ECCOMAS Congress 2016
VII European Congress on Computational Methods in Applied Sciences and Engineering
M. Papadrakakis, V. Papadopoulos, G. Stefanou, V. Plevris (eds.)
Crete Island, Greece, 5–10 June 2016

A NUMERICAL INVESTIGATION OF FLOW DYNAMICS OVER A TRAPEZOIDAL SMOOTH OPEN CHANNEL

Giacomo Viccione

Department of Civil Engineering, University of Salerno, 84084, Fisciano, Italy
e-mail: gviccion@unisa.it

Keywords: Computational Fluid Dynamics, Flow dynamics, Free Surface Flows, Smoothed Particle Hydrodynamics, Volume of Fluid, HEC-RAS.

Abstract. *The paper presents a numerical investigation of the flow dynamics in a laboratory flume. The adopted geometry consists of a U shaped trapezoidal smooth open channel with a fixed slope and unerodible bed. The branches, 3m of length each, are linked with a joint 1m long, realizing two 90 degrees bends. The system is fed upstream with a water discharge under critical conditions while a Cipolletti weir is set downstream to control flow profiles. Steady flow characteristics are deduced by means of three different softwares: a pure Lagrangian developed by the author, based on the Weakly Compressible Smoothed Particle hydrodynamics (WCSPH) technique, Flow3D® and HEC-RAS®. Depending on the specific boundary conditions being given, velocity profiles and water interfaces at certain cross sections are deducted by using the first two codes. Results are discussed then spatially averaged in terms of mean velocities and water depths respectively to make a comparison with the ones obtained with HEC-RAS, showing a satisfactory agreement.*

1 INTRODUCTION

Experimental investigations of free surface flows, either laboratory or full-scale based, are aimed at describing complex fluid dynamics. They represent a viable tool for the design of structures interacting with liquids as well as for the verification under variable operating conditions. Needless to say, the related literature is quite consistent. Just to mention some meaningful examples, related studies concern the analysis of the interaction between fluids and solid [1,2] or porous interfaces [3-5], the assessment of the solid transport, either of bed or suspended or both types [6-8], sea waves [9,10], granular flow movements [11,12],

Although the effectiveness of experimental studies has been demonstrated by several investigations, the search for numerical solutions [13] is considered as a valuable alternative, as well as a supporting option [12], because it is cost-effective, easily adaptable to the variation of operating condition, more and more convenient for the endless increase of computer power. Numerical methods, either Lagrangian [14], Eulerian [15] or mixed [16] have been extensively applied over the last century as practical tools to give response to engineering and scientific problems. Among Lagrangian types, an emerging method, known as Smoothed Particle Hydrodynamics [17, 18], eventually coupled with others to exploit related advantages as in [19,20], has been applied since the last two decades, see for instance [21, 22].

The aim of this paper is to check the feasibility of three numerical models to describe the flow dynamics in a complex laboratory flume. Over the past three decades, two-dimensional and three-dimensional algorithms have been widely used to describe free surface flows, see for instance [23].

A full three-dimensional investigation is carried by mean of a Lagrangian code, based on the Weakly Compressible Smoothed Particle hydrodynamics (WCSPH) technique [24, 25] and the Eulerian-based software Flow3D® [26, 27]. Related results are compared in terms of velocity distribution and water profiles at certain cross sections. Average fields are then compared with the ones obtained using the one-dimensional computer program HEC-RAS [28, 29]. The algorithm based on the WCSPH technique and Flow3D® basics [30,31] are already described in [32], therefore they are not discussed here. Related applications are given in [33-36]. HEC-RAS (River Analysis System) is a freeware software for modeling open channel flows with the presence of bridges, culverts, weirs and other structures. Developed by the Hydrologic Engineering Center (HEC) of the United States, it is commonly used for engineering design, flood hazard mapping and flood insurance studies. Governing equations are one-dimensional, i.e. there is no hydraulic dependence on the cross section shape or channel bends.

2 MATERIALS AND METHODS

The trapezoidal open channel flow shown in the next Fig.1 is adopted in the present investigation. It is actually used for experimental investigations @ the Environmental and Maritime Hydraulics Laboratory (LIDAM) of the University of Salerno. As the flume is made of stainless steel, the roughness is neglected when carrying three dimensional studies. Manning's roughness is set as $n=0.012 s/m^{1/3}$ [37] when using HEC-RAS instead. The branches, 3m of length each, are linked with a joint 1m long and of trapezoidal shape as well, realizing two 90 degrees bends. The slope of each branch is kept fixed at 1%. The system is fed upstream with a water flow rate Q under critical conditions while a Cipolletti weir of height h measured from the bottom is set downstream to control flow profiles. Steady conditions can be reached in the reality thank to the presence of a recirculating system, consisting of a centrifugal pump which connect the outlet and inlet tanks. Here, steady state conditions are reached after a transitory which is of the order of tens of seconds. During such initial phase the inflow volume takes the space behind the weir: the higher is the latter, the longer is the transitory.

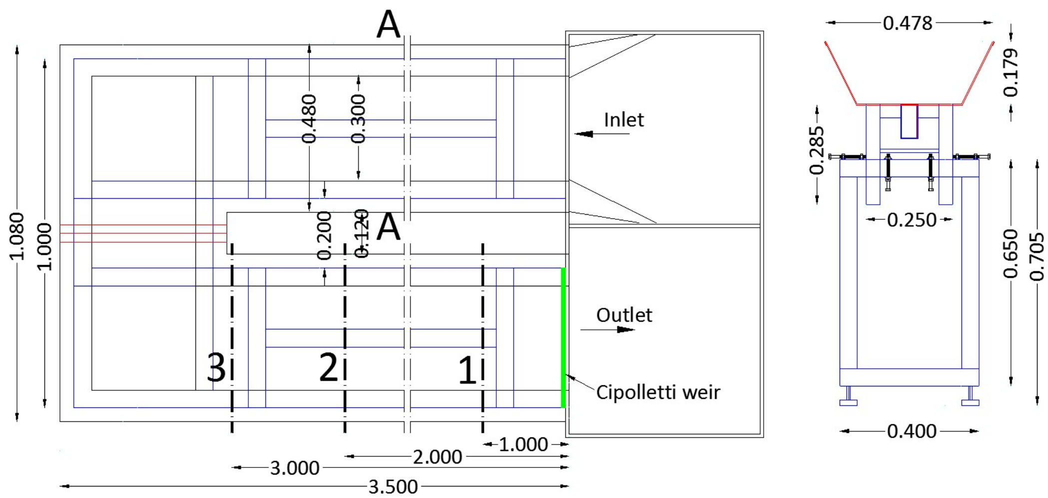


Figure 1: The open channel flow adopted in the present investigation. Left side: top view. Right side: vertical cross section A-A. 3D velocity field is extrapolated from vertical cross sections 1, 2 and 3.

Numerical simulations have been carried out taking the portion of the system between the Inlet and the Outlet zones, i.e. the flume itself. Related geometry was first created in a CAD environment as a collection of three-dimensional solids (Fig. 2, left side), then exported as a STereoLithography (stl) file, a standard compatible with Flow3D®. It basically contains a list of triangular surfaces that approximate the CAD model. The same file was finally converted in a text (xyz) file using TextPad®, obtaining the vertex coordinates of boundary particles (Fig. 2, right side), to be given as input to the WCSPH algorithm. The mesh size of the computing domain was set equal to 0,005m. The same value was adopted as reference for the edges of triangles listed in the stl file.

In the present study we consider the combination of two water discharges (Q_1=0.0085m^3/s, Q_2=0.0170m^3/s) set upstream and four weir's height (d_1=0.025m, d_2=0.05m, d_3=0.070m, d_4=0.010m) for a total of height scenarios. The upstream condition can be straightforwardly fixed in Flow3D® while in WCSPH it is set by fixing the velocity of the inflow particles.

3 RESULTS

This section yields the numerical results obtained with the WCSPH algorithm and the Flow3D® solver. To be concisely, a graphical comparison (Fig. 3) is given only for the simulations with Q_1=0.0085m^3/s and d_3=0.070m, in terms of velocity distribution over the cross sections 1,2 and 3 depicted in Fig. 1, left side. A general view of the dynamics is sketched in Fig. 4.

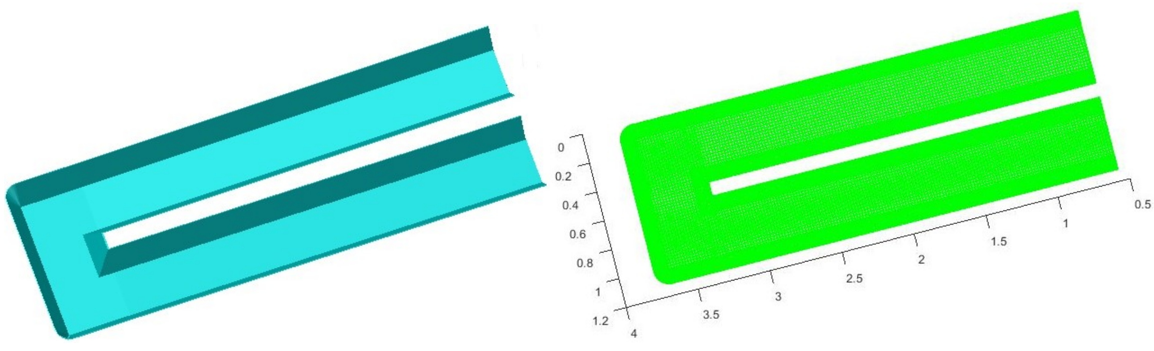
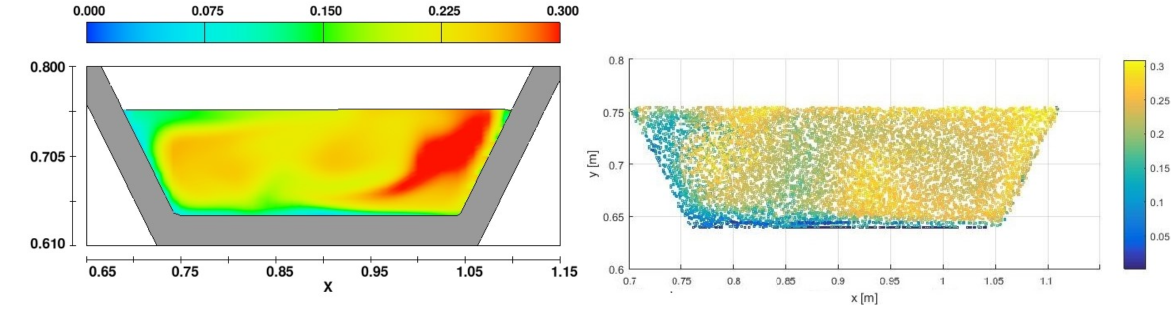
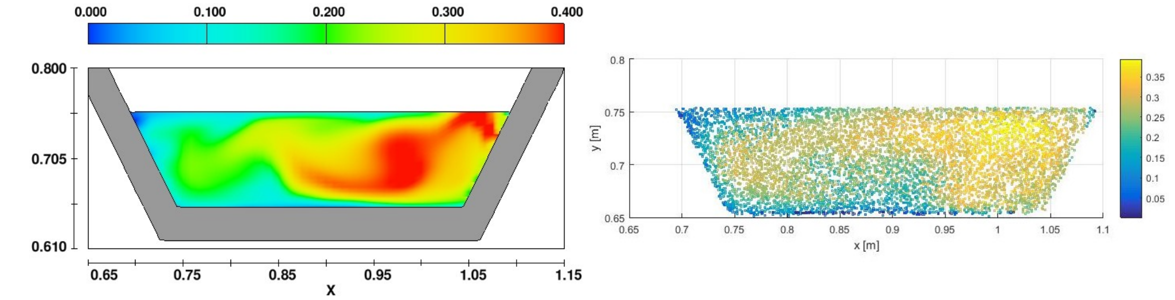


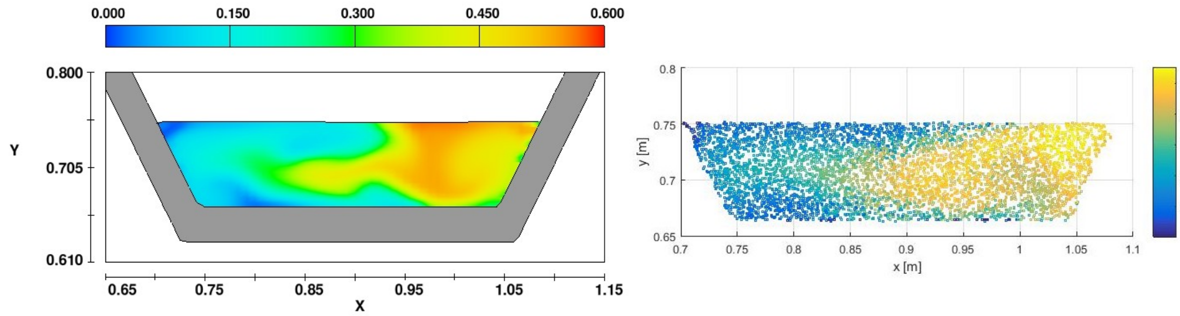
Figure 2: The flume was first created in a CAD environment (left side) then exported as STereoLithography file for Flow3D simulations and finally converted as boundary particles for the WCSPH algorithm.



Cross Section 1



Cross Section 2



Cross Section 3

Figure 3: The velocity distribution obtained with Flow3D® (left side) and WCSPH (right side) at vertical cross section 1, 2 and 3.

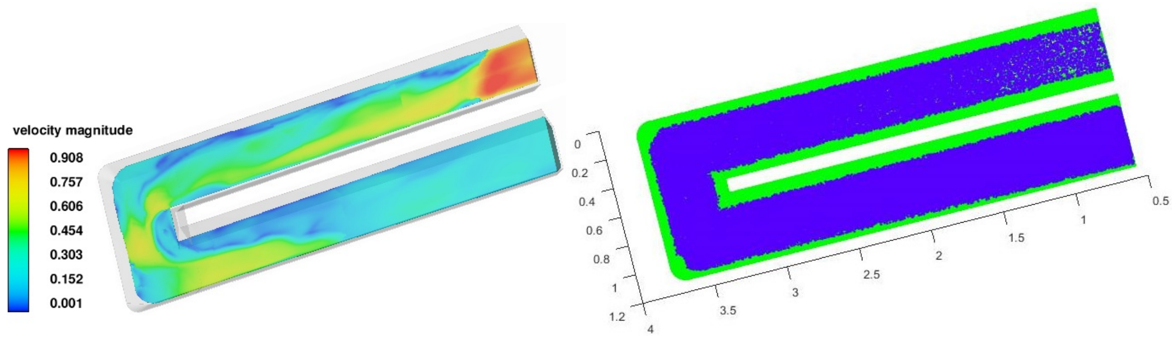


Figure 4: The flow dynamics for Q_1=0,0085m^3/s and h_3=0,070m. It is shown on the left side the velocity distribution obtained with Flow3D® while on the right side the particle disposition obtained with WCSPH.

From Fig. 3 and Fig. 4, left side, it can be observed that the trapezoidal joint influences the flow dynamics.

In particular, the velocity distribution at vertical cross sections is not symmetric respect to the vertical midplane of the branches. Higher velocity values are located on the right hand side of cross sections 1, 2 and 3 because the flow is constrained by the presence of the joint's corners. The phenomenon could be qualitatively compared to some extend to the liquid discharge issuing from a sharp orifice. Symmetry is partially recovered at Section 1 (i.e. the closest one to the weir) as it is the farthest section from what it can be thought as a disturbing region. The comparison of the velocity distribution and water table (the local free surface flow location) is satisfying. Both the numerical approaches basically reproduce the same flow dynamics.

As for the remaining simulations, the next summary Table 1, yield the velocities V_i and water surface elevations h_i, i=1,2,3 at cross sections (CS) 1,2 and 3, obtained by extracting the corresponding output values from HEC-RAS and by spatially averaging the local velocity distribution and the local free surface flow with reference to Flow3D and WCSPH simulations. Comparison among results is satisfying. It can be noted that the weir's height d_j, j=,1,2,3,4 effectively controls water profiles. Anyway, there is not a linear dependence of V_i and WS_i with d_j. This is basically due to the trapezoidal shape of the flume.

Q_1=8.5l/s	CS 1		CS 2		CS 3	
d_1=2.5cm	V_1	h_1	V_2	h_2	V_3	h_3
SPH	**0.43**	**0.72**	**0.75**	**0.71**	**0.75**	**0.70**
Flow3D	**0.42**	**0.72**	**0.74**	**0.71**	**0.74**	**0.71**
HEC-RAS	***0.44***	***0.71***	***0.75***	***0.70***	***0.75***	***0.71***

Q_2=17.0l/s	CS 1		CS 2		CS 3	
d_1=2.5cm	V_1	h_1	V_2	h_2	V_3	h_3
SPH	**0.56**	**0.74**	**0.67**	**0.76**	**0.62**	**0.75**
Flow3D	**0.57**	**0.74**	**0.68**	**0.76**	**0.61**	**0.76**
HEC-RAS	***0.58***	***0.73***	***0.70***	***0.74***	***0.63***	***0.75***

Q_1=8.5l/s	CS 1		CS 2		CS 3	
d_2=5.0cm	V_1	h_1	V_2	h_2	V_3	h_3
SPH	**0.29**	**0.74**	**0.36**	**0.74**	**0.43**	**0.74**
Flow3D	**0.28**	**0.74**	**0.37**	**0.74**	**0.43**	**0.74**
HEC-RAS	***0.29***	***0.73***	***0.37***	***0.73***	***0.44***	***0.73***

Q_2=17.0l/s	CS 1		CS 2		CS 3	
d_2=5.0cm	V_1	h_1	V_2	h_2	V_3	h_3
SPH	**0.40**	**0.76**	**0.46**	**0.76**	**0.54**	**0.76**
Flow3D	**0.40**	**0.76**	**0.48**	**0.77**	**0.54**	**0.77**
HEC-RAS	***0.41***	***0.76***	***0.48***	***0.76***	***0.55***	***0.76***

Q_1=8.5l/s	CS 1		CS 2		CS 3	
d_3=7.0cm	V_1	h_1	V_2	h_2	V_3	h_3
SPH	**0.22**	**0.76**	**0.27**	**0.76**	**0.31**	**0.75**
Flow3D	**0.23**	**0.76**	**0.28**	**0.76**	**0.33**	**0.75**
HEC-RAS	***0.23***	***0.75***	***0.28***	***0.75***	***0.32***	***0.75***

Q_2=17.0l/s	CS 1		CS 2		CS 3	
d_3=7.0cm	V_1	h_1	V_2	h_2	V_3	h_3
SPH	**0.34**	**0.78**	**0.39**	**0.79**	**0.44**	**0.78**
Flow3D	**0.34**	**0.79**	**0.39**	**0.78**	**0.45**	**0.79**
HEC-RAS	***0.35***	***0.78***	***0.40***	***0.78***	***0.45***	***0.78***

Q_1=8.5l/s	CS 1		CS 2		CS 3	
d_4=10.0cm	V_1	h_1	V_2	h_2	V_3	h_3
SPH	**0.17**	**0.79**	**0.19**	**0.77**	**0.22**	**0.79**
Flow3D	**0.16**	**0.79**	**0.19**	**0.77**	**0.21**	**0.79**
HEC-RAS	***0.17***	***0.78***	***0.20***	***0.78***	***0.22***	***0.78***

Q_2=17.0l/s	CS 1		CS 2		CS 3	
d_4=10.0cm	V_1	h_1	V_2	h_2	V_3	h_3
SPH	**0.27**	**0.82**	**0.30**	**0.82**	**0.33**	**0.81**
Flow3D	**0.28**	**0.82**	**0.30**	**0.81**	**0.33**	**0.82**
HEC-RAS	***0.28***	***0.81***	***0.31***	***0.81***	***0.34***	***0.81***

Table 1: Mean velocities [m/s] and water surface elevations [m] obtained from Flow3D and WCSPH results. A comparison is made with corresponding values (in italic) obtained with HEC-RAS, showing a satisfactory agreement.

4 CONCLUSIONS

The capability of three numerical approaches to reproduce the flow dynamics over a U shaped trapezoidal smooth open channel with a fixed slope and unerodible bed is here discussed. Three-dimensional results obtained by running Flow3D and WCSPH revealed that the velocity distribution at vertical cross sections is not symmetric respect to the vertical mid-plane of channel branches because it is influenced by the presence of the channel joint. Mean velocities and water levels at three cross sections were compared with corresponding values obtained with HEC-RAS, showing a satisfactory agreement.

REFERENCES

[1] G. Viccione, M. Genovese, F. Rossi, D. Guida, T.L.L. Lenza, Physical Modelling of Laboratory Debris Flows by Using the Sodium Carboxymethylcellulose (Na-CMC), *WSEAS Transactions on Fluid Mechanics*, 2015, **10**, 164-174.

[2] T. Mai, Z.Z. Hu, D. Greaves, A. Raby, Investigation of hydroelasticity: Wave impact on a truncated vertical wall, *Proceedings of the International Offshore and Polar Engineering Conference*, 647-654, 2015.

[3] P. Contestabile, F. Aristodemo, D. Vicinanza, P. Ciavola, Laboratory study on a beach drainage system, *Coastal Engineering*, **66**, 50-64, 2012.

[4] P. Ciavola, D. Vicinanza, F. Aristodemo, P. Contestabile, Large-scale morphodynamic experiments on a beach drainage system, *Journal of Hydraulic Research*, 2011, **49**(4), 523-528.

[5] K. Harada, F. Imamura, T. Hiraishi, Experimental Study on the Effect in Reducing Tsunami by the Coastal Permeable Structures, *Proceedings of the International Offshore and Polar Engineering Conference*, 652-658, 2002.

[6] D. Termini, River-bed erosion due to changing boundary conditions: Performance of a protective measure, *IAHS-AISH Proceedings and Reports*, **364**, 112-117, 2014,

[7] D. Termini, Bed scouring downstream of hydraulic structures under steady flow conditions: Experimental analysis of space and time scales and implications for mathematical modeling, *Catena*, **84**(3), 125-135, 2014.

[8] M. Guerrero, The investigation of sediment processes in rivers by means of the Acoustic Doppler Profiler, *IAHS-AISH Proceedings and Reports*, **364**, 368-373, 2014.

[9] D. Vicinanza, F. Dentale, D. Salerno, M. Buccino, Structural response of seawave Slot-Cone Generator (SSG) from Random Wave CFD simulations, *Proceedings of the International Offshore and Polar Engineering*, 985-991, 2015.

[10] L.F.C. Lopez, D. Salerno, F. Dentale, A. Capobianco, M. Buccino, Experimental campaign on the overtopping of the seawall Malecòn Tradicional, *Proceedings of the International Offshore and Polar Engineering*, 1499-1505, 2015.

[11] L. Sarno, M.N. Papa, R. Martino, Dam-break flows of dry granular material on gentle slopes, *International Conference on Debris-Flow Hazards Mitigation: Mechanics, Prediction, and Assessment*, 503-512, 2011.

[12] S. Evangelista, G. De Marinis, C. Di Cristo, A. Leopardi, Dam-break dry granular flows: Experimental and numerical analysis, WSEAS Transactions on Environment and Development, **10**(1), 382-392, 2014.

[13] R.W. Yeung, Numerical methods in free-surface flows, *Annual Review of Fluid Mechanics*, **14**, 395-442, 1982.

[14] G. Viccione, V. Bovolin, E. Pugliese Carratelli, Simulating Flows with SPH: Recent Developments and Applications, Book ch. in: *Intech Hydrodynamics - Optimizing Methods and Tools*, 69-84, 2011.

[15] F. Dentale, G. Donnarumma, E. Pugliese Carratelli, Numerical wave interaction with tetrapods breakwater, *International journal of naval architecture and ocean engineering*, **6**, 1-13, 2014.

[16] M. Souli, A. Ouahsine, L. Lewin, ALE formulation for fluid–structure interaction problems, *Computer Methods in Applied Mechanics and Engineering*, **190**(5–7), 659–675, 2000.

[17] J.J. Monaghan, Smoothed Particle Hydrodynamics, *Reports on Progress in Physics*, **68**(8), 1703–1759, 2005.

[18] D. Violeau, B.D. Rogers, Smoothed particle hydrodynamics (SPH) for free-surface flows: Past, present and future, Journal of Hydraulic Research, 54(1), 1-26, 2016.

[19] D.M. Robb, S.J. Gaskin, J.-C. Marongiu, SPH-DEM model for free-surface flows containing solids applied to river ice jams, *Journal of Hydraulic Research*, 54 (1), 27-40, 2016.

[20] C. Altomare, J.M. Domínguez, A.J.C. Crespo, (...), I. Caceres, M. Gómez-Gesteira, Hybridization of the Wave Propagation Model SWASH and the Meshfree Particle Method SPH for Real Coastal Applications, *Coastal Engineering Journal*, **57**(4), 1550024, 2015.

[21] M. Gómez-Gesteira, B.D. Rogers, R.A. Dalrymple, A.J.C. Crespo State-of-the-art of classical SPH for free-surface flows, *J. Hydraul. Res.*, **48** Extra Issue, 6–27, 2010.

[22] G. Viccione, V. Bovolin, E.P ugliese Carratelli, Defining and optimizing algorithms for neighbouring particle identification in SPH fluid simulations, *International Journal for Numerical Methods in Fluids*, **58**, 625-638, 2008.

[23] M.B. Kocyigit, R.A. Falconer, B. Lin, Three-dimensional numerical modelling of free surface flows with non-hydrostatic pressure, *International Journal for Numerical Methods in Fluids*, **40**(9) , 1145-1162, 2002.

[24] A. Amicarelli, R. Albano, D. Mirauda, (...), A. Sole, R. Guandalini, A Smoothed Particle Hydrodynamics model for 3D solid body transport in free surface flows, *Computers and Fluids*, **116**, 205-228, 2015.

[25] D. Le Touzé, A. Colagrossi, G. Colicchio, M. Greco, A critical investigation of smoothed particle hydrodynamics applied to problems with free-surfaces, *International Journal for Numerical Methods in Fluids*, **7**(7), 660-691, 2013.

[26] F. Dentale, G. Donnarumma, E. Pugliese Carratelli, Simulation of flow within armour blocks in a breakwater, *J. Coastal Res.*, **30**(3), 528-536, 2014.

[27] P. Ghadimi, A. Dashtimanesh, M. Farsi, S. Najafi, Investigation of free surface flow generated by a planing flat plate using smoothed particle hydrodynamics method and FLOW3D simulations, *Proceedings of the Institution of Mechanical Engineers Part M: Journal of Engineering for the Maritime Environment*, **22**(2), 125-135, 2013.

[28] G.W. Brunner, HEC-RAS, River Analysis System, User's Manual, version 3.0. US Army Corps of Engineers, Davis, California, USA , 2001.

[29] G. Di Baldassarre, A. Montanari, Uncertainty in river discharge observations: A quantitative analysis, Hydrology and Earth System Sciences, **13**(6), 913-921, 2009

[30] Violeau D., Fluid Mechanics and the SPH Method: Theory and Applications, Oxford University Press, 2012.

[31] Flow Sciences Inc. Flow-3D User Manual, release 9.4, Santa Fe, NM, USA, 2009.

[32] G. Viccione, E. Pugliese Carratelli, V. Bovolin, A numerical investigation of liquid impact on planar surfaces, VII European Congress on Computational Methods in Applied Sciences and Engineering, Crete Island, Greece, 5–10 June 2016, under review.

[33] E. Pugliese Carratelli, G. Viccione, V. Bovolin, Free surface flow impact on a vertical wall: a numerical assessment, *Theoretical and Computational Fluid Dynamics*, accepted for publication, 2016.

[34] G. Viccione, V. Bovolin, E. Pugliese Carratelli, Simulating fluid-structure interaction with SPH. In: *International Conference of Numerical Analysis and Applied Mathematics*, Kos, Greece, Vol.1479, 209-212, 2012.

[35] G. Viccione, V. Bovolin, Simulating Triggering and Evolution of Debris-Flows with Smoothed Particle Hydrodynamics (SPH), *Italian Journal of Engineering Geology and Environment*, 523-532, 2011.

[36] L. Cavallaro, F. Dentale, G. Donnarumma, E. Foti, R.E. Musumeci, E. Pugliese Carratelli, Rubble Mound reakwater Overtopping: Estimation of the reliability of a 3D Numerical Simulation, In: *ICCE 2012, International Conference on Coastal Engineering*, Santander, Spain, 2012.

[37] V.T. Chow, Open-Channel Hydraulics, McGraw-Hill, 1959.