XXXII Convegno Nazionale di Idraulica e Costruzioni Idrauliche
Palermo, 14-17 settembre 2010

SPH MODELLING OF HYDRAULIC JUMP WITH HIGH INFLOW FROUDE NUMBER

D. De Padova[1], M. Mossa[1], S. Sibilla[2], E. Torti[2]

(1) Dipartimento di Ingegneria per l'Ambiente e per lo Sviluppo Sostenibile D.I.A.S.S. Politecnico di Bari, Italia, e-mail: d.depadova@poliba.it; m.mossa @poliba.it.
(2) Dipartimento di Ingegneria Idraulica e Ambientale, University of Pavia, Pavia, Italy, e-mail: stefano.sibilla @unipv.it, emanuela.torti@unipv.it

ABSTRACT

The application of a 2-D pseudo-compressible XSPH scheme to the analysis of hydraulic jumps is discussed. The effect of velocity smoothing procedures of SPH schemes and of turbulence modelling on the solution is analysed. The numerical model is then applied to the simulation of an undular jump at Fr=8.3 generated in a very large channel of the Coastal Engineering Laboratory of the Water Engineering and Chemistry Department at Politecnico di Bari. The study makes particular reference to the velocity and free surface profile measured in the longitudinal central section of the channel, with the aim of analysing the hydraulic jump development. The agreement between the numerical results and laboratory measurements is satisfactory.

1 INTRODUCTION

Hydraulic jump occurs when a supercritical open-channel flow dissipates a large part of its kinetic energy and converts into a subcritical flow. Dissipation takes place usually through a strong turbulent roller which characterizes the jump. Different types of hydraulic jump (undular, weak, steady, strong. etc..) can occur depending on the value of the upstream Froude number, and several different hydraulic phenomena (strong turbulence effects, downstream wave propagation, unsteadiness, air entrainment) characterize the different jump types.

The present paper discusses the modelling of the development of high-Froude undular jumps by the SPH method. SPH is a purely Lagrangian method developed during seventies (Lucy, 1977; Gingold and Monaghan, 1977) in astrophysics to study the collision of galaxies and the impacts of bolides on planets. Although the traditional grid-based numerical methods such as the finite difference (FDM) and the finite element methods (FEM) are, in general, better established methods than SPH, they can show difficulties in representing complex free-surface unsteady phenomena. This motivated researchers to seek for alternatives to simulate these kind of flows, and meshless Lagrangian methods, such as SPH, proved to be one of the best choices.

In SPH the fluid is represented by particles which are free to move in space, carry all the computational information, and thus form the computational frame for solving the partial differential equations describing the conservation laws of the continuum fluid dynamics. The numerical method has been shown to be robust and applicable to a wide variety of fields. Among the variety of flows, the hydraulic jump appears to be a suitable test case to validate different SPH models developed to analyse unsteady turbulent open-channel flows, although only few SPH results of this type exist (Gallati and Braschi, 2003).

In a previous study (De Padova et al., 2009), SPH modelling was applied to the simulation of an undular jump at Fr = 3.9 and successfully tested using physical experiments on supercritical flow motion by Ben Meftah et al. (2007; 2008).

The present paper first investigates the influence of some of the SPH numerical parameters and of the possible turbulence models by comparing 2D SPH solutions of a weak jump at Fr = 2 with reference literature data (Chow, 1959).

The application of SPH modelling to the analysis of an undular jump with a high Froude number (Fr = 8.3) is then discussed. Results are compared with laboratory data obtained in a very large channel of the Coastal Engineering Laboratory of the Water Engineering and Chemistry Department at Politecnico di Bari (Ben Meftah et al, 2008).

The agreement of computed velocity profiles and water elevations with laboratory data confirms the adequacy of the SPH description.

2 EXPERIMENTAL SET UP

The experimental investigation was carried out at the Coastal Engineering Laboratory of Valenzano (L.I.C.) of the Water Engineering and Chemistry Department.

The system (Fig. 1) is made by a rectangular steel channel having the base and the lateral walls in transparent glass material of thickness 15 mm, connected and sealed internally with silicone rubber, watertight and also able to prevent thermal dilatation. The plant has a base surface 15 × 4 m and is placed at a distance of 0.96 m from the floor; the height of the walls, and therefore the maximum allowed depth of the channel was 0.4 m. A closed hydraulic circuit provided the proper flow rate into the channel.

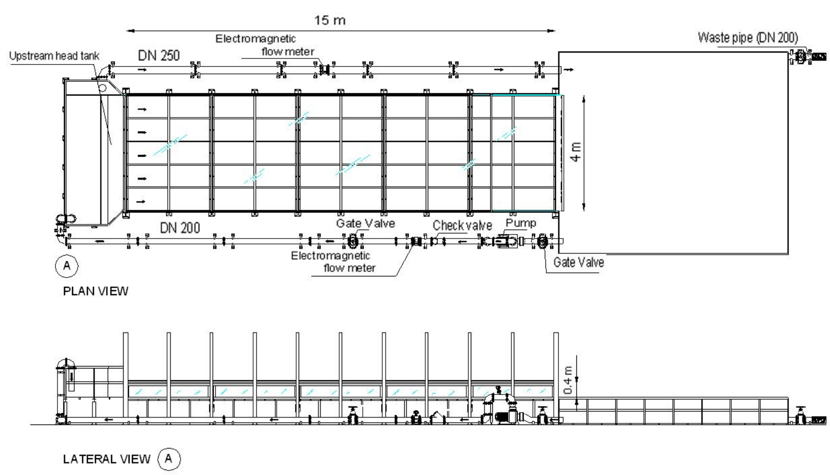

Figure 1. Representation of the channel used for the experimental study.

Fig. 2 shows a picture of the channel in which the upstream tank is clearly visible, together with the lateral pipe directed towards the downstream tank in order to regulate the total discharge flowing along the main channel.

For the measurement of the velocity the two-dimensional Nortek ADV (Acoustic Doppler Velocimeter) system was used, together with CollectV software for the data acquisition and ExploreV software for the data analysis, all of them produced by Nortek.

Water height was measured using an ultrasonic measuring system UltraLab ULS 20130 by General Acoustics, characterized by a resolution of 0.18 mm. Further details on the experimental tests are reported in Ben Meftah et al. (2007; 2008).

Figure 2. Picture of the channel.

The test analysed in the present paper is characterized by a constant flow rate of 0.1 m^3/s. Table 1 shows the main experimental parameters of the investigated hydraulic jump. Specifically, U_0 is mean flow velocity at the vena contracta, Fr_0 is the flow Froude number at the vena contracta, AR is the jump aspect ratio equal to B/h_0 , where B is the channel width and h_0 is the flow depth at the vena contracta, $Re = Q/(B\nu)$ is the Reynolds number calculated by using the kinematic viscosity evaluated for the measured water temperature, l is the longitudinal distance from upstream channel to the toe of the shockwave, L is the longitudinal length from the upstream channel gate to the hydraulic jump front normal to the upstream current and β is the angle between the lateral shockwave and the channel side wall.

TEST	Jump type	U_0 [m/s]	Fr_0 [-]	AR [-]	Re [-]	l [m]	L [m]	β [°]
1	Full developed	2.56	8.3	409.8	18046	4.6	6.1	40

Table 1. Experimental parameters characterizing the analysed hydraulic jump

Figure 3 also shows b_j as the length of hydraulic jump front normal to the upstream current. Since the channel is very large, the hydraulic jump front is trapezoidal, as shown in the sketch of Fig. 3, with presence of lateral shock waves.

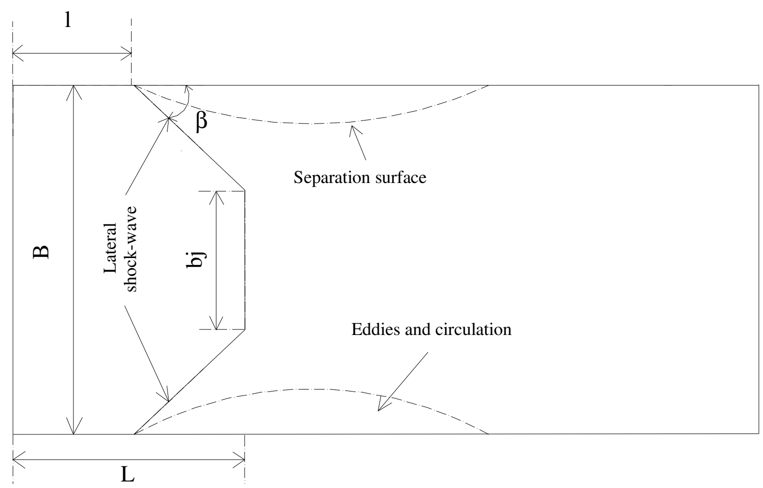

Figure 3. Main geometric parameters of the analysed undular jump.

3 SPH NUMERICAL METHOD

The SPH method is based on the discretization of the fluid volume in motion by N_p moving fluid particles, each having a constant mass *m*. The value of each flow variable in a generic point $\vec{x}$ is obtained by an interpolation based on the values in the surrounding particle centers, through an interpolation or *kernel* function $W(\vec{x},h)$. It is required that the kernel function be continuous, non-zero only within a circle of radius 2*h*, normalized by $\int_{\Re^n} W(\vec{x},h)d\vec{x} = 1$ and that it tends to the Dirac delta function when *h* tends to zero; *h* is defined as the *smoothing length*. When the SPH interpolation is applied to the equations of motion, the system of partial differential equations is reduced to an algebraic system in the unknown flow variables in the moving points (Monaghan, 1992a).

The jump flow is here simulated by adopting a weakly compressible flow formulation (WCSPH), where the incompressible fluid is approximated by a slightly compressible one having a suitable equation of state (Monaghan, 1992b). The flow motion is therefore here described by the Navier-Stokes equations for a weakly compressible fluid, which, in the SPH approximation, take the form:

$$\frac{D\rho_i}{Dt} = -\sum_j m_j \left(\vec{v}_j - \vec{v}_i\right) \cdot \vec{\nabla} W_{ij} \tag{1}$$

$$\frac{D\vec{v}_i}{Dt} = -\sum_j m_j \left(\frac{p_i}{{\rho_i}^2} + \frac{p_j}{{\rho_j}^2} \right) \vec{\nabla} W_{ij} + \sum_j \frac{m_j}{\rho_i \rho_j} \; (\bar{\bar{\tau}}_j - \bar{\bar{\tau}}_i) \cdot \vec{\nabla} W_{ij} + \vec{g} \tag{2}$$

where $\vec{v}$ is velocity, p pressure, ρ density, $\bar{\bar{\tau}}$ the total (viscous + turbulent) stress tensor, $\vec{g}$ gravity acceleration, $W_{ij} = W(\vec{x}_i - \vec{x}_j, h)$ and the summation is extended to all the fluid particles j of mass m_j located at a distance $\left|\vec{x}_j - \vec{x}_i\right| < 2h$ from the generic particle i where the flow variables are evaluated.

Two turbulence models have been tested. The first is based on the introduction of a mixing-length $l_m = f_i \min(\kappa z, l_{\max})$, where κ=0.41 is the Von Kármán constant, z is the distance from the wall, $l_{\max}$ is a cut-off maximum value and

e . (3)

is a damping function which is less than unity only near the free surface. The effect of f_i is to lower l_m near the free-surface, where a non-physical growth of the eddy viscosity may lead to numerical instabilities.

The second is an SPH version of the Standard k-ε turbulence model (Launder and Spalding, 1974):

$$\begin{aligned} \frac{Dk}{Dt} &= P_{k_i} + \frac{1}{\sigma_k} \sum_j m_j \frac{\nu_{T_i} + \nu_{T_j}}{\rho_i + \rho_j} \frac{k_i - k_j}{r_{ij}^2 + 0.01h^2} \mathbf{r}_{ij} \bullet \nabla W_{ij} - \varepsilon \\ \frac{D\varepsilon}{Dt} &= C_{\varepsilon_1} \frac{\varepsilon_i}{k_i} P_{k_i} + \frac{1}{\sigma_\varepsilon} \sum_j m_j \frac{\nu_{T_i} + \nu_{T_j}}{\rho_i + \rho_j} \frac{\varepsilon_i - \varepsilon_j}{r_{ij}^2 + 0.01h^2} \mathbf{r}_{ij} \bullet \nabla W_{ij} - C_{\varepsilon_2} \frac{\varepsilon_i^2}{k_i} \end{aligned} \tag{4}$$

where P_k is the production of turbulent kinetic energy depending on the local rate of deformation, ν_T is the eddy viscosity and σ_k = 1, σ_ε = 1.3, $C_{\varepsilon 1}$ = 1.44 and C_{e2} = 1.92 are model constants whose values are those proposed for the standard k-ε formulation.

At each time step, the updated velocity $\vec{v}^T$ of each particle is obtained by explicit integration of the momentum equation (2). A smoothed value $\vec{v}^S$ of velocity is then obtained by:

$$\vec{v}^S(\vec{x}_i) = (1-\varphi) \; \vec{v}^T(\vec{x}_i) + \varphi \frac{\sum_j \frac{m_j}{\rho_j} \vec{v}^T(\vec{x}_j) \; W_{ij}}{\sum_j \frac{m_j}{\rho_j} \; W_{ij}} \tag{5}$$

where φ is a smoothing parameter.

Time integration is obtained according to the XSPH scheme (Monaghan, 1992a), where the smoothed velocity $\vec{v}^T$ is integrated explicitly in time to obtain the updated position of the particles while the smoothed values $\vec{v}^S$ are used in the solution of the continuity equation (1).

Pressure is then computed by the equation of state for weakly compressible fluids:

$$p - p_0 = \frac{\varepsilon}{\rho}(\rho - \rho_0) \tag{6}$$

where ε is the compressibility modulus while the index 0 is relative to a reference state. Pressure values are also corrected by a smoothing procedure similar to (5), but applied only to the difference between the local and the hydrostatic pressure (Sibilla, 2008).
When WCSPH method is adopted, the time step is limited by the Courant condition:

$$CFL = \frac{\Delta t\sqrt{\varepsilon}}{h\sqrt{\rho}} < 1 \tag{7}$$

however, (7) leads to excessively small values of the time step when the real value of ε for water is adopted. Reasonable values for the time step Δt can be obtained only if a reduced compressibility modulus ε_R is adopted in the simulations. It has been verified by Monaghan (1992) that realistic SPH simulations of incompressible flows can be obtained in this way, provided that the local numeric Mach number $Ma = |\vec{v}|\sqrt{\rho/\varepsilon_R}$ be everywhere lower than 0.1.

Wall boundary conditions are applied by the "ghost particle" method, i.e. new particles are placed in a 2*h*-wide layer beyond the physical boundary by mirroring the positions of the internal particles. Flow variables are assigned to each ghost particles depending on the values pertaining to the related mirrored particle, in order to obtain the desired boundary condition at the wall (Takeda et al., 1994, Randles and Libersky, 1996).

4 RESULTS

A detailed analysis of the dependence of the numerical solution on the physical and numerical parameters of the model was first performed. Free surface profiles were computed for the case of a 2D weak jump at Fr = 2 and compared to reference experiments (Chow, 1959).

Some of the results are here summarized. Figure 4 shows that high values of the smoothing parameter φ can cause a too high dissipative effect in the simulated jump, leading to a steeper free surface profile and a shorter jump length: a smoothing coefficient $\varphi = 0.02$ appears to yield results closer to the reference data. The simulation appears to be less sensitive to the turbulence model: both the mixing length model with different l_{max}/h_1 values and the standard k-ε model yield similar results (fig. 5).

According to the preliminary results, simulations of the structure of a high-Froude undular jump in the mid-section of a wide channel were performed by adopting a smoothing coefficient $\varphi = 0.02$ and a mixing length turbulence model with l_{max}/h_1 = 0.25.

The computational domain consists in a rectangle 2 m long and 0.4 m high. The simulated domain is therefore shorter than the experimental channel in order to reduce computational costs: care was paid to verify that results were not altered by the domain length. Particles were initially placed on a staggered grid with zero initial velocity.

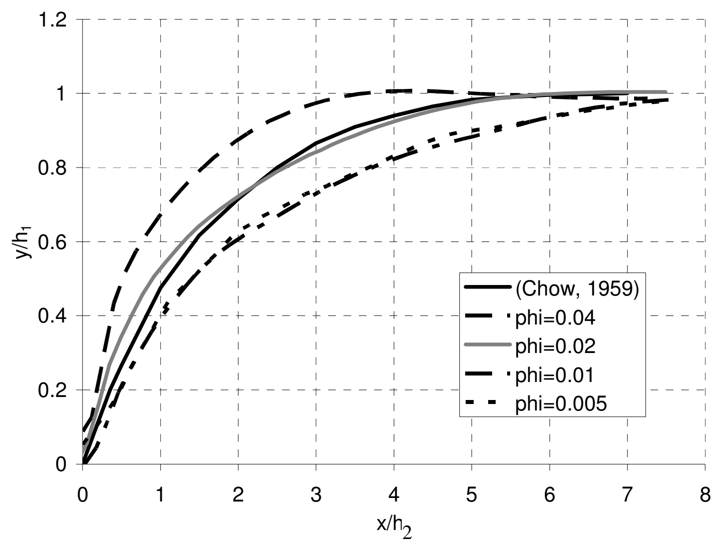

Figure 4. Effect of the smoothing parameter φ on the free surface profile in a weak jump simulation at Fr =2, obtained with a mixing length turbulence model (l_{max} = 0.8 h_1).

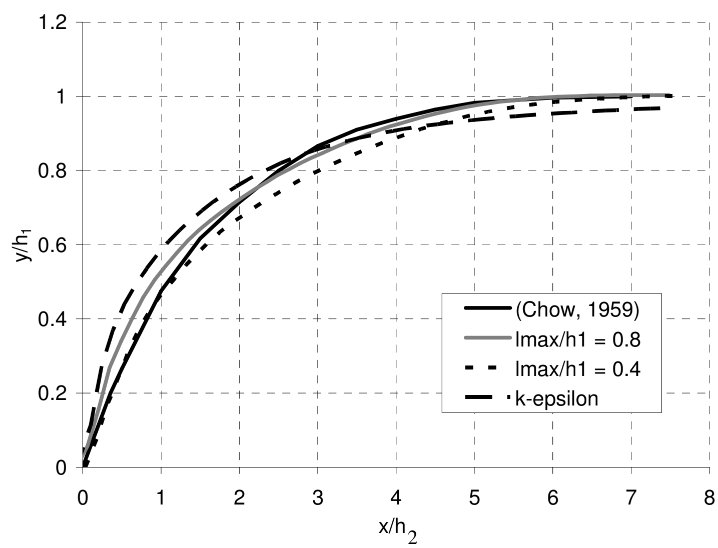

Figure 5. Effect of the turbulence model on the free surface profile in a weak jump simulation at Fr =2, , obtained with smoothing parameter φ = 0.02.

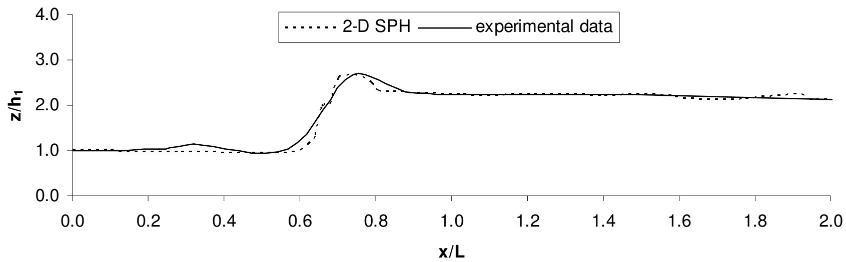

Figure 6. Comparison of the numerical and experimental elevation profile in the central longitudinal section of the wide channel.

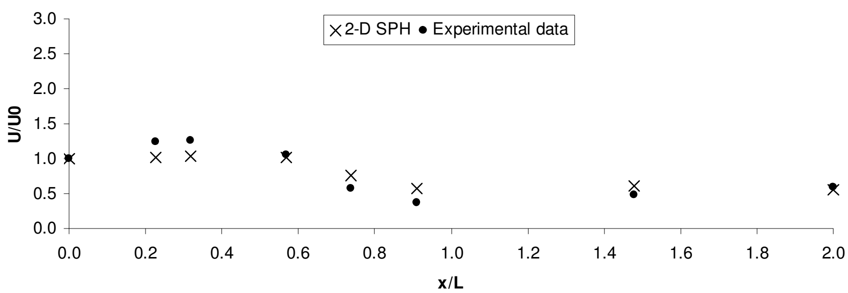

Figure 7. Time averaged horizontal velocity component of longitudinal velocity of the undular hydraulic jump in wide channel at *Fr*=8.3.

The choice of initial particles spacing $\Delta x=\Delta z$ depends on the physical process of the problem and the desired computational accuracy and efficiency. The particle spacing is taken as 0.002 m and thus approximately 26,000 particles are used . It can be shown that the efficiency of the SPH kernel is based on the choice of the $\Delta x/h$ term (De Padova et al., 2008). Here a smoothing length h=0.003 m and, thus, a value of $\Delta x/h$=0.67, was used. In the simulation, the upstream flow depth h_1 and the downstream flow depth h_2, were equal to 0.028 m and 0.06 m, respectively.

With the aim to analyse the hydraulic jump development and to compare the numerical results with the experimental data of Ben Meftah et al. (2007; 2008), the study made particular reference to the velocity and free surface profiles.

Regarding the distribution of the water elevation along the channel at *Fr*=8.3, figure 6 shows a comparison between the free surface profile of the hydraulic jump obtained by the 2D SPH model and the one measured in the longitudinal central section of the channel in the laboratory. It can be observed that the longitudinal profile obtained with the numerical simulations were in fairly good agreement with the experimental one obtained in the experimental measurements in both tests. In the zone of the formation of the hydraulic jump the agreement was weakly worse.

Figure 7 shows the comparison between the averaged horizontal velocity component obtained in the vertical sections along the longitudinal central section of the channel ($0 < x/L < 2$), at a point distant 0.01 m from the bottom for both tests. The term x is the longitudinal distance assumed equal to zero in the first section in the numerical code.

The numerical data are in good agreement with the values measured in the physical model except in the zone of the hydraulic jump formation (0.6< x/L <1.0), where the computational results are slightly overestimated. In can be nevertheless stated that the numerical model predicted the mean velocity along the channel with satisfactory agreement.

5 CONCLUSION

The SPH model was applied to the modelling of undular jump with an high Froude number (Fr=8.3) generated in a very large channel of the Coastal Engineering Laboratory of the Water Engineering and Chemistry Department at Politecnico di Bari.

A preliminary analysis on the influence of physical and numerical parameters of the model was carried out on 2D weak jump (Fr=2). The most reasonable behaviour of the free surface profile was found with smoothing parameter φ set to 0.02. Both the mixing length and k-ε turbulence models showed good agreement with reference experiments.

The SPH model was then used to simulate the high-Froude undular jump case and its results were verified by comparison with flow measurements taken in the aforementioned laboratory installation. The SPH model provided a good prediction in term of time-averaged water depths and time-averaged longitudinal velocity components. The free surface profile and the averaged horizontal velocities were predicted with satisfactory accuracy, except in the zone where a series of small rollers on the surface of the jump (0.6 < x/L < 1.0) where the agreement was weakly worse. Therefore, the SPH model reproduced the formation of the hydraulic jump in fairly agreement with the laboratory data.

As a further development of the present research, in order to analyse and reproduce also the lateral shock waves represented in Figure 3, the development of a 3D SPH model is planned.

REFERENCES

Ben Meftah M., De Serio F., Mossa M. and Pollio A, Analysis of the velocity field in a large rectangular channel with lateral shockwave. *Environmental Fluid Mechanics*, 7: 519-536, 2007.

M. Ben Meftah M., De Serio F., Mossa M. and Pollio A, Experimental study of recirculating flows generated by lateral shock waves in very large channels. *Environmental Fluid Mechanics*, 8: 215-238, 2008.

Chow V. T., Open Channel Hydraulics, McGraw-Hill Book Company, New York, 1959.

De Padova D, Dalrymple. R.A., Mossa M and Petrillo A.F., An analysis of SPH smoothing function modelling a regular breaking wave. Proc. 31° Convegno Nazionale di Idraulica e Costruzioni Idrauliche, Perugia, 182, 2008.

De Padova D., Mossa M. and Sibilla S., Laboratory experiments and SPH modelling of hydraulic jumps, proc. 4th Spheric Workshop, Nantes, 255-257, 2009.

Gallati M. and Braschi G., Numerical description of the jump formation over a sill via SPH metod. Proc. CMFF'03, 2: 845-852, Budapest, 2003.

Gingold R. A. and Monaghan J.J., Smoothed particle hydrodynamics: theory and application to nonspherical stars. *Monthly Not. Royal Astron. Soc.*, 181: 375–389, 1977.

Launder B.E. and Spalding D.B., The numerical computation of turbulent flows. *Comp. Meth. Appl. Mech. Eng.*, 3: 269-289, 1974.

Lucy L. .B, Numerical approach to testing the fission hypothesis. *Astronomical J.*, 82: 1013-1024, 1977.

Monaghan J.J., Smoothed particle hydrodynamics. *Ann. Rev. Astronomy Astrophysics*, 30: 543-574, 1992 a.

Monaghan J.J., Simulating free surface flows with SPH. *J. Computational Physics*, 110: 399-406, 1992 b.

Randles P.W. and Libersky L.D., Smoothed Particle Hydrodynamics – some recent improvements and applications. *Comput. Methods Appl. Mech. Eng.*, 138: 375-408,1996.

Sibilla S., SPH simulation of local scour processes. *ERCOFTAC Bulletin*, 76: 41-44, 2008.

Takeda H.T., Miyama S.M. and Sekiya M., Numerical simulation of viscous flow by smoothed particle hydrodynamics. *Progress of Theoretical Physics*, 92: 939-960, 1994.