i}\left[\frac{\partial}{\partial x_j}\left(h\frac{\partial u_j}{\partial t}\right)\right] - \ldots$$

$$\frac{\partial}{\partial x_i}\left[\frac{1}{2}(\delta\eta)^2\frac{\partial}{\partial x_j}\left(\frac{\partial u_j}{\partial t}\right) + \delta\eta\frac{\partial}{\partial x_j}\left(h\frac{\partial u_j}{\partial t}\right)\right] \tag{8}$$

$$K_i = \frac{\partial}{\partial x_i}\left\{(z-\delta\eta)\left(u_j\frac{\partial}{\partial x_j}\right)\left[\frac{\partial\left(hu_j\right)}{\partial x_j}\right] + \frac{1}{2}\left(z^2 - \delta^2\eta^2\right)\left(u_j\frac{\partial}{\partial x_j}\right)\left(\frac{\partial u_j}{\partial x_j}\right)\right\} + \ldots$$

$$\frac{1}{2}\frac{\partial}{\partial x_i}\left\{\left[\frac{\partial\left(hu_j\right)}{\partial x_j} + \delta\eta\frac{\partial u_j}{\partial x_j}\right]^2\right\} \tag{9}$$

where the level z is the reference depth for calculating the velocities, which may be given as:

$$z \approx -0.531\,h_0 \tag{10}$$

where:

η is the surface elevation,

h is the total water depth,

t is the time,

$\delta = a_0/h_0$ and $\mu = k_0 h_0$ are scales of nonlinearity and dispersion, respectively,

a_0, h_0 and k_0, are, in sequence, the typical wave amplitude, the depth of water at rest and the number of waves,

$u_i = (u, v)$ is the velocity at depth in the coordinated z: $u_i = \left(\frac{\partial\phi}{\partial x_i}\right)$, ϕ being the velocity potential.

To solve the partial differential equations that govern the movement of water, the numerical method employed by *FUNWAVE2D* 2.0 is the finite difference scheme. A detailed description of the model can be found in the *FUNWAVE2D* 2.0 manual (KIRBY *et al.*, 2005). Figure 5 shows the flowchart of the computational algorithm.

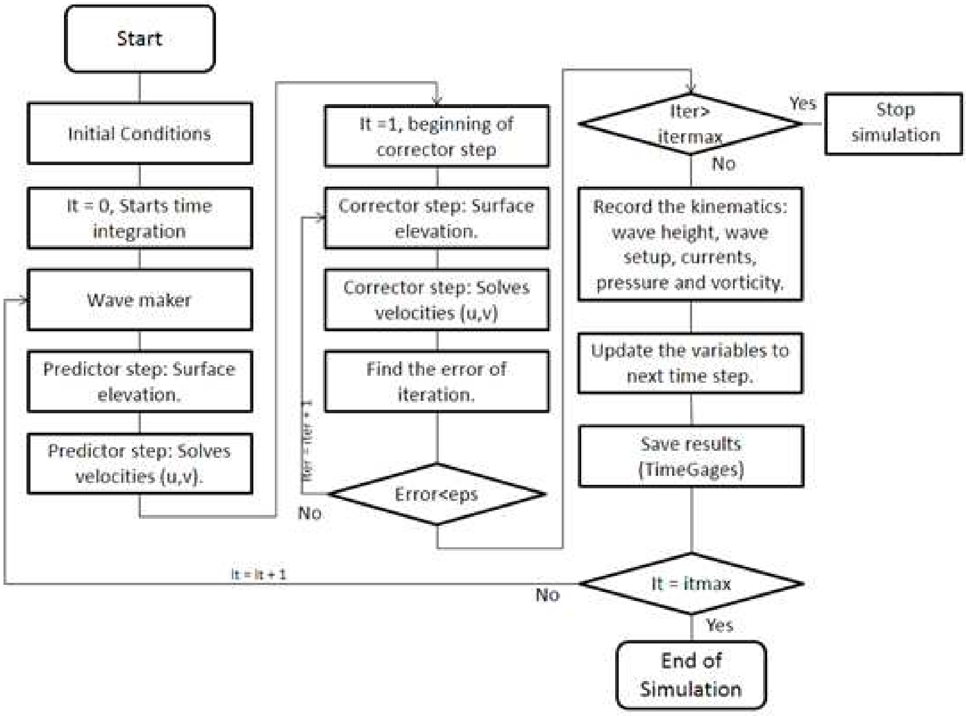

Figure 5. FUNWAVE2D flowchart software.

3. NUMERICAL SIMULATIONS

The updating of particle properties in time (position, velocity, energy, and others) was performed by numerical integration methods. These can be diverse, ranging from the Runge-Kutta first order (Euler) method and Leap-Frog to more elaborate algorithms such as predictor-corrector, Verlet, Beeman or Sympletic (Gomez-Gesteira *et al.*, 2010).

The stability of the methods applied depended on the choice of time step (Δt). The Courant-Friedrichs-Lewy numerical stability criterion (CFL) ensured the convergence of results (Courant, Friedrichs and Lewy, 1967).

4. CASE STUDY

In order to compare the results obtained from the SPH Lagrangian particle method with those of the Euler method a study of generation and wave propagation on a flat beach is presented.

It was simulated a beach domain some 2.75 m long in the inclined region, with an inclination angle of 4.2364°, amplitude and wave period of 0.01 and 1.4 s, respectively, and water level of 0.18 m. For the generation of waves a wavemaker was employed.

In the Eulerian simulation, a domain mesh with 156 nodes in the horizontal direction and 21 nodes in the vertical direction was used. The spacing between dots in the horizontal direction was 0.05 m with a total horizontal distance of 7.75 m. Vertically, the spacing varied with the total depth, given by $H = h + \eta$, where h is the mean sea level and η is the surface elevation, defining a sigma-type mesh. As the method of wave generation employs a generative source function, the model requires a region of constant depth away from obstacles. Because of this, the generating source was allocated in mesh node 41, at horizontal coordinate -2.00 m. The origins of the referentials were coincident in Eulerian and Lagrangian simulations. The dimensions of the Eulerian domain ranged from -4.00 m to 3.75 m. The Eulerian simulation period was 30.00s, with a time increment of 0.001 s during the simulations. The CFL number assumed a value of 0.20 in simulations.

A cubic spline kernel was used in Lagrangian simulations and 4221 fluid particles were employed to discretizing the domain of interest; from 0 to 3.75 m. The lateral distance between the center of mass of two neighboring particles was 0.01 m. The support radius was defined as 0.013 m (approximately 1.30 times the initial spacing between the centers of mass of the particles). The virtual particle technique was used to represent the fluid boundaries. An amount of 387 particles was used to represent the fixed bottom of the channel and an amount of 31 particles to represent the motion of the flap-type wavemaker. A time step of 4.5x10-5 s was used to simulate the generation e propagation of water waves over a time simulation of 30s. For the time integration, in both Eulerian and Lagrangian simulations, the predictor-corrector method was used in order to generate a water wave of 0.02 m amplitude.

4.1 Results and Discussion

The free surface elevation was obtained by Lagrangian and Euler models (SPHysics and FUNWAVE2D, respectively) in different simulation times. Figure 6 shows the numerical results at 28.0, 28.7 and 29.4 s.

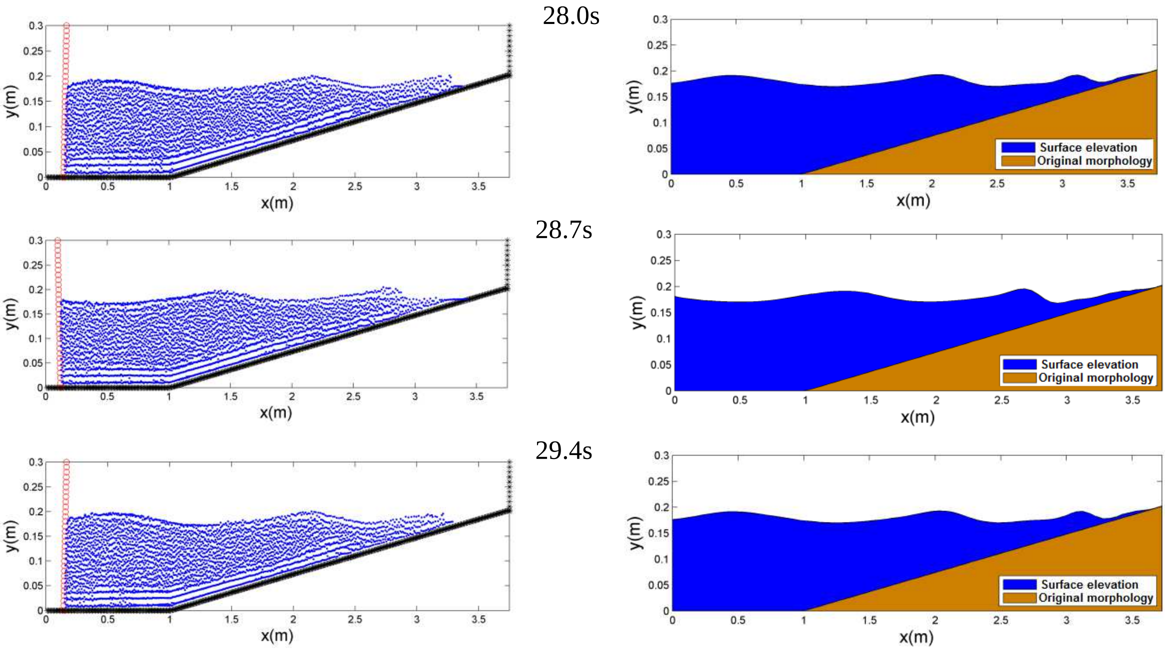

Figure 6. Simulations results. On the left, the results of the SPHysics model; on the right, the results of the FUNWAVE2D model.

From the analysis of Fig. 6, it can be concluded that the simulation results obtained from both models were consistent with the behavior of a propagating wave on a flat beach. Looking to the free surface comparisons in Fig. 6, the eulerian model shows higher agreement with lagrangian model before the wave break point.

In the Eulerian model, the wave breaking is treated using a formulation of artificial turbulent viscosity to promote a more realistic description of its beginning and end, in quantitative terms (related to the energy dissipated). A simple eddy viscosity-type formulation is used to calculate the wave energy dissipation caused by the wave breaking (Kennedy *et al*., 2000). It has been proven that the adopted wave breaking model can adequately predict the energy dispassion for both spilling and plunging breakers, but lacks accuracy in waveform. It is well-known that the depth-integrated Boussinesq-type models cannot describe the overturning of a free surface and the detailed wave breaking process.

The Lagrange model has the advantage of presenting the collapse of the wave in shallow water in a shape that is visually representative. During a simulation, the waveform (defined by the water particles whose positions change over time) is visible and consistent with the physical phenomenon that is taking place.

Figure 7 shows the results for two models at 28.0 s, in just a single graphical representation. It can be observed that both are in agreement outside the surf zone but show some discrepancy in the slope near the shore.

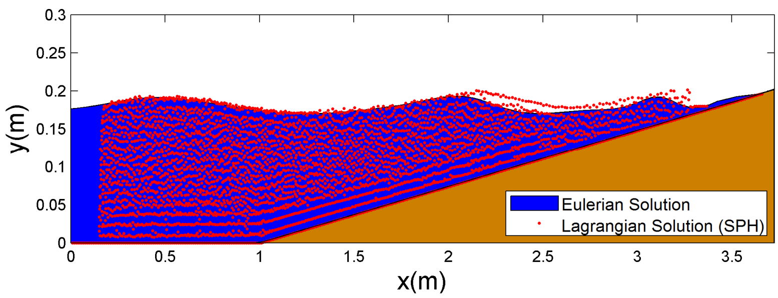

Figure 7. Graphical comparison of the results found for SPHysics and FUNWAVE models at simulation time of 28.0 s.

To test the coupling and joint use of the models, studies were conducted beforehand. The propagation of a solitary wave in a tank of constant depth was studied (Narayanaswamy *et al.*, 2010). The results obtained are shown in Fig. 8.

A solitary wave is generated in a first region by FUNWAVE and propagated until the beginning of the second region, where SPHysics is employed, being reflected on the wall that defines the end of the latter region. Then, the reflected wave returns to the FUNWAVE region.

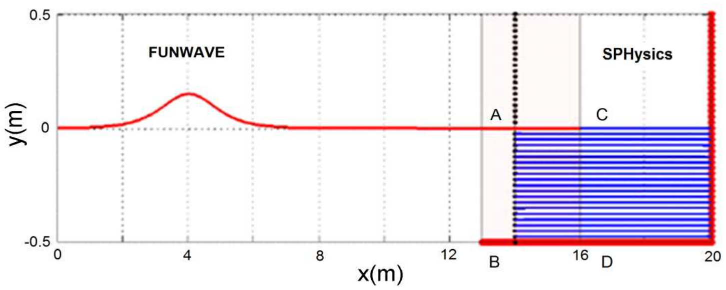

Figure 8. Initial configuration of the case study employing the coupling of the SPHysics and FUNWAVE models (Narayanaswamy *et al.*, 2010).

From the analysis of the numerical simulations performed, it was concluded that there was agreement between the results obtained by both models in regions of constant mean depth, which makes possible the coupling shown above.

5. CONCLUSIONS

The numerical results obtained from both models showed good agreement with regard to the *behavior* of the free surface elevation over the whole area of the beach. The difference between the results is limited to the surf zone, where the Eulerian model uses an approach to consider wave breaking and the Lagrangian model treats this phenomenon according to the equations of conservation and energy transport. Although the results found by the Lagrangian model are more consistent with the physics of the problem in the whole area studied, they should be confronted with data from measurements or analytical solutions for a better evaluation of the numerical solution obtained.

6. REFERENCES

Courant, R.; Friedrichs, K. and Lewy, H., 1967. “On the partial difference equations of mathematical physics”. *IBM Journal*, Vol. 11, p. 215-234.

Gomez-Gesteira, M.; Rogers, B. D.; Dalrymple R. A.; Crespo A. J. C. and Narayanaswamy M., 2010. *User Guide for SPHysics Code*. <https://wiki.manchester.ac.uk/sphysics/images/SPHysics_v2.2.000_GUIDE.pdf>

Gingold R. A. and Monaghan J. J., 1977. “Smoothed Particle Hydrodynamics: Theory and Application to Non-Spherical Stars”. *Monthly Notices of the Royal Astronomical Society*, Vol. 181, p. 375-338.

Kelager, M., 2006. “*Lagrangian Fluid Dynamics Using Smoothed Particle Hydrodynamics*”. Department of Computer Science, University of Copenhagen, Denmark.

Kennedy, A. B.; Chen, Q.; Kirby, J. T. and Dalrymple, R. A., 2000. “Boussinesq modeling of wave transformation, breaking and runup. I: 1D”. *Journal of Waterway, Port, Coastal, and Ocean Engineering*, Vol. 126, n. 1, p. 39–47.

Kirby, J. T.; Long, W. and Shi, F., 2005. *FUNWAVE 2.0: Fully Nonlinear Boussinesq Wave Model on Curvilinear Coordinates. Part I (Model Formulations) and Part II (User’s Manual).* University of Delaware, Center for Applied Coastal Research, Dept. of Civil & Environmental Engineering. Newark, Delaware.

Liu · M. B. and Liu G. R., 2003. “Smoothed Particle Hydrodynamics: A Meshfree Particle Method”. World Scientific, Singapore.

Liu · M. B. and Liu G. R., 2010. “Smoothed Particle Hydrodynamics (SPH): an Overview and Recent Developments”. *Arch. Comput. Methods Eng*., Vol. 17, p. 25-76.

Lucy L. B., 1977. “Numerical approach to testing the fission hypothesis”. *Astronomical Journal*, Vol. 82, p. 1013-1024.

Monaghan J. J., 1994. “Simulating free surface flows with SPH”. *J. Comput. Phys*., Vol. 110(2), p. 399-406.

Muller, M.; Charipar, D. and Gross M., 2003. “Particle-Based Fluid Simulation for Interactive Applications”. In *Proceedings of Eurographics/SIGGRAPH Symposium on Computer Animation*. San Diego, California, USA.

Narayanaswamy, M.; Crespo, A. J. C.; Gomez-Gesteira, M. and Dalrymple, R. A., 2010. “SPHysics-FUNWAVE hybrid model for coastal wave propagation”. *Journal of Hydraulic Research*, Vol. 48, p. 85-93.

Wei, G.; Kirby, J. T.; Grilli, S. T. and Subramanya, R., 1995. “A Fully Nonlinear Boussinesq Model Equations for Surface Waves. Part 1. Highly Nonlinear, Unsteady Waves”. *Journal of Fluid Mechanics*, Vol. 294, p. 71-92.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.