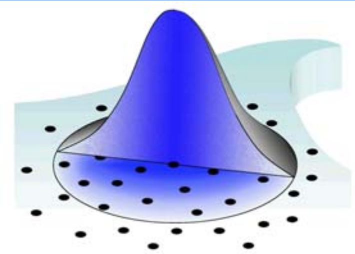
SPHERIC newsletter

10th issue – August 2010

SPH European Research Interest Community

http://wiki.manchester.ac.uk/spheric/

Contact: david.letouze@ec-nantes.fr



Fifth SPHERIC Workshop Editorial

*By **Benedict D. Rogers**, Chair of the Local Organising Committee*
University of Manchester, School of Mech. Aero & Civil Engineering, UK

The School of Mechanical, Aerospace and Civil Engineering at the University of Manchester was delighted to host the fifth international workshop organised by the Smoothed Particle Hydrodynamics European Research Interest Community. The workshop once again proved to be a very popular event with a record 80 abstracts submitted from which 59 were chosen to be presented during the event. Over 100 delegates attended this fifth consecutive workshop held from June 23rd to June 25th 2010, preceded by a training day June 22nd.

The workshop began with a training day attended by more than 30 participants. After an introduction to the SPH method by Damien Violeau (EDF, France) taking both beginners and experienced practitioners through the basics of SPH and how it is used for industrial applications within EDF, Jean-Christophe Marongiu (Andritz Hydro, Switzerland) gave a stimulating lecture on one of the most important areas of SPH for engineering simulations: boundary conditions. This was then followed by participants then being introduced to the open-source code SPHysics, under the guidance of B.D. Rogers (University of Manchester, UK) and A.J.C. Crespo (University of Vigo, Spain). In the afternoon, J. Biddiscombe (CSCS, Switzerland) gave a lecture on the application of ParaView-meshless developed at CSCS, dedicated to post-processing and visualization of meshless simulations.

Over the next three days, the 18 workshop sessions on various topics (see below) gave an excellent overview of the varied SPH activity occurring around the world. A brief opening speech was given by B.D. Rogers (University of Manchester, UK) introducing the delegates to the university with its role in the industrial revolution and the historical development of modern science. This was followed by an overview of School activities by Prof. Peter Stansby (Head of School) and finally an overview of SPHERIC activities by Damien Violeau (EDF, France, Chair of SPHERIC).

Two interesting keynote lectures on different aspects were given as part of the workshop: *(i)* on the finite volume particle method by Dr N. Quinlan from the National University of Ireland, Galway, and *(ii)* Multiresolution particle simulations by Prof. P. Koumoutsakos, ETHZ, Switzerland where we saw a simulation of 10 billion particles for the first time.

During the aperitif at the end of the first day, delegates were given two tours of experiments: the first one was a state-of-the-art investigation of the wave energy device championed here called the Manchester Bobber by Professor Peter Stansby, and the second was a demonstration by Prof. Dominique Laurence of the original experimental apparatus used by Osborne Reynolds in his investigation of turbulence leading to his famous number which is now taught to all students of fluid mechanics all over the world!

At the end of the second day, the banquet took place at the Lowry centre where presentations were made: *(i)* the Libersky student prize was awarded to Martin Ferrand of the University of Manchester and EDF who was awarded a brand new nVidia Fermi GPU card donated by nVidia, *(ii)* a presentation was made by new Chair of SPHERIC David Le Touzé (ECN, France) of the Steering Committee to Damien Violeau to thank him for his outstanding work as Chair of SPHERIC during the last five years.

The workshop organisers also wish to thank our generous sponsors: Lloyds Register, BAE Systems, Supermicro and nVidia. On behalf of the Local Organising Committee, we express our thanks for the efforts of everyone involved in this workshop and hope the delegates enjoyed the event.

Contact: benedict.rogers@manchester.ac.uk

DAY 1: Wednesday 23 June 2010

8:00	Registration and documentation
8:30	Opening of the 5th SPHERIC Workshop
8:45	**Keynote lecture**: Dr. N. Quinlan, National University of Ireland, Galway
9:40	Coffee break
10:00	**Session 1: Multi-Fluids 1**
11:00	**Session 2: Boundary Conditions 1**
12:00	Lunch
13:15	**Session 3: Free-Surface Flow & Bed modeling**
14:15	**Session 4: Multi-Fluids 2**
15:15	Coffee break
15:30	**Session 5: Boundary Conditions 2**
16:30	**Session 6: Free-Surface Techniques**
17:30	Aperitif

Keynote lecture: *Dr N. Quinlan, National University of Ireland, Galway*
The finite volume particle method for ALE simulation of flow around moving bodies

Session 1: Multi-Fluids 1
Chairman*: Andrea Colagrossi, INSEAN, Italy*

- *Improvement of multiphase model using preconditioned Riemann solvers*, Leduc, J., Marongiu, J.-C., Leboeuf, F., Lance, M., Parkinson, E.
- *Surface reconstruction by approximating the color function with application to Surface Tension*, Andersson, B., Jakobsson, S., Mark, A., Davidson, L., Edelvik, F.

- *3D drop deformation and breakup in simple shear flow considering the effect of insoluble surfactant*, Adami, S., Hu, X.Y., Adams, N.A.

Session 2: Boundary Conditions 1
***Chairman**: Damien Violeau, EDF, France*

- *An alternative approach to modelling complex Smoothed Particle Hydrodynamics boundaries*, Lehnart, A., Fleissner, F., Eberhard, P.
- *SPH no-slip BC implementation analysis at the continuous level*, Souto-Iglesias, A., Gonzalez, L.M., Colagrossi, A., Antuono, M.
- *Improved solid boundary treatment method for the solution of flow over an airfoil and square obstacle by SPH method*, Shadloo, M.S., Zainali, A., Sadek, S.M., Yildiz, M.

Session 3: Free-surface Flow & Bed modelling
***Chairman**: Jason Hughes, University of Plymouth, UK*

- *The SPH method to simulate the model test of a sandy river levee on seepage induced failures*, Mori, H., Saito, Y., Sasaki, T., Sago, K., Kuwano, R.
- *Hydraulic jump simulation by SPH*, De Padova, D., Mossa, M., Sibilla, S., Torti, E.
- *Prediction of Sediment Scouring through SPH*, Manenti S., Sibilla S., Gallati M., Agate, G., Guandalini, R.

Session 4: Multi-Fluids 2
***Chairman**: Xiangyu Hu, University of Munich, Germany*

- *SPH Modelling of water/soil-suspension flows*, Ulrich, C., Rung, T.
- *Modelling the flow of self-compacting concrete*, Kulasegaram, S., Karihaloo, B.L., Ghanbari, A.
- *Simulation of liquid impacts using a multiphase parallel SPH model*, Oger, G., Guilcher, P.M., Jacquin, E., Brosser, L., Grenier, N., LeTouzé, D.

Session 5: Boundary Conditions 2
***Chairman**: David Le Touzé, Ecole Centrale Nantes, France*

- *IB-SPH simulations of wave-body interactions*, Cherfils, J.M., Blonce, L., Pinon, G., Rivoalen, E.
- *Simulating free-surface channel flows through SPH*, Federico, I., Marrone, S., Colagrossi, A., Aristodemo, F., Veltri, P.
- *Improved time scheme integration approach for dealing with semi analytical wall boundary conditions in SPARTACUS2D*, Ferrand, M., Laurence, D., Rogers, B.D., Violeau, D.

Session 6: Free-Surface Techniques
***Chairman**: Peter K. Stansby, University of Manchester, UK*

- *SPH Shallow Water Equation Solver for real flooding simulation*, Vacondio, R., Mignosa, P., Rogers, B.D., Stansby, P.K.
- *Violent Fluid-Structure impacts solved through a δ-SPH model*, Marrone, S., Antuono, M., Colagrossi, A., Colicchio, G., Le Touzé, D., Graziani, G.
- *Propagation of gravity wave-packets through a δ-SPH method*, Antuono, M., Marrone, S. , Colagrossi, A., Lugni, C.

DAY 2: Thursday 24 June 2010

8:45	**Keynote lecture:** Prof. P. Koumoutsakos, ETHZ, Switzerland
9:40	Coffee break
10:00	**Session 7: Analysis of SPH Methodology 1: Issues with free-surface flows**
11:00	**Session 8: Analysis of SPH Methodology 2: Mathematical aspects and Intrinsic effects**
12:00	Lunch
13:15	**Session 9: Analysis of SPH Methodology 3: Mathematical aspects, Error and Visualisation**
14:15	**Session 10 : Poster session**
15:15	Coffee break
15:30	**Session 11: Hydraulic Applications**
16:30	**Session 12: Fluid-Structure Interaction 1**
17:30	Steering committee meeting
19:00	Banquet and Libersky student prize

Keynote lecture: *Prof. P. Koumoutsakos, ETHZ, Switzerland*
Multiresolution simulations using particles

Session 7: Analysis of SPH Methodology 1: Issues with free-surface flows
Chairman*: Moncho Gómez Gesteira, University of Vigo, Spain*

- *The influence of the truncated kernel to free surface predictions with ISPH and a new solution*, Xu, R., Stansby, P.K.
- *Theoretical analysis of SPH in simulating free-surface viscous flows*, Colagrossi, A., Antuono, M., Souto-Iglesias, A., Izaguirre-Alza, P., Le Touzé, D.
- *Three SPH novel benchmark test cases for free-surface flows*, Botia-Vera, E., Souto-Iglesias, A., Bulian, G., Lobovsky, L.

Session 8: Analysis of SPH Methodology 2: Mathematical aspects and Intrinsic effects
Chairman*: Nathan Quinlan, National University of Ireland, Galway*

- *Lyapunov stability analysis of semi-discretised SPH*, Vignjevic, R., Powell, S.
- *WSPH and ISPH calculations of a counter-rotating vortex dipole*, González, L.M., Sánchez, J.M., Macià, F., Souto-Iglesias, A., Duque, D., Gómez-Goñi, J., Rodríguez-Perez, M.A.
- *Spurious atomistic viscosities in Smoothed Particle Hydrodynamics*, Ellero, M., Espanol, P.

Session 9: Analysis of SPH Methodology 3: Mathematical aspects, Error and Visualisation
Chairman*: Rade Vignjevic, University of Cranfield, UK*

- *Resolution study on Smoothed Particle Hydrodynamics with mesoscopic thermal fluctuation*, Bian, X., Ellero, M., Adams, N.
- *SPH truncation error in 3D simulations*, Amicarelli, A., Marongiu, J.-C., Leboeuf, F., Leduc, J., Fang, L., Caro, J.
- *In-situ visualization and analysis of SPH data using a ParaView plugin and a distributed shared memory interface*, Soumagne, J., Biddiscombe, J., Clarke, J.
- *An SPH model with C1 consistency*, Xu, H., Dao M.-H., Chan, E.-S.

Session 10: Poster session
Chairman*: Benedict Rogers, University of Manchester, UK*

- *Advanced pre-processing for SPHysics*, Mayrhofer, A., Gómez Gesteira, M., Crespo, A.J.C., Rogers, B.D.
- *Evaluation of SPH capability in modeling internal transient and oscillating flow regimes*, Shahriari, S., Hassan, I., Kadem, L.
- *Infiltration induced collapse in coastal bluffs*, Vandamme, J., Zou, Q., Ellis, E.
- *Improving the performance of a trapezoidal sloshing absorber*, Kennan, S., Prakash, M., Semercigil, S.E., Turan, O.F.
- *Development of a Smoothed Particle Hydrodynamics code for the numerical prediction of primary atomization of fuel injecting nozzles*, Höfler, C., Koch, R., Bauer, H.-J.
- *Numerical study on Fluid-Structure Interaction using Smoothed Particle Hydrodynamics and Finite Element methods*, Yang, Q., Jones, V., McCue, L.
- *SPHysics code validation against a near-shore wave breaking exp.*, Makris, C.V., Krestenitis, Y.N., Memos, C.

Session 11: Hydraulic Applications
Chairman*: Dominique Laurence, EDF, France*

- *Application of SPH-ALE method to Pelton hydraulic turbines*, Marongiu, J.-C., Parkinson, E., Leboeuf, F., Leduc, J.
- *Flow modeling in a Turgo turbine using SPH*, Koukouvinis, F., Anagnostopoulos, J. S., Papantonis, D. E.
- *The use of 3D SPHERA code to support spillway design and safety evaluation of flood events*, Agate, G., Guandalini, R.

Session 12: Fluid-Structure Interaction 1
Chairman*: Jean-Christophe Marongiu, Andritz Hydro, Switzerland*

- *High-performance Fluide-Structure Interactions for impacts with fast dynamic Europlexus software*, Caleyron, F., Combescure, A., Faucher, V., Potapov, S., Fabis, J.

- *SPH for waves generated by a heaving cone using variable mass particle distribution*, Omidvar, P., Stansby, P.K., Rogers, B.D.
- *A rectangular sloshing absorber with designed obstructions to improve energy dissipation*, Grant, J., Prakash, M., Marsh, A.P., Semercigil, S.E., Turan, O.F.

DAY 3: Friday 25 June 2010

8:45	**Session 13: Solids and Materials**
9:45	**Session 14: Alternative Formulations**
10:40	Coffee break
11:00	**Session 15: Multi-Fluids 3**
12:00	**Session 16: Multi-Fluids 4 & Free-Surface**
13:00	Lunch
14:00	**Session 17: Fluid-Structure Interaction 2**
15:00	**Session 18: Hardware Acceleration**
15:45	End of the workshop

Session 13: Solids and Materials
Chairman: *Sivakumar Kulasegaram, Cardiff University, UK*

- *A SPH modeling of material damage and failure*, Owen, M.
- *Modelling thixotropy with SPH: Application to ceramic processing*, Wonisch, A., Kraft, T., Moseler, M., Riedel, H.
- *SPH Modelling of fragmentation in metals*, De Vuyst, T., Campbell, J.C., Vignjevic, R.

Session 14: Alternative Formulations
Chairman: *Thomas Rung, University of Hamburg, Germany*

- *Third Generation RSPH: Towards robust and simple integration with conventional SPH techniques*, Børve, S.
- *Discrete differential operators for Voronoi particle dynamics*, Duque, D., Espanol, P.
- *Convergence of the finite volume particle method for viscous flow*, Lobovsky, L., Nestor, R.M., Quinlan, N.

Session 15: Multi-Fluids 3
Chairman: *Paul Groenenboom, ESI BV, Netherlands*

- *A soft-tissue model coupled with fluid dynamics using SPH*, Adami, S., Hu, X.Y., Adams, N.A.
- *SPH simulations of advective-diffusion phenomena induced by pollutants in water*, Aristodemo, F., Federico, I., Veltri, P., Panizzo, A.
- *Simulation of surface tension by SPH method and its applications*, Zhang, M., Zhang, S.

Session 16: Multi-Fluids 4 & Free-Surface
Chairman: *Stefano Sibilla, University of Pavia, Italy*

- *A study of the matter of SPH application to saturated soil problems*, Bui, Ha H., Fukagawa, R., Sako, K.
- *Internal mechanical response of a tethered DNA in shear flow*, Litvinov, S., Ellero, M., Hu, X.Y., Adams, N.A.
- *SPH-FEM coupling to simulate Fluid-Structure Interactions with complex free-surface flows*, Fourey, G., Oger, G., Le Touzé, D., Alessandrini, B.

Session 17: Fluid-Structure Interaction 2
Chairman: *Antonio Souto-Iglesias, Technical University of Madrid, Spain*

- *Numer. predictions of ship flooding scenarios using SPH*, Marsh, A.P., Oger, G., Khaddaj-Mallat, C., Le Touzé, D.
- *SPH simulations of fish-like swimmers*, Kajtar, J., Monaghan, J.J.
- *Numer. simulation of the ditching of a helicopter with flotation devices*, Cartwright, B., Groenenboom, P., Chhor, A.

Session 18: Hardware Acceleration
Chairman: *John Biddiscombe, CSCS, Switzerland*

- *Parallel hybrid CPU/GPU acceleration of the 3-D parallel code SPH-flow*, Oger, G., Jacquin, E., Doring, M., Guilcher, P.M., Dolbeau, R., Cabelguen, P.L., Bertaux, L., LeTouzé, D.
- *Development of a dual CPU-GPU SPH model*, Crespo, A.J.C., Domínguez, J.M., Barreiro, A., Gómez-Gesteira, M.

Improved time scheme integration approach for dealing with semi analytical wall boundary conditions in SPARTACUS2D

***Martin Ferrand** & **Dominique Laurence** & **Benedict D. Rogers**, School of Mech., Aero. & Civil Engineering, University of Manchester, Manchester, UK*
***Damien Violeau**, Saint-Venant Laboratory for Hydraulics, Université Paris-Est, Chatou, France*

This work received the award of best student paper (Libersky Prize) at the 5th Int. SPHERIC Workshop, Manchester (UK), June 2010 (Ferrand et al., 2010).

Dealing with wall boundary conditions is one of the most challenging parts of the SPH method and many different approaches have been developed. Accurate boundary conditions are of great interest in fields such as studying turbulence close to the wall which is the overall aim of this current research effort. The present work is based on Kulasegaram *et al.* (2004) which consists of renormalizing the density field near a solid wall with respect to the missing kernel support area with the γ_a factor defined by:

$$\gamma_a \equiv \int_{\Omega\cup\Omega_a} w(|\mathbf{r}-\mathbf{r}_a|)d\mathbf{r}$$

This methodology, combined with the Lagrangian formalism, defines intrinsic gradient and divergence operators which are variationally consistent and ensure conservation properties. But as mentioned by De Leffe *et al.* (2009), the latter method defines an inaccurate gradient operator which provides non consistent behaviour. We have developed corrections of the discrete gradient operator for an arbitrary field $\{A_b\}$ as:

$$\mathbf{Grad}_a \equiv \frac{\rho_a}{\gamma_a}\sum_b\left(\frac{A_a}{\rho_a^2}+\frac{A_b}{\rho_b^2}\right)\nabla w_{ab} - \frac{1}{\gamma_a}\sum_s\left(\frac{A_a}{\rho_a}\rho_s+\frac{A_s}{\rho_s}\rho_a\right)\nabla\gamma_{as}$$

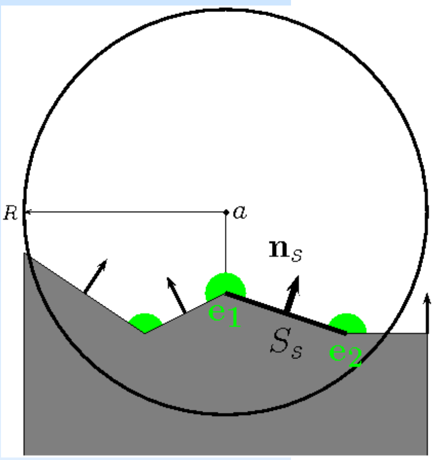


Figure 1 – Sketch of the shape of a boundary with edge particles *e* (in green) and segments *s* with surface S_s and inward normal $\mathbf{n}_s$.

We approximate the shape of the boundary $\partial\Omega$ of the domain Ω with straight segments in 2D denoted by the subscript $(.)_s$ which have a normal $\mathbf{n}_s$ and a surface area S_s (see Figure 1). Each segment is defined by two edge points denoted by the subscript $(.)_{e1}$ and $(.)_{e1}$. These edge particles (also called semi particles in this paper) are of particular interest for recording the pressure field at the solid boundary (and hence for fluid and structure coupling, for example). They are also useful to improve accuracy of the continuity equation, as they mimic a wet wall. The contribution of each segment s to $\nabla\gamma_a$, which represents a sort of normal to the wall, is denoted by $\nabla\gamma_{as}$ and defined by

$$\nabla\gamma_{as} \equiv \left[\int_{r_{e1}}^{r_{e2}} w(r)d\ell\right]\mathbf{n}_s$$

and then $\nabla\gamma_a$ can be decomposed in:

$$\nabla\gamma_a = \sum_s \nabla\gamma_{as}$$

In order to compute the kernel correction, Feldman and Bonet (2007) use an analytical value which is computationally expensive, whereas Kulasegaram *et al.* (2004) and De Leffe *et al.* (2009) use polynomial approximations which can be difficult to define for complex geometries. We propose a different method, the quantities $\nabla\gamma_{as}$ being either computed analytically, or approximated by

$$\nabla\gamma_a = \sum_s w_{as}S_s\mathbf{n}_s$$

As regards the γ_a's, they are computed using a dynamic equation. For a moving deformable wall (in the sense that each segment composing the wall is moving with its own velocity), this equation reads

$$\frac{d\gamma_a}{dt} = \sum_s \nabla\gamma_{as}\cdot\mathbf{u}_a^{R_s}$$
$$\gamma_a = 1 \quad \text{if} \quad \partial\Omega\cap\Omega_a = \varnothing$$

where $\mathbf{u}_a^{R_s}$ is the velocity of the particle a in a reference frame R_s where the segment s is fixed.

The time integration scheme used for the continuity equation requires particular attention, and as already mentioned by Vila (1999), we prove there is no point in using dependence in time of the particles' density if no kernel gradient corrections are added. Thus, by using a near-boundary kernel-corrected version of the time integration scheme proposed by Vila, we are able to simulate long-time simulations ideally suited for turbulent flow in a channel in the context of accurate boundary conditions.

A comparison is made between different models in a still water case and a dynamic case. Some of the developments to treat the solid boundaries suffer from an inability to reproduce correctly a still water case: here a basin of approximately 2 m large and 1 m height with a wedge in the bottom middle of the tank. We compare the results obtained when the basin is filled with 0.5 m of water for the Lennard-Jones repulsive forces, the traditional fictitious particles method and the present method. As expected, the repulsive forces give the worst results (see Figure 2) in the sense that particles keep sliding along vertical walls. That is due to the fact that the missing area in the kernel support is not compensated, and thus the gravity is not balanced enough. The plot of the pressure of particles against the depth is therefore very noisy and very badly reproduced next to the bottom. The fictitious particles method gives better results, but the pressure profile is still noisy. Moreover this approach is problematic to describe in complex geometries and requires additive particles to mimic the boundary, which increase the computation cost. The present method gives expected results: a linear pressure profile even near the bottom and a zero velocity field.

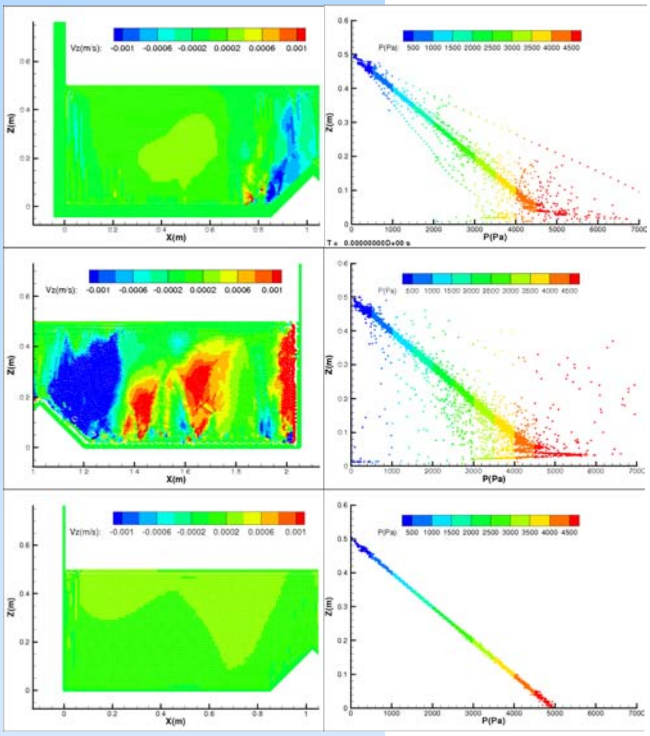


Figure 2 – Vertical velocity field and pressure profile for still water in a tank with a wedge after 20s, with different boundary conditions: from top to bottom, fictitious particles; Lennard-Jones forces; present method.

A simulation of a dam break with the same specific shape of the boundary has been performed for the two methods previously described and the present one. The water is initially a column of 1 m height and 0.5 m width on the left side of the basin. Snapshots of the pressure field at the same physical time are plotted on Figure 3. We can notice that all the models ensure impermeable boundaries, but both repulsive forces and fictitious particles methods give more noisy pressure fields. Furthermore, a refinement has been computed by reducing the initial particle spacing by a factor 2. With the present model for wall BC, the pressure field is even smoother.

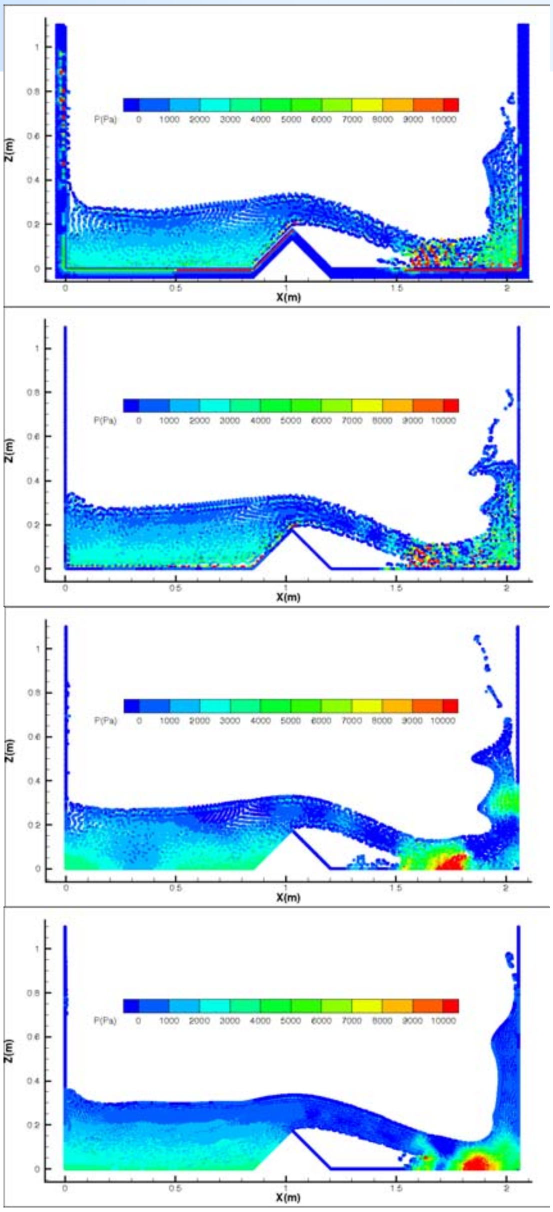


Figure 3 – Pressure field for a dam break test case in a tank with a wedge: from top to bottom, fictitious particles; Lennard-Jones forces; present method; present method with higher discretization.

Contact: martin.ferrand@ponts.org

References

Ferrand, M., Laurence, D., Rogers, B., Violeau, D. (2010), *Improved time scheme integration approach for dealing with semi-analytical wall boundary conditions in Spartacus2D*, Proc. 5th SPHERIC International Workshop, pp. 98–105.

De Leffe, M., Le Touzé, D., Alessandrini, B. (2009), *Normal flux method at the boundary for SPH*, Proc. 4th SPHERIC International Workshop, pp. 149–156.

Kulasegaram, S., Bonet, J., Lewis, R.W., Profit, M. (2004), *A variational formulation based contact algorithm for rigid boundaries in two-dimensional SPH applications*, Comput. Mech. **33**:316–325.

Improvement of multiphase model using preconditioned Riemann solvers

J. Leduc, F. Leboeuf, M. Lance, *Ecole Centrale de Lyon, University of Lyon, Ecully, France*
E.Parkinson, J.-C. Marongiu*, ANDRITZ Hydro, Vevey, Switzerland*

This work received the 2nd award of best student paper at the 5th Int. SPHERIC Workshop, Manchester (UK), June 2010 (Leduc et al., 2010).

Multiphase phenomena are an issue in terms of numerical developments when high density ratios and surface tension effects are present. Previous work on the SPH-ALE (Arbitrary Lagrange Euler) method brought improvement in term of precision, stability of single phase flows (Marongiu, 2008) and shows its ability to model multiphase flows (Leduc, 2009). This method is based on the work of Vila (1999) who introduced resolution of Riemann problems between pairs of particles in the ALE framework. As it is established for the finite volumes method, the introduction of Godunov scheme leads to excessive numerical diffusion which decreases the quality of the results (see also Murrone and Guillard, 2008).

A preconditioned Riemann solver is a way to decrease this numerical diffusion by acting on the mathematical properties of the Jacobian matrix for the Euler equations. Using the preconditioner of Turkel, the eigenvalues are better conditioned (using the mathematical definition). By testing this method on low Mach number multiphase test cases, this study evaluates the capability of preconditioning to reduce numerical diffusion.

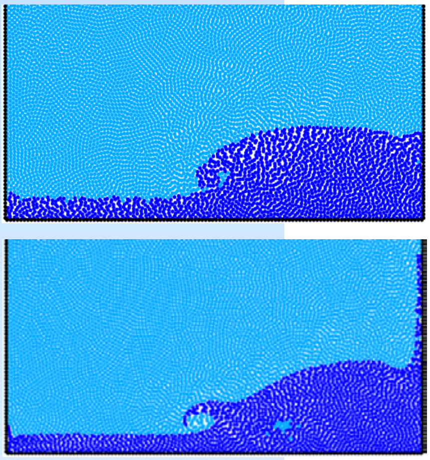


Figure 1 – Water dam break without (top) and with (bottom) preconditioning.

Figures 1 show the impact of preconditioning for the simulation of a water dam break test case. The density ratio is 1000, and 1800 particles are used to describe water. With preconditioning, the method is able to catch a thin layer of water which remains near the right sidewall. The model is also able to catch water entrapment which was not present without preconditioning.

The results of a gravity wave test case are shown on figure 2 for two different discretizations. The impact of preconditioning is clearly visible on the decrease of the oscillations of the kinetic energy (no viscosity term is present). Preconditioning has a higher influence for lower discretization. This technique has also an impact on the frequency of the gravity waves. For the 120×120 discretization, the error goes from 0.50% without preconditioning to 0.47% with preconditioning.

This technique was also applied to test cases with surface tension effects and comparable results were observed on static and oscillating 2D droplets. This study shows the interest of using preconditioning techniques in the frame of the SPH-ALE method to decrease the numerical diffusion.

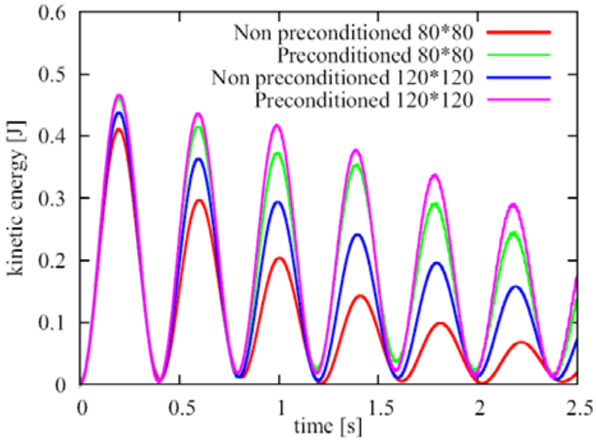


Figure 2 – Evolution of the kinetic energy for the gravity wave test case.

Contact: julien.leduc@ec-lyon.fr

References

Marongiu J.-C., Leboeuf F. and Parkinson E. (2008), *Riemann solvers and efficient boundary treatments: an hybrid SPH-finite volume numerical method*, Proc. 3rd Int. SPHERIC Workshop, Lausanne (Switzerland), pp. 101–108.

Murrone A. and Guillard H. (2008), *Behaviour of upwind scheme in the low Mach number limit: III. Preconditioned dissipation for a five equation two phase model*, Comput. Fluids **37**:1209–1224.

Leduc J., Marongiu J.-C., Leboeuf F., Lance M. and Parkinson E. (2009), *Multiphase SPH: A new model based on acoustic Riemann solver*, Proc. 4th Int. SPHERIC Workshop, Nantes (France), pp. 8–13.

Leduc J., Marongiu J.-C., Leboeuf F., Lance M. and Parkinson E. (2010), *Improvement of multiphase model using preconditioned Riemann*, Proc. 5th Int. SPHERIC Workshop, Manchester (UK), pp. 1–6.

Multi-phase SPH simulations including surfactant effects

S. Adami, X.Y. Hu & N.A. Adams, *Institute of Aerodynamics, Technical University of Munich*

This work received the 3rd award of best student paper at the 5th Int. SPHERIC Workshop, Manchester (UK), June 2010.

In multi-phase systems with small length scale surface tension effects may dominate the flow characteristics, thus play an important role. The overall effect of surface tension is to reduce the interfacial area and can be split into a normal (capillary force) and a tangential component (Marangoni force). Usually it is adequate to consider only the capillary forces, which depend on the local curvature of the interface and the pairing of the fluids in contact. But when the surface tension varies along the interface, the resulting surface tension gradients can induce tangential forces of relevant magnitude. Surface tension gradients develop mainly due to temperature gradients along the interface or due to the presence of surface active agents. These so-called "surfactants" replace fluid molecules at an interface and consequently change the strength and direction of the surface tension. Widely used in many technological applications, most important is the presence of surfactants in the human lung.

Due to the strong reduction of the surface tension at the fluid-air interface in the lung alveoli, surfactants lower the work required for respiration and are essential for the breathing process. Diseases like infant respiratory distress syndrome (IRDS) or adult respiratory distress syndrome (ARDS) are known to result from a dysfunction or a lack of surfactant (Hamm *et al.*, 1992). Interestingly, a surfactant replacement therapy (injection of artificial surfactant to the pulmonary alveoli) is restoring the IRDS whereas does not help to recover from an ARDS. Additionally, ventilator induced lung injuries (VILI) caused by artificial respiration in intensive care medicine are not fully understood and believed to be connected to the dynamics of the local surfactant distribution.

Besides its general necessity for the respiratory system, a profound knowledge of the effect of surfactants in the lung is still missing. With the aim of developing new artificial respiratory systems, we want to study the effect of surfactants in the lung to understand better the dominating phenomena in the pulmonary system. Experimentally almost impossible to investigate, simulations of the interaction of the epithelial lung cells with the surfactant enriched liquid lining layer and the respired air provide a detailed insight into this complex problem.

We have developed a three-dimensional multi-phase SPH method with surfactant effects (Adami *et al.*, 2010). Due to its Lagrangian nature, SPH is capable of simulating complex deforming geometries and is advantageous especially for multi-phase flows. In our model, the surfactant is treated as an active scalar, which changes the surface tension coefficient at an interface locally depending on the concentration. In our conservative scheme we consider surfactant diffusion on the interface as well as in the bulk phase. Furthermore, we have introduced transport from/to the bulk phase to the interface representing adsorption and desorption.

To show the effect of surfactants on deforming interfaces, here we present the drop deformation in a simple shear flow at Re=1 and Ca=0.4. In Figure 1 we show a snapshot of the concentration Γ at T=50. The upper and lower wall boundaries move with the velocity U_∞ in opposite directions, hence the initial spherical droplet deforms and finally breaks up into three smaller drops. At moderate diffusion rates surfactant gradients along the interface are smoothed out.

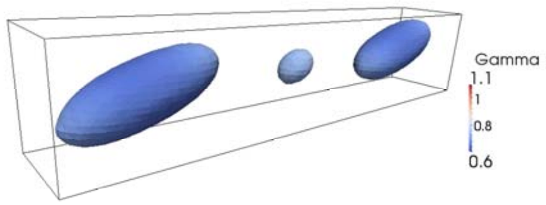


Figure 1 – Drop breakup in simple shear flow at Re=1, Ca=0.4 and Pe=1 at T=50.

At high Peclet numbers, where surface diffusion is negligible, the surfactant concentration increases at the tips and reduces the surface tension strongly. Consequently, a singularity develops at the tips and the so-called "tip-streaming" effect is observed, see Figure 2.

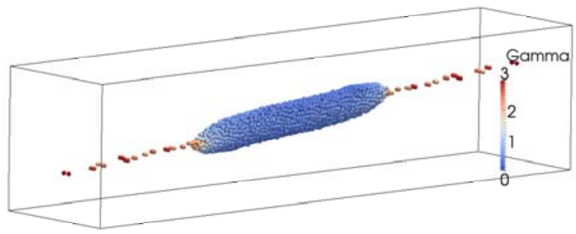


Figure 2 – Tip-streaming at Pe=100, other parameters compare Figure 1.

This simple example shows how surfactants can alter the dynamics of a multi-phase system. In order to study more realistic problems, we now couple our fluid solver with a soft-tissue model to simulate a fully coupled three-dimensional alveolar geometry.

Contact: stefan.adami@tum.de

References

Hamm, H., Fabel, H. and Bartsch W. (1992), *The surfactant system of the adult lung: physiology and clinical perspectives,* Clin. Investig. **70**:637–657

Adami, S., Hu, X.Y. and Adams, N.A. (2010), *A conservative SPH method for surfactant dynamics,* J. Comp. Phys. **229(5)**:1909.

On recent enhancements of Moving Particle Semi-implicit (MPS) method

***H. Gotoh** & **A. Khayyer**, Department of Civil and Earth Resources Engineering, Kyoto University, Kyoto, Japan*

Particle methods have been increasingly applied as powerful and versatile computational tools to simulate a wide variety of physical processes including incompressible free-surface fluid flows. For this class of problems, the MPS (Moving Particle Semi-implicit; see Koshizuka and Oka, 1996) method is a well-known projection-based particle method that has been successfully applied to a broad range of engineering applications.

Despite its capability, the original MPS method has a few drawbacks including non-conservation of momentum and existence of unphysical pressure fluctuations. To resolve these shortcomings, Khayyer and Gotoh (2008, 2009, 2010a) proposed revised versions of MPS, namely, CMPS (Corrected MPS), CMPS-HS (CMPS with Higher order Source term) and CMPS-HS-HL-HV (CMPS-HS with a Higher order Laplacian model applied for both Poisson pressure Equation and Viscous forces). Figure 1 shows two typical snapshots corresponding to a sloshing simulation by CMPS-HS and CMPS-HS-HL-HV methods.

Figure 2 depicts the application of CMPS-HS-BF (Bottom Friction) to simulation of a dam break on a wet bed (Janosi *et al.*, 2004) and its associated mixing processes. Detailed descriptions and comparisons with ISPH and SPH results are provided by Khayyer and Gotoh (2010b).

The improved MPS methods have also been extended to three-dimensions. GPU-based and parallelized codes have been developed for enhancement of computational efficiency. Figure 3 shows two typical snapshots illustrating a plunging wave breaking and its resultant splash-up simulated by 3D parallelized CMPS (Gotoh *et al.*, 2009).

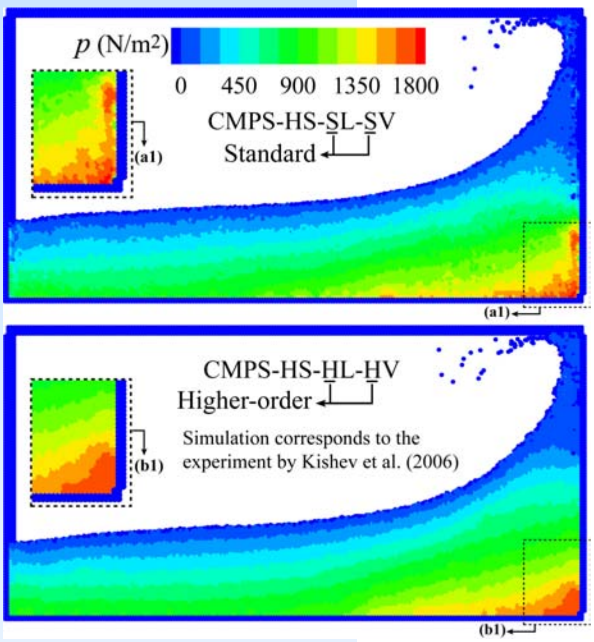


Figure 1 – Sloshing simulation by improved MPS methods (Khayyer and Gotoh, 2010a).

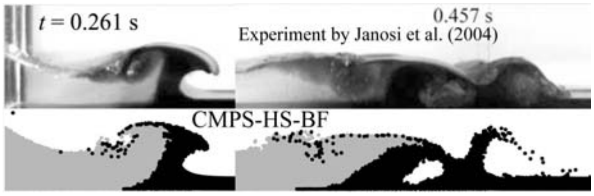


Figure 2 – Simulation of a dam break on a wet bed by an improved MPS (Khayyer and Gotoh, 2010b).

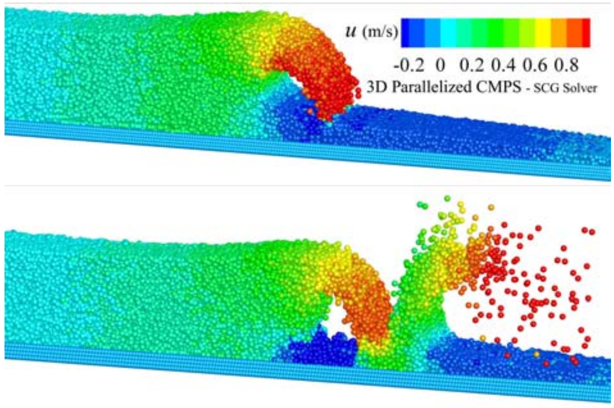


Figure 3 – Snapshots illustrating a plunging wave breaking and resultant splash-up (Gotoh *et al.*, 2009).

Contact: gotoh@particle.kuciv.kyoto-u.ac.jp
khayyer@particle.kuciv.kyoto-u.ac.jp

References

Gotoh, H., Khayyer, A., Ikari, H. and Hori, C. (2009), *Refined Reproduction of a Plunging Breaking Wave and Resultant Splash-up by 3D-CMPS Method*, Proc. ISOPE-2009, Osaka, Japan, pp. 518–524.

Janosi, I.M., Jan, D., Szabo, K.G., Tel, T. (2004), *Turbulent drag reduction in dam-break flows*, Exp. Fluids. **37**:219.

Khayyer, A. and Gotoh, H. (2008), *Development of CMPS method for accurate water-surface tracking in breaking waves*, Coastal Eng. J. **50**:179.

Khayyer, A. and Gotoh, H. (2009), *Modified Moving Particle Semi-implicit methods for the prediction of 2D wave impact pressure*, Coastal Eng. **56**:419.

Khayyer, A. and Gotoh, H. (2010a), *A Higher Order Laplacian Model for Enhancement and Stabilization of Pressure Calculation by the MPS Method*, Applied Ocean Research doi:10.1016/j.apor.2010.01.001.

Khayyer, A. and Gotoh, H. (2010b), *On particle-based simulation of a dam break over a wet bed*, Jour. Hydraulic Res., IAHR **48**:238.

Koshizuka, S. and Oka, Y. (1996), *Moving particle semi-implicit method for fragmentation of incom-pressible fluid*, Nuclear Sci. Eng. **123**:421.