AUGUST 2019

SPHERIC

NEWSLETTER

28th issue

SPH rEsearch and engineeRing International Community

SUMMARY

14th SPHERIC workshop

Pablo Loren-Aguilar
University of Exeter, United Kingdom

The 14th SPHERIC International Workshop took place in the University of Exeter (United Kingdom) from the 25th to the 27th of June 2019. A total of 113 delegates enjoyed three fascinating days at our campus, sharing once more the latest developments and applications of the SPH method.

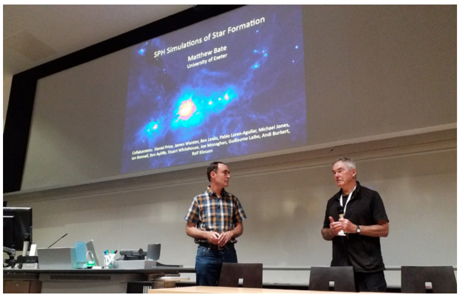
During the traditional Training Day we had two wonderful morning lectures. A general introduction to SPH provided by Dr. Guillaume Oger (Ecole Centrale Nantes), and a talk by Dr. Angelo Tafuni (New Jersey Institute of Technology) focused on the role of boundary conditions in SPH. In the afternoon session, a delegation of NEXTFLOW software (led by Dr. Matthiew de Leffe) offered a fantastic hands-on session showing the capacities of their SPH-Flow and SPH-Studio software.

The main workshop included this year a total of 53 talks and three keynote lectures. The first keynote lecture was delivered by Professor Matthew Bate from the University of Exeter, reviewing the role SPH has played in the field of Star Formation. The second keynote lecture was delivered by Dr. Salvatore Marrone reviewing the 2018 Monaghan Prize "Delta-SPH model for simulating violent impact flows" followed by a discussion panel with Salvatore Marrone (CNR-INSEAN), David Le Touzé (Ecole Centrale de Nantes), Damien Violeau (EDF R&D) and Xiangyu Hu (Technische Universitat Munchen). The final keynote lecture of this year's edition was delivered by Dr. Richard Southern from the National Centre for Computer Animation (Bournemouth University, UK). He reviewed in a vibrant talk the role particle-in-cell methods play in the visual effects industry.

As usual, all the presented papers had a huge quality and covered many topics, making the workshop incredibly rich. Interactions with solids, convergence, consistency and stability, multiphase flows, applications to industry, boundary conditions, astrophysics or incompressible flows were some of the topics covered in the different sessions, not leaving a minute to be bored. The Libersky Prize for the best student talk was delivered for the second year on a row to Thomas Fonty (EDF R&D) for the paper "A first air entrainment SPH model using a two-phase mixture formulation", co-authored by Agnes Leroy, Damien Voileau and Martin Ferrand. The second prize was delivered to Jacob Kegerreis (Durham University) for his work on "Efficient Spherical Initial Conditions and High Resolution Simulations of Giant Impacts", and the third prize was delivered to Joshua Borrow (Durham University) for his work "Choosing the best flavour of SPH for astrophysics problems : more complex may not be better". Given the amazing quality of all the presentations, the Scientific Committee had a really tough time selecting the winners.

2019 Libersky Prize winner Thomas Fonty with Professor Ben Rogers (left) and Dr. Pablo Loren-Aguilar (right)

During the workshop, the delegates had the chance to enjoy a good selection of social activities. We enjoyed a BBQ on Tuesday evening after a long and intense day of talks. On Wednesday afternoon the delegates had the chance to explore the city of Exeter with our local Red Coat tours guides, and to enjoy a wine tasting session in preparation for the traditional gala dinner that took place at Reed Hall. During the dinner, the two following SPHERIC workshops in Harbin (China) and New York (USA) were presented by Dr. Pegnan Sun (Ecole Centrale de Nantes and Harbin Engineering University) and Dr. Angelo Tafuni (New Jersey Institute of Technology) respectively. There was also time to pay a well deserved tribute to Dr. Andrea Colagrossi for his invaluable contribution as a SPHERIC steering committee member.



Prof. Professor Matthew Bate just about to kickstart the workshop with the 1st keynote

Dr. Savatore Marrone giving Tuesday´s second keynote lecture

Dr. Richard Southern giving the 3rd and final keynote lecture

Dr. SPHERIC Steering Committee chair Professor Ben Rogers acknowledging Dr. Andrea Colagrossi for this invaluable contribution as a SPHERIC Steering Committee member

A first air entrainment SPH model using a two-phase mixture formulation

Thomas Fonty*[1], Agnes Leroy[1], Damien Violeau[1], Martin Ferrand[1]
*thomas.fonty@edf.fr
[1] Eletricité de France R&D / LNHE & LHSV 6 quai Watier 78400 Chatou, France

The entrapment of air bubbles within flowing water, namely air entrainment, is a common feature of flows over hydraulic works (plunging jet, hydraulic jump, etc.). Its accurate modelling, pivotal to better design of such structures, still remains challenging both experimentally and numerically due to its multicomponent and multiscale nature. The air presence indeed modifies the flow dynamics, the turbulence, and can lead to a wide variety of two-phase regimes. The SPH method appears as a natural way to model such highly deformed flows. To avoid computationally prohibitive costs related to the full discretization of bubbles or drops and handle practical problems, multiphase averaged models were developed in the past decades as in Ishii and Hibiki (2011). Within this framework, a SPH mixture model for high density ratio flows relying on a volume-based formulation with relative velocity between phases has been developed and validated in Fonty et al. (2018, 2019). Instead of having a once and for all assigned phase as in multi-fluid SPH, each particle now carries both phases through their respective volume fractions as illustrated on Figure 1. The mixture model considers the flow as a single-fluid with one continuity and one momentum equation for mixture quantities, complemented by a mass conservation equation for one phase. The relative velocity between phases, closed by a physics-dependent expression, introduces additional terms. Multifluid SPH operators as described in Ghaïtanellis et al. (2018) are used within the Unified Semi-Analytical Wall boundary conditions framework of Ferrand et al. (2013). In this work, we aim at testing the performances of this SPH mixture model for air-water flows. Several challenges need therefore to be addressed.

Turbulence is often at the core of the entrainment phenomenon. It deforms the air-water interface and the subsequent irregularities can trap air bubbles. Its necessary modelling is done through a standard k-ε model. The downwards vertical velocity generated by the turbulence, if strong enough to exceed the terminal velocity caused by buoyancy of air bubbles, leads to diffusion of air in the bulk of the fluid.

This leads us to the second challenge, namely the relative velocity expression that encompasses this two-component behaviour of the flow modulated by the turbulence. Although the need to find an algebraic closure for the relative velocity enables the versatility of the mixture model (e.g. consider air-water or water-sediment flows), this closure remains flow-dependent. We try to keep a sufficiently general closure by writing it as a sum of a Stokesian drag term and a turbulent diffusion term linked to the turbulent viscosity arising from the k-ε computations. This model is applied to a laminar two-phase mixture Poiseuille flow (including diffusion of the phase through a constant diffusion coefficient, thus turbulence is killed in this case) for which an analytical solution was derived at steady state to check its correct implementation.

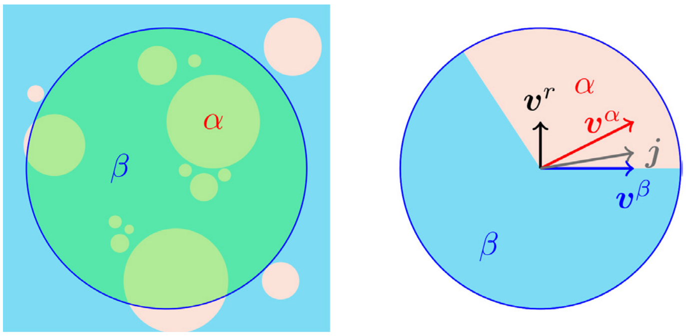


Figure 1 - Control volume (green area) in two-phase flow and the corresponding velocity fields

Eventually, as air entrainment cases usually involve inlets and outlets, the development of multifluid open boundaries is required to allow distinct phases or mixtures to enter and leave the domain. We extend the framework of Ferrand et al. (2017) to this mixture model by working with inverse volumes instead of densities in the continuity equation and writing a new 1D Riemann problem at boundaries to include the volume fraction equation.

All those items are then combined to model a schematic air entrainment test case: the flow going down a stepped spillway for Q=0.182 m^3/s is shown in Figure 2. The turbulent boundary layer developing from the walls reaches the free surface at the sixth step as shown in Figure 3. It is then activating the turbulent diffusion term of the relative velocity expression, counteracting the separating effect of the buoyant contribution. The volume fraction profiles compare well with the experiment by Chanson (2001) as highlighted in Figure 4. Some discrepancies appear on the longitudinal profile and require further investigations, especially the effect of neglecting mixture terms in the momentum equation. Ongoing work is performed to include buoyancy effects in the turbulence model and first 3D applications of this model to industrial test cases are on the way.

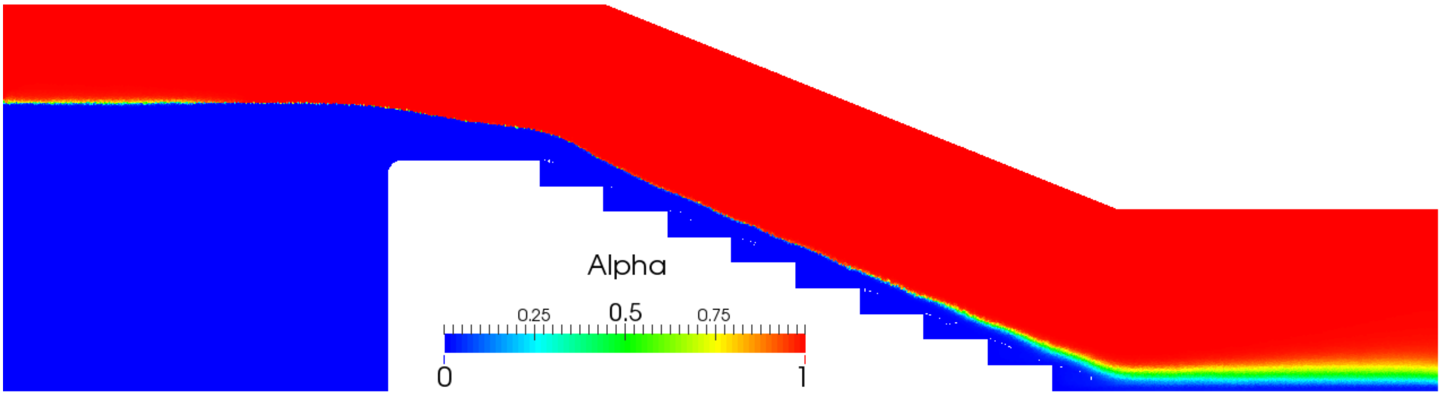


Figure 2 - Stepped spillway: flow over the steps colored by volume fraction.

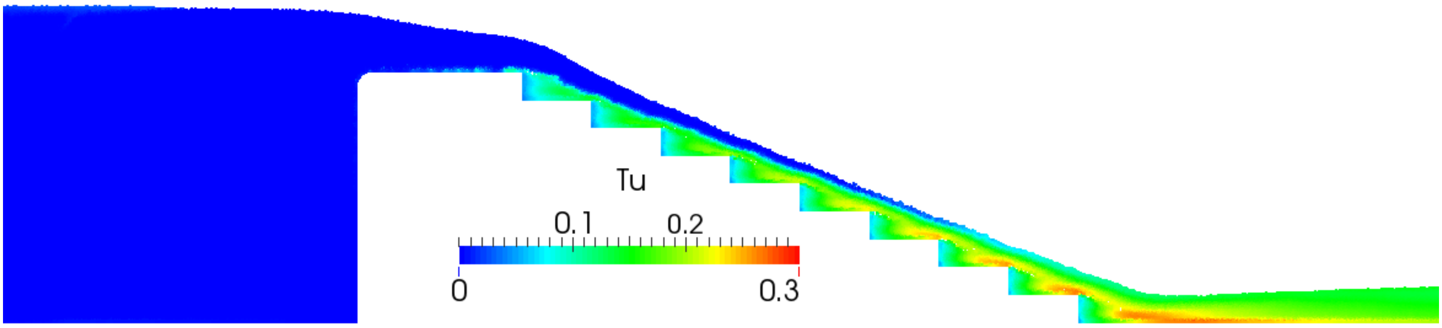


Figure 3 - Stepped spillway: flow over the steps for particles with less than 90% of air colored by the turbulent intensity.

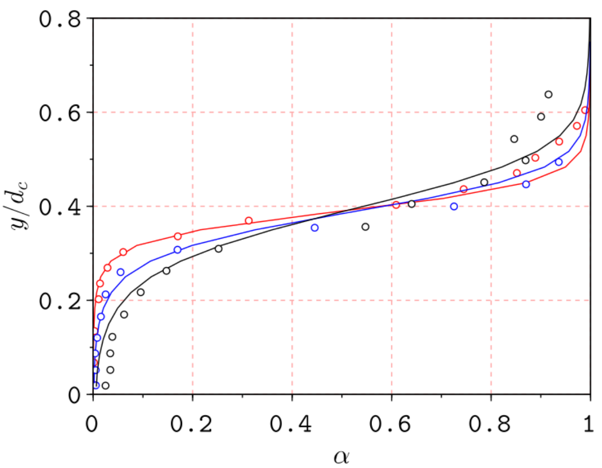


Figure 4 - Stepped spillway: distribution of the volume fraction above the last three steps (steps 6, 7 and 8 in red, blue and black respectively). Symbols: experimental data by Chanson et al. (2001), lines: present SPH simulation

[1] Chanson, H. (2001) Caractéristiques diphasiques des écoulements sur les coursiers en marches d'escalier. La Houille Blanche 8.
[2] Ferrand, M., Laurence, D. R., Rogers, B. D., Violeau, D. & Kassiotis, C. (2013) Unified semi-analytical wall boundary conditions for inviscid, laminar or turbulent flows in the meshless SPH method. International Journal for Numerical Methods in Fluids. 71: 446.
[3] Ferrand, M., Joly, A., Kassiotis, C., Violeau, D., Leroy, A., Morel, F.-X., & Rogers, B. D. (2017), Unsteady open boundaries for SPH using semi analytical conditions and Riemann solver in 2D. Comp. Phys. Com. 210: 29.
[4] Fonty, T., Ferrand, M., Leroy, A., Joly, A. & Violeau, D. (2018) An upwind scheme for conservative, realizable two-phase mixture SPH model with high density ratios. Proceedings of 13th international SPHERIC workshop, Galway, Ireland.
[5] Fonty, T., Ferrand, M., Leroy, A., Joly, A., & Violeau, D. (2019) Mixture model for two-phase flows with high density ratios: a conservative and realizable SPH formulation. International Journal of Multiphase Flow. 111: 158.
[6] Ghaïtanellis, A., Violeau, D., Ferrand, M., El Kadi Abderrezzak, K., Leroy, A. and Joly, A. (2018) A SPH elastic-viscoplastic model for granular flows and bed-load transport. Advances in Water Resources. 111: 156.
[7] Ishii, M., & Hibiki, T. (2011), Thermo-fluid dynamics of two-phase flow - Second Edition. Springer

Choosing the best flavour of SPH for astrophysics problems: more complex may not be better

Josh Borrow*[1], Bert Vandenbroucke[2], Matthieu Shaller[3]
*joshua.borrow@durham.ac.uk
[1] Institute for Computational Cosmology Durham University Durham, DH1 3LE, UK
[2] SUPA School of Physics and Astronomy, Unversity of St. Andrews, St. Andrews, KY169SS, UK
[3] Leiden Observatory, Leiden University, NL-2300 RA Leiden, NL

Cosmological simulations have been used in astronomy for decades to provide predictions for observables based on underlying theoretical models. With these simulations, astronomers can predict the abundances, sizes, and formation times for many classes of astronomical objects, from small populations of stars all the way up to large groups and clusters of galaxies. Hydrodynamics introduced to these gravity solvers by the TreeSPH code (Hernquist & Katz 1989), however at this time computational resources were not developed enough to run a full hydrodynamical model of a representative portion of the Universe. A revolution in the field took place in the early 20's, with Horizon-AGN, Illustris, and EAGLE [2,7,9] running cosmological hydrodynamics simulations that were able to accurately capture the huge dynamic range of 10^{11} in density required to resolve both the large-scale structure of the Universe and the formation of individual galaxies simultaneously [1]. In these simulations that use around 1000^3 – 2000^3 resolution elements for the whole computational domain, bound structures (galaxies) are resolved by around 10^{4-5} particles.

In cosmology, as in other fields, there has been a push in recent years to adopt more complex hydrodynamical models. Early simulations used basic Density-Entropy SPH [8]. The focus in recent years has shifted to SPH-ALE methods with a finite mass solver [4]. It is still unclear, however, that these methods present any tangible improvement in hydrodynamical accuracy, considering both their increased computational cost (which could be spent integrating more resolution elements) and the low particle count with which the actual structures that are studied in the simulation are typically resolved. For comparison, we use a scheme similar to the one presented in [5], with an extra limiter [7] for the conduction term.

To consider if these SPH-ALE methods are suitable for use in a cosmological context, we need two things: relevant test problems, and a code that is able to run both the ALE scheme and basic SPH schemes fairly. For the former, we choose three main tests: the Sedov-Taylor blastwave, the Evrard collapse, and the Gresho-Chan vortex. For the latter, we use the Swift cosmological simulation code [6], which uses the same kernel (here the Wendland-C2 with $\eta = 1.2$) and neighbour-finding structures, allowing the differences in run-times between simulations to be dominated by the different interaction costs for the various schemes.

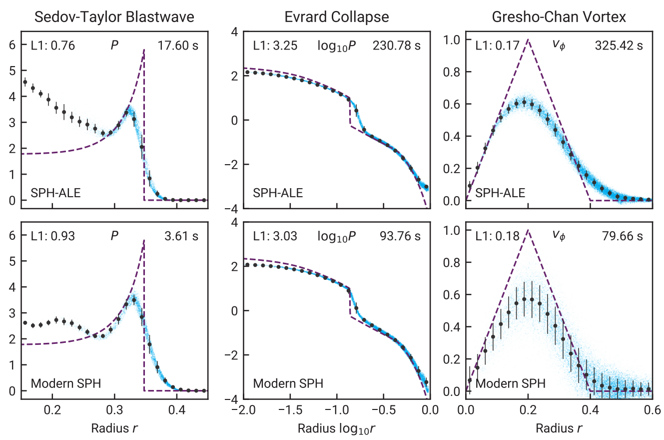


Figure 4 - The performance of the schemes on the various tests. The purple dashed line shows the ideal solution, blue points show all particles, and the filled black circles show binned quantities with radius. In the top left we give the total wall-clock time on this problem, and top right shows the L1 norm. Note how similarly both the schemes perform at this resolution, despite the significant increases in wall-clock time.

[1] Borrow, Josh, Bower, Richard G., Draper, Peter W., Gonnet, Pedro, & Schaller, Matthieu. 2018. Proceedings of the 13th SPHERIC International Workshop, Galway, Ireland, June 26-28 2018, Jul, 44–51.
[2] Dubois, Y., et al. . 2014. MNRAS, 444(2), 1453–1468.
[3] Hernquist, L., & Katz, N. 1989. ApJS, 70(June), 419–446.
[4] Hopkins, P. F. 2015. MNRAS, 450(June), 53–110.
[5] Price, D. J., et al. . 2017. Phantom: A smoothed particle hydrodynamics and magnetohydrodynamics code for astrophysics.
[6] Schaller, M., Gonnet, P., Chalk, A. B. G., & Draper, P. W. 2016. In: Proceedings of the PASC Conference.
[7] Schaye, Joop, et al. . 2015. MNRAS, 446(1), 521–554.
[8] Springel, V. 2005. The cosmological simulation code GADGET-2. MNRAS, 364(Dec.), 1105–1134.
[9] Vogelsberger, Mark, et al. . 2014. MNRAS, 444(2), 1518–1547.

Building and Breaking Planets: Efficient Spherical Initial Conditions and High-Resolution Simulations of Giant Impacts

Jacob Kegerreis*[1], Vincent Eke[1], Richard[1] Massey[1], Pedro Gonnet[2], Donald Korycansky[3], Matthieu Schaller[4], Luis Teodoro[5]

* jacob.kegerreis@durham.ac.uk
[1] Institute for Computational Cosmology Durham University Durham, UK
[2] Google AI Perception, Google Switzerland, Zurich, Switzerland
[3] CODEP, Earth Sciences, University of California, Santa Cruz, USA
[4] Leiden Observatory. Leiden University, Leiden, Netherlands
[5] BAERI/NASA Ames Research Center Moffett Field, USA

Giant impacts dominate many planets' late accretion and evolution. We see the consequences of these violent events throughout our solar system, from the formation of Earth's Moon to the odd obliquity of Uranus spinning on its side. These complicated processes are most commonly studied using smoothed particle hydrodynamics (SPH) simulations [1].

The highly non-linear nature of giant impacts requires many particles to resolve features of interest. Limited work has been done to study convergence of simulation results, partly due to the computational challenge of performing such tests.

To this end, we developed two open-source tools: the simple SEA scheme to create optimal spherical arrangements of particles [4] (github.com/jkeger/seagen); and the SPH-plus-gravity code SWIFT, presented previously at SPHERIC for its parallelisation and scalability [2,3] (swift.dur.ac.uk).

The stretched equal-area (SEA) algorithm creates near-equilibrium, spherically symmetric initial conditions of particles. It ensures that every particle has an SPH density within 1% of the desired value. This mitigates the need for expensive computation that is otherwise needed to relax initial conditions.

We then modelled giant impacts onto Uranus using 10^8 SPH particles to test various quantities with increasing particle number. We find that even large-scale results such as the planet's rotation rate are not converged with standard simulations of 10^5 and 10^6 particles, as shown in Fig. 1. The overall behaviour is similar, but variations in the initial debris that falls back have a significant effect on the planet and its rotation, which appears well-converged with 10^7 and 10^8 particles. Similar convergence is seen for the ejected mass, while the low mass of rock placed into orbit has not converged by 10^7 particles.

Increasing resolution is only one important challenge for developing realistic simulations. We have used a simple implementation of SPH with a focus on simply increasing the number of particles. Future studies must test high resolutions with more sophisticated equations of state and SPH formulations.

We conclude that simulations with $<10^7$ SPH particles can be unreliable even for large-scale results of planetary impacts. 10^7 and 10^8 particles appear to pass the threshold of resolving the major processes. However, different collisions and other simulation outputs will depend more or less strongly on the behaviour of small structures, with correspondingly different requirements for convergence.

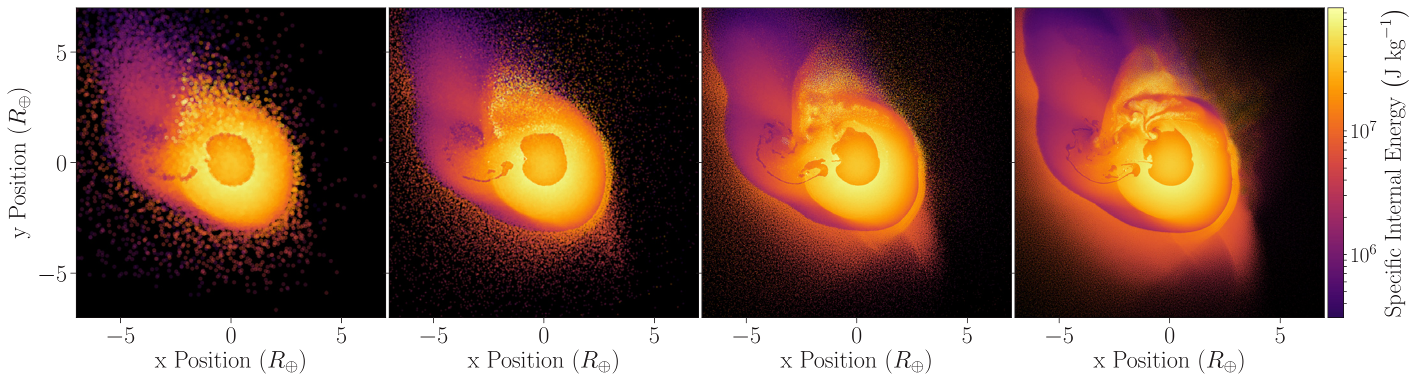


Figure 1 - The early time evolution of the planet's angular momentum within the Roche radius for the different resolution simulations.

[1] Benz, W., Slattery, W. L., and Cameron, A. G. W. Icarus, 66:515-535, 1986.
[2] Borrow, J. et al., Proc. 13th SPHERIC Intl. Wksh., 2018.
[3] Gonnet, P. et al., Proc. 8th SPHERIC Intl. Wksh., 2013.
[4] Kegerreis, J. A., et al. Mon. Not. R. Astron. Soc., 487:5029-5040, 2019.

Simulations of aircraft ditching through the H2020 project "SARAH"

G. Oger*[1], A. Vergnaud[1], I. Hammani[1], B. Bouscasse[1], D. Le Touzé[1], M. de Leffe[2], A. Bannier[2], S. Marrone[3], A. Colagrossi[3], Y. Jus[4], M. Garnier[4]

* guillaume.oger@ec-nantes.fr
[1] Ecole Centrale Nantes, Nantes, France
[2] Nextflow Software, Nantes, France
[3] CNR-INM, Roma, Italy
[4] BV Solutions, Nantes, France

SARAH (Increased Safety & Robust Certification for ditching of Aircraft & Helicopters) is a Horizon 2020 collaborative project aiming at establishing novel holistic, simulation-based approaches to the analysis of aircraft and helicopter ditching. Results of SARAH are expected to support the trustworthiness of aviation services, by proposing fast and accurate simulation tools for the emergency ditching. These tools are crucial for the design phase of aircrafts (fixed-wing aircrafts as well as helicopters), guaranteeing increased safety while preserving flight performances. Specifically, two types of numerical tools are required: "low-fidelity" models delivering results with moderate accuracy but with very short computational times, which represents an important property for the designers; "high-fidelity" tools proposing a much more accurate prediction of the different physical phenomena involved, but with more significant computational times.

It was decided to calibrate the low-fidelity models through the results obtained using the high-fidelity tools. Another important part of the project concerns experiments involving free surface impacts of fixed-wing aircrafts (carried out at the CNR-INM in Rome) and helicopters (carried out in the large wave basin of Ecole Centrale Nantes). To ensure the validity of the numerical models, the simulation results are systematically compared with experimental ones (impact forces, local pressures, trajectories etc...) .Among the selected high-fidelity tools, SPH-flow software - property of NextFlow Software and Ecole Centrale Nantes, co-developed with the support of CNR-INM - is used to simulate these violent free surface impacts [1,3].

For the airplane ditching studies, the SPH simulations are dedicated to reproduce some experimental impacts performed in the towing tank of the CNR-INM in Rome [2]. The time evolution of forces and local pressures are studied, and special attention is paid to the detection of cavitation pockets which seem to occur during high speed impacts, strongly affecting the resulting flow. For the helicopter ditching study, the simulations deal with various impact configurations and for different wave conditions. The numerical solutions are systematically compared with the experimental results (impact forces, local pressures, trajectories etc...) obtained at the wave basin of Ecole Centrale Nantes.

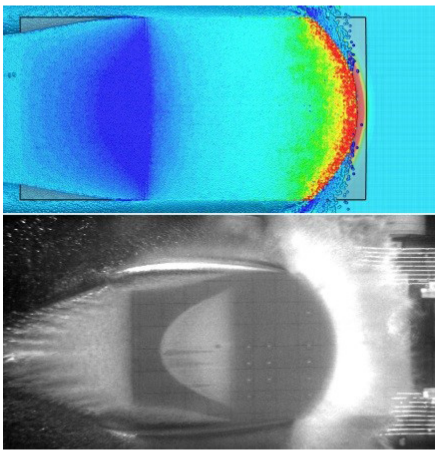


Figure 1 - Numerical (top) and experimental (bottom) view of a double curvature profile (rear of airplane) impacting the free surface and forming a cavitation pocket.

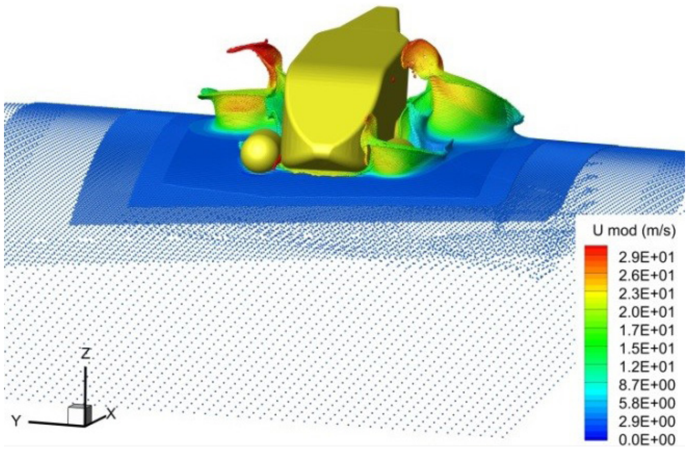


Figure 2 - SPH simulation of helicopter impact at a wave crest, reproducing an experimental case performed at the ECN's wave basin

[1] Chiron, L. and Oger, G. and De Leffe, M. and Le Touzé, D. (2018), J. Comp. Phys. 354:552.
[2[Iafrati, A. and Grizzi, S. and Siemann, M.H. and Benítez Montañés, L. (2015), J. Fluids Struct. 55: 501.
[3] Oger, G. and Le Touzé, D. and Ducrozet, G. and Candelier, J. and Guilcher, P.-M. (2014), 33rd OMAE.

72nd Annual Meeting of the APS Division of Fluid Dynamics Seattle WA (November 23-26, 2019)

Dr. Angelo Tafuni
atafuni@njit.edu
School of Applied Engineering and Technology, Department of Mechanical and Industrial Engineering, New Jersey Institute of Technology, Newark, NJ 07103

Important Dates

Abstract Submission Deadline: August 1, 2019

Travel Grant Application Deadline: August 1, 2019

Child Care Grant Application Deadline: August 1, 2019

Early Registration Rate: on or before September 19, 2019

Regular Registration Rate: September 20 – October 24, 2019

On-Site Registration Rate: October 25 – November 26, 2019

Cancellation Deadline (no registration refunds past this date): November 8, 2019

Meeting Dates: Saturday, November 23, 2019 - Tuesday, November 26, 2019

The 72nd Annual Meeting of the American Physical Society's Division of Fluid Dynamics (DFD) will be held in Seattle, WA on November 23-26, 2019. The DFD annual meeting is one of the largest conferences in fluid dynamics, with 3000 or more attendees expected from around the world. The objective is to promote the advancement and dissemination of knowledge in all areas of fluid dynamics. Undergraduate and graduate students, postdoctoral researchers, university faculty, and researchers across government and industry are encouraged to share the latest developments in the field.

This year's meeting is being hosted by the University of Washington, the University of Washington – Applied Physics Lab, Northwest Research Associates, Oregon State University, Portland State University, Saint Martin's University, the University of British of Columbia, and the University of Idaho. The scientific program will include four award lectures, along with twelve invited lectures, a series of minisymposia, and numerous focus sessions. Among these, a focus session will be dedicated to "SPH and Meshfree Methods", category 8.10 in list of abstract sorting categories. This year's DFD meeting will also pioneer Flash Presentations as a new presentation modality. Submissions for Oral Presentations will fall along two choices: the traditional 12-minute presentation, or a 1-minute oral "Flash Presentation" with an accompanying poster presentation.

You can submit an abstract by August 1, 2019 at the following address: http://abstracts.aps.org, please be sure to select "72nd Annual Meeting of the APS Division of Fluid Dynamics, Seattle, Washington".

Find more info at: https://apsdfd2019.org

2020 SPHERIC Harbin International Workshop (SPHERIC Harbin 2020)

Prof. A-Man Zhang
College of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China

Important Dates

Abstract Submission Deadline: 10 September 2019

Announcement of Selected Abstracts: 30 September 2019

Early Registration Deadline: 31 October 2019

Full Paper Submission Deadline: 31 October 2019

Regular Registration Deadline: 30 November 2019

Training Day: 13 January 2020

Workshop: 14-16 January 2020

The 2020 SPHERIC Harbin International Workshop (SPHERIC Harbin 2020) will be held in Harbin by Harbin Engineering University, China, during January 13-16, 2020. This is the second time that the SPHERIC Workshop is held outside Europe. SPHERIC Harbin aims to enable experienced researchers and Ph.D. students using SPH or other particle-based methods to share and contribute to the development and applications of these methods.

Abstract and paper submission

Abstracts should be one page long and must include one illustrative figure outlining the main results. After notification of abstract acceptance, authors are required to prepare the full 8-page paper. Abstract and full paper templates are available on the workshop website.

Conference venue

SPHERIC Harbin 2020 will take place in the Golden Valley Building, Harbin, Heilongjiang, China. Golden Valley Building is located in the most prosperous historical street in Harbin - Central Street Pedestrian Street, adjacent to the beautiful Songhua River and Zhaolin Park, overlooking the beautiful Sun Island and the “Ice and Snow World”, with convenient transportation and beautiful environment. Harbin is heralded as the Ice City for its well-known winter tourism and recreations. The annual Harbin International Ice and Snow Sculpture Festival is one of the four largest ice and snow festivals in the world. Enormous snow sculptures at Sun Island and the “Ice and Snow World” with full-size ice castles and massive sculptures with splendid illumination.

Official Website

More detailed information on the conference venue, abstract and paper submission, registration, travel and accommodation will be available on the workshop website:
http://spheric.hrbeu.edu.cn/

15th SPHERIC International Workshop, Newark NJ (June 16-18, 2020)

Dr. Angelo Tafuni

atafuni@njit.edu
School of Applied Engineering and Technology, Department of Mechanical and Industrial Engineering, New Jersey Institute of Technology, Newark, NJ 07103

Important Dates

Training Day (with the DualSPHysics team): Monday, June 15, 2020

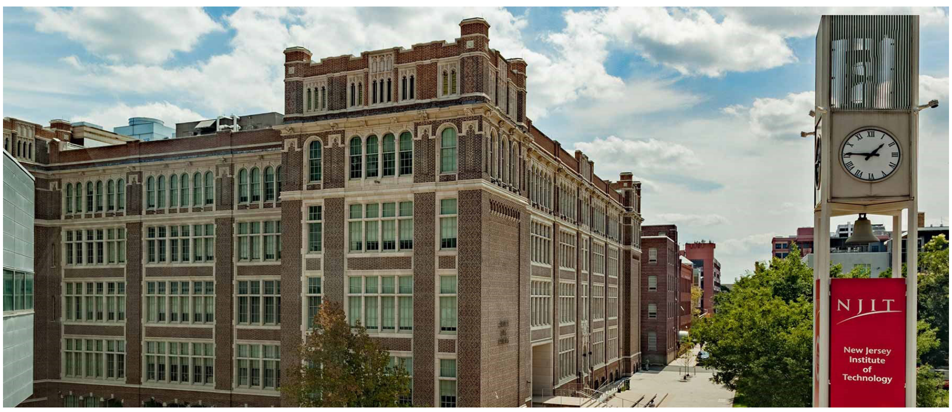
SPHERIC Workshop: Tuesday, June 16, 2020 – Thursday, June 18, 2020

The SPHERIC International Workshop is the annual global forum for development and applications of Smoothed Particle Hydrodynamics (SPH) and related methods. During the workshop, the latest advances in the SPH method are presented. In addition, the optional training day offered the day before the start of the workshop offers an intensive introduction to the theory and application of SPH.

For the first time in its history, the SPHERIC International Workshop will take place in the United States of America in June 2020. The 15th edition of the workshop will be hosted by the Newark College of Engineering at New Jersey Institute of Technology in Newark (NJ), led by assistant professor Dr. Angelo Tafuni. Newark is located in the heart of New Jersey's Gateway Region, approximately 8 miles (13 km) west of New York City. The city of Newark is a major hub of air, road, rail, and ship traffic, making it a significant gateway into the New York metropolitan area and the mid-Atlantic United States. The city is also home to Newark Liberty International Airport, the second-busiest airport in the New York metro area and the 15th-busiest in the United States in terms of passenger traffic.

More info will soon become available at www.spheric2020.com

We look forward to welcoming you in Newark and sharing a successful and enjoyable meeting with you!



UK Fluids Network SPH Special Interest Group (Cambridge Granta Park on 10th July 2019)

Dr. Stephen Longshaw

stephen.longshaw@stfc.ac.uk
UK Research and Innovation – Science and Technologies Facilities Council

SPH-SIG members

Brunel University London;

UKRI Science and Technology Facilities Council (STFC);

Centre for Modelling and Simulation (CFMS);

The University of Bristol;

The University of Cambridge;

Altair;

National Institute of Geophysics and Volcanology (INGV, Italy);

Heriot-Watt University;

Arup Group;

Nextflow Software;

The University of Sheffield;

The University of Manchester;

Simpact Engineering Ltd;

TWI Ltd;

ESI Group

On the 10th of July the UK Fluids Network SPH SIG met at the Grant Centre in Cambridge in the United Kingdom and was hosted by Brunel University.

This meeting had the express purpose of bringing industry and those working on SPH as a research topic together to find common ground and goals. Attendees from the SIG were present as well as a number from industrial research backgrounds interested in the method. To this end there were also a number of representatives of the various SPH software packages available to give overviews of current capabilities and future directions.

Presentations were made both by technical area leaders from UK institutions as well as those from commercial software houses. Talks ranged from Multi-physics SPH for complex problems – From fundamental physics to the development and application of new paradigms by Prof. Benedict Rogers of the University of Manchester through to Practical uses of SPH in automotive, aerospace and marine industrial applications by Amaury Bannier of Nextflow Software. Full details and a selection of presentations can be downloaded from the SPH SIG website: https://fluids.ac.uk/sig/SPH

This meeting differed from past SPH SIG events in that it aimed to provide a contextual overview of SPH technical developments by SIG members with a view to defining new and interesting industrially-driven problems.
The event was well attended and provided an in-depth overview of the state of SPH as both a practical method and research topic. The inclusion of current software implementations of the method ensured that its potential use was more than just theoretical, with direct pathways to adoption shown.
The meeting made it clear that SPH has a place in both industrial and academic work-flows, with examples showing applications to fluid sloshing, high-velocity impacts, oiling, entrained media transport and many others. Final group discussions ultimately concluded that the SPH community, through organisations like the SPH SIG and SPHERIC, should aim to drive a set of industrially relevant benchmarks to allow the different solvers to prove their capabilities

SPHERIC
NEWSLETTER

SPH rEsearch and engineeRing International Community
http://spheric-sph.org/
Contact: benedict.rogers@manchester.ac.uk