SPHERIC newsletter

SPH European Research Interest Community

http://cfd.me.umist.ac.uk/sph
Contact: damien.violeau@edf.fr

D. Violeau, Chairman
N. Quinlan, Secretary
B. Rogers, Webmaster

Steering Committee
EDF R&D / LNHE (France)
National University of Ireland, Galway
University of Manchester (UK)
University of Vigo (Spain)
L'Aquila University (Italy)
Ecole Centrale de Lyon (France)
VA TECH Hydro (Switzerland)

Members
Novosibirsk State University (Russia)
University of Sydney (Australia)
Ecole Centrale de Nantes (France)
INSEAN (Italy)
Universidad Nacional de Educacion a
    Distancia (Spain)
John Hopkins University (USA)
University of Nottingam (UK)
University of Plymouth (UK)
Swiss Center for Scientific Computing
North China Electric Power University
University of Bristol (UK)
Université de Montpellier (France)
Technical University of Madrid
    (Spain)
Groupe-Conseil LaSalle (Canada)
CSIRO Mathematical and Information
    Sciences (Australia)

Editorial

The *Smoothed Particle Hydrodynamics* (SPH) numerical method is nowadays successfully used in academic research and appears to be attractive for industrial applications in fluid dynamics and continuous media. Recent publications in recognised journals and congress proceedings show that an increasing number of academic or private institutions contribute to this development all around the world and particularly in Europe. Collaborations between institutes of various countries are also becoming more and more numerous, meaning that SPH is reaching its maturity in applied and industrial sciences.

At the occasion of a summerschool managed by the Ecole Centrale de Lyon (France), VA TECH Hydro and EDF R&D (respectively Swiss and French companies) mentioned the opportunity to establish a European Special Interest Group (SIG) on SPH within the Ercoftac community (*European Research Community On Flow, Turbulence And Combustion*). Seven institutions agreed to be members of the steering committee and attended a launch meeting in EDF laboratories in Chatou, near Paris in October 2005. Two months later, 15 more institutes reached the SIG, bringing the number of members to 22 laboratories from ten countries (inside *and outside* Europe), and the SIG received approval from Ercoftac. New partners are still welcome and should contact us to enter the group.
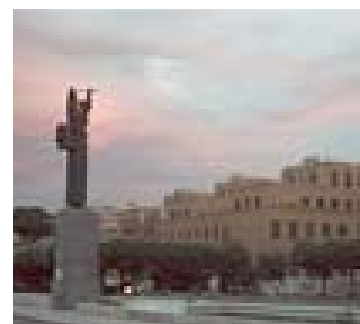
SPHERIC, or SPH European Research Interest Community (to call the working group by its name) aims at promoting the SPH method in both academic and industrial fields and enhance collaborations between countries and institutes. In this sense, this newsletter highlights the main on-going developments in this promising numerical area, in order to make SPH more and more attractive to scientists working in the field of continuous media. It should be published twice a year from now on.
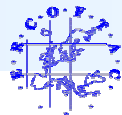
First SPHERIC workshop in Rome, 2006

The first SPHERIC workshop is planned for Rome in May 2006, to be organized by the University of Rome La Sapienza. As the first event managed by SPHERIC, this two-day meeting will allow SPH developers to discuss their applications and codes, through a series of sessions based on benchmark test cases. Prof. J.J. Monaghan, as one of the creators of SPH and a major World specialist of this method, will give a keynote adress.

We hope that the 2006 SPHERIC workshop will highlight the quality and range of SPH research, benefit all the participants, and attract new laboratories into our circle. Additional information regarding the programme will be provided very soon through the SPHERIC website.

Contact: panizzo@tiscali.it

Modelling laminar and turbulent flows

***R. Issa** & **D. Violeau**, Laboratoire National d'Hydraulique et Environnement, EDF R&D, Chatou, France*
***E.S. Lee**, University of Manchester, UK*

Validation is a key stage in developing an SPH code. EDF's Spartacus-2D and Spartacus-3D codes, designed to simulate laminar and turbulent flows with traditional SPH equations including turbulent closures, have been extensively used on test cases allowing such validation (open-channel flows, steady periodic hill flow, etc.). At moderate Reynolds numbers, the hill flow established in the framework of ERCOFTAC benchmark cases (http://tmdb.ws.tn.tudelft.nl/workshop10/case 9.2/case9.2.html), was simulated with both Spartacus-2D and Code_Saturne, EDF's finite volume code solving 3D RANS equations. Comparisons prove that SPH is capable of reproducing steady confined laminar flows with a very satisfactory accuracy, as shown in **figure 1** (Issa *et al*., 2004).

The case of turbulent flows has been deeply investigated to develop appropriate turbulent closures for SPH. Although most of them are eddy-viscosity-based models – in particular the well known k–ε model (Violeau, 2004) – more advanced closures have been tested, such as PDF (Probability Density Function) methods and LES (Large Eddy Simulation, see Issa *et al*., 2005). The latter model was used to simulate the three-dimensional collapse of a water column in a tank, and compared with an experimental flow carried out at the University of Delft (**figure 2**).

Ongoing developments concern non-linear eddy-viscosity models for modelling complex turbulent flows, accounting for the effect of a free surface on turbulence in SPH.

Contact: reza.issa@edf.fr

a

b

References

Issa, R., Lee, E.S., Violeau, D. and Laurence, D. (2004), *Incompressible separated flows simulations with the Smoothed Particle Hydrodynamics gridless method*, Int. Journ. Num. Meth. Fluids **47**:10-11, 1101-1106.
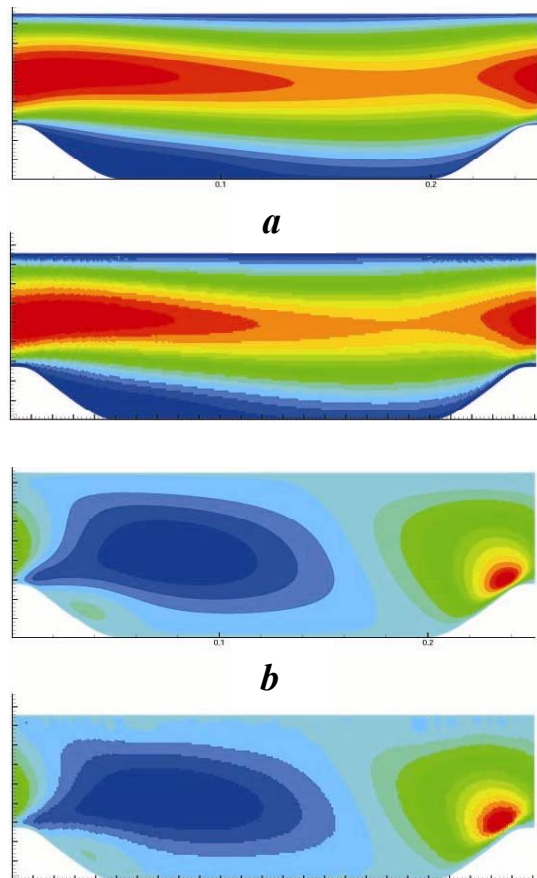
Violeau, D. (2004), *One and two-equations turbulent closures for Smoothed Particle Hydrodynamics*, Proc. VIth International Conference on Hydroinformatics, Singapore, 87-94.
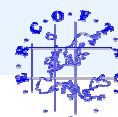
Issa, R., Violeau, D. and Laurence, D. (2005), *A first attempt to adapt 3D Large Eddy Simulation to the Smoothed Particle Hydrodynamics gridless method*, Proc. International Conference on Computational and Experimental Engineering and Sciences, 1st Symposium on Meshless Methods, June 2005, Stara-Lesna, Slovakia.

Violeau, D. and Issa, R. (2005), *Numerical modelling of complex turbulent free surface flows with the SPH Lagrangian method*, Submitted to the Int. Journ. Num. Meth. Fluids.

Figure 1 (top) – Axial (*a*) and vertical (*b*) velocities in a 2D laminar periodic hill flow simulated with the Code_Saturne finite volume code (top) and the SPH Spartacus code (bottom).

Figure 2 (bottom) – Collapse of a water column: comparison between Spartacus-3D with LES closure and laboratory experiments.
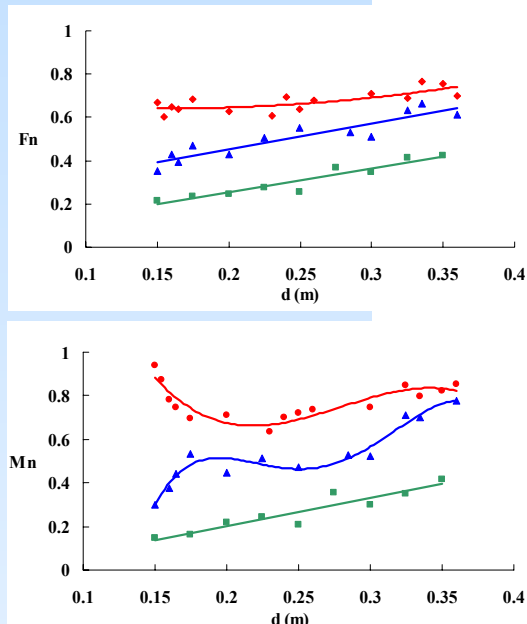
Modelling interaction between waves and coastal structures

M. Gómez-Gesteira & **A.J.C. Crespo**, *Grupo de Física de la Atmósfera y del Océano,*
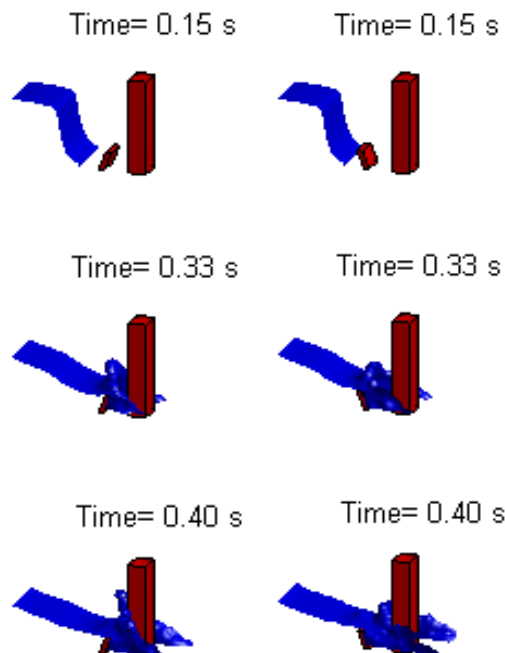*Facultad de Ciencias, Universidad de Vigo, Ourense, Spain*
R.A. Dalrymple, *Department of Civil Engineering, Johns Hopkins University, Baltimore, USA*

Models based on SPH are an option to address coastal processes, particularly the interaction between waves and coastal structures (Gomez-Gesteira and Dalrymple 2004, Gomez-Gesteira *et al.* 2005). The comparison of SPH results with laboratory experiments has shown to be a reasonable issue, since the simulation of real phenomena is an arduous task, not only due to computational limitations, but also to the lack of reliable data. Thus, SPH has been compared to experimental data obtained by Yeh and Petroff at the University of Washington. Their experiment has been referred to as a 'bore in a box' which was a dam break problem confined within a rectangular box, with square column placed in the middle, representing a coastal structure. Details about the experimental setup are given in Gomez-Gesteira and Dalrymple (2004).

The impact of the wave on the structure can be mitigated by means of dike placed in front of the structure (Crespo *et al.*, 2005). **Figure 1** shows the normalized force and momentum exerted on the structure for different values of the dike height H_D and dike-structure distance d (key parameters controlling the degree of mitigation under the same incoming wave).

Figure 1 – Protection degree for different distances between the dike and the structure (d) and different normalized dike heights H_D (ratio between the dike height and the dam break height); $H_D = 0.17$ (red line); $H_D = 0.30$ (blue line); $H_D = 0.42$ (green line).

Figure 2 – Different instants of wave collision with a structure protected by a dike sloped landward (left) or seaward (right).
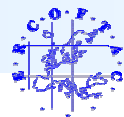
In general, force increases continuously with distance, due to flow reconstruction after surpassing the dike. On the other hand, momentum shows different regimes depending on the way water surpasses the dike: creating a jet, by gushes or surrounding it. Apart from these parameters, the dike slope can also play a key role in protection as shown in **figure 2**. Thus, even a small deviation from the vertical (± 15 degrees) has a significant influence on the impact, in such a way that both the force and the momentum exerted on the structure are around 20% higher when using a dike sloped landward than when sloped seaward.

<u>Contact</u>: mggesteira@uvigo.es

References

Crespo, A.J.C., Gómez-Gesteira, M. and Dalrymple, R.A. (2005), *Mitigation of Tsunamis using SPH3D Method*, Proceedings of Ocean Waves Measurement and Analysis, V[th] Int. Symposium WAVES 2005, paper number 87, edited by B. Edge and J.C. Santas, published by CEDEX, Madrid.

Gómez-Gesteira, M. and Dalrymple, R.A. (2004), *Using a 3D SPH Method for Wave Impact on a Tall Structure*, Journ. Waterway., Port, Coastal and Ocean Eng., **130**:2, 63-69.

Gómez-Gesteira, M., Cerqueiro, D., Crespo, C. and Dalrymple, R.A. (2005), *Green water overtopping analyzed with a SPH model*, Ocean Eng., **32**:223-238.

Modelling wave overtopping on a sloping seawall

S. Shao, *North China Electric Power University, Beijing, China*

Wave overtopping is a practical problem in coastal engineering. It involves complicated free surface deformations and velocity structures. The SPH model is an ideal approach to simulate such a process due to its completely mesh-free numerical scheme. The following figures show the particle snapshots and velocity structures of a plunging wave breaking, running up and overtopping on a seawall. An incompressible SPH numerical scheme together with the Large Eddy Simulation (LES) model is employed as the modelling approach. In the incompressible SPH formulations, the pressure is computed implicitly from a pressure Poisson equation obtained from the combinations of the mass and momentum equations, in contrast with the original weakly compressible SPH, in which the pressure is obtained explicitly from an equation of state. The incompressible SPH code was developed and improved from the Moving Particle Semi-implicit (MPS) program provided by Professor Seiichi Koshizuka at The University of Tokyo.

Further development of the present incompressible version of SPH model is ongoing. The future improvement will involve: (1) Extending the current 2-D code into 3-D, which can deal with the practical problems in a more realistic way. However, the solution of the pressure Poisson equation in the incompressible SPH algorithm is time-consuming, and thus needs more advanced computing techniques such as parallel computing. (2) Extending the current one-phase code into two-phase or multi-phase code, which will be able to treat the influence of air bubbles during the wave breaking and the sediment motion under waves. This can be achieved by modifying the one-phase algorithm and considering the interactions of different phases. (3) Comparisons of the incompressible and weakly compressible SPH methods. One may admit that the incompressible SPH has some disadvantages as compared with the original weakly compressible SPH, such as the fact that the former needs more computational time due to the solution of the pressure Poisson equation, in spite of the possible improvement of numerical stability. A future SPH model combining the merits of both is to be expected.

Contact: songdongshao@hotmail.com

References

Shao, S.D. and Gotoh, H. (2005), *Turbulence particle models for tracking free surfaces*, Journ. Hydraulic Research, **43**:3, 276-289.

Violeau, D., Piccon, S. and Chabard, J.-P. (2001), *Two attempts of turbulence modeling in smoothed particle hydrodynamics*, Proc. 8th Int. Sym. Flow Modeling and Turbulence Measurements, Tokyo, Japan, 4-6 December.
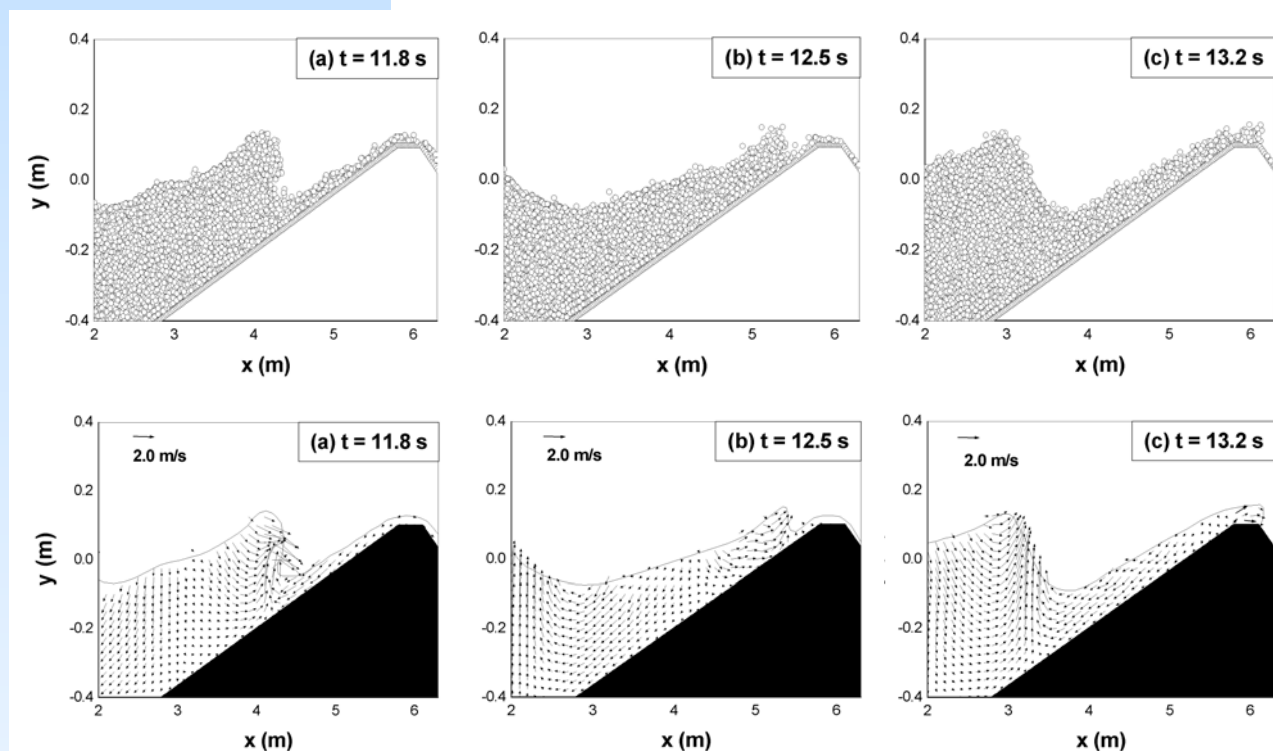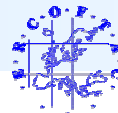
Figure 1 – Particle snapshots and velocity fields during wave breaking and overtopping of a sloping seawall.

Modelling of water waves generated by landslides

A. Panizzo, *University of Rome La Sapienza, Italy*
G. Cuomo, *University of RomaTre, Italy*

The forecasting of tsunami and of their effects need joint efforts in the fields of geophysics, geology, geotechnics and hydraulics. To improve the knowledge on this important topic, since 1998, thanks to the funding of the National Dams Authority and the Civil Protection Agency of the Italian Government, researchers of the University of L'Aquila, University of Roma TRE, Tor Vergata and La Sapienza, started to study tsunami waves. Results, publications and more information about current researches can be found on www.tsunamis.it.

In this framework, experimental studies and 2D/3D SPH numerical models have been used to study the features of water waves generated by both subaerial and underwater landslides. The vertical drop of a weighted box into water, well known as the Scott Russell wave generator, has been reproduced using a physical model study and then the SPH approach (**figure 1**). The important cases of tsunami waves along a straight coast and around a conical island are currently studied using both large scale physical models and 3D SPH simulations (**figure 2**).
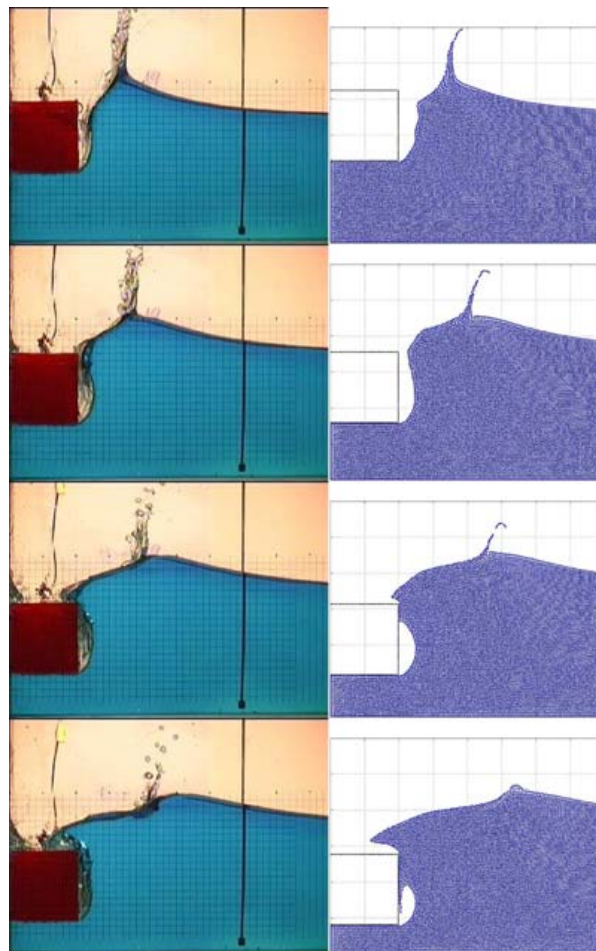
Contact: panizzo@ing.univaq.it

References

Panizzo, A. (2004), *Physical and numerical modelling of subaerial landslide generated waves*, PhD thesis, Università degli studi di L'Aquila, L'Aquila, 2004.
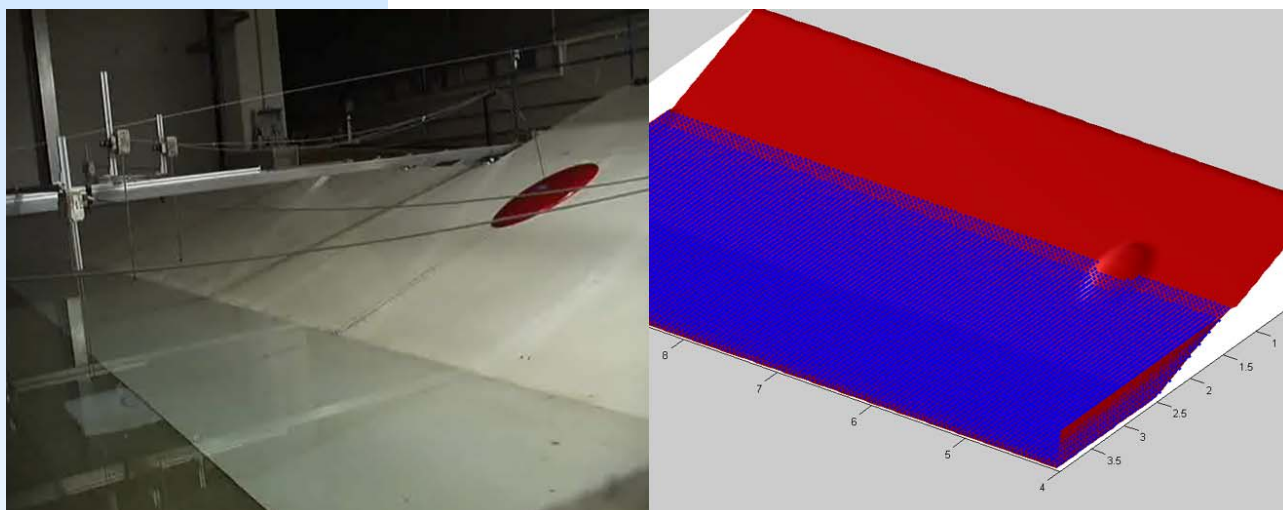
Panizzo, A. and Dalrymple, R.A. (2004), *SPH modelling of underwater landslide generated waves*, Proc. Int. Conf. Coastal Eng. 2004, ASCE.
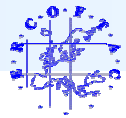
Panizzo, A., De Girolamo, A. and Petaccia, A. (2005), *Forecasting impulse waves generated by subaerial landslides*, Journ. Geophysical Research (in press).

Figure 1 (top) – Comparison of the water fields generated in the experiments and modelled with SPH.

Figure 2 (below) – Physical model of an elliptical landslide on a plane slope, and the SPH 3D grid.

Ongoing developments at NUI, Galway

N. Quinlan, **M. Basa** & **M. Lastiwka**, *National University of Ireland, Galway*

The aim of the SPH research group at NUI, Galway is to explore the potential of SPH for engineering fluid dynamics, and in particular for biomedical fluid flows, which involve complex geometry and moving walls. Before tackling such difficult problems, however, we are investigating more fundamental and general questions.

The resolution for any particular simulation is dependent on the distribution of computational particles (like the nodes of a conventional mesh-based method), and can be controlled to some degree by the particles' initial distribution. However, as the solution evolves, the particle distribution is determined by the flow and local control of resolution is lost. In adaptive particle distribution (Lastiwka *et al*., 2005), particles are inserted and removed as required, depending on local gradients. This process is analogous to adaptive mesh refinement, but is simplified by the mesh-free nature of SPH. For validation cases in shock tube flow, it has been possible to improve shock capturing accuracy significantly (**figures 1** and **2**). To evaluate methods of computing viscous stress (requiring second derivatives of velocity), transient two-dimensional viscous channel flows have been modelled from startup to steady state (Poiseuille flow). Two-pass methods, in general, yield better results either than direct double differentiation of kernel functions or hybrid SPH/finite-difference methods, but are more costly (Basa *et al*., 2005). It is also advantageous to use consistency-corrected kernels and, in low Mach number flow, to use a reduced pressure (for example, by subtracting out a global average pressure).
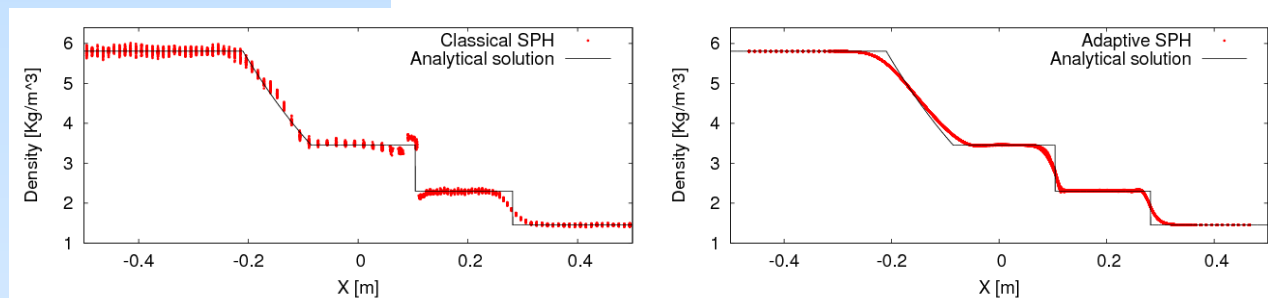
In other work, we are investigating the fundamental accuracy of SPH, implementation of boundary conditions, and incompressible flow modelling. Work is beginning on a project to develop an SPH tool for blood flow, in collaboration with vascular surgeons at University College Hospital, Galway.
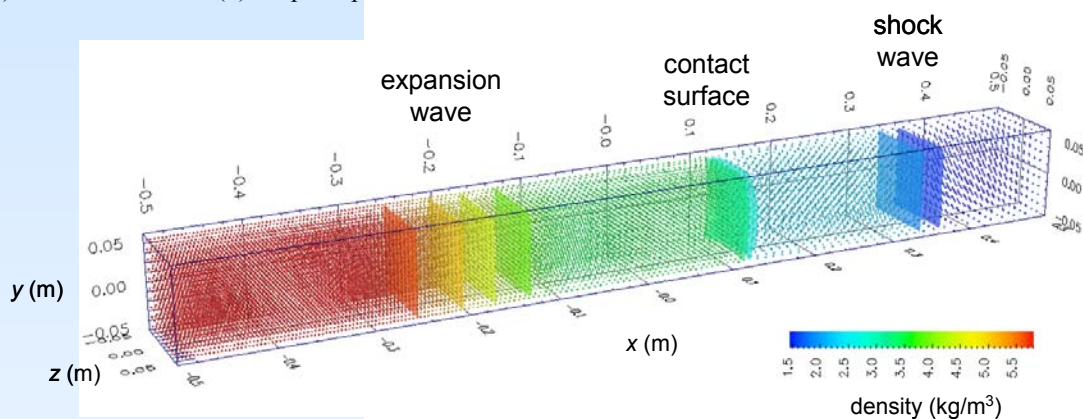
Contact: nathan.quinlan@nuigalway.ie

References

Basa M., Lastiwka M. and Quinlan N.J. (2005), *Poiseuille Flow with SPH: effects of viscosity and boundary implementation*, in Meshless 2005: Proceedings of the ECCOMAS Thematic Conference on Meshless Methods, Leitão V.M.A., Alves C.J.S., Duarte C.A. (eds.), DM-IST, Lisbon.

Lastiwka M., Quinlan N.J. and Basa M. (2005), *Adaptive particle distribution for Smoothed Particle Hydrodynamics*, Int. Journ. Num. Meth. Fluids **47**(10–11):1403–1409.
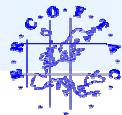

Figure 1 – Results for nominally 1D shock tube flow modelled in 3D, with a randomized initial particle distribution for (a) standard SPH and (b) adaptive particle distribution.

Figure 2 – Visualisation of a shock tube flow modelled in 3D, with particles and isosurfaces coloured by density.
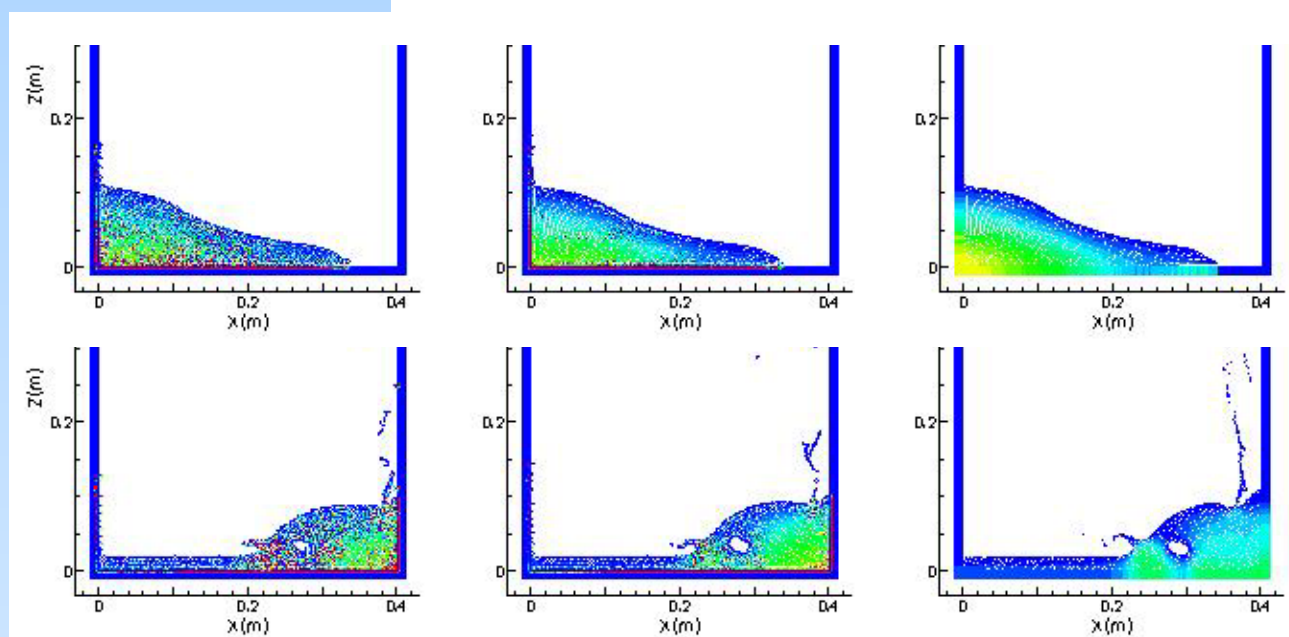
Ongoing developments at the University of Manchester

B. Rogers & **E.S. Lee**, *University of Manchester, UK*

At the University of Manchester, the SPH research group consists of two professors, Dominique Laurence & Peter Stansby, a research assistant, Ben Rogers and a PhD research student, Eun-sug Lee. The group's interests lie in two distinct but overlapping areas: modelling violent free-surface flow such as breaking waves, and turbulence. The group predominantly uses the EDF SPH code Spartacus-2D.

Recent activities by the group include the development of a 2-D incompressible SPH solver for free-surface flows based on the papers by Cummins & Rudman (1999) and Lo & Shao (2003). **Figure 1** shows two snapshots for a comparison between using the standard weakly compressible SPH method, a filtered version of weakly compressible method and the incompressible SPH (I-SPH) for a 2D schematic dam break. It can be seen already that the I-SPH solver leads to improved pressure contours and different results for a highly violent process. At this very moment, with the help of Damien Violeau at EDF, the University is concentrating on developing a tracking technique to identify the surface particles correctly for the incompressible solver that will lead to better numerical results.
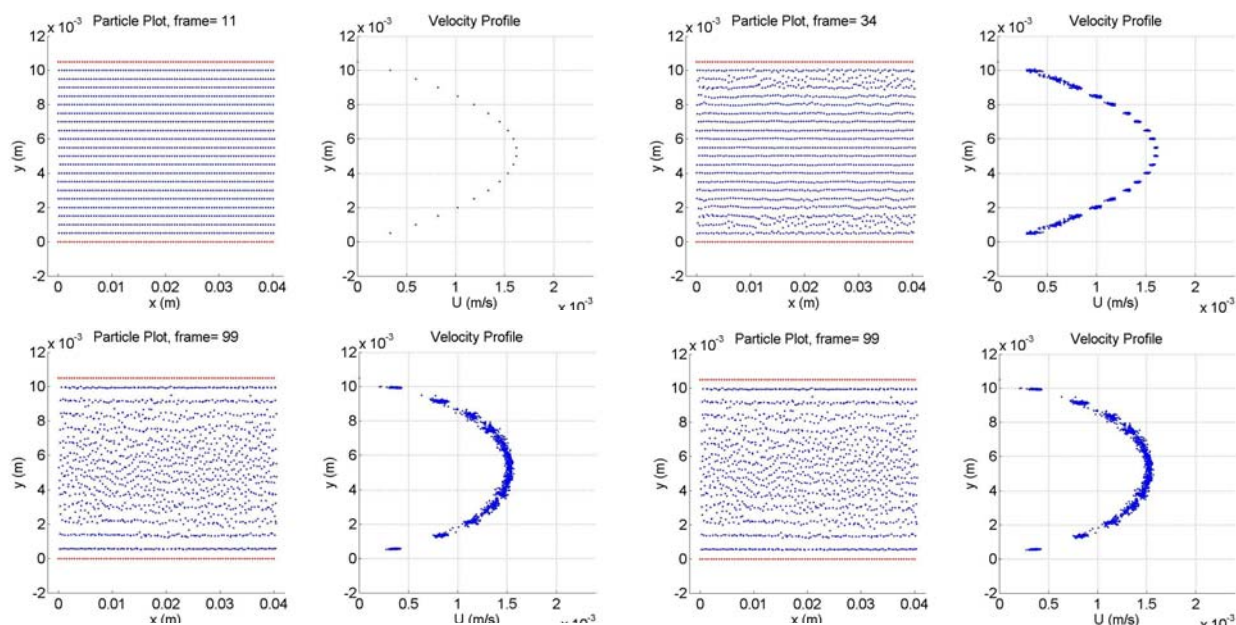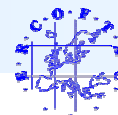

Figure 1 – Two snapshots showing density colour contour plot for 2D dam break: (i) weakly compressible, (ii) filtered weakly compressible, (iii) truly incompressible.

We are also studying the apparently random-like motion of the SPH particles that quite often seems to be inherent in the method itself (Colagrossi, 2004). **Figure 2** (next page) displays snapshots that show the typical break up of a very simple periodic laminar flow in a 2D channel at Re = 10. It has long been noticed, especially with the implementation by many researchers of turbulence models, that the random motion of the particles and occasional clumping that is sometimes observed is unphysical. We can observe that this is first noticeable in the second snapshot. While the reasons for these are generally known and understood (*i.e.* problems and inaccuracies in the kernel summation process), and can be mitigated using kernel correction methods (*e.g.* Dilts, 1999), at the University of Manchester we are researching how this can be used.

Contact: benedict.rogers@manchester.ac.uk

References

Cummins S.J., Rudman M.J. (1999), *An SPH projection method*, Journal of Computational Physics **152**:584-607.

Shao S., Lo E.Y.M. (2003), *Incompressible SPH method for simulating Newtonian and non-Newtonian flows with a free surface*, Advances in Water Resources **26**: 787-800.

Colagrossi, A. (2004), *A meshless Lagrangian method for free-surface and interfacial flows with fragmentations*, PhD Thesis, Universita di Roma, La Sapienza, Italy.

Dilts, G.A. (1999), *Moving least squares hydrodynamics: consistency and stability*, Int. J. Numer. Meth. Eng., **44**: 1115-1155.
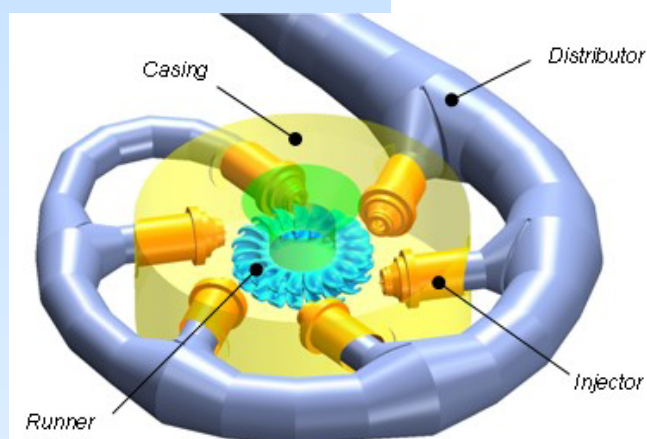
Figure 2 – Initiation of random particle motion with SPH in a 2D pipe flow.

Modelling the flow in a Pelton turbine

J.-C. Marongiu & P. Maruzewski, Ecole Centrale de Lyon, France
E. Parkinson, VA TECH HYDRO, Switzerland

VA TECH HYDRO is a world wide global supplier of hydraulic turbines. As such, an intensive use of CFD is done for design and analysis of existing turbine components. Pelton turbines are among the most complex to handle because flow patterns and hydraulic losses are very difficult to observe and quantify, due to the very complex occurring processes, including pressure losses, secondary flows, jets, film flow, free surfaces, spray formation, ventilation losses, unsteadiness, and complex interaction between components (**figures 1** and **2**).
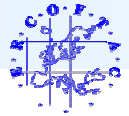

Figure 1 – Definition sketch of a Pelton turbine.

Figure 2 – Typical flow in the vicinity of the buckets.

Considering the actual state of the art in refurbishment technology, the potential for further efficiency improvements in Pelton turbine technology is high, thanks to a better comprehension of the flow features and the loss mechanisms. However, the full upgrading potential is not met if great care is not taken in the integration of the new runner, even if of high efficiency, in its environment. The inlet piping system may not be optimal and thus induce poor quality jets. The casing could also not manage properly the evacuation of the water sheets to avoid any disturbances on the incoming jets and the buckets. The adaptation of the environment to the new runner is thus a key step in a refurbishment process. It is also the case for the fine tuning of a runner design for a new installation. The numerical simulation of all these interactions, especially the influence of casing on the performances is extremely complex to handle with classical
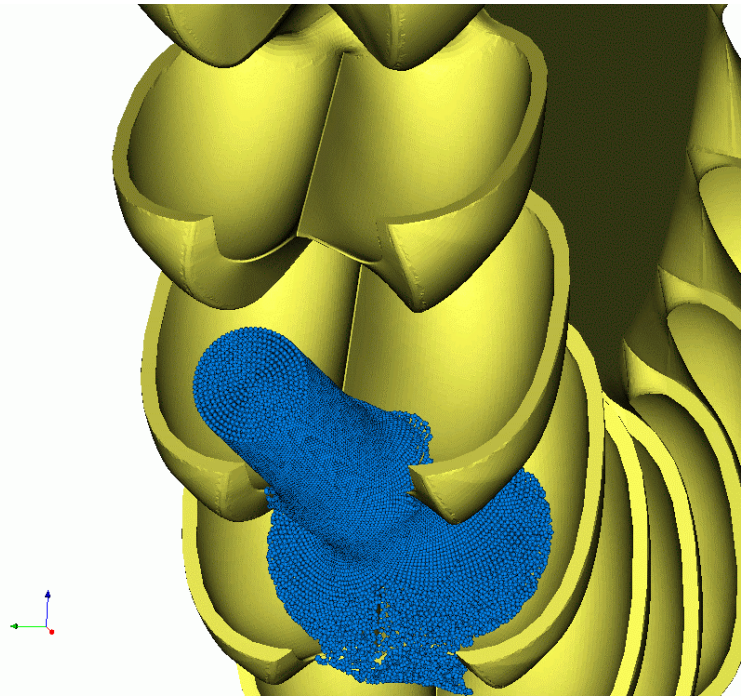
mesh-based approaches because of the mesh definition and the numerical process. With respect to these points, the meshless SPH approach appears as a very promising solution to simulate properly the water sheets patterns in a Pelton turbine.

Figure 3 – A Pelton turbine in its casing.   **Figure 4** – First SPH simulation of the impact on a bucket.

Developing our SPH tool with the Ecole Centrale de Lyon, one of the key point for us was to find a convenient (and affordable) way to handle complex geometries. We decided to use the very simple model based on wall particles exerting repulsive forces on fluid particles (Monaghan, 1994). Thanks to an interface with CAD tools, it is now easy to handle geometries with curved surfaces an sharp edges. An other important issue is the way we discretize the water jet. We found out that the initial distribution of the particles in the jet could lead to unphysical results where the water sheets were not symmetric. This strange behavior seems to result from a regular pileup of the particles leading to peculiar directions and non symmetrical interpolation. We correct this by slightly shifting the initial distribution between each injection of new particles.

After validation test cases including the impact of a jet on a flat plate, a first realistic jet on a real Pelton bucket was simulated (**figures 3** and **4**). Despite satisfying results, the computational time was prohibitive for an industrial use of the code. It is well known that SPH computation are time consuming compared to Eulerian methods. We decided to develop a parallel SPH algorithm, based on the MPI library and a very general domain decomposition method. However, unlike many other CFD applications, we can't assume here a regular load distribution in the calculation domain. Indeed, the number of fluid particles increases as the calculation pushes on, and the position of the fluid in the domain rapidly changes due to rotation and reflection on wall boundaries. Hence the methods using static decomposition based on geometry splitting are inadequate. In order to ensure the load balance, the decomposition is performed at each time step.

With these developments, our code, NEMO, has the potential to run realistic flow calculations, especially the impact of a water jet on a complete Pelton runner rotating in its casing. Further work will focus on validation through experimental measurements.

Contact: etienne.parkinson@vatech-hydro.ch

References

Monaghan, J.J. (1992), *Smoothed Particle Hydrodynamics*, Annual Review of Astrophysics **30**:543–74.

Monaghan, J.J. (1994), *Simulating free surface flows with SPH*, Journal of Computational Physics **110**: 399–406.