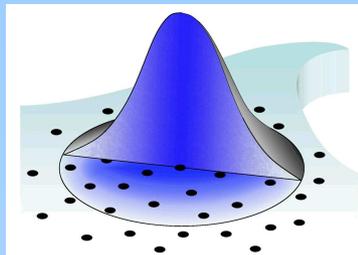


SPH European Research Interest Community

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

Contact: david.letouze@ec-nantes.fr



D. Le Touzé, Chairman
N. Quinlan, Secretary
B. Rogers, Webmaster
D. Violleau, Newsletter

Steering Committee

Ecole Centrale de Nantes (France)
National University of Ireland, Galway
University of Manchester (UK)
University of Vigo (Spain)
L'Aquila University (Italy)
EDF R&D / LNHE (France)
VA TECH Hydro (Switzerland)
University of Plymouth (UK)
Swiss National Supercomputing Centre
INSEAN (Italy)
ESI-Group Netherlands
Technical University of Munich (Germany)

Members

Ecole Centrale de Lyon (France)
Novosibirsk State University (Russia)
Technische Universitaet Muenchen (Germany)
Univer. Nacional de Educacion a Distancia (Spain)
Johns Hopkins University (USA)
University of Nottingham (UK)
University of Bradford (UK)
University of Lancaster (UK)
Université de Montpellier (France)
Technical University of Madrid (Spain)
Shanghai Jiao Tong University (China)
Ecole Polytechnique Fédérale de Lausanne (Switzerland)
Université du Havre (France)
Swiss Federal Inst. of Technology (Switzerland)
ERSE SpA (Italy)
CIMNE Barcelona (Spain)
University of Palermo (Italy)
University of Genova (Italy)
CEDEX (Spain)
University of Pavia (Italy)
Dublin Institute of Technology (Ireland)
Imperial College London (UK)
Institute for Plasma Research (India)
BAE Systems (UK)
University of Umeå (Sweden)
Institut Français du Pétrole (France)
University of West Bohemia (Czech Republic)
Tarbiat Modares University (Iran)
University of West Bohemia (Czech Republic)
Universidade de Santiago de Compostela (Spain)
CUGRI (Italy)
University of Hamburg (Germany)
Iran University of Science & Technology
University of Cambridge (UK)
ASR Limited (New Zealand)
Cranfield University (UK)
City University London (UK)
HydroOcean (France)
Laboratório Nacional de Engenharia Civil (Portugal)
Aristotle University of Thessaloniki (Greece)
Catholic University Leuven (Belgium)
University of Calabria (Italy)
University of Ljubljana (Slovenia)
Virginia Tech (USA)
SINTEF Materials and Chemistry (Norway)
Manchester Metropolitan University (UK)
Aberystwyth University (UK)
J P KENNY Sdn. Bhd. (Malaysia)
Politecnico di Milano (Italy)
Bangladesh Univ. of Engineering and Technology
Ritsumeikan University (Japan)
Institut fuer Thermische Stroemungsmaschinen (Germany)
Sulzer Markets & Technology Ltd (Switzerland)
University of Heidelberg (Germany)
Fraunhofer-Chalmers Res. Centre for Ind. Math. (Sweden)
Simulation Innovation Lab (South Korea)
Mustafa Kemal University (Turkey)
National Cheng Kung University (China)
Amir Kabir University of Technology (Iran)
University Technology Malaysia
National Nuclear Laboratory (UK)
Texas A&M University (USA)
Istituto Nazionale di Geofisica e Vulcanologia (Italy)
Glasgow Caledonian University (UK)
National Technical University of Athens (Greece)
University of Exeter (UK)
University of Parma (Italy)
Kyoto University (Japan)
CRS4 (Italy)
University of Regina (Canada)
University of Auckland (New-Zealand)
Alstom Hydro (France)
Mara University of Technology (Malaysia)
Institute of Mechanics (Vietnam)
Instituto Superior Tecnico (Portugal)
Kitware (USA)

Editorial: 7th SPHERIC workshop

The 7th international workshop organized by the Smoothed Particle Hydrodynamics European Research Interest Community will be the major international event of 2012 focused entirely on SPH techniques and applications. It will be held at Prato (Italy), a campus of Monash University and jointly organized by Monash University (Australia), the University of Pavia (Italy) and CNR-INSEAN (Italy).

The Workshop will be held on May 29th–31st with a training day on 1st June. Prato is a small but thriving Tuscan city, with a beautiful historic centre, and a vibrant cultural and economic life famous for its textile industry. Prato has a rich historical and artistic heritage, including a mid-13th century castle built by the Hohenstaufen Emperors. The centre of the city is enclosed by well preserved medieval walls. Prato is well connected by train to Florence (30 mins), Bologna (1 hour), Pisa (1 ½ hours), Lucca (1 hour), Rome (2 hours), Venice and Milan (3 hours).

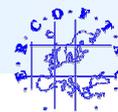


The Monash University Prato Centre (see above) is located on the ground and first floors of the elegant 18th century Palazzo Vaj on Via Pugliesi in the historic centre of Prato. Further information about the Prato Centre can be found on the Monash University Prato Centre website which is linked to the Workshop website (see below).

The aim of this scientific workshop is to enable experienced researchers using SPH to share and contribute to the development and applications of the SPH method, and enable PhD students to present their work in a favourable atmosphere. The topics will include a large range of applications and technical improvements to SPH including:

- Solids and Structure;
- Multiple fluid and Multi-Phase Flows;
- Boundary Conditions;
- Viscosity and Turbulence;
- Incompressible Flows;
- Free Surface and Moving Boundaries;
- Voronoi-particle and FV-particle methods, adaptive SPH;
- Shallow-water SPH method;
- High-Performance Computing;
- Hydraulic Applications;
- Maritime and Naval Architecture Applications;
- Process Engineering;
- Geotechnical Applications;
- Micro Fluidics;

(continued next page)



- Astrophysics;
- Solids and Fracture Mechanics;
- Biomechanics;
- Disaster Simulations;
- Virtual Surgery;
- Virtual Reality Applications.

The number of papers presented will be limited so there will be no parallel sessions. The papers presented will be selected from the abstracts. There will be no posters. The Libersky prize will be awarded to the best student judged on the presentation and paper. A feature of this Workshop will be open discussions of a special SPH

topic at the end of the first and second day. The topics of the discussions will be announced later.

The important dates are (approximately):

- Abstract submission: mid-February 2012;
- Announcement of selected abstracts: mid-March;
- Early registration deadline: late March;
- Final papers submission: late April.

Further details can be found on the Website:

<http://spheric7.monash.edu/>

Joseph Monaghan
Stefano Sibilla
Andrea Colagrossi

Selected Recent Publications and Ph.D. Theses on SPH

Below is a small selection of references recently added to SPHERIC's online catalogue of SPH literature at <http://www.citeulike.org/group/3462>. Anybody can access and contribute to this database.

Amicarelli, A., Marongiu, J.-C., Leboeuf, F., Leduc, J., Neuhauser, M., Fang, L., Caro, J. (2011). *SPH truncation error in estimating a 3D derivative*. Int. J. Num. Meth. Eng. **87**(7):677–700. <http://dx.doi.org/10.1002/nme.3131>

Maciá, F., Antuono, M., González, L. M., Colagrossi, A. (2011). *Theoretical analysis of the No-Slip boundary condition enforcement in SPH methods*. Progr. Theor. Phys. **125**(6):1091–1121. <http://dx.doi.org/10.1143/PTP.125.1091>

Hess, S. Springel, V. (2010). *Particle hydrodynamics with tessellation techniques*. Mon. Not. R. Astron. Soc. **406**(4):2289–2311. <http://dx.doi.org/10.1111/j.1365-2966.2010.16892.x>

Vacondio, R., Rogers, B. D., Stansby, P. K. (2011). *Accurate particle splitting for smoothed particle hydrodynamics in shallow water with shock capturing*. Int. J. Num. Meth. Fluids. <http://dx.doi.org/10.1002/fld.2646>

Gomez-Gesteira, M., Rogers, B. D., Dalrymple, R. A., Crespo, A. J. C. (2010). *State-of-the-art of classical SPH for free-surface flows*. J. Hydr. Res. **48**(S1):6-27. <http://dx.doi.org/10.1080/00221686.2010.9641242>

Jiang (2011). *A corrected smoothed particle hydrodynamics method for solving transient viscoelastic fluid flows*. App. Math. Mod. **35**(8):3833–3853. <http://dx.doi.org/doi:10.1016/j.apm.2011.02.014>

Adami, S., Hu, X. Y., Adams, N. A. (2010). *A new surface tension formulation for multi-phase SPH using a reproducing divergence approximation*. J. Comput. Phys. <http://dx.doi.org/10.1016/j.jcp.2010.03.022>

Fatehi, R. Manzari, M. T. (2010). *A remedy for numerical oscillations in weakly compressible smoothed particle hydrodynamics*. Int. J. Numer. Meth. Fluids, **67**(9):1100–1114. <http://dx.doi.org/10.1002/fld.2406>

<http://dx.doi.org/10.1002/fld.2406>

Ellero, M., Español, P., Adams, N. A. (2010). *Implicit atomistic viscosities in smoothed particle hydrodynamics*. Phys. Rev. E **82**(4):046702. <http://dx.doi.org/10.1103/PhysRevE.82.046702>

Crespo, A. C., Dominguez, J. M., Barreiro, A., Gómez-Gesteira, M., Rogers, B. D. (2011). *GPUs, a new tool of acceleration in CFD: Efficiency and reliability on smoothed particle hydrodynamics methods*. PLoS ONE **6**(6):e20685. <http://dx.doi.org/10.1371/journal.pone.0020685>

The following theses are available to download in full at <http://wiki.man.ac.uk/spheric>:

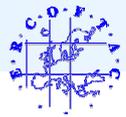
J.B. Kajtar: Smooth lattice general relativity, and SPH simulations of swimming linked bodies, Monash University.

R. Vacondio: Shallow Water and Navier-Stokes SPH-like numerical modelling of rapidly varying free-surface flows, Università degli Studi di Parma Facoltà di Ingegneria.

R. Xu: An Improved Incompressible Smoothed Particle Hydrodynamics Method and Its Application in Free-Surface Simulations, University of Manchester.

I. Federico: Simulating Open-channel Flows and Advective Diffusion Phenomena through SPH Model, Università della Calabria.

M. Basa: Improvements in the accuracy and performance of smoothed particle hydrodynamics for fluid dynamics applications, National University of Ireland Galway.



Augmenting meshless methods using the Voronoi tessellation

J. Michael Owen, Lawrence Livermore National Lab, M/S L-38, P.O. Box 808, Livermore, CA, 94550, USA

The Lagrangian nature of SPH combined with its robustness makes it an attractive tool for modelling problems with multiple materials and/or solids undergoing large deformations. However, the fact that SPH does not have a clear definition of interfaces between materials (or their surfaces in general) raises a host of issues. One example of this difficulty arises in evolving the mass density of materials with surfaces. The well-known summation definition for the SPH density:

$$\rho_i = \sum m_j W_{ij} \quad W_{ij} \equiv W(x_j - x_i, h_i)$$

yields large errors near surfaces due to the implicit volume normalization of the SPH sum, i.e., the assumption

$$\sum_j \frac{m_j}{\rho_j} W_{ij} \approx 1$$

fails at surfaces. Some researchers have tried to come up with surface corrections, but common practice to date seems to be to employ the continuity equation:

$$\frac{D\rho_i}{Dt} = -\rho_i (\partial_\alpha v_\alpha)_i \approx \sum_j m_j (v_i^\alpha - v_j^\alpha) \partial_\alpha W_{ij}$$

(Gray, 2001) in modelling solids with surfaces. However, this approach is generally less accurate than the summation definition.

Recently we have been investigating (Owen, 2011) how we might use the Voronoi tessellation in order to improve our meshless calculations. The Voronoi tessellation offers many interesting properties we might exploit: well defined convex cell volumes for each generator, planar faces, continuous cell volume change with moving generators, robust automatic generation, etc. As an initial application we have looked at using the unique Voronoi cell volumes V_i associated with each SPH point i in order to construct a corrected summation density. The formulation we have had the most success with is

$$\rho_i = \frac{\sum_j m_j W_{ij}}{\sum_j V_j W_{ij}}$$

This definition explicitly corrects the standard SPH summation density with the weighted Voronoi cell volume of the points. Assuming the Voronoi tessellation respects the surfaces of the material, this naturally accounts for the presence of those surfaces and avoids the normalization difficulty of the summation density.

Convergence studies using analytic test problems show this volume corrected density equation is at least as accurate as the ordinary SPH definition, and much more accurate than the continuity equation. Interestingly the

volume corrections make some difference on the interiors of problems as well, particularly near strong density gradients (such as at strong shock transitions). This is not surprising: as the local SPH node distribution becomes increasingly non-uniform the standard assumptions under which SPH is derived begin to break down –surfaces are simply an extreme example.

Figure 1 shows snapshots of the density evolution in a simulation of a two-material experiment (Haas, 1987); in this experiment a shock in air is propagated over a He bubble, resulting in a great deal of vorticity by late time. The density contrast between the air and He (roughly a factor of 5.5) makes evolution at the interface challenging: the model employing the continuity equation fails to complete, while the uncorrected SPH sum density demonstrates many artefacts near the material boundary. By contrast the corrected density algorithm (shown in figure 1) completes the simulation without difficulty and demonstrates reasonable convergence properties with refinement. Note in figure 1 we have rendered the density pseudo-color on the Voronoi mesh.

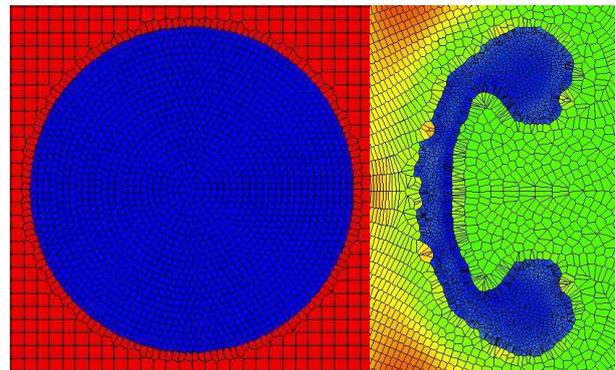
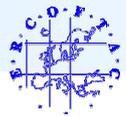


Figure 1 – Simulated mass density in the shock-bubble experiment: initial unshocked configuration at $t = 0$ s (left) and density well after the shock passage at $t = 0.002$ s (right).

Contact: mikeowen@llnl.gov

References

- Gray, J.P., Monaghan, J.J., Swift, R.P. (2001), *SPH elastic dynamics*, *Comput. Methods Appl. Mech. Engrg.* **190**:6641.
- Haas, J.F., Sturtevant, B. (1987), *Interaction of weak shock waves*, *J. Fluid Mech.*, 181:41.
- Owen, J.M. (2011), *Applications of the Voronoi tessellation for mesh-free methods*, presented at the 2011 Multimaterial Hydrodynamics conf., Arcachon, France, Sept 5–9.



GPU computing on SPH models: DualSPHysics code

A.J.C. Crespo, J.M. Dominguez, M.Gómez-Gesteira, EPhyslab, Universidade de Vigo, Ourense, Spain
B.D. Rogers, School of Mechanical, Aerospace and Civil Engineering, University of Manchester, Manchester, U.K.

The open-source code SPHysics (www.sphysics.org) allows modelling problems using fine description, but the main problem is the high computational runtime, so that SPHysics is hardly applied over large domains. Hardware acceleration and parallel computing are required to make SPHysics more useful and versatile.

Graphics Processing Units (GPUs) appear as a cheap alternative to handle High Performance Computing for numerical modelling. GPUs are designed to manage huge amounts of data and their computing power has increased in the last years much faster than central processing units (CPUs) have evolved. The Compute Unified Device Architecture (CUDA) is a parallel programming framework and language for the multi-core computations on the GPU device which is effectively the C/C++ language with some extensions. Researchers and engineers of different fields are achieving high speedups implementing their codes with the CUDA language. The parallel power computing of GPUs can be also applied to SPH methods where the same loops for each particle along the simulation can be parallelised.

The code DualSPHysics (www.dual.sphysics.org) has been developed starting from the SPH formulation implemented in SPHysics (Gómez-Gesteira *et al.*, 2010). This FORTRAN code is robust and reliable but is not properly optimised for huge simulations. DualSPHysics is implemented in both the C++ and CUDA languages to carry out simulations on the CPU and GPU, respectively.

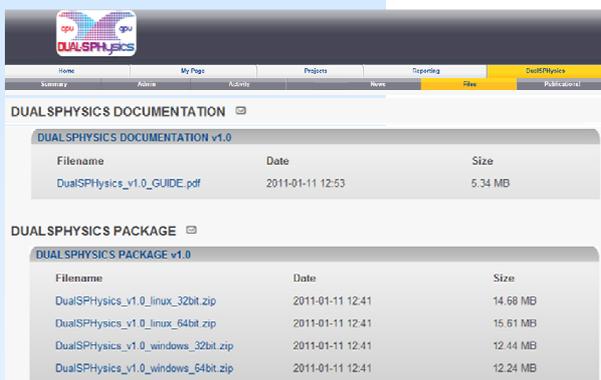


Figure 1 – Snapshot of the DualSPHysics website: www.dual.sphysics.org.

To demonstrate the reliability and efficiency of using GPU-based computing for SPH models, the DualSPHysics code has been used to reproduce the experimental data of the well-known SPHERIC benchmark validation test number 2: <http://wiki.manchester.ac.uk/spheric/index.php/Test2>.

DualSPHysics shows good agreement when comparing the experimental and numerical results. Figure 2 compares visually different instants of the simulation with real images from the experiment. The good agreement between experimental and numerical wave heights and pressures has been demonstrated in Crespo *et al.* (2011).

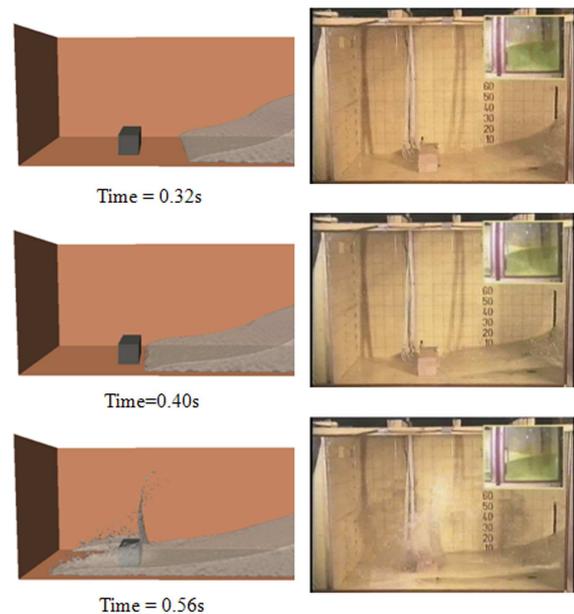


Figure 2 – 3D dam break problem proposed by Kleefsman *et al.* (2005).

A simulation of 70,000 steps using more than 1 million particles takes more than 5 days on a single CPU (Intel® Core™ i7 940) and less than 2 hours on the GPU (GTX 480), so the new code is 64 times faster for this case and 16 times faster if the four cores of the CPU are used. Therefore, the speedup obtained in Crespo *et al.* (2011) demonstrated the possibility to study real-life engineering problems at a reasonable computational cost.

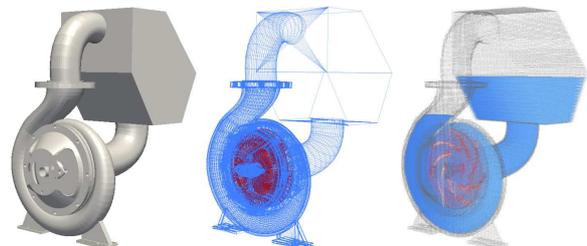


Figure 3 – CAD file is converted into SPH particles using the pre-processing code of DualSPHysics.

The binaries of DualSPHysics can be freely downloaded from www.dual.sphysics.org. The package includes not only the SPH solver that also pre-processing and post-processing codes. Any complex geometry can be loaded

from different format files such as .cad, .3ds, .stl, .ply, .dwg, .dxf, .shp, .igs, .vtk, .csv., etc., and then converted into SPH particles. In Figure 3, a CAD file is converted into particles representing the boundary starting from a triangulation of the object's surface, followed by a filling algorithm.

The post-processing tools allow us to compute magnitudes of interest such as vorticity at different planes, forces exerted on different objects, maximum wave heights or just plotting the different physical quantities of the particles. In addition, new codes let us improve the quality of the visualization using isosurfaces that represent the free-surface of the fluid such as it is shown in Figure 4.

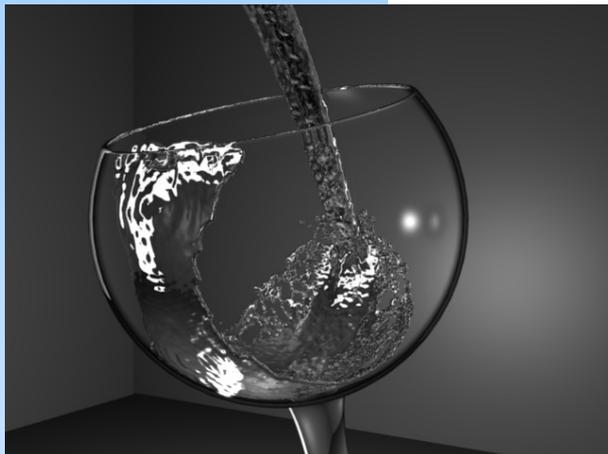


Figure 4 – New visualization using the isosurfaces of the fluid and the open-source software Blender.

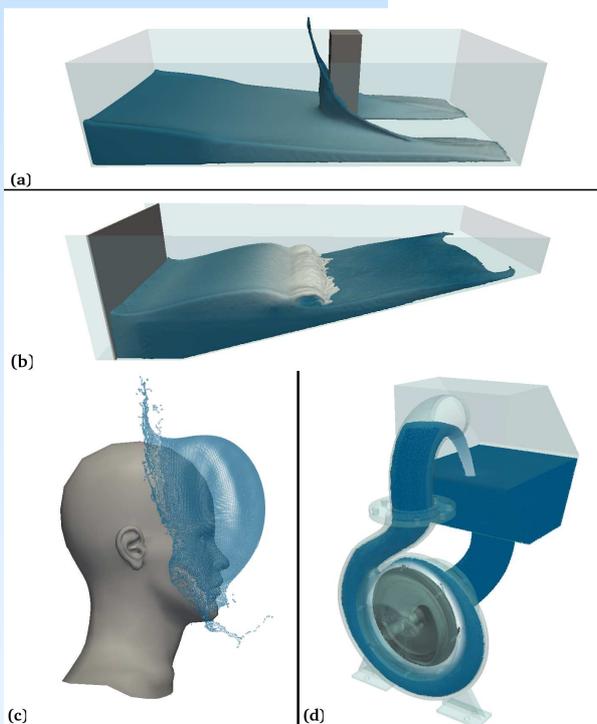


Figure 5 – Test cases: dam break (a), wave maker (b), real face (c) and pump (d).

The package also provides the tools and documentation necessary to create the user's own simulation cases. A

guide with descriptions of the codes is available and extra documentation explains how to convert different format files with the external geometry into the input files that DualSPHysics uses. Some demonstration cases are included with the DualSPHysics package (Figure 5); a dam break interacting with an obstacle in a tank (a), a wave maker in a beach (b), the real geometry of a human face interacting with impinging fluid (c) and a pump mechanism that translates water within a storage tank (d). Table 1 summarizes the performance obtained using DualSPHysics and the speedup when using the GPU version instead of the CPU one for the test cases.

Table 1 – Computational runtimes of the test cases and speedups achieved with the GPU computing.

Test Case	# of particles	CPU time (min)	GPU time (min)	Speedup
Dambreak	300k	1072	11	97
Wavemaker	200k	1235	20	64
Realface	1M	1180	18	66
Pump	1M	>1month	712	

Finally, for simulations requiring several million particles the memory requirements of a single GPU are still limited, so the immediate future for GPU computing is focused on implementing the DualSPHysics across multiple GPUs. The SPHysics team is working nowadays in this direction following the first steps described in Valdez-Balderas *et al.* (2011).

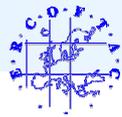
NEWS: The source files of the SPH solver (C++ and CUDA files) will be released in January 2012 to encourage SPH users to contribute to the project.

The 2012 SPHERIC Workshop Training Day will feature a hands-on session of using the code.

Contact: alexbebe@uvigo.es

References

- Crespo, A.J.C., Dominguez, J.M., Gómez-Gesteira, M., Barreiro, A., Rogers, B.D. (2011). *User Guide for the DualSPHysics Code v1.2*.
- Crespo, A.J.C., Dominguez, J.M., Barreiro, A., Gómez-Gesteira, M., Rogers, B.D. (2011). *GPUs, a new tool of acceleration in CFD: Efficiency and reliability on Smoothed Particle Hydrodynamics methods*. PLoS ONE 6(6):e20685. doi:10.1371/journal.pone.0020685.
- Gómez-Gesteira, M., Rogers, B.D., Dalrymple, R.A., Crespo, A.J.C., Narayanaswamy, M., (2010). *User Guide for the SPHysics Code v2.0*.
- Kleefsman, K.M.T., Fekken, G., Veldman, A.E.P, Iwanowski, B., Buchner, B., (2005). *A volume of fluid based simulation method for wave impact problems*. J. Comp. Phys. 206:363–393.
- Valdez-Balderas, D., Dominguez, J.M., Crespo, A.J.C., Rogers, B.D. (2011). *Developing massively parallel SPH simulations on multi-GPU clusters*. Proc. 6th SPHERIC International Workshop, pp. 340–347.



SPH for Internal Aerodynamic of Combustion Engine

Yohan Blacodon & Julien Bohbot, IFP Energies Nouvelles, 1&4 Avenue du Bois Préau, Rueil-Malmaison, France

The main problem encountered by engineers in the computational simulation domain applied to automotive engines are the set of efficient modelling to deal with moving complex geometries. This difficulty represents an important part in the time spent by engineer during a computational study. Thus, the SPH method appears as a good solution to solve the meshing step problem. The aim of a thesis work (Blacodon, 2011) was to adapt an existing SPH solver, SPARTACUS-3D (Issa, 2005), in order to model the internal aerodynamics of a combustion engine. Thus, a basic piston engine test case, based on experimental device from IMFT, has been chosen to validate the development and make comparison with PIV results. This basic test case highlights several numerical difficulties, such as complex and moving geometry and strong compressibility, which appear in engine simulations.

Due to the necessity of modelling compressible fluids, the Navier-Stokes compressible equations are used, considering a perfect gas. Thus, the Riemann solver mathematical framework (Van Leer, 1979) is used to adapt the SPH method to internal compressible flows. This method consists of solving a Riemann problem between two neighbour particles. The left and right states involved in the Riemann problem are thus defined by the states of interacting particles. The pressures of particles are substituted by the resultant pressure P^* obtained by solving the Riemann problem.

The Godunov Particle Method (GPH) is extended to the second order by applying a Monotone Upstream-centred Schemes for Conservation Laws (MUSCL) approach. It brings more stability to the SPH solver. A hybrid solid boundary condition is also implemented in the SPARTACUS code in order to deal with rigid moving walls in presence of a compressible fluid and strong density gradients. It consists in:

- The addition of a mirror term in the kernel approximation, which consists of the addition of the weight of neighbour particle throughout the boundary.
- The resolution of a relaxation problem near boundaries, in order to determine the proper conditions to impose to the boundary particles, considering the moving nature of the wall.
- The resolution of a partial Riemann problem between a fluid particle and a boundary particle, using the result of the previously resolved relaxation problem for the state of the boundary particle.

This "hybrid" boundary condition ensures the impermeability of solid boundaries, modelled by a simple line of boundary particles. It allows dealing with strong

pressure gradient in a combustion engine, with also the possibility to refine boundaries. Moreover, complex geometries (curved wall, angles, etc.) can be easily handled using SPH method.

In order to validate those numerous developments and implementations realized, the test case based on the experimental device is computed, i.e a compressed tumble. As illustrated in Figure 1, the PIV results and SPH simulations are in good agreement. It is more relevant when the discretization is increased (i.e initial inter-particle spacing is reduced). The extension to the third dimension also improves the calculation result. It highlights the importance of three-dimensional effects in a combustion chamber.

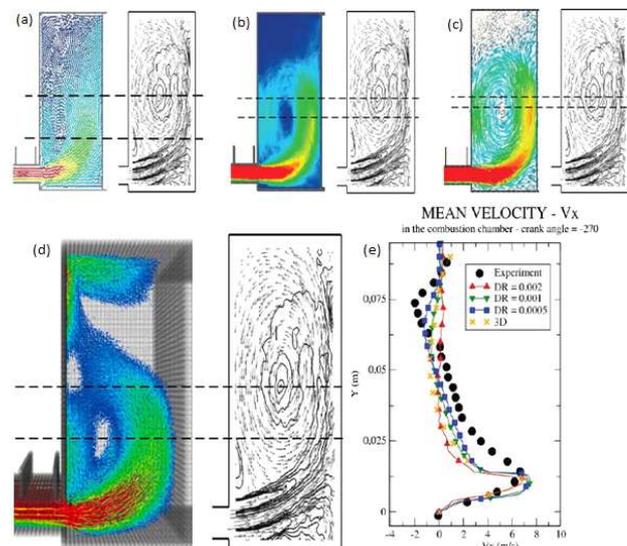
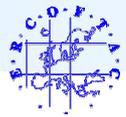


Figure 1 – (a, b & c): 2D visualization of the evolution of the tumble at crank angle 270° Before Top Dead Center (BTDC) with increase of discretization ($dr = 2\text{mm}$, 1mm , 0.5mm); (d): 3D visualization of the tumble at angle 270° BTDC and (e): Streamsize velocity profiles.

Contact: yohan.blacodon@ifpen.fr

References

- Blacodon, Y. (2011), *Development of meshless methods and application to numerical simulation for internal aerodynamic of combustion engine*, PhD thesis, IMFT.
- Issa, R. (2005), *Numerical assessment of the Smoothed Particle Hydrodynamics gridless method for incompressible flows and its extension to turbulent flows*, PhD thesis, the University of Manchester.
- Van Leer, B. (1979), *Towards the ultimate conservative difference scheme. V. A second-order to Godunov's method*. J. Comput. Phys. **32**:101–136.



Simulations of rotor-stator interactions with SPH-ALE

Neuhauser M. & Leboeuf F., Ecole Centrale de Lyon, Ecully, France
Marongiu J.-C. & Dodier A., ANDRITZ HYDRO, Vevey, Switzerland

In order to analyse transient flow in turbomachinery, SPH has the advantage of being able to capture highly dynamic phenomena. Because of its mesh-free nature, it allows simulating rotor-stator systems without a rotor-stator interface. SPH-ALE was introduced in 1999 by the work of Vila and was among others further developed in Marongiu *et al.* (2008). This method leads to 1D moving Riemann problems between pairs of neighbouring particles which can be solved by Godunov methods. Due to the ALE formalism, the particle velocity is a free parameter and can be chosen independently of the flow velocity.

In Figure 1 the pressure field of the 2D flow around a symmetric 4-digit NACA airfoil can be seen. It shows the comparison of the SPH-ALE code and an in-house Euler code which is an inviscid finite volume solver based on Gehrler (1999). In this simulation the inlet is situated at the left hand side of the figure while at the top and the bottom periodic boundary conditions were applied. Inlet and outlet are nonreflecting, inspired by Ghidaglia *et al.* (2005). In the SPH-ALE simulation the particles do not move. The initial distribution of the fluid particles around the NACA airfoil was obtained with the particle packing algorithm of Bouscasse *et al.* (2011). It can be observed that the SPH-ALE solution is in good agreement with the FV solution, especially around the blades.

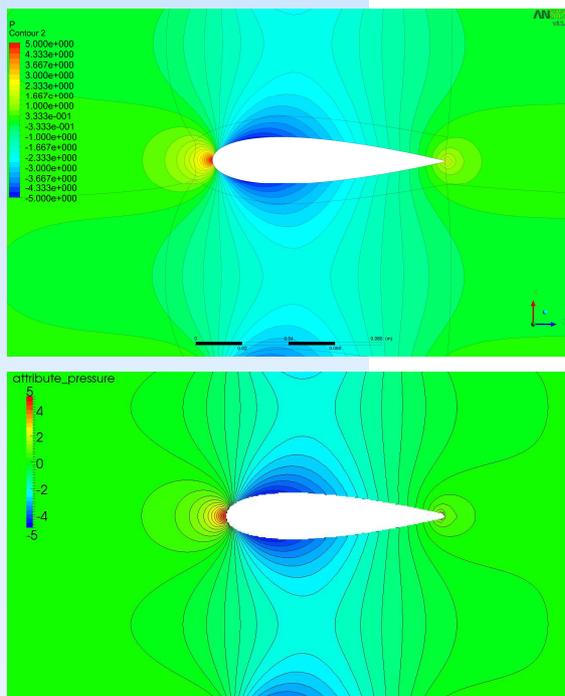


Figure 1 – Pressure field of the flow around a symmetric 4-digit NACA airfoil. Solution from an in-house Finite Volume code (top) and SPH-ALE (bottom).

Figure 2 shows the velocity field of the 2D case of a rotor and a stator computed with SPH-ALE. Here the left blade moves in minus z direction and the right blade is static. The boundary conditions were set as in the first simulation. Making use of the possibility to choose the particle velocity freely, we have blocks of fluid particles with different values of the particle velocity. The particles in a block around the rotating blade move with the same velocity as the rotor, while the other particles do not move. Figure 2 clearly shows the wake of the blades which is propagated downstream, as well as the influence of the stagnation point that propagates upstream.

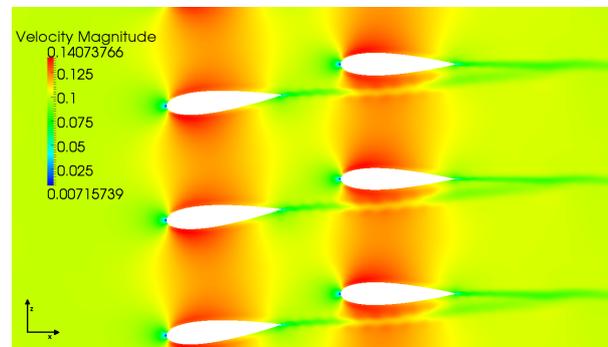


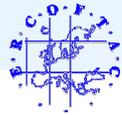
Figure 2 – Velocity field of the flow around a rotating (left) and a static (right) symmetric 4-digit NACA airfoil with SPH-ALE.

These simple 2D examples are the first steps of using SPH-ALE for rotor-stator interactions in turbomachinery.

Contact: Magdalena.Neuhauser@ec-lyon.fr

References

- Bouscasse, B., Antuono, M., Marrone, S., Colagrossi, A. (2011), *Particle initialization through a novel packing algorithm*, Proc. 6th Int. SPHERIC Workshop, Hamburg (Germany), pp. 109–116.
- Gehrler, A. (1999), *Entwicklung eines 3D-Navier-Stokes Codes zur numerischen Berechnung der Turbomaschinenstroemung*, Dissertation, Technical University Graz (in German).
- Ghidaglia, J.-M., Pascal, F. (2005), *The normal flux method at the boundary of multidimensional finite volume approximations in CFD*, European Journal of Mechanics B/Fluids **24**:117.
- Marongiu, J.-C., Leboeuf, F., 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.



Three new benchmark test cases in the SPHERIC database

Souto-Iglesias A. & Botia E., Department of Sciences Applied to Naval Engineering, Technical University of Madrid

The SPHERIC Steering Committee meeting on November the 25th agreed to accept three new well-suited test cases to validate SPH free surface computations. This note pretends to provide an insight to the SPH community into these new test cases, briefly described hereafter.

The first proposed test case focuses on both the lateral wall and roof wave impact problems from a sloshing flow subjected to a harmonic roll motion in a LNG tank's longitudinal section (scale 1/50). This case is particularly interesting due to the fact that the resulting dynamics of the problem involve wave breaking and air trapping phenomena as seen in Figure 1.

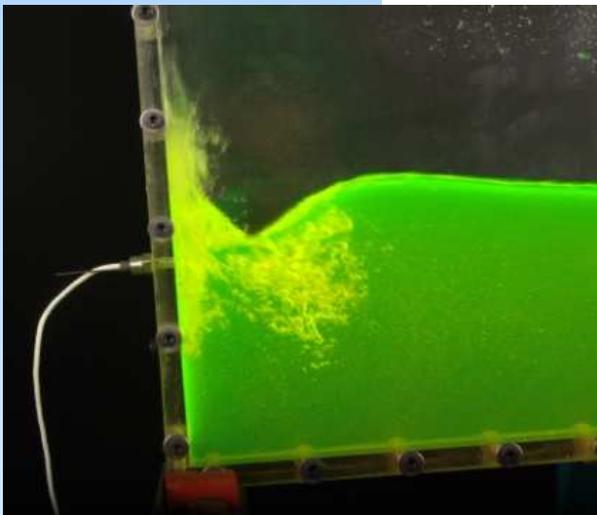


Figure 1 – New test case #1: lateral impact.

The second proposed case involves the coupling of the sloshing produced in a LNG tank section (scale 1/50) and a structural system with a single degree of freedom, generally denoted as a tuned liquid damper (TLD). The tank is free to roll and its motion is driven by an externally mounted periodically moving mass. This creates an angular moment, which coupled with the fluid's sloshing motion, dampens the roll motion of the tank. This test case aims to show to what extent the breaking waves and sloshing dynamics affect the damping characteristics of a TLD. Figure 2 shows the typical dynamics of the low filling resonance case with water acting as a damper of the free roll motion.

The last case presented here focuses on the study of the interaction between a free surface sloshing flow and an elastic body, commonly denoted as fluid structure interaction (FSI) problem. The sloshing flow in a rectangular tank produces a free surface motion which interacts with a deformable elastic body clamped in either the bottom or top center of the tank (Figure 3). In order to make it more useful for SPH validations, the

experiments have also been run with oil, thus covering a wider range of Reynolds numbers.



Figure 2 – New test case #2: Coupling in low filling level with water.

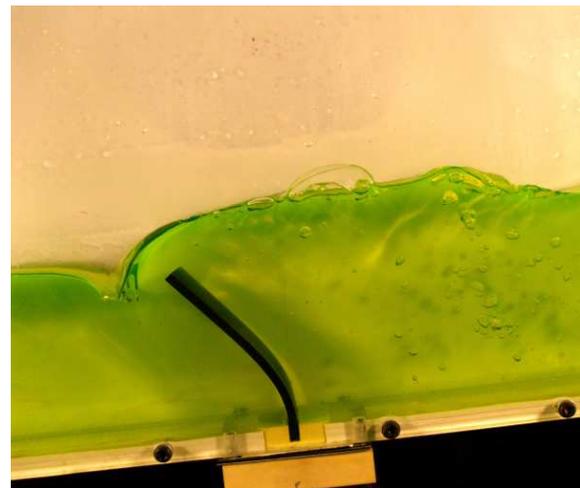


Figure 3 – New test case #3: FSI problem, low filling level and water.

The detailed information necessary for the implementation and posterior validation assessments are available at the following link:

http://canal.etsin.upm.es/ftp/SPHERIC_BENCHMARKS/

The related paper (Botia-Vera *et al.*, 2010) can be found at the following link:

http://canal.etsin.upm.es/files/botia_et_al_spheric10.pdf

Contact: antonio.souto@upm.es

References

Botia-Vera, E., Souto-Iglesias, A., Bulian, G., Lobovsky, L. (2010), *Three SPH Novel Benchmark Test Cases for free surface flows*, Proc. 5th Int. SPHERIC Workshop, Manchester (UK), pp. 146–153.