Peking University, China

中国北京

2017 SPHERIC Beijing International Workshop

Handbook

October 17-20, 2017

WELCOME MESSAGE

Dear Delegates

The College of Engineering of Peking University is delighted to host the 2017 SPHERIC Beijing International Workshop (or **SPHERIC Beijing 2017**). This is an important event of 2017 in the field of Smoothed Particle Hydrodynamics (SPH) and related particle-based methods.

The SPH European Research Interest Community (SPHERIC) was founded in 2005 as a Special Interest Group of the ERCOFTAC community and aims at encouraging and facilitating the spread of the method throughout Europe and the wider international community. Since that time, the SPHERIC community continues both to grow and to play an important role in helping the development of SPH for academia, industry and government organizations. SPH is one of the most exciting new areas in the field of computational methods and is opening up the possibility of research into fields that were beyond any modelling capability.

The SPHERIC Beijing 2017 organization committee received abstracts from China, France, Germany, UK, Italy, Spain, Switzerland, Ireland, USA, Japan and Australia, while 56 abstracts were selected to present in the SPHERIC Beijing 2017. This demonstrates just how active the field is, with works ranging from traditional hydrodynamics to solids, fluid-structure interaction, high performance computing and industrial applications.

The SPHERIC Beijing 2017 has been supported by the National Natural Science Foundation of China (NSFC), the Chinese Society of Theoretical and Applied Mechanics (CSTAM), Beijing Innovation Centre for Engineering Science and Advanced Technology (BIC-ESAT), Institute of Ocean Research and State Key Laboratory for Turbulence and Complex Systems of Peking University, and Beijing Paratera Technology Co. Ltd.

It is a great pleasure to welcome you all to Beijing, and share a successful and enjoyable meeting with you.

Moubin Liu

Professor, Peking University
Chair, Local Organization Committee
SPHERIC Beijing 2017

ACKNOWLEDGEMENT

SPHERIC Beijing 2017 has been supported by

National Natural Science Foundation of China (NSFC)

Chinese Society of Theoretical and Applied Mechanics (CSTAM)

Institute of Ocean Research, Peking University

Beijing Innovation Centre for Engineering Science and Advanced Technology (BIC-ESAT), Peking University

State Key Laboratory for Turbulence and Complex Systems, Peking University

Beijing Paratera Technology Co. Ltd.

CONTENTS

ABOUT SPHERIC

SPHERIC, or "SPH European Research Interest Community", is the international organisation representing the community of researchers and industrial users of Smoothed Particle Hydrodynamics (SPH), aims to promote the SPH method in both academic and industrial fields and enhance collaborations between countries and institutes. It was recognized as a Special Interest Group (SIG) within Ercoftac in January 2006.

As a purely Lagrangian technique, SPH enables the simulation of highly distorting fluids and solids. Fields including free-surface flows, solid mechanics, multi-phase, fluid-structure interaction and astrophysics where Eulerian methods can be difficult to apply represent ideal applications of this meshless method.

The SPH method was developed to study non-axisymmetric phenomena in astrophysics in the 1970s, but its application to engineering emerged in the 1990s and early 2000s. In the past twenty years the method has developed rapidly in many fields of application from impacts to fracture to breaking waves and fluid-structure interaction.

Following the impulse generated by a collection of local initiatives in 2005 (France, UK, Italy...), a need to foster and collaborate efforts and developments was identified. Since then, the SPHERIC organisation has gone on to push the development of the method forward providing a network of researchers and industrial users around the world as a means to communicate and collaborate.

For more information about SPHERIC, please visit `http://spheric-sph.org`

SPHERIC Beijing 2017

You are cordially invited to attend the 2017 SPHERIC Beijing International Workshop (SPHERIC Beijing 2017) to be held at Peking University (PKU) in Beijing China, from October, 17-20, 2017. This is the first time that the SPHERIC Workshop is held outside Europe.

The SPHERIC workshops are the only worldwide events which exclusively focus on the Smoothed Particle Hydrodynamics (SPH) methodology and related simulation approaches. SPH has recently gained enhanced attention in the area of scientific computing. Exemplary applications refer to the development of galaxies in astrophysics, environmental engineering, applied solid mechanics, marine and coastal engineering, nuclear power engineering, medical engineering or geotechnical problems.

The successful concept of SPHERIC is due to a methodological focus in an interdisciplinary application environment, integrating the know-how of physicists, mathematicians, IT experts and engineers from academia and industry. On behalf of the organizing team, it is a pleasure and honor to us to invite scientists and researchers to the SPHERIC 2017 at PKU, in Beijing, China.

For more information about SPHERIC Beijing 2017, please visit `http://ocean.pku.edu.cn/SPHERIC_Beijing/index.php.htm`

ABOUT PEKING UNIVERSITY

General Information

Peking University is a comprehensive and national key university. The campus, known as ”Yan Yuan”（the garden of Yan）, is situated at Haidian District in the western suburb of Beijing, with a total area of 2,743,532 square metres (or 274 hectares). It stands near to the Yuanmingyuan Garden and the Summer Palace.

Peking University is proud of its outstanding faculty, including 53 members of the Chinese Academy of Sciences (CAS), 7 members of the Chinese Academy of Engineering (CAE), and 14 members of the Third World Academy of Sciences (TWAS).

The university has effectively combined research on important scientific subjects with the training of personnel with a high level of specialized knowledge and professional skill as demanded by the country's socialist modernization. It strives not only for improvements in teaching and research work, but also for the promotion of interaction and mutual promotion among various disciplines.

Thus Peking University has become a center for teaching and research and a university of a new type, embracing diverse branches of learning such as basic and applied sciences, social sciences and the humanities, and sciences of medicine, management, and education. Its aim is to rank among the world's best universities in the future.

History

Founded in 1898, Peking University was originally known as the Imperial University of Peking. It was the first national university covering comprehensive disciplines in China, and has been a leading institution of higher education in China since its establishment. It also served as the highest administration for education at the beginning of its founding.

In 1912, the university adopted its present name. At the end of the 20th century, the Chinese government put Peking University at the top of its agenda for promoting higher education, with the aim to build a world-class university in the 21st Century. After merging with Beijing Medical University in 2000, Peking University once again was strengthened in its disciplinary structure.

Peking University has continually played the essential role of pioneers in the course of China's modernization. The university's traditional emphasis on patriotism, progress, democracy, and science, together with its educational standards of diligence, precision, factualism, and innovation, have been passed down from generation to generation.

WORKSHOP DETAILS

Committees

Scientific Committee

- Prof. David Le Touzé (Ecole Centrale de Nantes, France)
- Dr. Damien Violeau (Electricité de France, France)
- Dr. Nathan Quinlan (National Univ. of Ireland, Ireland)
- Dr. Ben Rogers (University of Manchester, UK)
- Prof. Stefano Sibilla (University of Pavia, Italy)
- Dr. Jean-Christophe Marongiu (ANDRITZ Hydro, France)
- Dr. Alex Crespo (Universidade de Vigo, Spain)
- Dr. Andrea Colagrossi (INSEAN, Italy)
- Dr. Xiangyu Hu (Technical University of Munich, Germany)
- Prof. Rade Vignjevic (Brunel University of London, UK)
- Prof. Thomas Rung (Technical University of Hamburg-Harburg, Germany)
- Dr. Antonio Souto-Iglesias (Technical University of Madrid, Spain)
- Dr. Renato Vacondio (University of Parma, Italy)
- Dr. Matthieu De Leffe (Nextflow Software, France)
- Prof. Moncho Gómez-Gesteira (Universidade de Vigo, Spain)
- Dr. Abbas Khayyer (University of Kyoto, Japan)
- Prof. Walter Dehnen (University of Leicester, UK)
- Dr. Raj Das (University of Auckland, New Zealand)
- Prof. Robert A. Dalrymple (Johns Hopkins University , USA)
- Prof. Alexis Hérault (Conservatoire National des Arts et Métiers, France)
- Prof. Joe Monaghan (Monash University, Australia)
- Prof. Peter Eberhard (University of Stuttgart, Germany)
- Prof. Moubin Liu (Peking University, China)
- Dr Mehmet Yildiz (Sabanci University, Turkey)

Organizing Committee

Chair

- Prof. M. B. Liu, Peking University

Co-Chairs

- Prof. H. F. Qiang, Xi'an Hi-Tech Institute, China
- Prof. A. M. Zhang, Harbin Engineering University, China
- Prof. L. Zou, Dalian University of Technology, China
- Prof. D. A. Hu, Hunan University, China
- Prof. F. Xu, Northwestern Polytechnical University, China
- Prof. Z. R. Li, Wenzhou University, China

Keynote Speakers

Prof. David Le Touzé

Ecole Centrale Nantes
Deputy Head, LHEEA research dept. (ECN and CNRS)
Head, H2I research group of LHEEA
Head, Centrale Nantes - Bureau Veritas Chair
Head, IRT Jules Verne SimAvHy Chair

Title: Smoothed Particle Hydrodynamics, fact checking: from theory to applications

Bio: Prof. David LE TOUZÉ is 40 years old. He got his MSc in Hydrodynamics and Ocean Engineering from Ecole Centrale Nantes (Nantes, France) in 2000. Ecole Centrale Nantes is a highly competitive French « Grande Ecole » which awards MScs and PhDs only. He then got his PhD with honors in 2003 from the same institute, whose topic was modeling gravity wave generation and propagation by spectral methods. He spent 2 years of post-doc at CNR-INSEAN (Rome, Italy) in 2004-05 where he started working in SPH. He came back to Ecole Centrale Nantes in 2006 and became Assistant Professor in 2007, Associate Professor in 2010 and Full Professor in 2012. His researches revolve mainly around free-surface flows. He is leading since 2012 a research group on Hydrodynamics, Interfaces and Interactions (H2i) which counts 8 professors and researchers, 14 PhD students, and 6 post-docs. His current research topics cover different numerical methods and techniques: SPH (Smoothed Particle Hydrodynamics), incompressible (OpenFOAM) and weakly-compressible (WCCH) Finite Volumes, Adaptive Mesh Refinement (AMR), Immersed Boundary Method (IBM), Vortex Method (DVH), Lattice-Boltzmann Method (LBM). He is also working on different method couplings: potential (waves) to Navier-Stokes Finite Volume Method for wave-structure interactions, SPH to Finite Element Method (FEM) for fluid-structure interaction, SPH to Finite Volume Method for efficient solutions of complex flows. Main applications of his research are in the fields of marine engineering (many naval, offshore and marine renewable energy topics), automotive (aquaplaning, gear boxes), aeronautics (ditching) and health (cardio-vascular flows). He is currency leading 7 industrial projects (over 5M contracts). He is the author of 30+ journal publications, with a google h-index of 22. He is also Deputy Head of his research department (LHEEA, 140 staff) which is a joint research unit between Ecole Centrale Nantes and CNRS.

Prof. J. S. Chen

William Prager Chair Professor, Structural Engineering Department, Director, Center for Extreme Events Research, University of California, San Diego

Title: An Implicit Gradient Reproducing Kernel Particle Method: Theory and Applications

Bio: J. S. Chen earned his undergraduate degree from National Central University (1978-1982) in Taiwan, and received master's (1986) and Ph.D. (1989) from Northwestern University. He worked in GenCorp's Research Division from 1989 to 1994. From 1994 to 2001, he held a faculty position in the Mechanical Engineering Department of The University of Iowa before moving to UCLA in 2001, where he served as the Chair of Civil & Environmental Engineering Department from 2007 to 2012. He was the Chancellor's Professor in the Civil & Environmental Engineering Department at UCLA and also Professor of Mechanical & Aerospace Engineering Department and Mathematics Department. In 2013, he joined the Structural Engineering Department of UCSD as the inaugural holder of the William Prager Endowed Chair. He also is the director of the Center for Extreme Events Research at the Jacobs School of Engineering at UC San Diego.

Conference Venue

The SPHERIC Beijing 2017 will be held at Peking University, Beijing, China.

- Address: No.5 Yiheyuan Road Haidian District, Beijing, P.R.China
- Tel: +86 1062 766 982
- Website: `http://ocean.pku.edu.cn/SPHERIC_Beijing/index.php.htm`

Accommodation

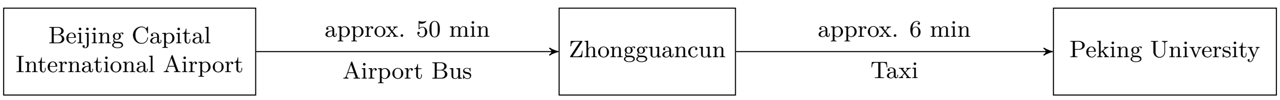
For participants outside China, the committee has reserved rooms at **ZhongGuanYuan Global Village PKU** at discounted prices. It should be noted that the registration fee does not include accomodation. All of the hotels below include breakfast and wifi access.

- Address: No.126 ZhongGuanCun North Street, Haidian District, Beijing, China
- Tel: +86 10 62752288
- Website: `http://www.pkugv.com`

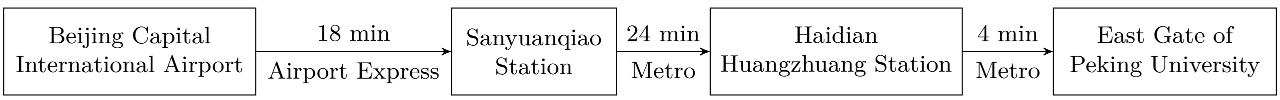
Transportation Information

TAXI Taxi is the most convenient transportation to the University from the airport. As the Capital Airport is located 40 km northeast from the campus, it will cost you around 100-130 RMB (expressway fee of 15 RMB included) to get to the university from the airport. It takes approximately 1-1.5 hours to arrive at PKU from the airport, depending on the traffic.

Getting to PKU from the Airport BUS You may also first travel to ZhongGuanCun via the airport shuttle bus and then take a taxi from ZhongGuanCun.



You may also take the subway. Take the Line ‘Airport Express’ from Terminal 2 or Terminal 3, and transfer to Line 10 at the ‘Sanyuanqiao’ station, and then transfer to the Line 4 at ‘Haidian Huangzhuang’ station, and finally get off at ‘The East Gate of Peking University’ station.



Registration/Information Desk

The registration desk at Room 104 in the Peking University Overseas Exchange Center, will be open from 8:30-18:00 on Tuesday 17th October.

Instructions for Presenters

- According to SPHERIC Workshop Presentation Style, each presenter will have 13 minutes strictly to present their work, followed by the successive presenter. After all the presentations in a specific session, all the presenters will be asked to stand in front of the conference room and answer possible questions in a Group.
- There is no need to explain the very basics of SPH to an SPH specialist audience, and please emphasize what is new and novel in method or application.

SPH Training Day

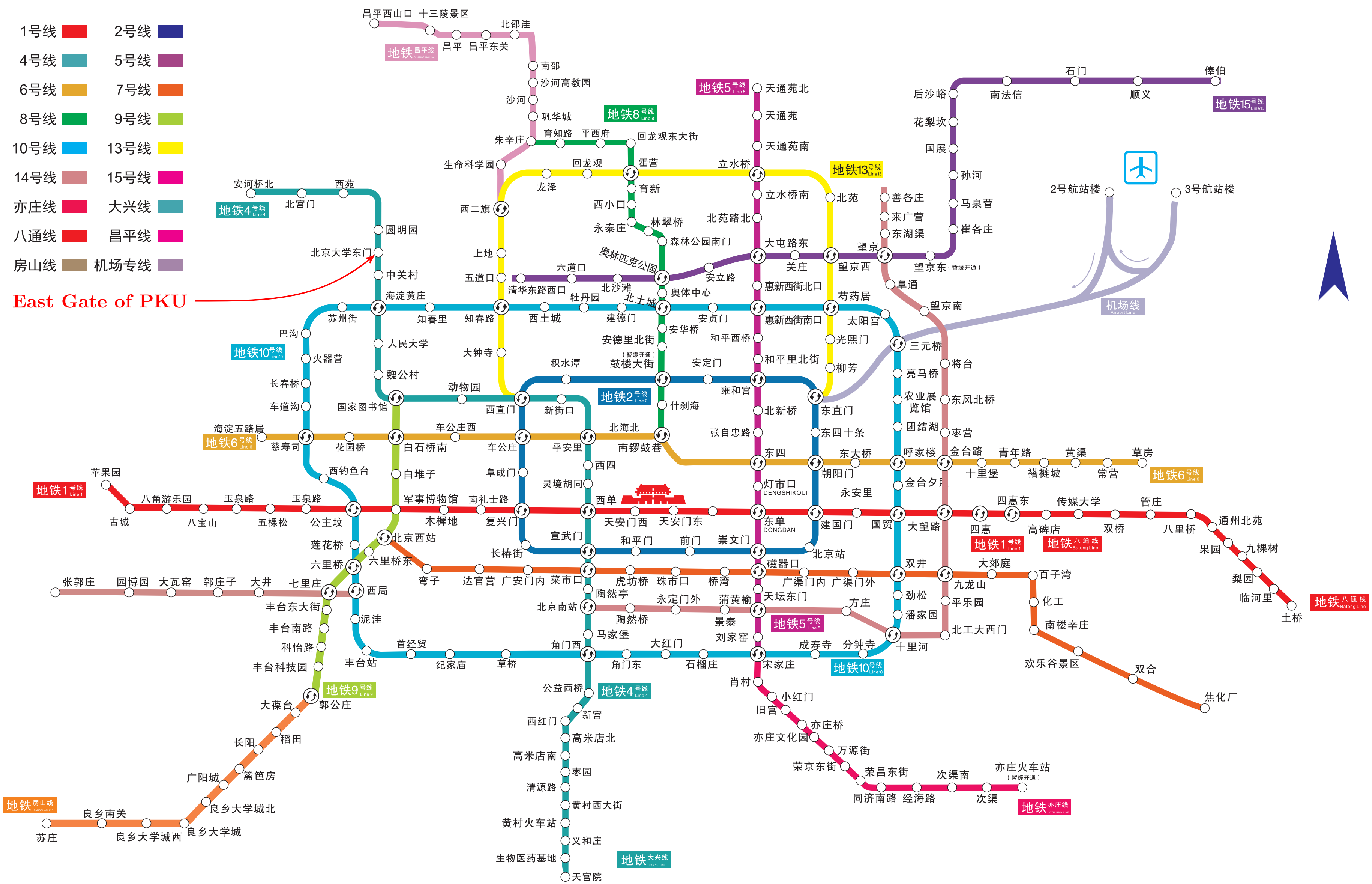
Supplementary to the workshop, an SPH training day will be offered on 17 October 2017. The training is most suitable for researchers who are familiar with the principles of SPH but are beginning their work in the field. More experienced SPH developers and users may find that the training day is a useful opportunity for sharing insights and ideas. The SPH training day will also take place at the Peking University.

Other Tips

Free wifi connection: TBD, EDUROAM is also available.

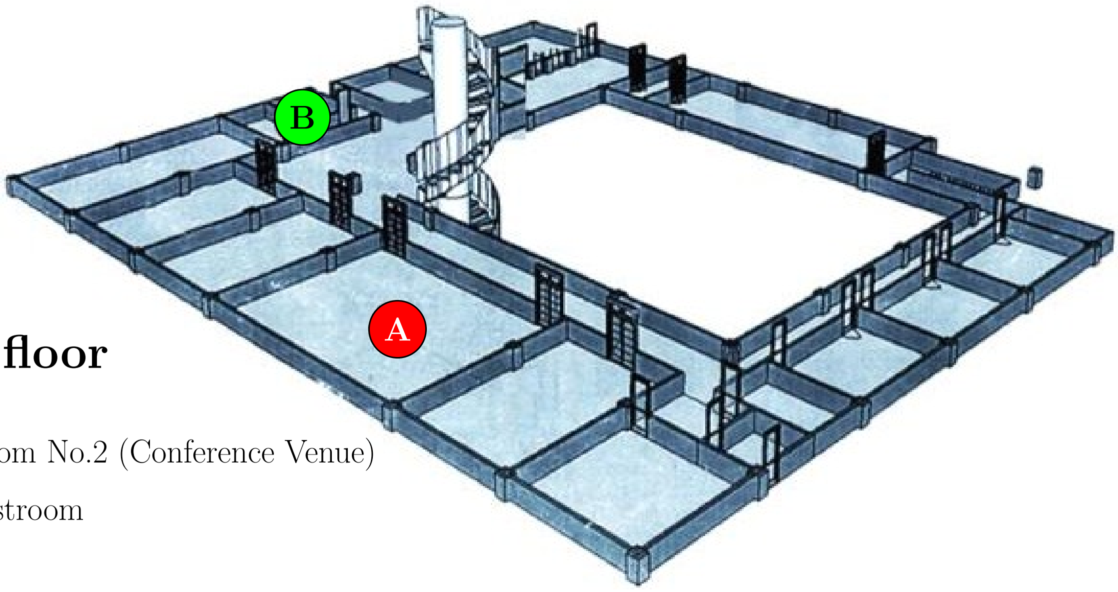
Name tags: Name tags are required for entry to all conference events. Please wear them at all times.

Beijing Subway Map



Map of Peking University Campus

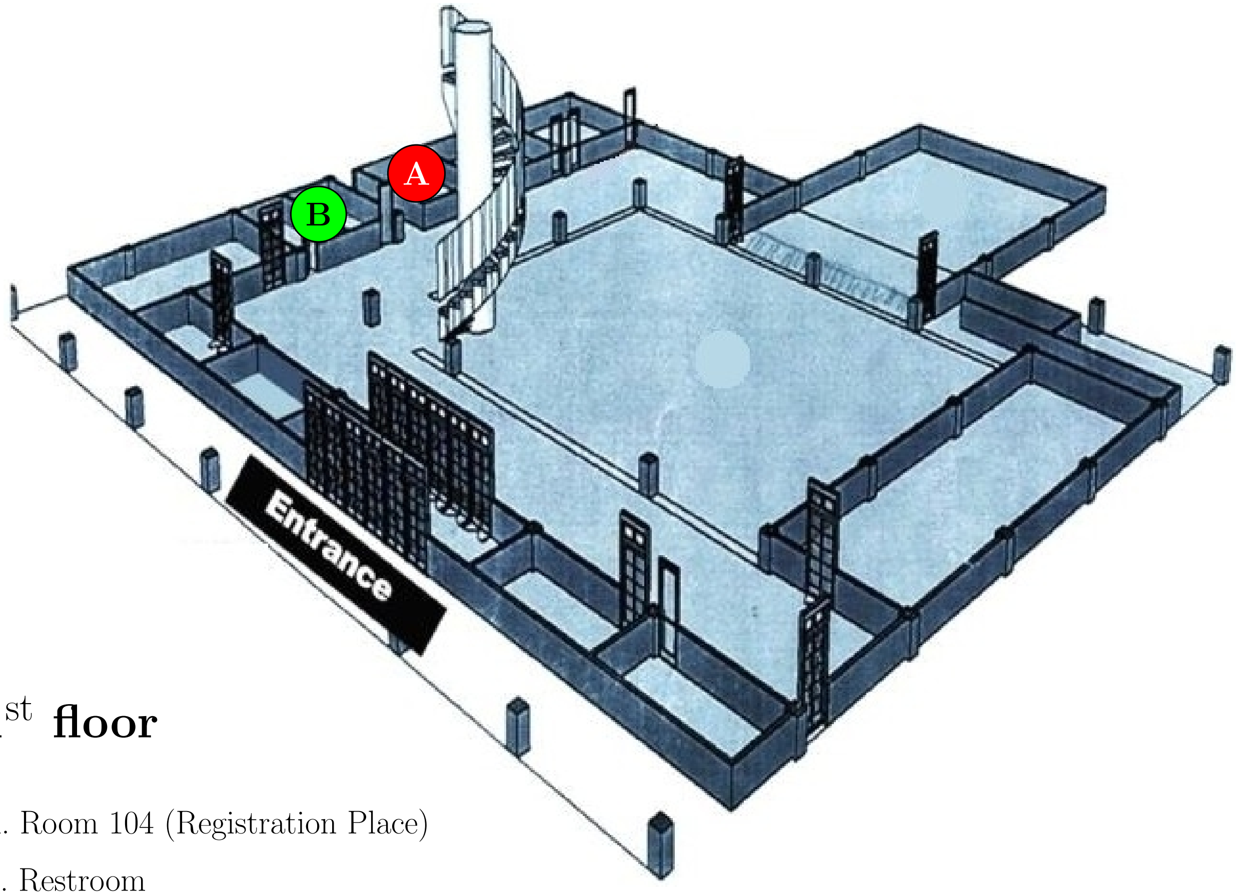
Floor Plan of Yingjie Exchange Center, Peking University



2nd floor

A. Room No.2 (Conference Venue)

B. Restroom



1st floor

A. Room 104 (Registration Place)

B. Restroom

Program Overview

	17 Oct. (Tue.)	18 Oct. (Wed.)	19 Oct. (Thu.)	20 Oct. (Fri.)
08:00 - 09:00		Registration 8:00 – 8:45 Opening 8:45 – 9:15		
09:00 - 10:00	Training Session 1 9:00 – 10:30	Keynote 1 9:15 – 10:10	Keynote 2 9:00 – 9:55 Session 6 9:55 – 11:00	Session 10 9:00 – 10:05
10:00 - 11:00	Coffee: 10:30–10:50 Training Session 2 10:50 – 12:20	Session 1 10:10 – 11:00		Session 11 10:05 – 11:10
11:00 - 12:00		Coffee: 11:00–11:20 Session 2 11:20 – 12:25	Coffee: 11:00–11:20 Session 7 11:20 – 12:10	Coffee: 11:10–11:30 Session 12 11:30 – 12:35
12:00 - 13:00	Lunch 12:20 – 13:15	Lunch 12:25 – 14:00	Lunch 12:10 – 14:00	Lunch 12:35 – 14:00
13:00 - 14:00	Training Session 3 13:15 – 13:45 Training Session 4 13:45 – 14:15			
14:00 - 15:00	Training Session 5I 14:15 – 15:30	Session 3 14:00 – 15:05	Session 8 14:00 – 15:05	Session 13 14:00 – 15:05
15:00 - 16:00	Coffee: 15:30–15:45 Training Session 5II 15:45 – 17:30	Session 4 15:05 – 16:10	Coffee: 15:05–15:25 Session 9 15:25 – 16:30	Session 14 15:05 – 16:10
16:00 - 17:00		Coffee: 16:10–16:30 Session 5 16:30 – 17:35	Take Bus To Yu Xian Du 16:30–17:00	Coffee: 16:10–16:30 Session 15 16:30 – 17:20
17:00 - 18:00	Dinner 17:30	Dinner 17:35	Visit YXD Food Museum 17:00 – 18:00	Dinner 17:20
18:00 - 19:00			Banquet & Award 18:00 – 21:00	

SPH Training Day Program

Venue: Room 1338W, Level 3, #1 Science Blk, Peking University

地点：北京大学理科一号楼三层 1338W 房间（计算机中心 8 号机房）

Training day	17 October (Tuesday)	
9:00-10:30	Training Session 1: Theory and Application of SPH - Part 1: An Introduction to Multi-Phase Modelling in SPH	Dr. Xiangyu Hu
10:30-10:50	Coffee	
10:50-12:20	Training Session 2: Advanced SPH/CG(Coarse-Grained) modelling for biomechanics and biomedical systems	Prof. Y. T. Gu
12:20-13:15	Lunch	Box lunch
13:15-13:45	Training Session 3: Simulation with DualSPHysics	Dr. Ben Rogers
13:45-14:15	Training Session 4: Pre- and Post-Processing (Visualisation) with DualSPHysics	Dr Jose Dominguez
14:15-15:30	Training Session 5: Practical hands-on session with DualSPHysics I	Dr. Ben Rogers, Dr. Jose Dominguez, Dr. Jose González-Cao, Mr. Feng Zhang
15:30-15:45	Coffee	
15:45-17:30	Training Session 5: Practical hands-on session with DualSPHysics II	
17:30	Dinner	@PKU canteen with card

- Full day registration (8:30AM to 18:00PM) at Room 104 in the Peking University Overseas Exchange Center
  10 月 17 日全天上午 8:30 到下午 18:00 亦在北京大学英杰交流中心 104 室注册

SPHERIC Beijing 2017 Workshop Program

Venue: Conference Room #2, Peking University Overseas Exchange Centre

地点：北京大学英杰交流中心第二会议室

1st DAY	**18 October (Wednesday)**		
08:00-08:45	Registration		
08:45-09:15	Opening	Speech by: B. Rogers Speech by: J. X. Wang, Vice Dean of CoE, PKU	Chair: M. B. Liu
09:15-10:10	Keynote 1: Smoothed particle hydrodynamics, fact checking: from theory to applications by Prof. David Le Touzé, Ecole Centrale de Nantes		Chair: J. X. Wang
10:10-11:00	Session 1: Maritime and Naval Architecture Applications		Chair: A. Colagrossi
	1.1 "DualSPHysics: a numerical tool to simulate real breakwaters" F. Zhang[Student Prize], S. P. Shang, Alejandro Crespo, José Dominguez, Moncho Gomez-Gesteira, Corrado Altomare, Andrea Marzeddu 1.2 "High speed water impacts of fat plates in different ditching configurations through a Riemann-ALE SPH model" S. Marrone, A. Colagrossi, M. De Leffe, L. Chiron, D. Le Touze 1.3 "Application of improved SPH solid-wall boundary model in missile water exiting" H. L. Zheng, H. F. Qiang, F. Z. Chen, C. Shi		
11:00-11:20	Coffee		
11:00-12:25	Session 2: Multiple Continua and Multi-Phase Flows		Chair: X. Y. Hu
	2.1 "Multiphase Godunov-typed smoothed particle hydrodynamics method with approximate Riemann solvers" Z. W. Cai, Z. Zong, L. Zhou, Z. Chen, C. Tiao 2.2 "A two-phase SPH model for sediment laden flows" H. B. Shi, X. P. Yu 2.3 "Numerical simulation of water-entry problems using an improved multiphase SPH method" H. Cheng[Student Prize], A. M. Zhang, F. R. Ming 2.4 "Stable sharp interface method for SPH" M. Y. Zhang		
12:25-14:00	Lunch (Buffet lunch @ ZhongGuanYuan Global Village PKU)		
14:00-15:05	Session 3: Impacts with Fluids or Solids		Chair: X. H. Guo

	3.1 "Aircraft tire water spray simulation using SPH" Y. K. Hu, Y. F. Rong, D. X. Leng, F. Xu, X. Y. Gao, R. G. Cao, W. Ding, J. Lv 3.2 "Numerical simulation of the damage of multi-floor buildings by conical projectile with SPH method" H. F. Qiang, X. Y. Sun, F. Z. Chen, G. X. Zhang 3.3 "Corrected smoothed particle hydrodynamics for simulating failure progress of model-scale ice" X. Zheng, N. B. Zhang, Q. W. Ma 3.4 "Numerical study of the mechanism of explosive/impact welding using an improved SPH method" Z. L. Zhang[Student Prize], M. B. Liu	
15:05-16:10	Session 4: Free Surface and Moving Boundaries Applications	Chair: X. F. Yang
	4.1 "SPH numerical investigation of oscillating characteristics of hydraulic jumps at an abrupt drop" Diana De Padova, Michele Mossa, Stefano Sibilla 4.2 "An SPH simulation of bubble cavity evolution on underwater movement" J. R. Shao, M. B. Liu 4.3 "The δALE-SPH model: an improved δ-SPH scheme containing particle shifting and ALE formulation" P. N. Sun[Student Prize], A. M. Zhang, A. Colagrossi, S. Marrone, M. Antuono 4.4 "SPH numerical simulation of lift-off by impact of sand particles on flat sand bed" J. Zhao, A. F. Jin, Maimtimin Geni, X. J. Ma	
16:10-16:30	Coffee	
16:30-17:35	Session 5: Geotechnical Applications	Chair: J. S. Wu
	5.1 "A comparative study of SPH and MPM in modeling mixed-mode failure in rocks" Sam Raymond[Student Prize], Bruce Jones, John Williams 5.2 "A SPH investigation of soil plastic behaviour with Mohr-Coulomb constitutive model" S. H. Zhao, Ha H. Bui, Vincent Lemiale, Giang D. Nguyen 5.3 "A robust approach to model rock fracture with SPH" Y. N. Wang, Ha H. Bui, Giang D. Nguyen, P. G. Ranjith 5.4 "An elasto-plastic-μ(I) SPH model for landslide induced debris flow" W. T. Zhang, Y. An, Q. Q. Liu	
17:35	**Dinner** @PKU canteen with card	

2nd DAY	**19 October (Thursday)**	
09:00-09:55	Keynote 2: An implicit gradient reproducing kernel particle method: theory and applications by Prof. J. S. Chen, University of California	Chair: P. Chen, Head, Division of Computational Mechanics , PKU
09:55-11:00	Session 6: Hydraulic Applications I	Chair: S. Marrone
	6.1 "Overview of SPH-ALE applications for hydraulic turbines in ANDRITZ Hydro" Jean-Christophe Marongiu, Magdalena Neuhauser, Martin Rentschler, Etienne Parkinson 6.2 "Numerical and experimental investigation of two porous wave - breaking structures" W. Q. Hu, Q. Fan Student Prize, J. M. Zhan, W. H. Cai 6.3 "SPH for the interaction between tsunami wave and upright cylindrical groups" J. J. Li Student Prize, L. Tian, Y. S. Yang, L. C. Qiu, Y. Han 6.4 "Hydrodynamics characteristics of land hinged oscillating wave surge converter with SPH method" D. H. Zhang, Y. X. Shi, C. Huang, Y. L. Si, B. Huang, and W. Li	
11:00-11:20	Coffee	
11:20-12:10	Session 7: Adaptivity (variable resolution)	Chair: L. C. Qiu
	7.1 "The study on SPH method with space variable smoothing length and its applications to multi-phase flow" W. K. Shi Student Prize, Y. M. Shen , J. Q. Chen 7.2 "A dynamic refinement strategy in SPH for simulating the water entry of an elastomer" L. Wang, F. Xu, Y. Yang 7.3 "Adaptive particle splitting in the finite volume particle method" Nathan J. Quinlan	
12:10-14:00	Lunch @PKU canteen with card	
14:00-15:05	Session 8: New applications of SPH	Chair: David Le Touzé

	8.1 " Modeling the melting process of quartz glass using SPH method" Z. Y. Liu[Student Prize], Q. L. Ma, H. S. Fang 8.2 "Study on dynamic behaviors of liquid-filled flexible multibody systems under the low-gravity environment" W. Z. Kong, Q. Tian 8.3 "A SPH model for the root system of plants" Matthias Mimault, Lionel Dupuy, Mariya Ptashnyk 8.4 "SPH simulation of drop impact on a hot wall with vaporization effects" X. F. Yang, S. C. Kong, M. B. Liu	
15:05-15:25	Coffee	
15:25-16:30	Session 9: High-Performance Computing	Chair: B. Rogers
	9.1 "Developing an extensible, portable, scalable toolkit for massively parallel incompressible smoothed particle hydrodynamics (ISPH)" X. H. Guo, Benedict D. Rogers, Steven Lind, Peter K. Stansby 9.2 "Three-dimensional sloshing simulations by using GPU-based MPS method" X. Chen, X. Wen, D. C. Wan 9.3 "GPU-based SPH modeling of flood with floating bodies in urban layouts including underground spaces" J. S. Wu, N. Li, W. Y. Liu, H. Zhang 9.4 "Improve the effectively of computational fluid dynamics work based on supercomputing cloud" N. Qiao	
16:30-17:00	Take Bus To Yu Xian Du (YXD)	
17:00-18:00	Visit the YXD Food Museum	
18:00-21:00	Conference Banquet & Award	

3rd DAY	20 October (Friday)	
09:00-10:05	Session 10: Numerical Aspects of SPH	Chair: N. Quinlan
	10.1 "SPH energy balance during the generation and propagation of gravity waves" Domenico Davide Meringolo, Y. Liu, A. Colagrossi 10.2 "Water hammer analysis using SPH in density summation form" D. Q. Hou, C. Y. Huang, M. L. Wang, H. F. Duan 10.3 "Particle trajectory calculation in SPH" J. Y. Shen, W. H. Lu, D. Q. Hou, Arris S. Tijsseling 10.4 "Simulating shock waves with corrective smoothed particle method (CSPM)" C. Y. Huang, J. Deng, D. Q. Hou, Arris S. Tijsseling	
10:05-11:10	Session 11: Fluid Structure Interaction	Chair: M. De Leffe
	11.1 "SPH modeling of fluid-structure interaction (FSI)" L. H. Han, X. Y. Hu 11.2 "Numerical modeling of 2D complex movement patterns to FSI problems using smoothed particle hydrodynamics" C. Zhuang, D. A. Hu, T. Long, G. Yang 11.3 "Implement of the MPS-FEM coupled method for the FSI simulation of the 3-D dam-break problem" Y. L. Zhang, D. C. Wan 11.4 "A new numerical method for SPH fluid-solid coupling simulation and its preliminary verification" X. J. Ma, Geni Mamtimin, A. F. Jin	
11:10-11:30	Coffee	
11:30-12:35	Session 12: Modelling of Incompressible Flows	Chair: A. M. Zhang
	12.1 "An enhanced ISPH-SPH coupled method for incompressible fluid-elastic structure interactions" Abbas Khayyer, Hitoshi Gotoh, Yuma Shimizu, Hosein Falahaty 12.2 "Interaction between solitary wave and flexible plate based on MPS-FEM coupled method" C. P. Rao, D. C. Wan 12.3 "Modeling of single film bubble and numerical study of the plateau structure in foam system" Z. G. Sun, N. Ni, Y. J. Sun, G. Xi 12.4 "Numerical simulation of Rayleigh-Taylor instability by MPS multiphase method" X. Wen, D. C. Wan	
12:35-14:00	Lunch (Buffet lunch @ ZhongGuanYuan Global Village PKU)	

14:00-15:05	Session 13: Alternative Formulations and Particle-Based Simulation Techniques	Chair: M. B. Liu
	13.1 "Numerical simulation of particle collision and breakup behavior by SDPH-FVM coupling method" H. F. Qiang, F. Z. Chen 13.2 "A physics evoked meshfree method" Z. B. Ma, Y. Z. Zhao 13.3 "Suppression of non-physical voids in the finite volume particle method" Mohsen H. Moghimi, Nathan J. Quinlan 13.4 "The Hermit-type RRKPM for piezoelectric materials" J. C. Ma, G. F. Wei	
15:05-16:10	Session 14: Other applications of SPH	Chair: Stefano Sibilla
	14.1 "A development of a SPH model for simulation of abrasive-water-jet impacting on a metallic surface" X. W. Dong, Z. L. Li, J. L. Liu 14.2 "SPH simulation of Couette flow with sinusoidally moving solid boundary" H. Q. Li, H. T. Liu, J. Z. Chang 14.3 "Application of particle-based computational acoustics to sound propagation and scattering" Y. O. Zhang 14.4 "Image processing with the SPH method" C. Y. Huang, W. H. Lu, D. Q. Hou, X. Cheng	
16:10-16:30	Coffee	
16:30-17:20	Session 15: Hydraulic Applications II	Chair: M. Rentschler
	15.1 "Analysis of the hydrological safety of dams using numerical tools: Iber and DualSPHysics" J. González-Cao, O. García-Feal, A. J. C. Crespo, J. M. Domínguez, M. Gómez-Gesteira 15.2 "Construction of two-dimensional SPH numerical wave tank" J. Y. Wang, F. Xu, Y. Yang 15.3 "An SPH numerical wave-current tank" M. He, H. S. Wang, X. F. Gao, W. H. Xu, Y. Shi	
17:20	Dinner @PKU canteen with card	

ABSTRACTS

Contents

DualSPHysics: a numerical tool to simulate real breakwaters

Feng ZHANG†[1], Shaoping SHANG[1], Alejandro CRESPO[2], José DOMÍNGUEZ[2], Moncho GÓMEZ-GESTEIRA[2], Corrado ALTOMARE[3], and Andrea MARZEDDU[4]
[1]Xiamen University, China
[2]Universidade de Vigo, Spain
[3]Flanders Hydraulic Research & Ghent University, Belgium
[4]Universitat Politècnica de Catalunya, Spain
zhangfeng@stu.xmu.edu.cn†

Abstract: DualSPHysics [1] is an SPH-based model conceived to be an efficient and user-friendly numerical technique for a wide range of application in the field of hydraulic, naval and coastal engineering. The model is open source and can be freely downloaded from `http://www.dual.sphysics.org`. Thanks to the power of GPUs (graphics cards with powerful parallel computing), real engineering problems can be simulated with DualSPHysics using high resolution at a reasonable time. When applied to coastal engineering, the model has been demonstrated to accurately reproduce wave propagation and transformation and wave-structure interaction phenomena. The code is devised to mimic an experimental facility (wave flume or wave basin) and therefore implements automatic wave generation and integrated active wave absorption (AWAS) techniques [2]. Moving boundaries are used to mimic the displacement of the wavemaker used in a physical facility. In the present study, a piston-type wavemaker that moves with a pre-imposed displacement is considered to generate regular wave trains. Â The main objectives of the work are:

1. To validate DualSPHysics in terms of wave run-up on a breakwater with a two layer cubic blocks armor. The numerical results are compared with experimental data of a smooth dike (Figure 1) and a dike with 2 layers of cubic blocks (Figure 2).

2. To apply the validated SPH model to study the run-up on a dike using the real dimensions, bathymetry and waves conditions from the coast of Chongwu (China).

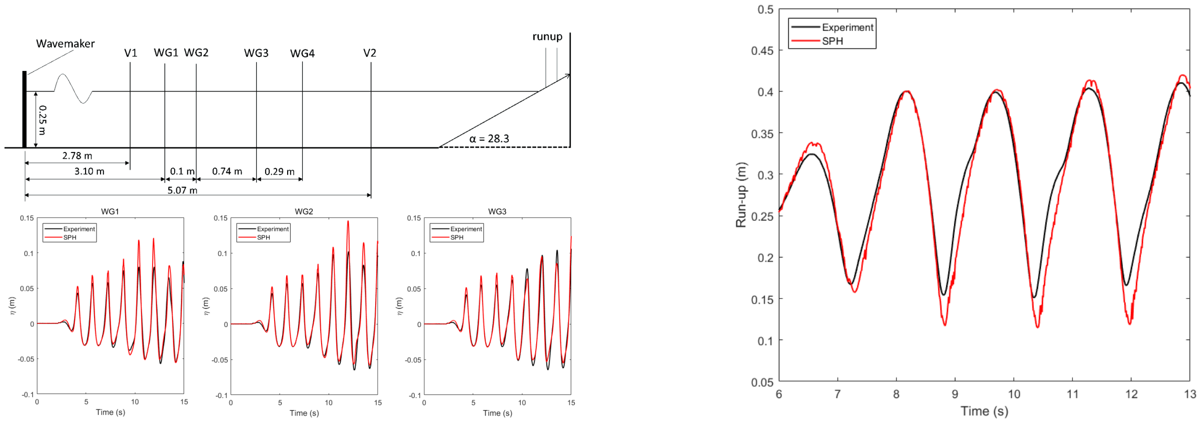


Figure 1: Validation of DualSPHysics comparing with experiments of the smooth dike and waves of $H = 0.1$ m, $T = 1.56$ s, $d = 0.25$ m.

As first **novelty**, a proper validation of run-up is here performed since we have compared the numerical and experimental time series of water surface elevation and time series of wave run-up. Previous work [3] presented a validation for run-up, but only a maximum value for different incoming waves was compared with experimental and literature data. The time series of the experimental wavemaker is assigned to the numerical one. Figure 1 and 2 shows the result of the validation with experiments using the smooth and the porous dike, respectively. Note that run-up for the armor block dike is numerically computed at 52 different positions along the width of the channel to catch the three-dimensional behavior. Several different wave conditions are simulated and overall **good accuracy** is obtained for both wave surface elevation and time series of the run-up.

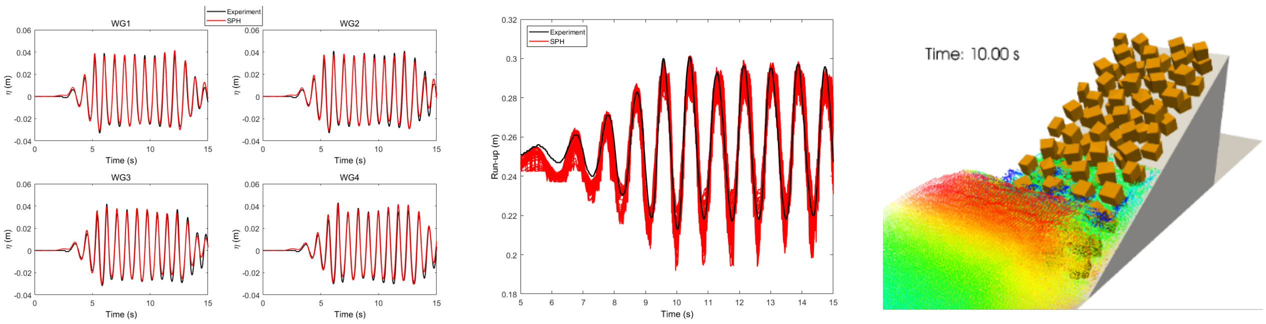


Figure 2: Validation of DualSPHysics comparing with experiments of the breakwater with a two layer cubic blocks armor and waves of $H = 0.08$ m, $T = 0.87$ s, $d = 0.25$ m.

The second **novelty** is the application of the SPH model to a real problem using the dimensions of a dike in China. In this case wave conditions are imposed based on real wave condition in situ and AWAS [2] is employed to compensate the wave reflection at the numerical wavemaker. This is mandatory to mimic the real open sea. Therefore, once the model has been properly validated with experiments **it can be applied to study real situations** in the coast of Chongwu.

References

[1] Alejandro JC Crespo, José M Domínguez, Benedict D Rogers, Moncho Gómez-Gesteira, S Longshaw, R Canelas, Renato Vacondio, A Barreiro, and O García-Feal. Dualsphysics: Open-source parallel cfd solver based on smoothed particle hydrodynamics (sph). *Computer Physics Communications*, 187:204–216, 2015.

[2] C Altomare, JM Domínguez, AJC Crespo, J González-Cao, T Suzuki, M Gómez-Gesteira, and P Troch. Long-crested wave generation and absorption for sph-based dualsphysics model. *Coastal Engineering*, 127:37–54, 2017.

[3] Corrado Altomare, Alejandro JC Crespo, BD Rogers, JM Domínguez, X Gironella, and M Gómez-Gesteira. Numerical modelling of armour block sea breakwater with smoothed particle hydrodynamics. *Computers & Structures*, 130:34–45, 2014.

High speed water impacts of flat plates in different ditching configurations through a Riemann-ALE SPH model

S. Marrone[1], A. Colagrossi[1,3], M. De Leffe[2], L. Chiron[2], and D. Le Touzé[3]
[1]CNR-INSEAN, - Marine Tech. Research Inst., Rome, Italy
[2]NEXTFLOW Software, Nantes, France
[3]École Centrale Nantes, LHEEA research dept., Nantes, France

Abstract: The violent water entry of flat plates is investigated through a Riemann-ALE SPH model. The test conditions are of interest for problems related to aircraft and helicopter emergency landing in water. Three main parameters are considered: the horizontal velocity, the approach angle (i.e. vertical to horizontal velocity ratio) and the pitch angle, α. Regarding the latter, small angles are considered in this study. As described in the theoretical work by Zhao and Faltinsen (1993), for small α a very thin, high-speed jet of water is formed, and the time-spatial gradients of the pressure field are extremely high. Further, air-entrainment can take place making even more complex the loading process of the plate. These test conditions are very challenging for numerical solvers. In the present study an enhanced SPH model [1] is firstly tested on a purely vertical impact with deadrise angle $\alpha = 4^\circ$ (Fig. 1). An in-depth validation against analytical solutions and experimental results is carried out, highlighting the several critical aspects of the numerical modelling of this kind of flow, especially when pressure peaks are to be captured (left plot of Fig. 2). A discussion on the main difficulties when comparing to model scale experiments is also provided. Then, the more realistic case of a plate with both horizontal and vertical velocity components is discussed and compared to ditching experiments recently carried out at CNR-INSEAN [2]. In this case both single and two-phase models are considered to take into account possible air-cushion effects (right plot of Fig. 2).

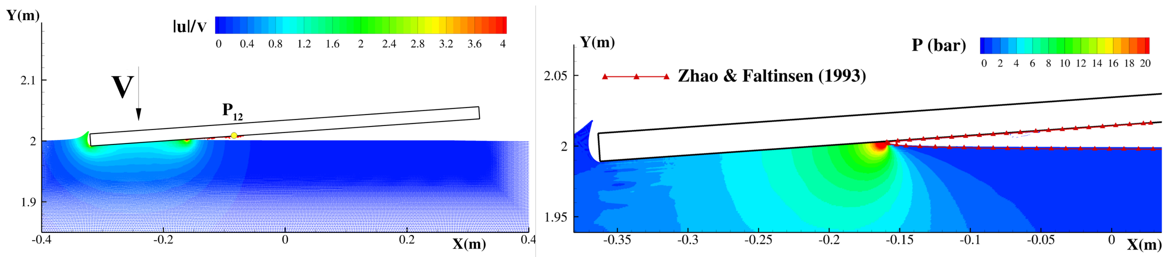


Figure 1: Water entry of a plate at $U = 0$, $V = -6$ m/s with 4° pitch angle. Left: contour plot of the flow velocity magnitude. Right: contour plot of the pressure field.

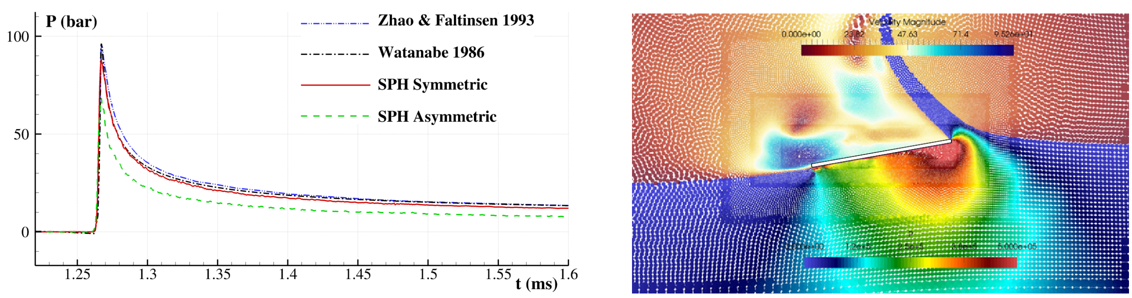


Figure 2: Left: pressure measured at probe P_{12} (see Fig. 1) with SPH in asymmetric and symmetric configurations and comparison with analytical solutions. Right: Two-phase water entry of a plate at $U = 40$, V= -1.5 m/s with 10° pitch angle.

References

[1] G Oger, Salvatore Marrone, David Le Touzé, and Matthieu De Leffe. Sph accuracy improvement through the combination of a quasi-lagrangian shifting transport velocity and consistent ale formalisms. *Journal of Computational Physics*, 313:76–98, 2016.

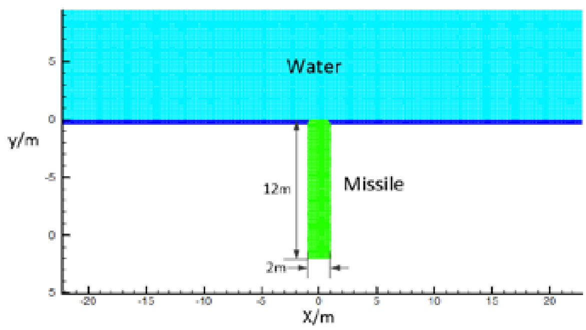
[2] A Iafrati. Experimental investigation of the water entry of a rectangular plate at high horizontal velocity. *Journal of Fluid Mechanics*, 799:637–672, 2016.

Application of Improved SPH Solid-Wall Boundary Model in Missile Waterexiting

Zheng Hua-lin, Qiang Hong-fu, Chen Fu-zhen, and Shi Chao
Department of Power Engineering, Xi'an Hi-Tech Institute, China
zero2one@aliyun.com

Abstract: For its meshfree nature, smoothed particle hydrodynamics (SPH) faces challenges on solving rigid-fluid interaction problems. By correcting the condition and force direction between fluid particles and boundary particles, an improved SPH solid-wall boundary model is proposed by Liu [1], which can apply boundary condition effectively. The classical dambreak problem are simulated, which are respectively compared with the experimental results and another simulation results.

And with this treatment, the 2-D missile waterexiting problem is simulated, which explores the influence of water depth and velocity at the ignition point on the process of missile waterexiting. The present study expands application of SPH method in solving rigid-fluid interaction problems, and with the help of it, the stable flow field, smooth velocity and pressure fields could be obtained.



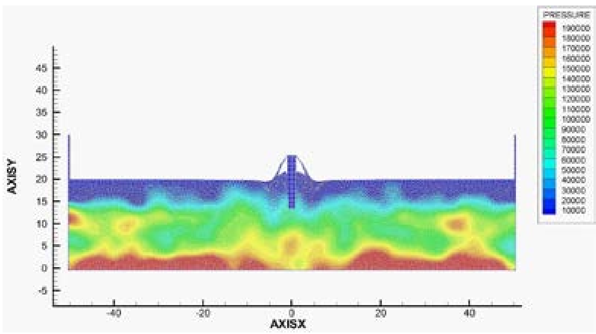


Figure 1: model of missile waterexiting (left) and schematic diagram of the process of missile waterexiting (right).

References

[1] Liu Hu, Qiang Hong-Fu, Chen Fu-Zhen, Han Ya-Wei, and Fan Shu-Jia. A new boundary treatment method in smoothed particle hydrodynamics. *ACTA PHYSICA SINICA*, 64(9), 2015.

Multiphase Godunov-typed Smoothed Particle Hydrodynamics Method with Approximate Riemann Solvers

Z.W.Cai[†1], Z. Zong[2], L. Zhou[2], Z. Chen[3], and C.Tiao[1]
[1]Department of Offshore Structures, China Ship Scientific Research Center, China
[2]School of Naval Architecture, Dalian University of Technology, China
[1]Department of Mechanical Engineering, National University of Singapore, Singapore
caizhiwen0904@163.com[†]

Abstract: In this paper, we propose a Multiphase Godunov-typed Smoothed Particle Hydrodynamics (MGSPH) method for simulating multi-fluid Riemann problems. In this method, different EOSs are applied on different materials; and interfacial approximate Riemann solvers are introduced on the interfacial particle pairs to deal with the transition between different EOSs. Various combinations of five kinds of single-phase approximate Riemann solvers (LLXF, ROE, HLLE, HLLC, DUCO) and three types of interface approximation Riemann solvers (ROE, LRS, RRS) are comparatively studied in three numerical tests. It turns out that LLXF and HLLE give worse results than other approximate Riemann solvers; and pressure instabilities are observed when applying RRS on interfacial particle pairs. In general, the combinations of DUCO+ROE and DUCO+LRS may be the suitable choices for MGSPH in simulating multiphase Riemann problems.

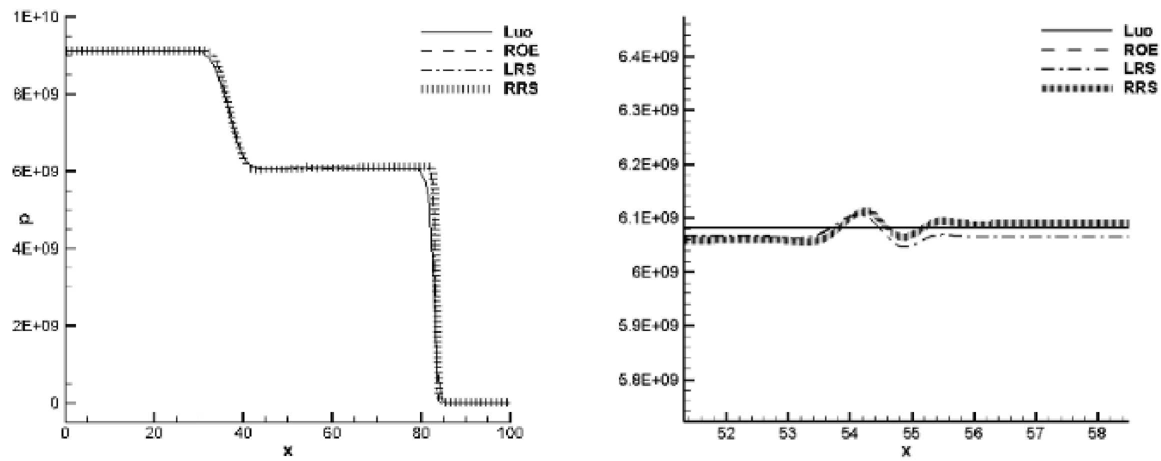


Figure 1: Gas-water shock tube problem. The solutions of $t = 0.000156$. Left: pressure; right: broken features

A Two-Phase SPH Model for Sediment Laden Flows

Huabin Shi† and Xiping Yu
Tsinghua University, Beijing, China
shihuabin@tsinghua.edu.cn†

Abstract: A two-phase SPH model based on a general volume fraction formulation for solid-fluid flows is proposed for sediment motion in free surface flows. Different from the existing SPH two-phase models, in the present model the water and the sediment are treated as two miscible fluids, and the two-fluid system is discretized by a single set of SPH particles, which move with the water velocity and carry properties of the two phases. Turbulence in both the water and sediment phases are dealt with and the particle-turbulence interaction are taken into account. The water is assumed to be weakly compressible while the sediment phase is incompressible. A new equation of state is proposed for the hydrodynamic pressure in the sand-water mixture and a matching strategy of Shepard filtering is adopted to damp the oscillation in pressure.

The model is validated and applied to sand dumping from a line source into a water tank and bed erosion under dam break flows. The computed results are in good agreement with the experimental data. Figure 1 shows the distributions of sediment concentration carried by SPH particles in the dumping of fine and coarse sand. Figure 2 compares the computational and experimental water free surface and bed profiles under dam break flows. It is shown in the applications that the proposed model promises to be an effective tool for sediment transport in free surface flows.

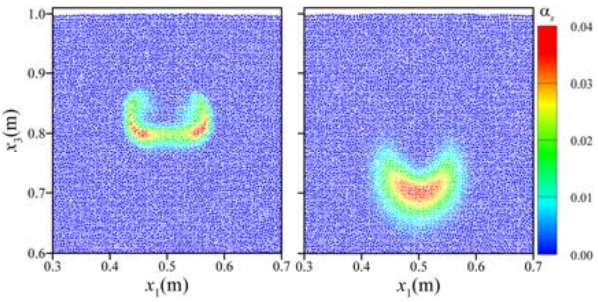


Figure 1: Computed sediment concentrations of the SPH particles. Left: Fine sand with a diameter of 0.8 mm; Right: Coarse sand with a diameter of 5.0 mm.

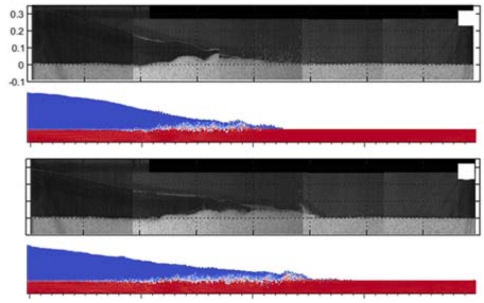


Figure 2: Comparisons between the computed and measured water free surface and bed profiles under dam break flows.

References

[1] Moncho Gomez-Gesteira, Benedict D Rogers, Alejandro JC Crespo, Robert A Dalrymple, Muthukumar Narayanaswamy, and José M Dominguez. Sphysics–development of a free-surface fluid solver–part 1: Theory and formulations. *Computers & Geosciences*, 48:289–299, 2012.

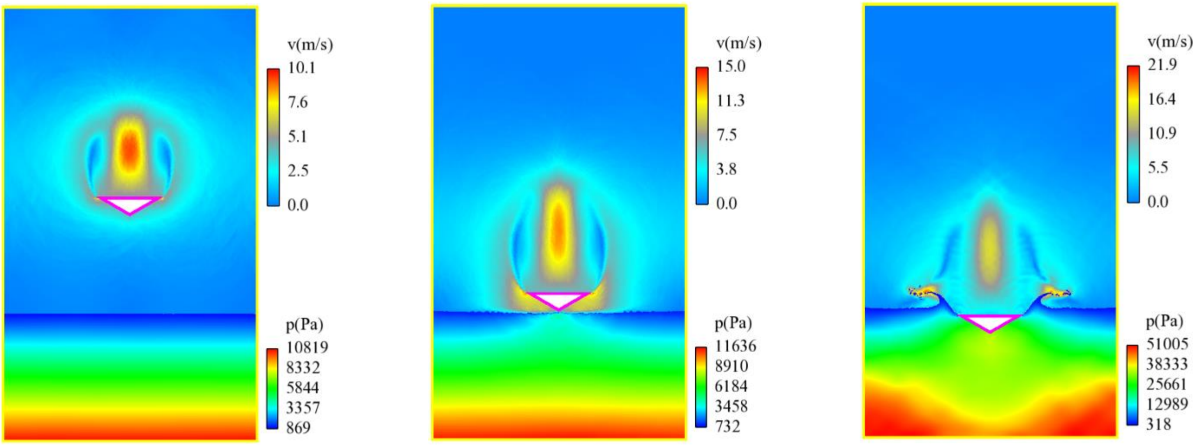
[2] Huabin Shi, Xiping Yu, and Robert A Dalrymple. Development of a two-phase sph model for sediment laden flows. *Computer Physics Communications*, 2017.

Numerical simulation of water-entry problems using an improved multiphase SPH method

H. Cheng, A.M. Zhang, and F.R. Ming†
Harbin Engineering University, China
mingfuren2006@126.com†

Abstract: Water entry is a complex fluid-solid interaction problems that be focused in many aspects, i.e., the shipbuilding and aerospace. The impacting loads are hard to be predicted especially for the solid with a small deadrise angle, beacuse the effect of the air should be taken into account. In the paper, an improved multiphase SPH model is established to simulate the water entry of wedge with different deadrise angles. The Sub-Grid Scale (SGS) model with a multiphase form is adopted in the SPH scheme to represent the effect of the turbulence. What's more, the traditional shifting algorithm is improved for multiphase problems. Based on this, firstly, the water entry of the wedge and the plate are simulated and compared to the experimental data to validate the feasibility and accuracy of the SPH model in the paper. Then, the numerical simulations with variant deadrise angles are carried out, and the results are compared to the single-phase SPH model to investigate the effect of the air.

The images of the water entry of wedge with deadrise angle of 30° is presented in Figure 1, where the wedge free falls from 1.93 m away from the water surface. The velocity field of the air and the pressure field of the fluid are represented respectively. At $t = 0.62$ s, before the wedge contacts with water, the fluid pressure has been affected by the high-speed air flow around the wedge. The whole flow field during the impact is steady which verifies the viablility of the SPH model in the paper.



$t = 0.30$ s $t = 0.62$ s $t = 0.65$ s

Figure 1: The images of water entry of wedge with deadrise angle of 30°.

References

[1] R Zhao, O Faltinsen, and J Aarsnes. Water entry of arbitrary two-dimensional sections with and without flow separation. In *Proceedings of the 21st symposium on naval hydrodynamics*, pages 408–423. Trondheim, Norway, National Academy Press, Washington, DC, USA, 1996.

[2] MB Liu and GR Liu. Smoothed particle hydrodynamics (sph): an overview and recent developments. *Archives of computational methods in engineering*, 17(1):25–76, 2010.

Stable sharp interface method for SPH

Stable sharp interface method for SPH
Institute of Applied Physics and Computational Mathematics, Beijing, China
zhang_mingyu@iapcm.ac.cn

Abstract: Interface is a key issue in the study of multiphase flows existing in our daily life and industrial applications. Many mesh methods have been developed to treat the interface, including Level set method (LSM), volume of fluid (VOF), front tracking method (FT), and phase field method, et al. As a meshfree method, Smoothed Particle Hydrodynamics (SPH) can easily handle complex flows with interface.

In Zhang and Deng's previous work [1], a sharp interface method (SIM) based on level set method and ghost fluid method is presented for SPH. The level set method is used to describe the interface. Based on level set method, the calculation of the interface normal vector and curvature has higher performance. And the ghost fluid method is introduced to handle the discontinuity. The interface states are obtained by using of the jump conditions at the interface and are extended to the ghost fluid particles. The ghost fluid method improves the stability and smoothness of calculation around the interface.

Based on the Lagrangian nature of SPH, a new method for locating the interface is presented. Because the Lagrangian nature of SPH guarantees mass conservation of each phase around the interface, the new interface method for SPH is more efficient and stable.

In this talk, the new developed interface method for SPH is presented. The numerical behavior of the new method is shown by benchmark tests. The new method and its future version can be used to study the complex flows.

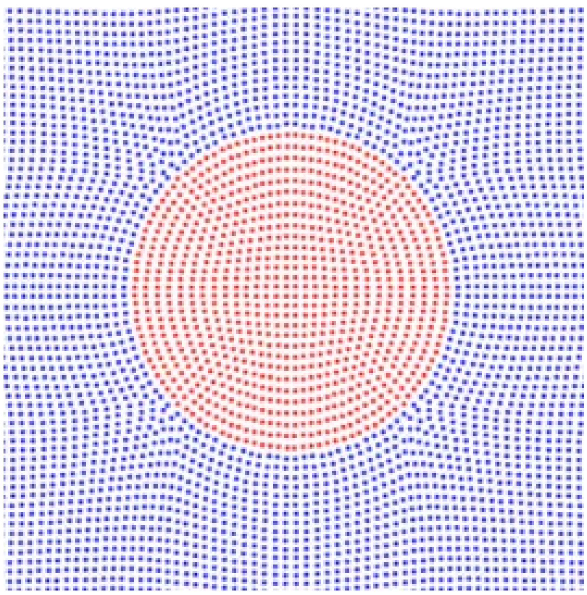


Figure 1: Square drop at stable state.

References

[1] Mingyu Zhang and Xiao-Long Deng. A sharp interface method for sph. *Journal of Computational Physics*, 302:469–484, 2015.

Aircraft tire water spray simulation using SPH

Yongkang HU†[1], Yingfei RONG[1], Dexin LENG[1], Fei XU[2], Xiangyang GAO[2], Rengang CAO[3], Wei DING[3], and Jun LV[3]
[1]Triangle Tire R&D, China
[2]Northwest Polytechnical University, China
[3]COMAC, China
huyongkang@triangle.com.cn†

Abstract: The sprays produced by aircraft tire running in water are complex and depends on the aircraft speed, tire shape and load, also the water depth. The serious consequence of water spray ingestion is the loss of engine power which will impact on the take-off and landing operations.

The present paper aims to investigate the chine design parameters of aircraft tire in function of water spray performance. Detailed aircraft tire construction models are being built with different chine design parameters using Abaqus/Explicit. The interaction between the aircraft and water is modeled using general contact of SPH elements on tire structure elements.

The numerical simulations at different loading and aircraft speeds are performed with different chine design models. Figure below illustrates clearly the impact of chine shape on water spray pattern, position and speed.

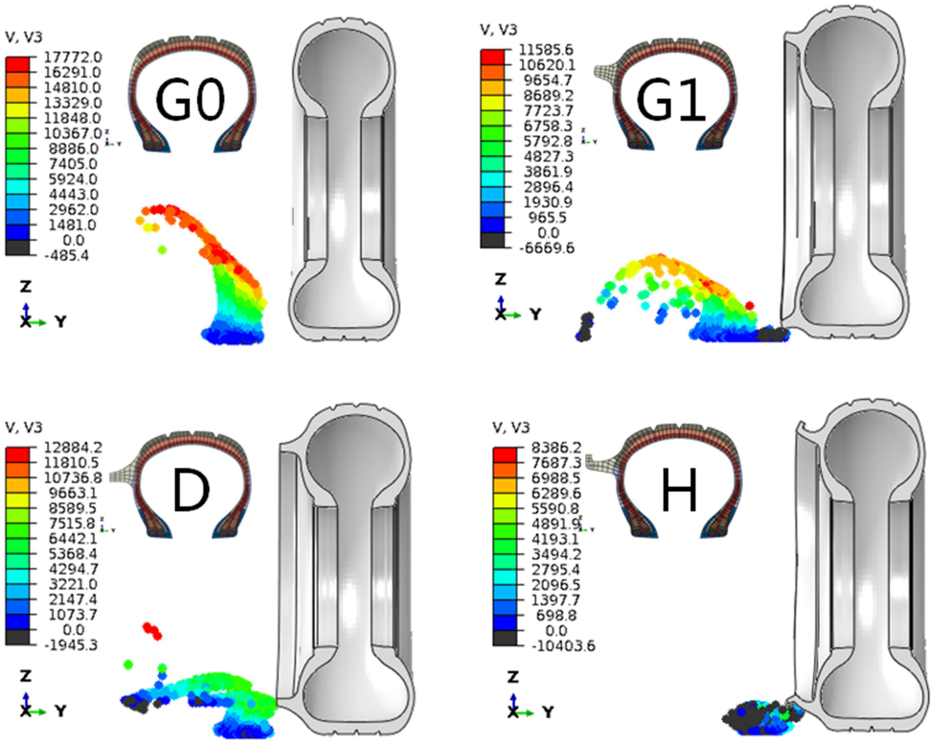


Figure 1: water spray pattern at 90 knots (Loads = 22266.4 N, Pressure = 0.869 MPa)

References

[1] DJ Mitchell. Esdu data item 83042 estimation of spray patterns generated from the side of aircraft tyres running in water or slush.

Numerical Simulation of the Damage of Multi-floor Buildings by Conical Projectile With SPH Method

Qiang Hong-fu, Sun Xin Ya, Chen Fu-zhen, and Zhang Guo Xing
Department of Power Engineering, Xi'an Hi-Tech Institute, China)
1430167246@qq.com

Abstract: In the modern war, it is an important means to win the war by effectively breaking the enemy important military facilities, such as the command center of the ground floor. Based on SPH method with fully variable smoothing lengths, the HJC constitutive model and SDPH method are used to deal with the deformation and damage of concrete slab under impact load and the free scattering of dead concrete particles, and the numerical simulation of the process of striking the multi-floor building. The damage degree of multi-floor buildings under different speed and penetration angles is analyzed, and the damage calculation and evaluation model of high-speed penetration of multi-storey buildings is established. Through the theoretical analysis, it is reasonable to show that the numerical simulation results are reasonable, which has some guiding value for predicting the impact performance of the missile and the effective fire strike.

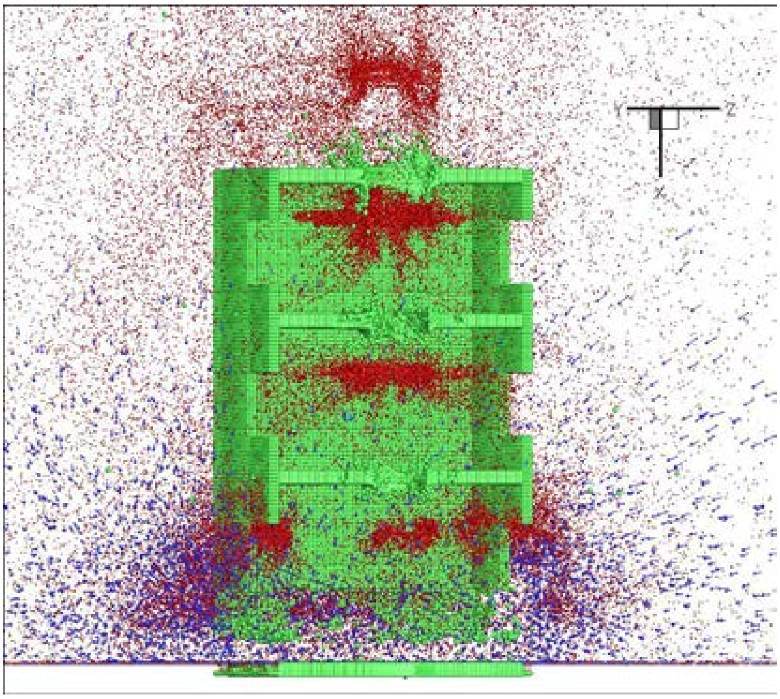


Figure 1: Collapsing of multi-floor buildings after penetrated and impact

References

[1] MB Liu, GR Liu, KY Lam, and Z Zong. Meshfree particle simulation of the detonation process for high explosives in shaped charge unlined cavity configurations. *Shock Waves*, 12(6):509–520, 2003.

[2] Larry D Libersky, Albert G Petschek, Theodore C Carney, Jim R Hipp, and Firooz A Allahdadi. High strain lagrangian hydrodynamics: a three-dimensional sph code for dynamic material response. *Journal of computational physics*, 109(1):67–75, 1993.

Corrected smoothed particle hydrodynamics for simulating failure progress of model-scale ice

Xing ZHENG[†1], Ningbo ZHANG[1], and Qingwei MA[2]
[1]Harbin Engineering University, China
[2]City University London, UK
zhengxing@hrbeu.edu.cn[†]

Abstract: A corrected SPH method based on simplified finite difference interpolation method (SPH_SFDI) is presented to simulate the failure process of ice. The Drucker-Prager model is applied into the SPH code to simulate four point bending and uniaxial compression failure of sea ice in model scale. To validate the SPH_SFDI method, the numerical results of SPH_SFDI are compared with SPH and experimental results. The good agreement with experimental data has demonstrated that the presented SPH_SFDI procedure can be a useful numerical tool for the simulation of failure progress of ice.

SPH_SFDI model which including the use of the elasto-plastic cohesion softening model is presented to simulate the bending and compression failure process of mode-scale ice. The predicted force in four-point bending and axial stress of the uniaxial compressive test are in good agreement with the experimental results. The predictions of fracture pattern are reasonable and acceptable. The conducted studies confirmed that the SPH_SFDI method can be effectively used simulate physical destruction phenomena which occur during the failure process of ice. According to the comparisons between the numerical results conducted by SPH and SPH_SFDI methods, and the experimental data, the performance of SPH_SFDI is found better satisfactory than that of original SPH in view of accuracy and stability for simulating the bending and compression failure process of mode-scale ice.

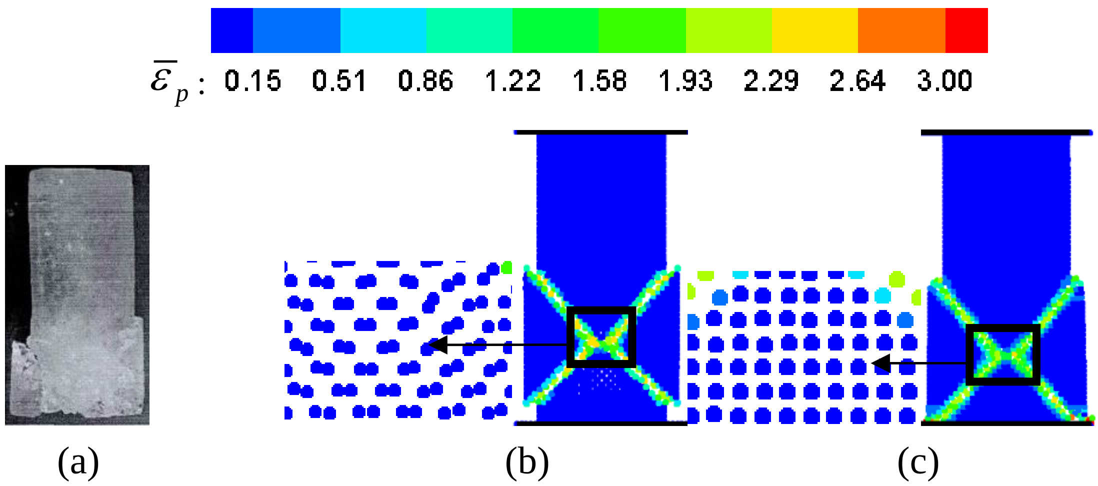


Figure 1: Comparison of the SPH (b) and SPH_SFDI (c) simulation (contours of accumulated plastic strain) with a typical experimental bulge fracture pattern (a).

References

[1] ZHENG Xing, ZHANG Ningbo, and MA Qingwei. CSPH for simulating failure progress of ice, proceeding of isope. *Proceeding of ISOPE, San Francisco (USA)*, pages 25–29, 2017.

Numerical study of the mechanism of explosive/impact welding using an improved SPH method

M. B. Liu[†1,2] and Z. L. Zhang[1]
[1]BIC-ESAT, College of Engineering, Peking University, Beijing 100187, China
[2]State Key Laboratory for Turbulence and Complex Systems, Peking University, Beijing 100871, China
mbliu@pku.edu.cn[†]

Abstract: Explosive welding (EXW) involves processes like the detonation of explosive, impact of metal structures and strong fluid-structure interaction, while the whole process of explosive welding has not been well modeled before [1]. In this paper, a novel smoothed particle hydrodynamics (SPH) model is developed to simulate explosive welding [2, 3]. In the SPH model, a kernel gradient correction algorithm is used to achieve better computational accuracy. A density adapting technique which can effectively treat large density ratio is also proposed. Typical phenomena in EXW such as the wavy interface, jetting formation, temperature and pressure distribution at the interfaces and melting voids are investigated by the present SPH simulations (shown in Fig. 1, Fig. 2 and Fig. 3), which are usually difficult for grid based methods. The mechanisms of wave formation are investigated, specially, two well-known mechanisms namely, the jet indentation mechanism and the vortex shedding mechanism are studied with the present simulations. Based on the well captured interfacial morphologies, the weldability windows for the impact welding (IMW) are given and are compared with the experimental and theoretical results. Furthermore, the weldability windows for EXW with respect to explosive quantity and initial welding angle are obtained. Meanwhile, welding limits and effective explosive quantity for EXW are discussed in detail.

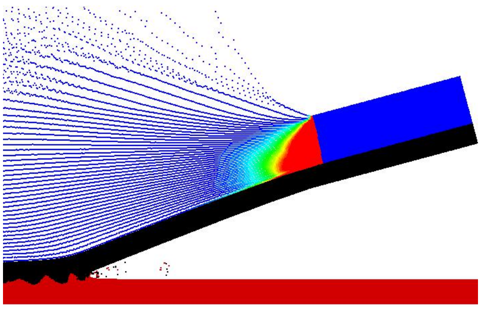


Figure 1: Spread of the explosion wave in the explosives.

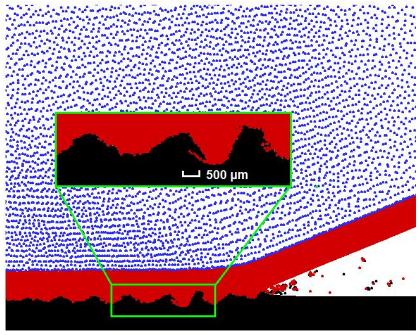


Figure 2: Wavy interface and jetting produced by the EXW.

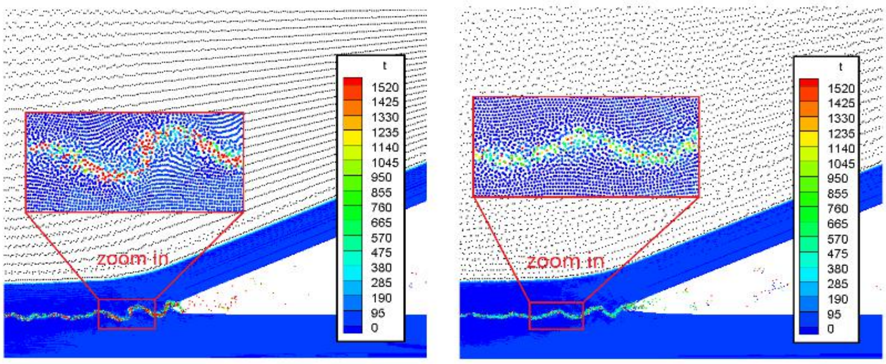


Figure 3: Temperature distribution in EXW with $M_{\text{explosive}}/M_{\text{plate}} = 0.1596$ (left) and 0.1915 (right), the welding angle is 15°.

References

[1] AA Akbari Mousavi and STS Al-Hassani. Numerical and experimental studies of the mechanism of the wavy interface formations in explosive/impact welding. *Journal of the Mechanics and Physics of Solids*, 53(11):2501–2528, 2005.

[2] MB Liu and GR Liu. Smoothed particle hydrodynamics (sph): an overview and recent developments. *Archives of computational methods in engineering*, 17(1):25–76, 2010.

[3] MB Liu, ZL Zhang, and DL Feng. A density-adaptive sph method with kernel gradient correction for modeling explosive welding. *Computational Mechanics*, pages 1–17, 2017.

SPH numerical investigation of oscillating characteristics of hydraulic jumps at an abrupt drop

Diana De Padova[1], Michele Mossa[1], and Stefano Sibilla[2]

[1]Department of Civil, Environmental, Building Engineering and Chemistry, Technical University of Bari, Italy

[2]Department of Civil Engineering and Architecture, University of Pavia, Italy

Abstract: This paper shows the results of the SPH modelling of the transition from supercritical to subcritical flow at an abrupt drop based on the laboratory experiments by Mossa et al. [1]. At an abrupt drop the transition from supercritical to subcritical flow is characterised by several flow patterns depending upon the inflow and tailwater flow conditions. SPH simulations are obtained by a pseudo-compressible XSPH scheme with pressure smoothing; turbulent stresses are represented either by an algebraic mixing- length model, or by a two-equation $k - \varepsilon$ model. The numerical model is applied to analyze the occurrence of oscillatory flow conditions between two different jump types characterised by quasi-periodic oscillation, and the results are compared with experiments performed at the hydraulics laboratory of Bari Technical University. Figure 1 shows an example of an oscillation cycle between a B-type jump (a,c) and a wave jump (b,d). The purpose of this paper is to obtain a deeper understanding of the physical features of a flow which is in general difficult to be reproduced numerically, owing to its unstable character. In particular, relying on previous SPH analyses of vorticity-dominated flows [2], vorticity fields, velocity, water depth and pressure spectra downstream of the jump (fig. 2), and velocity and pressure cross-correlations can be computed and analysed.

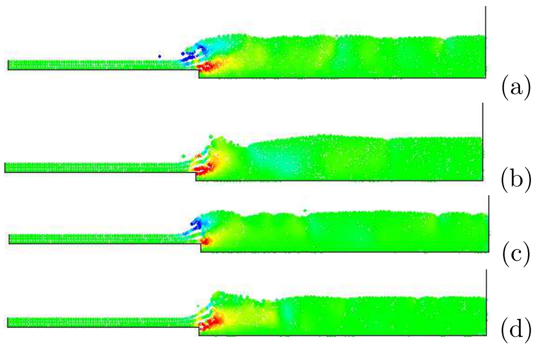


Figure 1: Instantaneous vorticity fields in the SPH simulation: a) $t = 15$ s; b) $t = 21$ s; c) $t = 26$ s; d) $t = 30$ s.

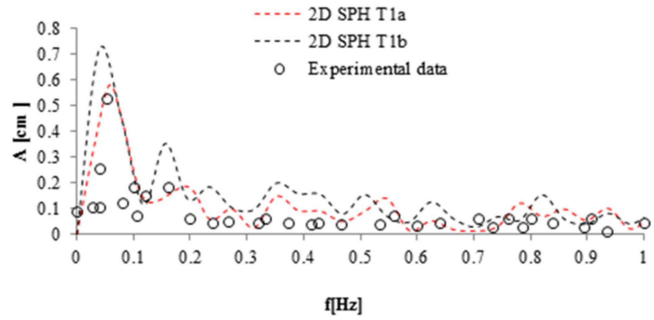


Figure 2: Amplitude spectra of pressure fluctuations under the jump for SPH simulations with ML turbulence model (T1a), with $k - \varepsilon$ model (T1b) and experiments

References

[1] Michele Mossa, Antonio Petrillo, and Hubert Chanson. Tailwater level effects on flow conditions at an abrupt drop. *Journal of Hydraulic Research*, 41(1):39–51, 2003.

[2] Diana De Padova, Michele Mossa, and Stefano Sibilla. Sph numerical investigation of the velocity field and vorticity generation within a hydrofoil-induced spilling breaker. *Environmental Fluid Mechanics*, 16(1):267–287, 2016.

An SPH simulation of bubble cavity evolution on underwater movement

Jiaru Shao[1] and Moubin Liu[†2]
[1]College of Mechanical Engineering, Chongqing University of Technology, Chongqing, China
[2]College of Engineering, Peking University, Beijing, China
mbliu@pku.edu.cn[†]

Abstract: Bubble cavities are very common in underwater movement problems. When objects move with a relatively high speed, bubble cavities will be generated, and the behaviour of moving objects will also be affected conversely. In this paper, the evolutions of bubble cavities in different cases are studied using smoothed particle hydrodynamics (SPH) method, which has special advantages in modelling free surfaces, moving interfaces and deformable boundaries. Firstly, the effectiveness of SPH method is validated by a water entry example firstly, which obtain similar results with experiment. Then the underwater movement of some objects with different velocities and shapes are researched, and the formation, growth and collapse of the bubble cavities are analyzed. It is found that the velocities, angles and shapes of the moving objects will affect the bubble cavities greatly, and the SPH model can give optimal predication of these corresponding conditions, which can decrease the impact force on the moving bodies sharply.

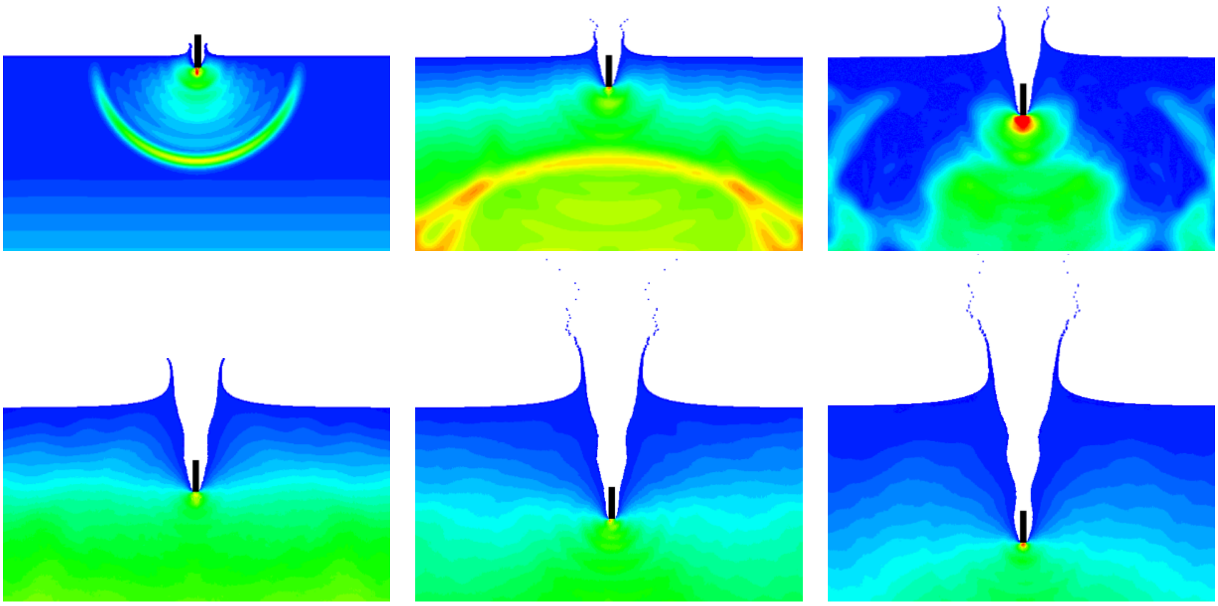


Figure 1: Pressure evolution at 0.01, 0.03, 0.06, 0.09, 0.12, 0.15 s

The δALE-SPH model: an improved δ-SPH scheme containing particle shifting and ALE formulation

P.N. Sun†[1], A.M. Zhang*[1], A. Colagrossi‡[2], S. Marrone[2], and M. Antuono[2]
[1]College of Shipbuilding Engineering, Harbin Engineering University, Harbin, China
[2]CNR-INSEAN, Marine Technology Research Institute, Rome, Italy
sunpengnan@yeah.net†, zhangaman@hrbeu.edu.cn*, andrea.colagrossi@cnr.it‡

Abstract: In the present work we derive a novel model, named δALE-SPH scheme, by merging the δ+SPH scheme [1] and the Arbitrary Lagrangian Eulerian (ALE) formulation [2]. Differently from the δ^+SPH scheme, the use of the ALE framework allows for a consistent inclusion of the Particle Shifting Technique (PST) and, consequently, for recovering the conservation of mass and linear momentum. In the proposed scheme, a diffusive term is included in the density equations to ensure a regular pressure field. Furthermore, different constraints on the mass flux equation are investigated. Indeed we discovered that the accuracy of the solution as well as the properties of the scheme strongly depend on how this equation is numerically handled.

Suitable algorithms for the numerical treatment near the free surface and on the solid wall boundary are implemented. These treatments improve the particle distribution and the pressure evaluation close to the fluid boundary. Finally, the δALE-SPH scheme is tested against several challenging benchmark test cases, proving to be more robust and accurate than the other SPH schemes.

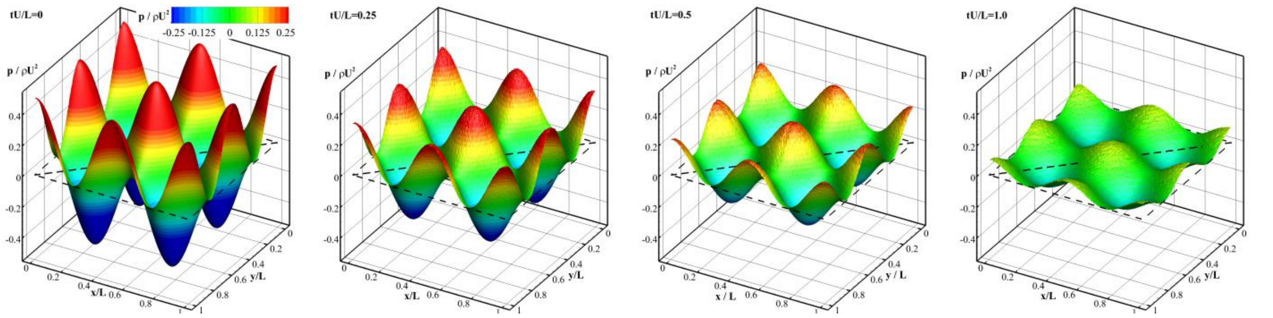


Figure 1: Pressure distribution on the fluid domain occupied by the Taylor Green vortices at four time instants when Re=100: the vertical axis shows the pressure amplitude and the two horizontal axes show the positions.

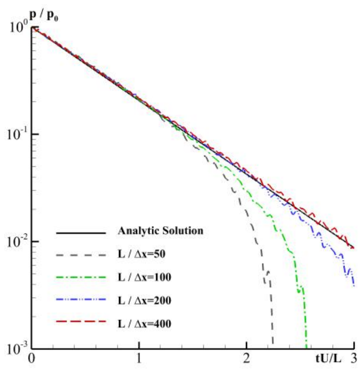


Figure 2: Time evolution of the pressure measured on the fluid center at Re = 100.

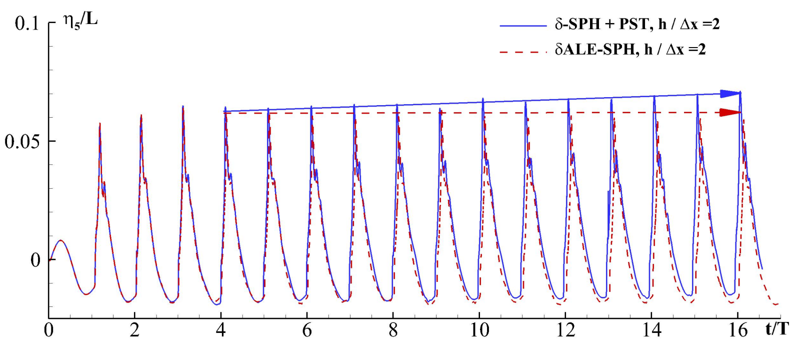


Figure 3: Time evolution of the wave height measured on a fixed position in a sloshing tank with horizontal oscillations. Results between δALE-SPH scheme and δ-SPH with PST are compared.

Taking the benchmark test Taylor-Green Vortices as example, the pressure field at different time instants are depicted in Figure 1. The absolute value of the pressure field gradually converges to zero (zero pressure is marked by the dashed plane in the four subplots in Figure 1). The pressure evolutions measured on the center of the fluid are plotted in Figure 2 for four different particle resolutions which show a fair convergence of the results to the analytic solution. Another example is the sloshing test as shown in Figure 3. The result of simply implementing

PST in δ-SPH shows a free surface rising up after a long time. The free surface rising up is mainly due to the non-conservation of momentum after the particle shifting. While in the result of δALE-SPH, the maximum height of the free surface has been almost constant, see the red arrow in Figure 3. Further numerical studies by other benchmarks will be presented in the full-length paper.

References

[1] PN Sun, A Colagrossi, S Marrone, and AM Zhang. The δplus-sph model: simple procedures for a further improvement of the sph scheme. *Computer Methods in Applied Mechanics and Engineering*, 315:25–49, 2017.

[2] G Oger, Salvatore Marrone, David Le Touzé, and Matthieu De Leffe. Sph accuracy improvement through the combination of a quasi-lagrangian shifting transport velocity and consistent ale formalisms. *Journal of Computational Physics*, 313:76–98, 2016.

SPH numerical simulation of lift-off by impact of sand particles on flat sand bed

Zhao Jie, Jin A. Fang, Maimtimin Geni, and Ma. Xiaojing
College of Mechanical Engineering, Xinjiang University, Urumqi Xinjiang 830047 China

Abstract: The study of Aeolian processes can offer insights into past and present geographies and climatic conditions. Previous models of the collision between sand particles are complex and require enormous amounts of calculation.In this paper, we use SPH method to simulate the lift-off phenomenon of sand particles in air flows which impact the surface of a grain bed. And the approach in this paper avoids in dealing with large deformation and the complexities of generating grids with complicated shapes and areas.Our study provides a better simulation method for research on the micro-movements of wind-blown sand.

The paper considers atmospheric boundary layer as a two-dimensional space,and use the Boussinesq assumptions to simplify the control equation (Bagnold 1941) ,then, using the SPH method,the whole computational domain is broken down into discrete particles using control equations .These discrete particles are consistent with natural sand particles in terms of shape. Therefore, using the SPH method has particular advantages in processing collision problems of sand particles. We carry out statistical analyses and compare our results with previous studies, and our simulated results show that the collision effect is very important for the take-off of sand particles in wind-blown sand movement and that it dictates the entire process of sand movement. The collision effect of sand particles can stir up several bigger and heavier sand particles. The simulated results demonstrate the micro-collision process more dynamically and precisely.

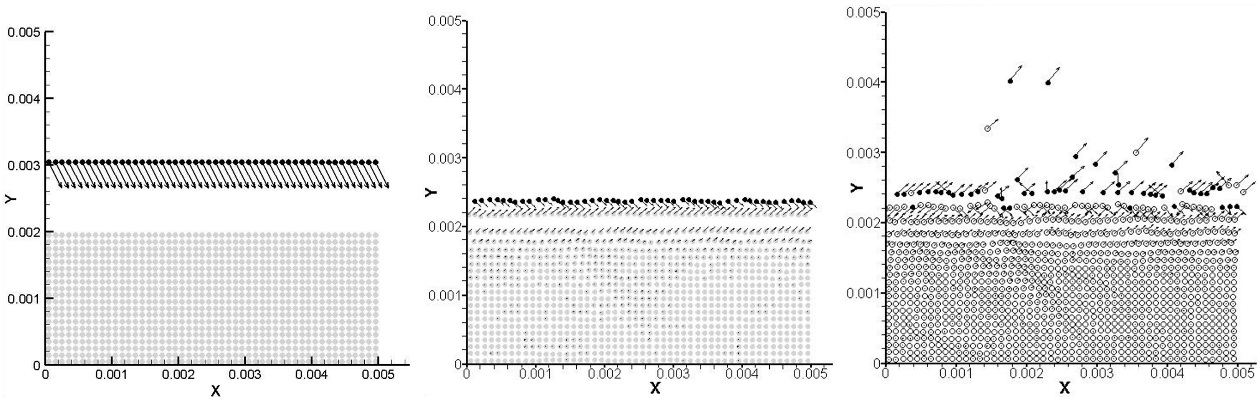


Figure 1: Dynamic process of sand impacting bed surface:initial distribution of sand particles before collision when step is 0(left).Position and velocity of sand particles at step is 4000 (middle).Velocity and direction of sand particles after collision when step is 5000 (right).

References

[1] Feng Shi and Ning Huang. Measurement and simulation of sand saltation movement under fluctuating wind in a natural field environment. *Physica A: Statistical Mechanics and its Applications*, 391(3):474–484, 2012.

Project supported by the National Natural Science Foundation of China(Grant No.11662019) and University Scientific Research Plan of Xinjiang Uygur Autonomous Region (Grant No.XJEDU2016S031)

A comparative study of SPH and MPM in modeling mixed-mode failure in rocks

Sam Raymond†, Bruce Jones, and John Williams
Geonumerics Group, Department of Civil and Environmental Engineering, MIT, USA
sjr@mit.edu†

Abstract: In this work coupled Drucker-Prager plasticity and Grady-Kipp damage models are used to simulate mode I and mode II failure in rocks, with differing computational approaches. A previous study found that this combination of constitutive models, and the smoothed particle hydrodynamics (SPH) method, compares well with experimental results [1]. Here we compare this previous approach to a new framework which takes advantage of the Material Point Method (MPM) instead. MPM is a hybrid grid-particle method, which shares many similarities with SPH and has been gaining in popularity. However, as a numerical method it is still in development and much of its abilities have yet to be fully examined. SPH, being a much more developed and more widely adopted technique, is an ideal candidate for comparison. A series of different configurations of flaws within a rock specimen are modeled using the two methods with the same coupled plasticity and damage framework. These specimens are compressed to failure and the failure paths, both tensile and shear, are tracked. The two methods are compared against experimental results and the implications are discussed.

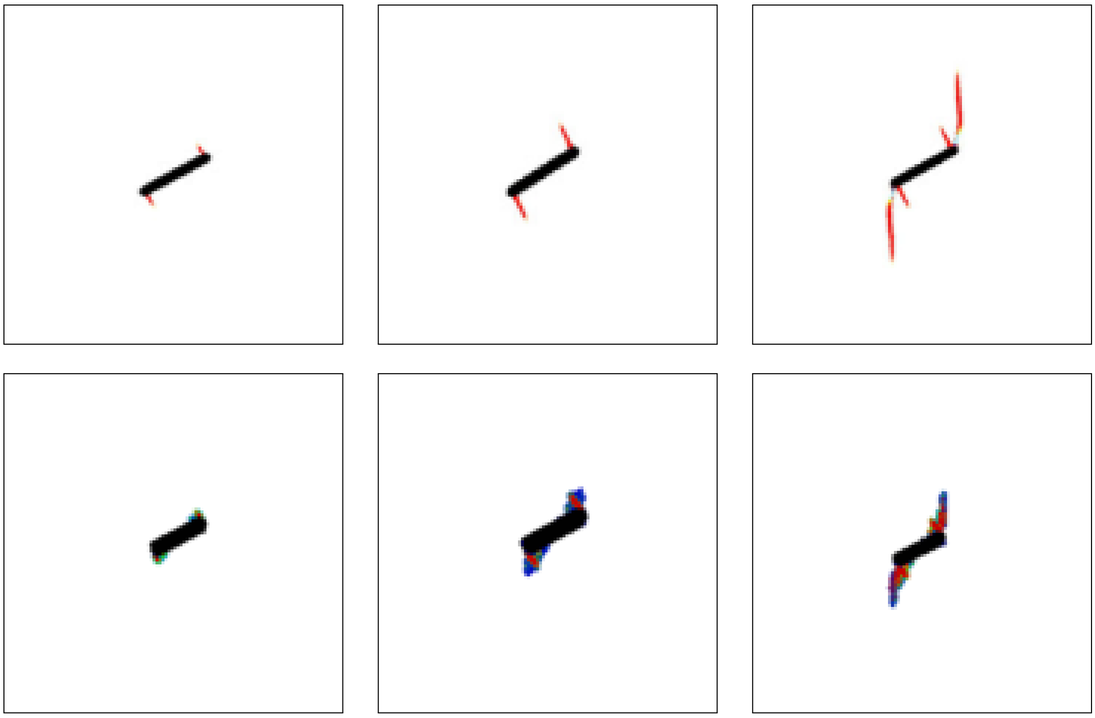


Figure 1: A single-flaw model using SPH(above) and MPM (below), colored by damage.

References

[1] Thomas Douillet-Grellier, Bruce D Jones, Ranjan Pramanik, Kai Pan, Abdulaziz Albaiz, and John R Williams. Mixed-mode fracture modeling with smoothed particle hydrodynamics. *Computers and Geotechnics*, 79:73–85, 2016.

A SPH investigation of soil plastic behaviour with Mohr-Coulomb constitutive model

Shaohan Zhao[1], Ha H. Bui[1], Vincent Lemiale[2], and Giang D. Nguyen[3]
[1]MCG Lab, Department of Civil Engineering, Monash University, Australia
[2]Data61, CSIRO, Clayton South, Vic 3169, Australia
[3]School of Civil Environmental and Mining Engineering, The University of Adelaide, Australia

Abstract: In the geotechnical engineering field, the prediction of soil plastic behaviour is essential as it signals the failure of earth structures, and eventually results in catastrophic consequences due to the very large deformation of soil. To study this phenomenon, the finite element analysis has been regarded as a standard method due to its stability and accuracy. However, the well-known mesh pathology hinders its applications to model large deformation. To resolve this problem, the mesh-free SPH method has been recently adopted and proved to be a powerful tool for solving large deformation and failure behaviour of soils without suffering from mesh related issues [1]. To further enhance the application of SPH in geotechnical area, this paper presents for the first time the numerical implementation of an elasto-plastic constitutive model with Mohr-Coulomb yield criterion in SPH. The Mohr-Coulomb constitutive model has been widely used in engineering practices to predict soil behaviour owing to its reliability and eases of specifying soil properties. However, its numerical implementation requires specific treatment to singularities at edges of the yield function. In this work, a simple approach that makes use of the Drucker-Prager yield function is proposed to facilitate the numerical implementation of the Mohr-Coulomb model in SPH. To verify the numerical implementation, a biaxial test under plane strain condition is conducted and results are compared to finite element solutions (see Figure 1). The results show that the proposed SPH numerical framework agrees well with finite element solutions for small deformation range, while being superior in the large deformation and post-failure prediction. This suggests that the combined SPH and Mohr-Coulomb model could be to study soil plastic behaviour.

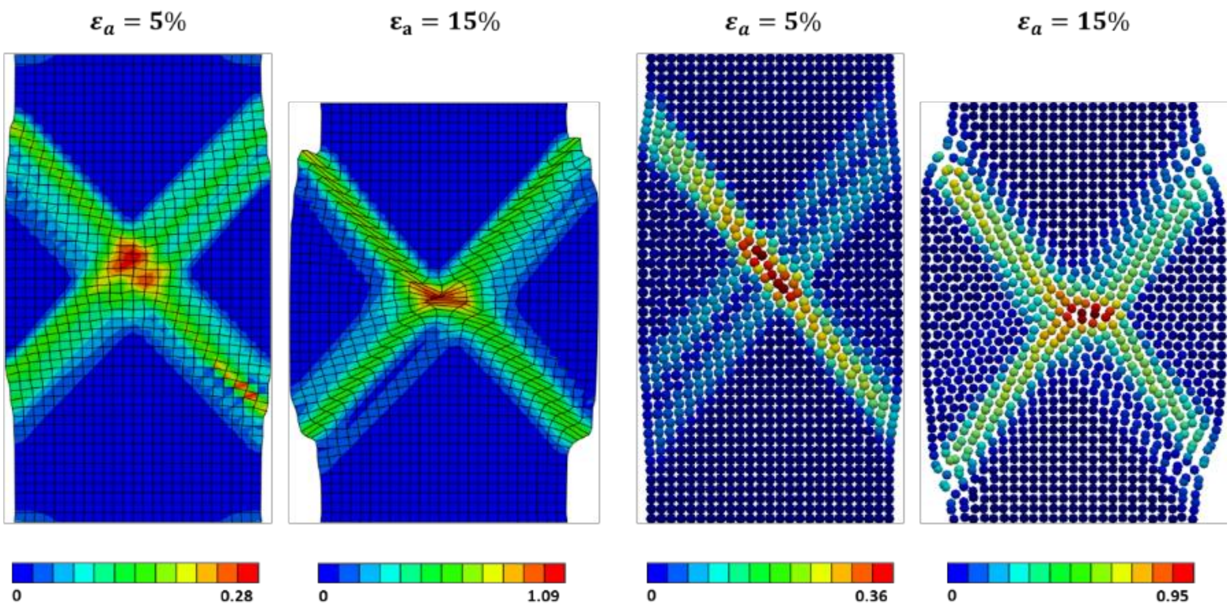


Figure 1: Biaxial test results from FEM (left) and SPH (right).

References

[1] Bui H Ha, R Fukagawa, K Sako, and S Ohno. Lagrangian mesh-free particle method (sph) for large deformation and postfailure of geomaterial using elastic-plastic soil constitutive model. *International Journal for Numerical and Analytical Methods in Geomechanics*, 2007.

A robust approach to model rock fracture with SPH

Yingnan Wang†1, Ha H. Bui*1, Giang D. Nguyen2, and P.G. Ranjith1
1Department of Civil Engineering, Monash University, Clayton, Vic 3800, Australia
2School of Civil, Environmental and Mining Engineering, The University of Adelaide, SA 5005, Australia
Yingnan.Wang@monash.edu†, Ha.Bui@monash.edu*

Abstract: Rock fractures often occur in both civil and mining engineering applications such as rock excavation and energy extraction, therefore the better understanding of rock fracture process could contribute to the reduction of both financial input into engineering projects and risks associated with construction activities. One approach to simulate rock fracture in computational models is to use the finite element method (FEM) with continuum constitutive models that relate the stress and strain through classical plasticity and damage theories [1]. However, FEM suffers from the ill-posed initial boundary condition and mesh-dependency when encountered with fracture problems. Furthermore, classical constitutive models could not possess a length scale parameter, resulting in the fact that they are unable to account for the length scale effect observed in experiments. Another approach to tackle this problem is discontinuous-based method, among which Discrete Element Method (DEM) is the most popular and widely used tools [2]. Even it has great capacity to deal with large deformation and complete detachment problem, high computational cost makes it very difficult to simulate rock fracture in real life engineering problems. In this paper, to overcome the above problems, a continuum size-dependent constitutive model with embedded cohesive fracture law is implemented into Taylor Smoothed Particle Hydrodynamics (Taylor-SPH) framework, forming a new computational approach for simulating rock fracture problems. Brazilian disc and notched semi-circular bending tests have been conducted to verify the applicability of this model. Figure 1 shows the SPH simulation results of Johnstone fracture process in semi-circular bending test, indicating that the proposed approach is capable of accurately predicting rock fracture behaviours.

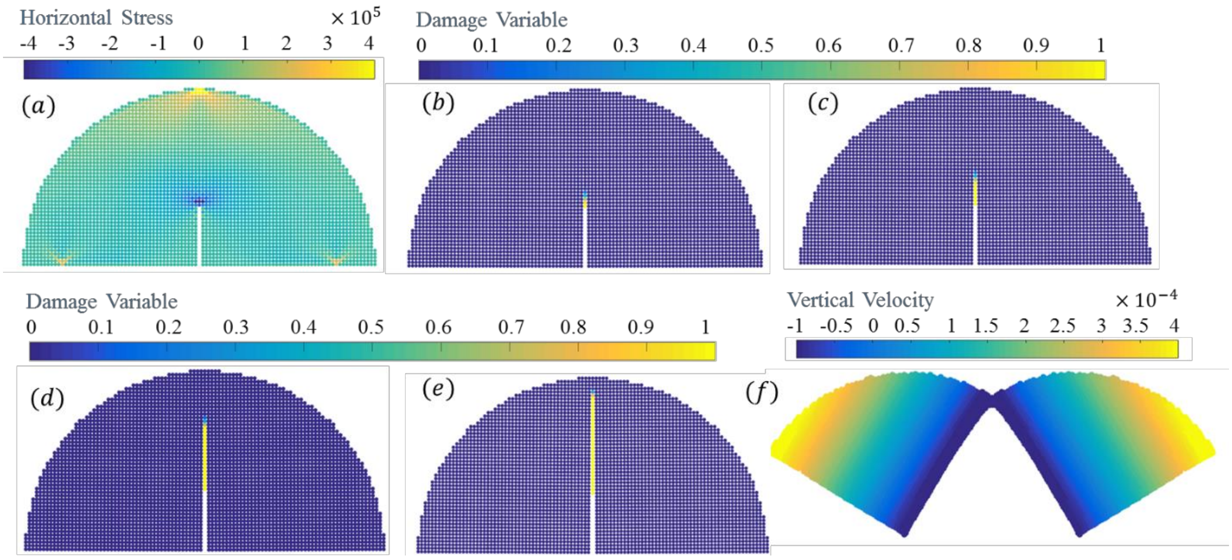


Figure 1: SPH simulation of Johnstone fracture in semi-circular bending test: (a) Horizontal stress profile before fracture; (b-e) Fracture initiation, propagation and final fracture pattern; (f) Profile of vertical velocity at the failure pattern.

References

[1] H Molladavoodi and A Mortazavi. A damage-based numerical analysis of brittle rocks failure mechanism. *Finite Elements in Analysis and Design*, 47(9):991–1003, 2011.

[2] Peter A Cundall and Otto DL Strack. A discrete numerical model for granular assemblies. *geotechnique*, 29(1):47–65, 1979.

An elasto-plastic-μ(I) SPH model for landslide induced debris flow

Wentao ZHANG†[1], Yi AN[1], and Qingquan LIU[2]
[1]Institute of Mechanics, Chinese Academy of Sciences, P.R. China
[1]Beijing Institute of Technology, P.R. China
zhangwentao@imech.ac.cn†

Abstract: In the landslide induced debris flow problem, the soil slope experiences four stages: slope instability, large deformation, debris flow and deposition. Although the SPH method has been applied in geotechnical problems such as slope instability since Bui et al. [1], the numerical simulation of this whole process, esp. which includes phase change between solid and fluid, is very few. In this study, the elasto-plastic viscous implementation with Drucker-Prager yield criterion for solid stage and μ(I) rheology for fluid stage is developed based on DualSPHysics. The yield behave of the soil is similar with Bui [1] while the after yielding flow is described with μ(I) rheology borrowed from granular flow. Comparing with Jutzi & Asphaug's model [2] which also introduced μ(I) rheology into SPH with isotropic pressure, the elasto-plastic stage is resolved more soundly in this study, especially for slope failure problems.

The model is validated with a set of laboratory dry-granular dam break experiments with different initial aspect ratio of the reservoir. Figure 1 shows the shape and velocity field of the case with aspect ratio of 2.5. Good agreement is found between the simulated results and the laboratory data on both the shape evolution and the velocity field. Moreover, the simulation could also provide the shear band distribution (Figure 1a) which is essential for understanding of the mechanism in this problem. Almost all parameters involved in the simulation have their physical meaning and could be obtained from experiment. Some artificial parameters focusing on numerical stability are chosen according to previous studies without tuning. This model could provide a powerful tool for the landslide induced debris flow study.

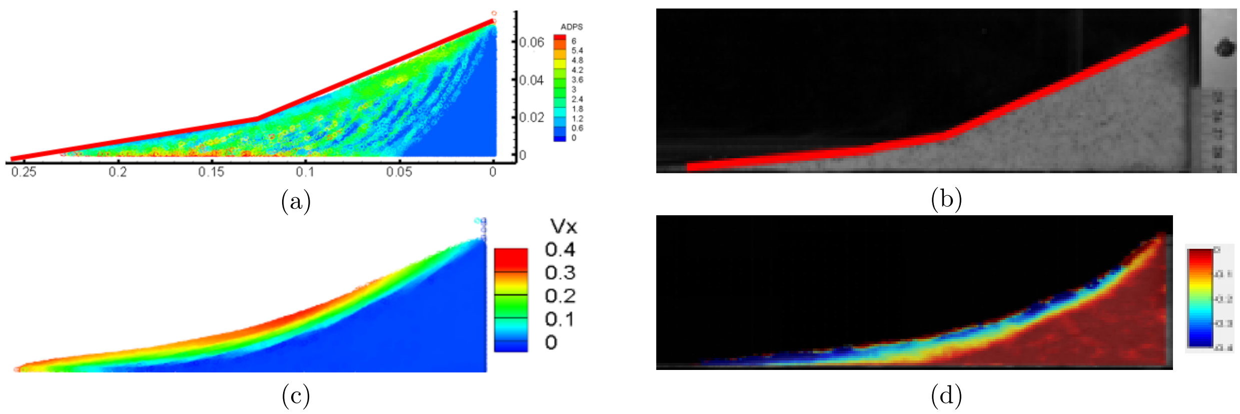


Figure 1: Comparison of shape and velocity between numerical and experimental results (a. simulated shape while the red solid line is final shape observed in experiment, b. observed shape, c. simulated x velocity, d. x velocity observed in experiment)

References

[1] Ha H Bui, Ryoichi Fukagawa, Kazunari Sako, and Shintaro Ohno. Lagrangian meshfree particles method (sph) for large deformation and failure flows of geomaterial using elastic–plastic soil constitutive model. *International Journal for Numerical and Analytical Methods in Geomechanics*, 32(12):1537–1570, 2008.

[2] Martin Jutzi and Erik Asphaug. Mega-ejecta on asteroid vesta. *Geophysical Research Letters*, 38(1), 2011.

Overview of SPH-ALE applications for hydraulic turbines in ANDRITZ Hydro

Jean-Christophe Marongiu†, Magdalena Neuhauser, Martin R ntschler, and Etienne Parkinson
ANDRITZ Hydro R&D, Switzerland
jean-christophe.marongiu@andritz.com†

Abstract: Over the past 13 years, ANDRITZ Hydro has developed an in-house tool based on the SPH-ALE method for applications in flow simulations in hydraulic turbines. The initial motivation is related to the challenging simulation of free surface flows in Pelton turbines, where highly dynamic water jets interact with rotating buckets, creating thin water jets traveling inside the housing and possibly causing disturbances on the runner. The SPH-ALE method, introduced by Vila [1], was selected because it significantly improves the stability and accuracy of weakly compressible the standard SPH method. It was further improved then. A novel treatment of boundary conditions based on surface integral terms and partial Riemann solvers was introduced [2], allowing treatment of real geometries with complex shapes. Accuracy was improved with higher order numerical schemes [3]. A coupling between SPH-ALE and Finite Volumes was developed, allowing to efficiently capturing flow features around olid bodies at a much reduced computational cost [4].

The present paper proposes an overview of industrial applications allowed by the developed tool, including design evaluation of Pelton runners and casings, transient operation surface flows in hydraulic structures and start-up of a Francis turbine. of Pelton units, free surface flows in hydraulic structures and start-up of a Francis turbine.

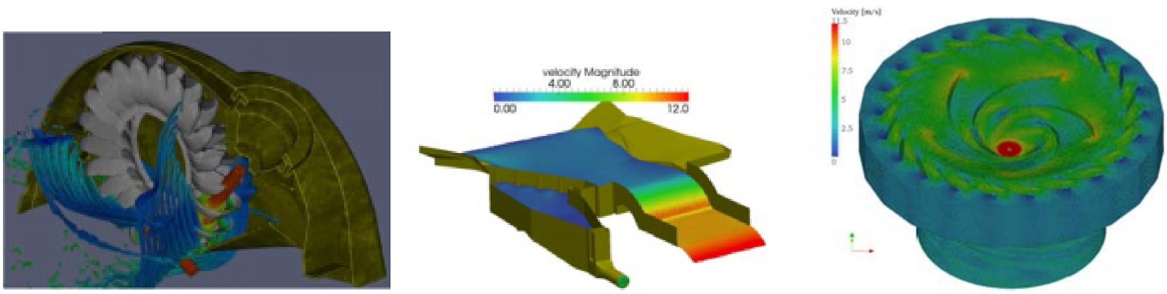


Figure 1: Left: free surface flows in an horizontal shaft Pelton casing. Center: free surface flows in a water intake. Right: confined transient flows during start-up of a simplified Francis turbine.

References

[1] JP Vila. On particle weighted methods and smooth particle hydrodynamics. *Mathematical models and methods in applied sciences*, 9(02):161–209, 1999.

[2] Jean-Christophe Marongiu, Francis Leboeuf, JoËlle Caro, and Etienne Parkinson. Free surface flows simulations in pelton turbines using an hybrid sph-ale method. *Journal of Hydraulic Research*, 48(S1):40–49, 2010.

[3] GA Renaut, JC Marongiu, and S Aubert. High-order and adaptive procedures for sph-ale simulations based on moving least squares method. In *Proceedings 10th International SPHERIC Workshob*, 2015.

[4] M Neuhauser and JC Marongiu. Coupling of a sph-ale and a finite volume method-extension to 2d and 3d. In *9th international SPHERIC workshop, Paris, France*, 2014.

Numerical and Experimental Investigation of Two Porous Wave-breaking Structures

HU Wenqing, FAN Qing, ZHAN Jiemin†, and CAI Wenhao
Sun Yat-sen University, Guangzhou, China
stszjm@mail.sysu.edu.cn†

Abstract: Compared with the traditional breakwater, floating breakwater has the advantages of simple construction and low cost. The design of the porous structure has been paid more attention because it can effectively reduce the loss of the structure by wave and maintain the wave absorbing ability. In this paper, the difference of the wave absorbing ability between two kinds of porous structures with the same porosity under the periodic wave is investigated by combining experimental measurement and SPH method. Furthermore, the computational results by SPH method have been compared with that by VOF (Volume of Fluid) method. The results show that the SPH method is suitable for the real complicated engineering application.

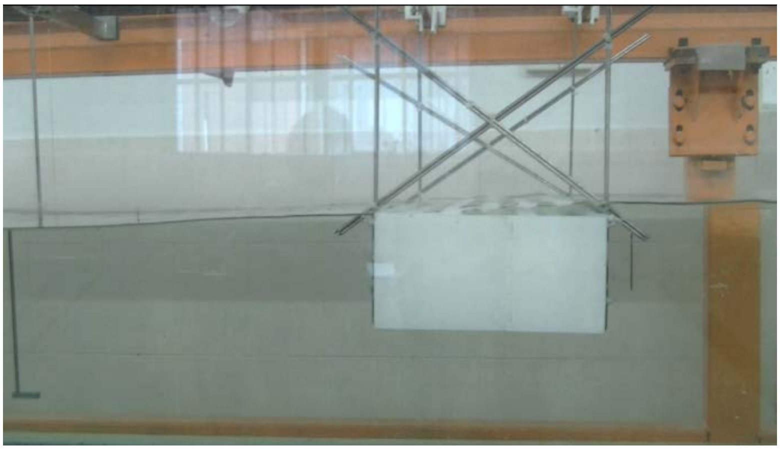


Figure 1: Wave around porous breakwater

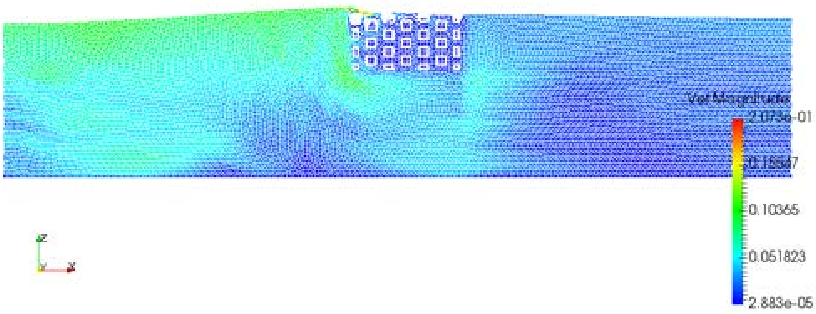


Figure 2: Velocity Field around porous breakwater (SPH)

SPH for the interaction between tsunami wave and upright cylindrical groups

Jing-jun Li†, Lei Tian, Yongsen Yang, Liu-Chao Qiu, and Yu Han
China Agricultural University, China
LiJingjunedu@163.com†

Abstract: The Smoothed Particle Hydrodynamics (SPH) method has proven to have great potential in dealing with the wave-structure interactions since it can deal with the large amplitude and easily captures the free surface. Based on the sph method, this paper verifies the accuracy of a piston-type wave maker and its generated wave force by simulating two experiments, which gives very satisfactory results. The focus of the study is the wave interaction with upright cylindrical groups. The weakening effect of the upright cylindrical group on the waves is found though the simulation by SPH. And with the increase in the number of cylindrical layers, the ability to weaken the waves is enhanced.

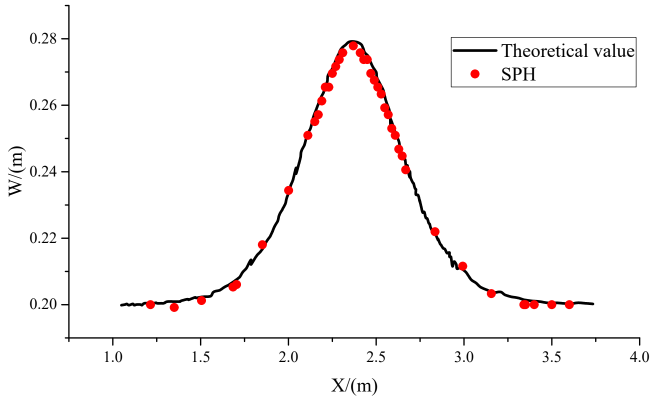


Figure 1: Validation of piston-type wave maker condition in SPH

The comparison between the theoretical and simulated values of the wave height verifies the piston-type wave maker by sph is accurate.

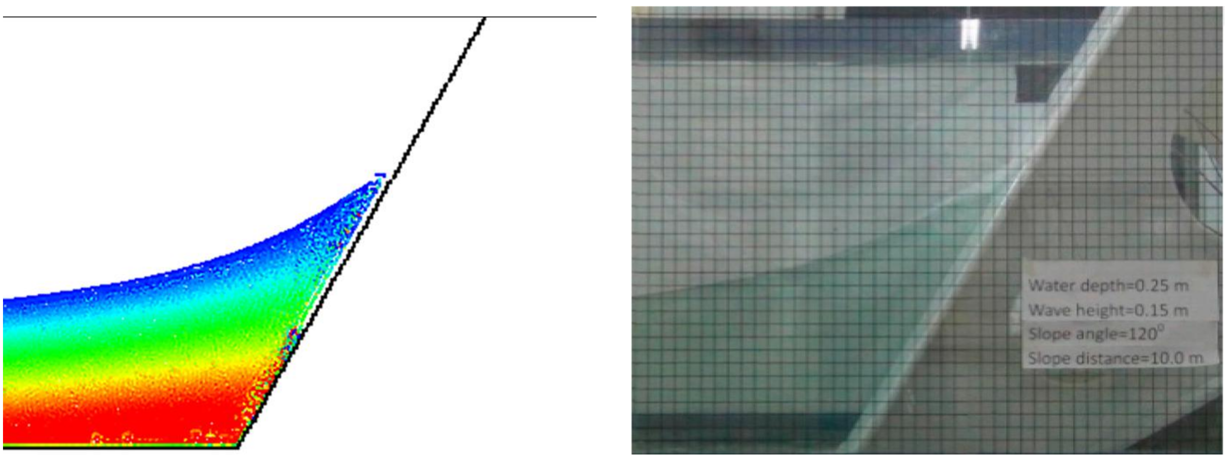


Figure 2: The interaction between waves and slopes: Comparison of SPH (left) to experimental value (right)

References

[1] Xing Zheng, Qingwei Ma, Songdong Shao, and Abbas Khayyer. Modelling of violent water wave propagation and impact by incompressible sph with first-order consistent kernel interpolation scheme. *Water*, 9(6):400, 2017.

Hydrodynamics characteristics of land hinged Oscillating Wave Surge Converter with SPH method

D. H. Zhang, Y. X. Shi†, C. Huang, Y. L. Si, B. Huang, and W. Li
Ocean College, Zhejiang University, Zhou Shan, Zhejiang province, China
yingxuan_shi@163.com

Abstract: Wave energy is an abundant and dense form of renewable energy. One of the most promising Wave Energy Converters (WECs) is the bottom hinged Oscillating Wave Surge Converter (OWSC), such as Oyster (Figure 1) which consists of a large buoyant flap hinged near the seabed. The flap oscillates back and forth under the action of the incident waves, and the kinetic energy of the flap is converted into electrical energy by pumping high pressure water ashore to drive a hydro-electric turbine. This design is good but when it is mounted on the sea bottom, several problems will appear, such as: difficulty in maintenance; corrosion by sea water; and oil leakage pollution [1]. To avoid these problems, some researchers [2, 3] designed the land hinged OWSC (Figure 2) whose hinged joints and hydraulic device can be placed above the water or on the coast. Although the land hinged OWSC is convenient in maintenance and avoids corrosion by sea water, it is still necessary to investigate its hydrodynamics characteristics which are directly related to wave power capturing efficiency.

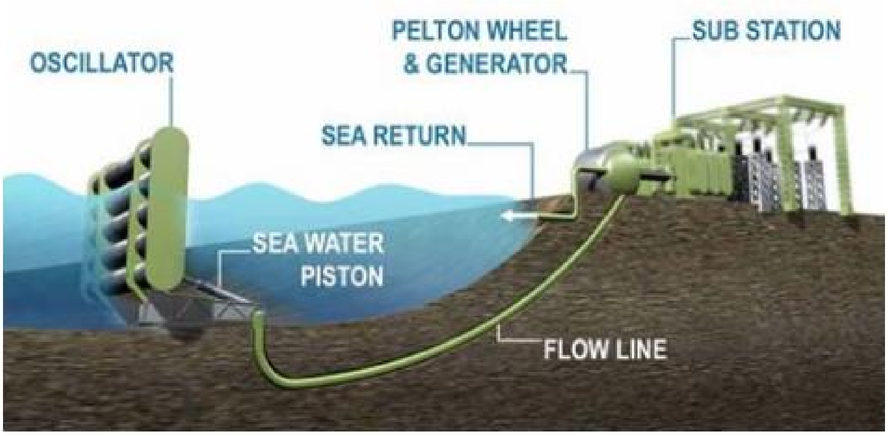


Figure 1: The sketch of Oyster

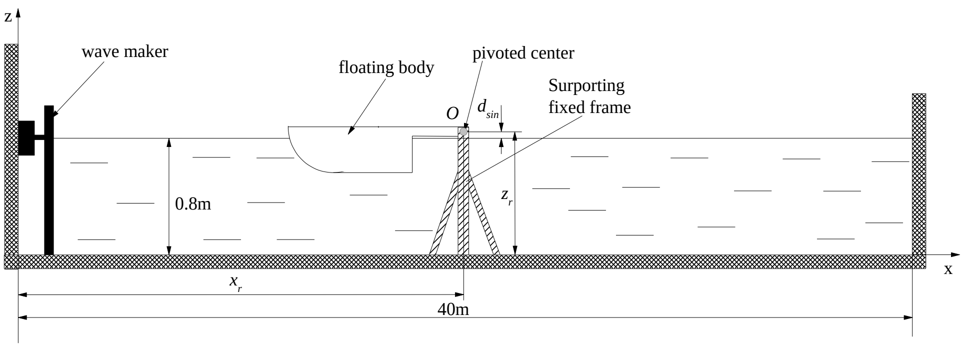


Figure 2: The sketch of single distributed absorbers

In this paper, Smoothed Particle Hydrodynamics (SPH) [4, 5] is used to the hydrodynamics characteristics of land hinged OWSC. The density diffusion model [6] is used to remove the spurious high-frequency oscillations. Boundary force model [2] is used to avoid the penetration of fluid particles across the wall. The classic equations of rigid body dynamics are used to control the motions of OWSC. Stand waves, regular waves, and bottom hinged OWSC are simulated to validate accuracy of SPH method. The results of SPH are compared with the analytical solutions or reference results, and good agreements are achieved. These results demonstrate that SPH method presented in this paper can give acceptable results in the simulations of voilent waves.

Finally, the hydrodynamics characteristics of land hinged OWSC are investigated. Figure 2 shows the single land hinged OWSC model. The piston-type wavemaker is located on the left end of NWTs. A pivoted absorber is fixed and semi-immersed. Following harmonic wave loadings, absorber swings up and down around the rotation center O. As shown in Figure 3(a) and Figure 3(b), different absorber models are taken into consideration to study the effect of geometry profile on wave energy capturing efficiency. As shown in Figure 4, two distributed absorbers with the fixed distances D_b are simulated to investigate effect of D_b on wave energy capturing efficiency. The results show that the active power of land hinged OWSC strongly depends on both the PTO damping coefficients and the wave periods. The optimized geometry profile may improve the efficiency of land hinged OWSC capturing wave energy. The distance of two distributed absorbers has important effect on wave energy capturing efficiency.

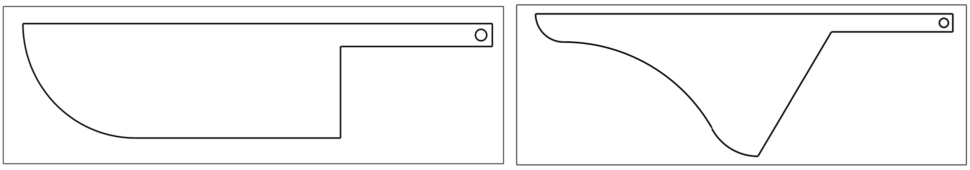


Figure 3: The geometry profile of pivoted absorber

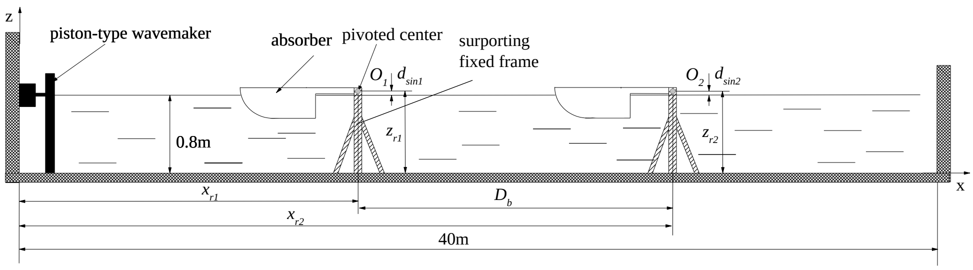


Figure 4: The sketch of two distributed absorbers

References

[1] Hoang-Thinh Do, Quang-Truong Dinh, Minh-Tri Nguyen, Cong-Binh Phan, Tri-Dung Dang, Seyoung Lee, Hyung-Gyu Park, and Kyoung-Kwan Ahn. Effects of non-vertical linear motions of a hemispherical-float wave energy converter. *Ocean Engineering*, 109:430–438, 2015.

[2] Rico H Hansen, Morten M Kramer, and Enrique Vidal. Discrete displacement hydraulic power take-off system for the wavestar wave energy converter. *Energies*, 6(8):4001–4044, 2013.

[3] Andrew Stephen Zurkinden, Francesco Ferri, S Beatty, Jens Peter Kofoed, and MM Kramer. Non-linear numerical modeling and experimental testing of a point absorber wave energy converter. *Ocean Engineering*, 78:11–21, 2014.

[4] MB Liu and GR Liu. Smoothed particle hydrodynamics (sph): an overview and recent developments. *Archives of computational methods in engineering*, 17(1):25–76, 2010.

[5] Joe J Monaghan. Smoothed particle hydrodynamics. *Reports on progress in physics*, 68(8):1703, 2005.

[6] Salvatore Marrone, M Antuono, A Colagrossi, G Colicchio, D Le Touzé, and G Graziani. δ-sph model for simulating violent impact flows. *Computer Methods in Applied Mechanics and Engineering*, 200(13):1526–1542, 2011.

[7] Ya-Wei Han, Hong-Fu Qiang, Jiu-Ling Zhao, and Wei-Ran Gao. A new repulsive model for solid boundary condition in smoothed particle hydrodynamics. *Acta Physica Sinica*, 62:221–229, 2013.

The study on SPH method with space variable smoothing length and its applications to multi-phase flow

Shi Wenkui, Shen Yanming, and Chen Jianqiang†
China Aerodynamics Research and Development Center, Sichuan Mianyang 621000, China
helloswk@126.com†

Abstract: In the past, most studies about variable smoothing length were aimed to improve computation accuracy. Smoothing length is connected to density, varying with time. Besides, most of its applications were focused on astrophysics and detonations [1].

This paper aims to improve the computational efficiency for solving multiphase flow by using space variable smoothing length, while the numerical accuracy is still held. Firstly, an initial particle diffusion distribution model is adopted. Then, in order to ensure the particle numbers in support domain for each particle almost unchanged, different particles are assigned to an independent smoothing lengths and masses. Meanwhile, some techniques are introduced to keep the symmetry of particle interactions.

Main results are presented in Figure 1. In air bubble rising case, the bubble shapes and position at different times obtained by irregular particle distribution are consistent with those achieved by regular particle distribution. In addition, in asymmetric wedge water entry case, the impact acceleration computed by irregular particle distribution with variable smoothing length agrees well with that achieved by regular particle distribution. It's worth noting that the particle numbers of both cases are decreased by about 1/4 compared to regular particle distribution, and the computation times are reduced by 25% (Table 1). It indicates that this method is suitable to simulate the problems of complex engineering such as three-dimensional multi-phase flow and water impact.

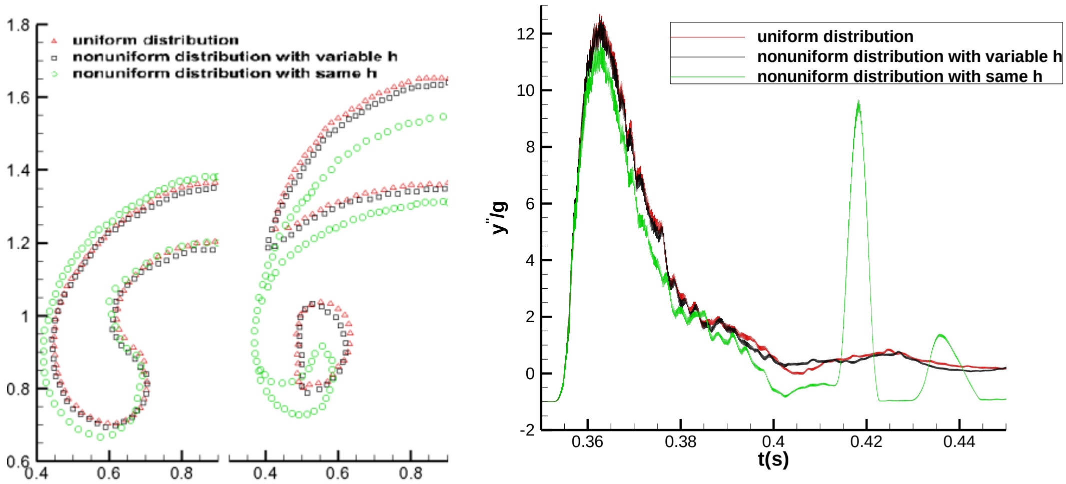


Figure 1: Left: Air bubble rising ($t\sqrt{g/R} = 3.2,\ 4.8$); Right: Wedge water entry.

Table 1: Comparison of computational efficiency

models / applications	Irregular particle distribution	Regular particle distribution	Particle ratio	Time ratio
Air bubble rising (1000 time steps)	37.56 s	50.59 s	19120:24940	75.4%
Wedge water entry (100 time steps)	38.99 s	50.86 s	199029:265277	73.6%

References

[1] Hong-fu QIANG, Kun-peng WANG, and Wei-ran GAO. Numerical simulation of high explosive detonation process using sph method with fully variable smoothing lengths. *Chinese Journal of Energetic Materials*, 1:012, 2009.

A dynamic refinement strategy in SPH for simulating the water entry of an elastomer

Lu Wang, Fei Xu†, and Yang Yang
School of Aeronautics, Northwestern Polytechnical University, 710072, Xi'an, P. R. China
xufei@nwpu.edu.cn†

Abstract: As a fully Lagrangian particle method, SPH's major advantage for the FSI (Fluid-Structure Interaction) issue is that the highly nonlinear behavior of the motion of the interface can be implicitly captured with a sharp interface. Naturally, extensive computation is needed due to the massive particles used to capture the interface precisely. In the Eulerian and Lagrangian mesh methods, the variable resolution that preserves high computational efficiency and accuracy has been implemented in both structured and unstructured meshes. Similarly, achieving variable resolution has been an important step in the development of the SPH method. It can not only describe the physical field more minutely but also decrease the computational cost because of the finer spatial resolution and the local refinement. To decrease the cost and to capture the interface between the fluid and structure, this paper provides a general dynamic refinement strategy including refinement pattern and relevant parameters as shown in Fig. 1, and a new refinement criterion aiming at the FSI problems is proposed to catch the interface more exactly.

When we simulate a FSI problem by SPH method, the structure is generally treated as a rigid body rather than an elastomer. There is not much research on interaction between elastomers and fluid in SPH. Therefore, we apply the previous proposed dynamic refinement strategy to the water entry of an elastic beam. By measuring the energy, acceleration and surface pressure of the elastic beam, the results indicate that this new strategy can obtain remarkable accuracy and efficiency. In addition, in order to deal with the interface accurately, we apply an improved FPM method (the developed method of SPH) to the vicinity of the interface, which shows remarkable results.

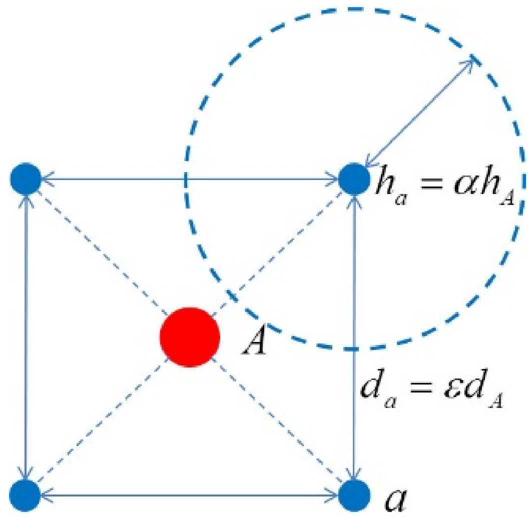


Figure 1: Splitting pattern in dynamic refinement strategy

References

[1] MouBin Liu, JiaRu Shao, and JianZhong Chang. On the treatment of solid boundary in smoothed particle hydrodynamics. *Science China Technological Sciences*, 55(1):244–254, 2012.

[2] Joe J Monaghan and Jules B Kajtar. Sph particle boundary forces for arbitrary boundaries. *Computer Physics Communications*, 180(10):1811–1820, 2009.

[3] Martin Lastiwka, Nathan Quinlan, and Mihai Basa. Adaptive particle distribution for smoothed particle hydrodynamics. *International Journal for Numerical Methods in Fluids*, 47(10-11):1403–1409, 2005.

[4] J Feldman and J Bonet. Dynamic refinement and boundary contact forces in sph with applications in fluid flow problems. *International Journal for Numerical Methods in Engineering*, 72(3):295–324, 2007.

Adaptive particle splitting in the Finite Volume Particle Method

Nathan J. Quinlan
Mechanical Engineering, National University of Ireland Galway

Abstract: The Finite Volume Particle Method (FVPM) is based on overlapping compactly support variable-mass particles (as in ALE-SPH) and employs interparticle area analogous to intercell area in the classical mesh-based finite volume method. It maintains exact conservation and zero-order consistency in non-uniform particle size distribution. This property makes it a strong candidate for implementation of adaptive particle distribution by particle splitting, which has the potential to reduce computational costs by orders of magnitude in complex problems. Adaptive particle splitting has previously been developed in SPH by several authors [1, 2, 3] and in FVPM [4].

In this paper, particle splitting will be developed for 2D FVPM, with results for various benchmark cases. Square particles are used, with the advantage that a parent may be split into four children which align with its support. It will be shown that volume, mass and momentum can be exactly conserved as a result. Field variables must be interpolated from the parent to the new particles. Analysis will be presented to show that the conservative scheme with minimum error is simply a zero-order interpolation. This introduces a first-order error at the site of splitting, which must be balanced against the reduction in error achieved in regions of refined particles.

Sample results are shown in Figure 1 for the dambreak case described by Barcarolo et al. [3]. When a particle approaches the right wall, it is split into 4 new particles of size equal to 0.55 of the parent size. The evolution of kinetic energy is shown for a static coarse particle distribution, a distribution refined by dynamic splitting, and a static fine (hence, expensive) distribution. There is excellent agreement between the results for the static fine distribution and the dynamically refined distribution with splitting. This indicates that splitting enables accuracy to be maintained or enhanced with significant reductions in computational cost.

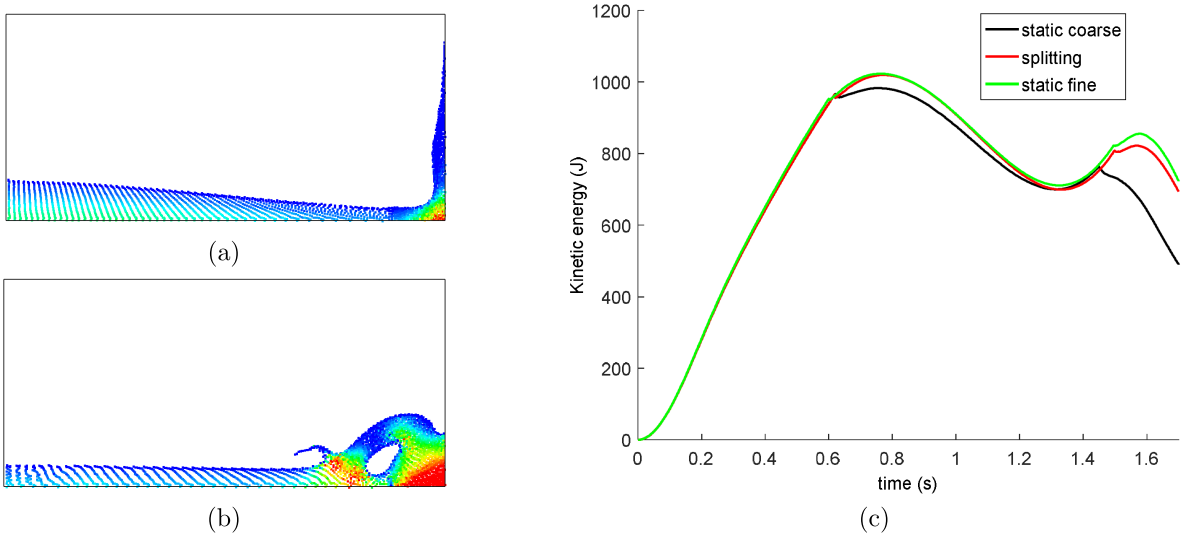


Figure 1: (a) and (b) instantaneous pressure fields in the dambreak flow. Particle splitting occurs at the dashed line. (c) Evolution of total kinetic energy.

References

[1] Martin Lastiwka, Nathan Quinlan, and Mihai Basa. Adaptive particle distribution for smoothed particle hydrodynamics. *International Journal for Numerical Methods in Fluids*, 47(10-11):1403–1409, 2005.

[2] Daniel A Barcarolo, David Le Touzé, Guillaume Oger, and Florian De Vuyst. Adaptive particle refinement and derefinement applied to the smoothed particle hydrodynamics method. *Journal of Computational Physics*, 273:640–657, 2014.

[3] Yaidel Reyes López, Dirk Roose, and Carlos Recarey Morfa. Dynamic particle refinement in sph: application to free surface flow and non-cohesive soil simulations. *Computational Mechanics*, 51(5):731–741, 2013.

[4] Ebrahim Jahanbakhsh, Christian Vessaz, Audrey Maertens, and François Avellan. Development of a finite volume particle method for 3-d fluid flow simulations. *Computer Methods in Applied Mechanics and Engineering*, 298:80–107, 2016.

Modeling the melting process of quartz glass using SPH method

Zhongyi Liu, Qianli Ma, and Haisheng Fang†
Huazhong University of Science & Technology, P.R. China
hafang@hust.edu.cn†

Abstract: There are various complex phenomena coupled with heat transfer in melting process of quartz glass, such as large surface deformation, free surface flow and fluid-solid interaction. Smoothed Particles Hydrodynamics (SPH), a mesh-free Lagrange method, has been proved an effective tool to handle such problems. The Lagrangian framework is powerful in tracing the free interfaces by adopting the mesh-free technique to overcome the time-comsuming remeshing process in the traditional grid-based CFD method.

The goal of the present paper is to extend applications of SPH method into research of quartz glass synthesis. The process is characterized by phase transition of quartz coupled with thermal transfer and highly viscous flow. The major challenging is treatment of radiative heat transfer in SPH. In the current modeling, a strategy to simplify the radiative heat transfer is developed. During melting of quartz glass, the heater is placed at the top of the furnace. Radiative heat transfers from the heater to the high-reflective crucible inner walls, and reaches the quart ingot surfaces simultaneously. Then, the quartz ingot is heated up, and the whole surface softens. As revealed in CFD modeling, temperature at the whole surface of the quartz ingot, except the bottom, rises up almost in the same time. The radiation heating is, therefore, simplified through a heat source at the ingot surface, which depends on the view factor from the surface to the heater and the crucible inner walls, evaluated by the temperature distributions calculated in CFD models.

The Fourier law in SPH format is used to calculate thermal conduction in the system. The radiation in participating media is approximated by an effective thermal conductivity from Rosseland model. Isotherm of the softening temperature (ST) is considered as the criteria of interface of fluid and solid. In the fluid region, the physical viscosity proposed by Morris et al. (1997) is used to model low-Reynolds incompressible flows, and the Monaghan artificial viscosity is applied to solve the viscous force, respectively. Temperature-dependent viscosities are derived to denote the flow characterization of molten quartz. With the proposed models, the melting process of quartz glass is predicted, which provides an optimization tool of the quartz glass synthesis.

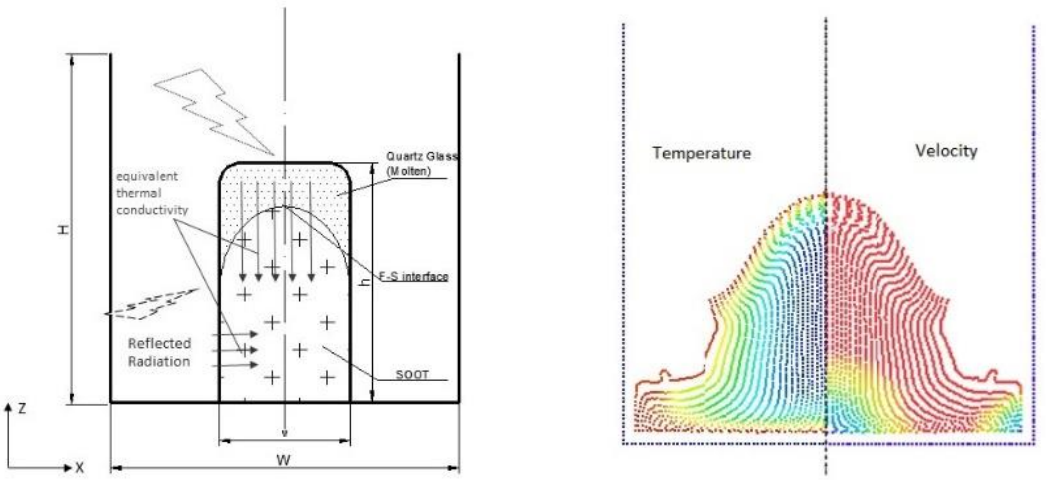


Figure 1: -Melting process simulation in SPH. Left is a sketch of melting system, and right is distribution of temperature and velocity in a moment.

Study on Dynamic Behaviors of Liquid-filled Flexible Multibody Systems under the Low-gravity Environment

Weizhen Kong† and Qiang Tian
Beijing Institute of Technology, China
weizhen_kong@aliyun.com

Abstract: There are vast kinds of liquid-filled flexible multibody systems in engineering fields, especially the aerospace field. In early studies, the coupled dynamic behaviors of the liquid-filled flexible multibody systems were mainly studied by using equivalent methods, in which the liquid is generally simplified as a rigid body system consisting of some lumped masses with springs or some pendulums, both representing the main slosh modes. However, these approaches are not able to reflect the liquid splashing phenomenon. A computation method is used to study the coupled dynamics of a partially liquid-filled flexible multibody system, where the liquid is modeled by using SPH method and the flexible bodies are described by using the Absolute Nodal Coordinate Formulation (ANCF). The fluid behavior is quite different in the absence of gravity since the inertia force is not the dominant force. The CSF model is used to introduce the surface tension force into the momentum equation.

A simulation is presented: a liquid-filled spacecraft with flexible manipulator arms grabbing a liquid-filled container. As shown in Figure 1, all the flexible components are described by element of ANCF and all rigid bodies in the system are modeled via the Natural Coordinates Formulations (NCF), like the mechanical claw and the handle connected to the container. SPH particles are used to describe the liquid and virtual particles are introduced for the coupling between finite elements of ANCF and particles of SPH.

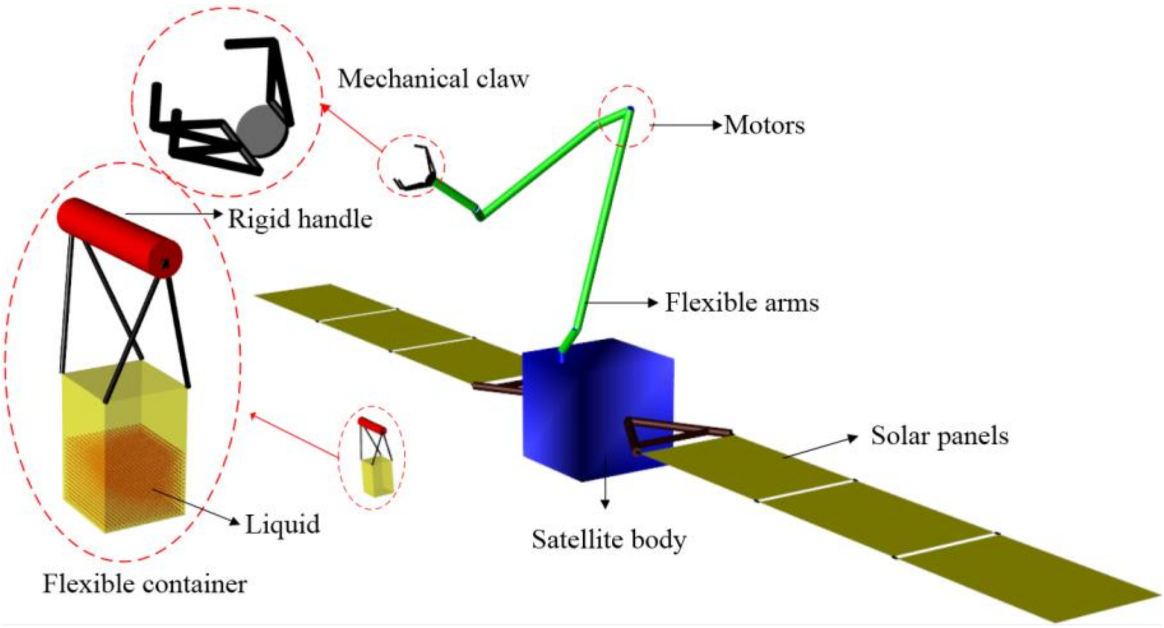


Figure 1: Liquid-filled spacecraft with flexible arms

A SPH model for the root system of plants

Matthias MIMAULT†1, Lionel DUPUY[1], and Mariya PTASHNYK[2]
[1]James Hutton Institute, UK
[2]University of Dundee, UK
matthias.mimault@hutton.ac.uk†

Abstract: Feeding more than 9 billion people will require a more reasonable use of water fertilizers and a good understanding of the mechanics of plant growth in soil. Unfortunately, experimenting biology and physics of growth in soil is difficult because live in situ observation of roots is extremely limited. Methods for measurement of roots are often destructive and time consuming, and knowledge of the process limiting growth in soil is lacking [1]. Mathematical modelling proved a useful approach to bridge this knowledge gap, but current models fail to incorporate the role of cellular processes at the organ level [2]. The objective of this work is to develop a cellular model for root growth solved at the macroscopic level using the Smoothed Particle Hydrodynamics method (SPH).

Our SPH model is based on the structural similarity of the root cells with the SPH particles, to solve macroscopic physical processes while modelling autonomous biological process at the level of the cell. We identify the cellular elongation and division as principal drivers of the growth. As the deformation of tissues occurring during growth is conducted by the Turgor pressure, we define our growth model in the framework of the poroelastic theory. Here the pore pressure is a consequence of cellular activity and subsequent chemical gradient within cells. Along with it, we include cell division with particle splitting. This division is controlled biologically, according to the chemical level distribution and located at the tip of the root, called the “meristem”. Once divided, the cells are moving away from the meristem and growth therefore is dominated by elongation. We discuss here the effect of those mechanisms in a poroelastic model and illustrate this approach with study cases. We also show how such models integrate within the SENSOIL project that combines innovative imaging systems and experiments.

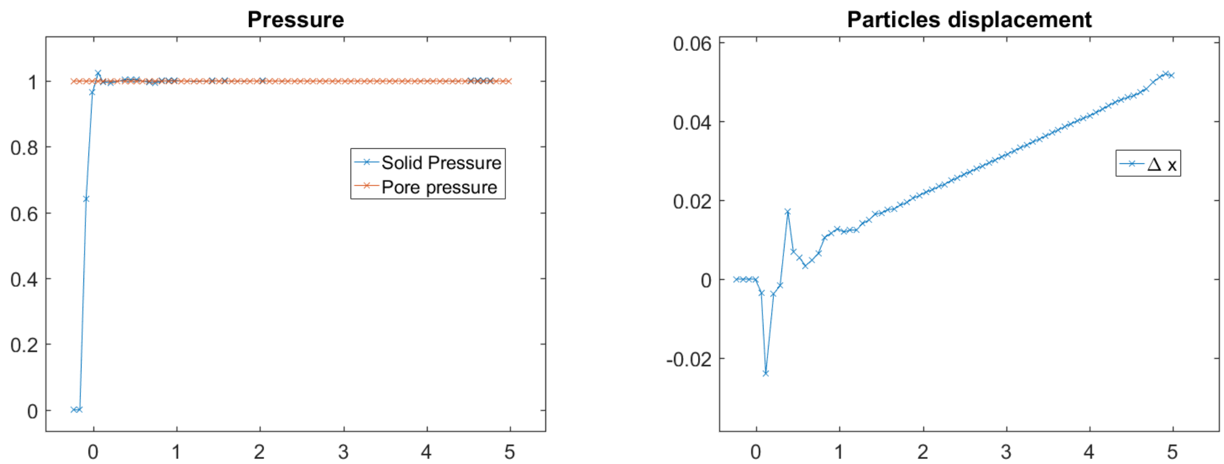


Figure 1: Pressure and displacement of particles in a 1D sample at equilibrium with imposition of a constant pore pressure and fixed left boundary. 200 particles for a cell size of 2.5 μm. Values in mm and kPa.

References

[1] Albert L Smit, A Glyn Bengough, Christof Engels, M van Noordwijk, Sylvain Pellerin, and SC van de Geijn. *Root methods: a handbook*. Springer Science & Business Media, 2013.

[2] Richard S Smith, Soazig Guyomarc'h, Therese Mandel, Didier Reinhardt, Cris Kuhlemeier, and Przemyslaw Prusinkiewicz. A plausible model of phyllotaxis. *Proceedings of the National Academy of Sciences*, 103(5):1301–1306, 2006.

SPH simulation of drop impact on a hot wall with vaporization effects

Xiufeng YANG[1], Song-Charng KONG[†1], and Moubin LIU[*2]
[1]Iowa State University, USA
[2]Peking University, China
kong@iastate.edu[†], mbliu@pku.edu.cn[*]

Abstract: Drop impact on a hot solid wall is encountered in a number of industrial applications, such as spray cooling of hot surfaces and fuel spray in internal combustion engines [1, 2]. The process of drop impact on a hot wall involves heat transfer, liquid-gas interface, vaporization, and diffusion of vapor species in the gas phase. Due to the complexity of the problem, it is a challenging work to model the interaction of a droplet and a hot wall.

The purpose of this paper is to present an SPH method for drop impact on a hot wall. By introducing an evaporation model to SPH, the phase change and mass transfer across the liquid-gas interface are simulated. The conservation equation of vapor species is used to simulate the diffusion process of the vapor species in the gas phase. Due to phase change, the mass of an SPH particle at the interface is allowed to change. To avoid large mass difference between SPH particles, particle splitting and merging techniques are developed.

The numerical method is tested by simulating a droplet impacting on a solid wall at different temperatures. The results show that the drop may spread, splash, or breakup on the wall when the wall temperature is lower than the Leidenfrost temperature as shown in figure 1 a-c. However, when the wall temperature is higher than the Leidenfrost temperature [3], the drop cannot stay on the wall as shown in figure 1 d-f.

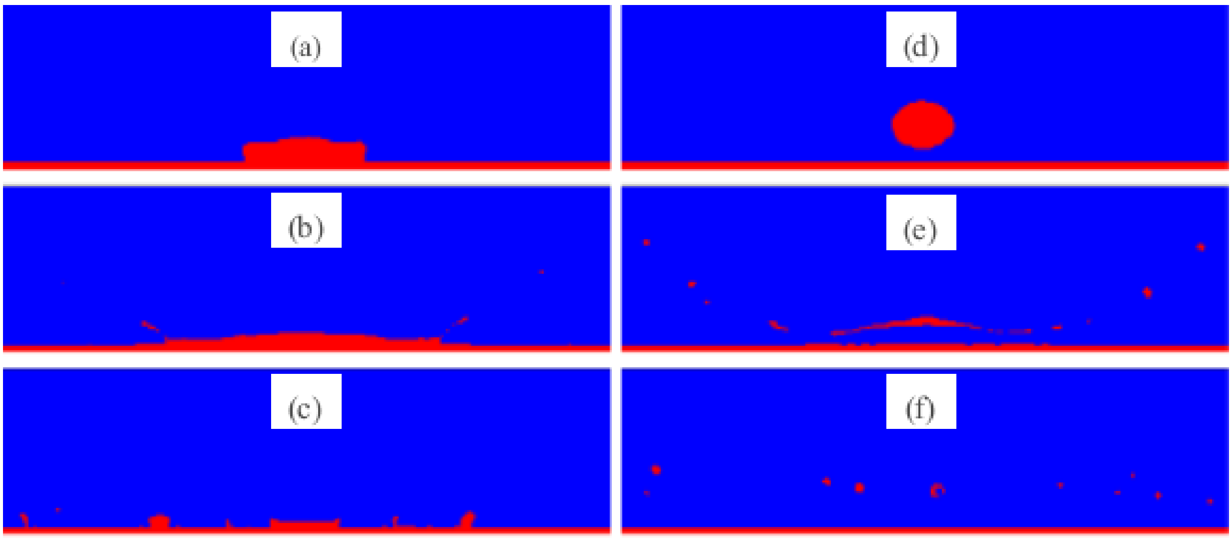


Figure 1: Different outcomes of drop-wall interaction: (a) spread, (b) splash, (c) breakup, (d) rebound, (e) and (f) Leidenfrost phenomena.

References

[1] AS Moita and ALN Moreira. Drop impacts onto cold and heated rigid surfaces: morphological comparisons, disintegration limits and secondary atomization. *International Journal of Heat and Fluid Flow*, 28(4):735–752, 2007.

[2] Xiufeng Yang, Lu Dai, and Song-Charng Kong. Simulation of liquid drop impact on dry and wet surfaces using sph method. *Proceedings of the Combustion Institute*, 36(2):2393–2399, 2017.

[3] David Quéré. Leidenfrost dynamics. *Annual Review of Fluid Mechanics*, 45:197–215, 2013.

Developing an Extensible, Portable, Scalable Toolkit for massively parallel Incompressible Smoothed Particle Hydrodynamics (ISPH)

Xiaohu Guo[†1], Benedict D. Rogers[2], Steven Lind[2], and Peter K. Stansby[2]
[1]Hartree Centre, STFC, Daresbury Laboratory, WA4 4AD, UK
[2]School of Mechanical, Aerospace and Civil Engineering, University of Manchester, M13 9PL, UK
xiaohu.guo@stfc.ac.uk[†]

Abstract: The stability, accuracy, energy conservation, boundary conditions of the projection based particle method such as ISPH [1] have been greatly improved [2]. However, for applications requiring 100s of millions of particles from the perspective of computation and high performance software implementation [3], there are still many challenges compared with other particle based methods. These may potentially hinder the use and exploitation of these methods for the large-scale real engineering applications which generally involve highly complex, nonlinear and distorted flow. Solving such applications requires a large number of particles and this in turn demands distributed computing with accelerating paradigms. The domain decomposition and dynamic load balancing using the message passing interface (MPI) for irregular particles distribution and computational domains with complex geometries are extremely challenging to implement. An appropriate assignment of particles to processors and grouping physically-close particles within a single processor can greatly reduce communication overhead and improve the software scalability. The additional distinct challenge for projection-based particle methods is solving pressure Poisson equations (PPE). The added complexity for solving sparse linear equations for projection-based particle methods such as ISPH is that the sparsity of the matrix is changing every time step due to the particle movement and hence changing connectivity of the computation points.

In this work, we present our ISPH toolkit for massively parallel simulations which aims to resolve the above challenges and facilitate the projection-based particle-methods software development. We provide all the major performance critical 'software kernels' such as the nearest neighbour list searching kernel using cell linked list approach, the domain decomposition, dynamic load balancing and particles ordering kernels using Hilbert Space Filling Curve and the PPE solver using the open-source high-performance computing (HPC) library PETSc. The innovations of the new ISPH toolkit are summarised in the following:

- Data structures designed to organise multiple kinds of fluid particles together with different boundary particles to maximise data locality and cache reuse. This enables software extensibility and communication between highly irregular subdomains.
- Domain decomposition and dynamic load balancing particularly for irregular particle distributions and computational domains with complex geometries. Figure 1 gives example of initial domain decomposition for the SPHERIC test case 2 Kleefsman's dambreaking test-case.
- Flexible parallel communication algorithms to manage efficiently specific tasks for user software development including updating halos, boundaries in the form of a single velocity vector, or new variables defined by users.
- Particle ordering in order to group physically-close particles which improves performance on cache-based computing architectures.
- Improved ISPH boundary conditions with general applicability and attractive accuracy versus performance which facilitates application development.
- Flexibility using different sparse linear solvers for different applications (PETSc) by improving the structure of the PPE matrix by preconditioning with particle reordering.

Detailed performance analysis using ISPH for each performance software kernels demonstrate our toolkit scalability (see Figure 2) of using tens of thousands cores to solve complex, nonlinear and distorted flow with ISPH. The analysis has revealed the parts of the code that are highly efficient and those that become a potential bottle neck as the number of processes increases beyond 10,000. The implementation details are intended to form future guidance for the new projection-based particle application development.

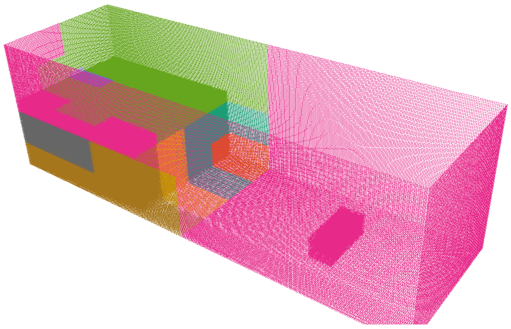


Figure 1: 3D domain decomposition with HSFC with 8 MPI Tasks, different colours represent different partitions

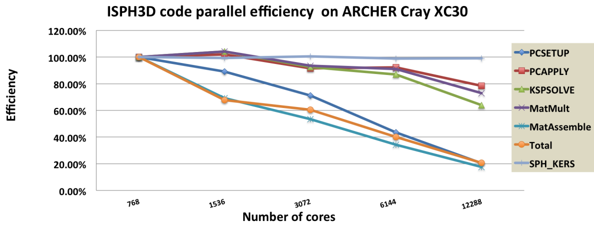


Figure 2: Efficiency Performance Analysis for ISPH3D

References

[1] SJ Lind, Rui Xu, PK Stansby, and Benedict D Rogers. Incompressible smoothed particle hydrodynamics for free-surface flows: A generalised diffusion-based algorithm for stability and validations for impulsive flows and propagating waves. *Journal of Computational Physics*, 231(4):1499–1523, 2012.

[2] Hitoshi Gotoh and Abbas Khayyer. Current achievements and future perspectives for projection-based particle methods with applications in ocean engineering. *Journal of Ocean Engineering and Marine Energy*, 2(3):251–278, 2016.

[3] Jose M Domínguez, Alejandro JC Crespo, and Moncho Gómez-Gesteira. Optimization strategies for cpu and gpu implementations of a smoothed particle hydrodynamics method. *Computer Physics Communications*, 184(3):617–627, 2013.

Three-Dimensional Sloshing Simulations by Using GPU-based MPS Method

Xiang Chen, Xiao Wen, and Decheng Wan†
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai 200240, China
dcwan@sjtu.edu.cn†

Abstract: The liquid sloshing is a complicated and nonlinear phenomenon in a partially filled tank under external excitations, which will destroy the structure of tank walls and the stability of the movement. The moving particle semi-implicit method (MPS) is a Lagrangian particle method. Because the flow field is presented by particles that contain the information of mass, velocity and so on, MPS method can simulate flow with large deformation and nonlinear fragmentation of free surface effectively.

However, MPS suffers from high computation cost with the increase of particle number. This significantly limits its applications in 3D flows which include a large number of particles. The GPU (Graphics Processing Unit) is a multi-processor designed to optimize for the execution of massive number of threads. Therefore, GPU is a better choice for high parallel MPS method. Based on modified MPS, our group developed MPS-GPU-SJTU solver by using GPU acceleration.

It is well known that the most computation time of MPS is consumed to solve PPE. One optimization strategy to reduce the storage and computation time of PPE is introduced. In addition, the convergent validation is carried out to verify the accuracy of GPU solver. And the 3-D sloshing problems are simulated by GPU and CPU solvers at the same time. The numerical impact pressure on tank walls is agreeable with experimental data. Nonlinear deformation of free surface, such as breaking wave, splashing, hitting the roof of tank and so on, can be observed. And it is also shown that the results of GPU solver show a good agreement with CPU and a large amount of computation time is reduced by GPU.

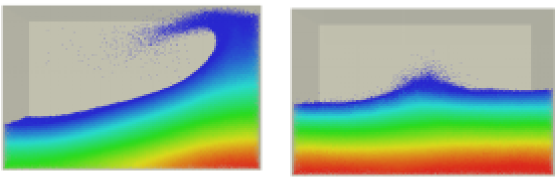


Figure 1: The flow fields of CPU

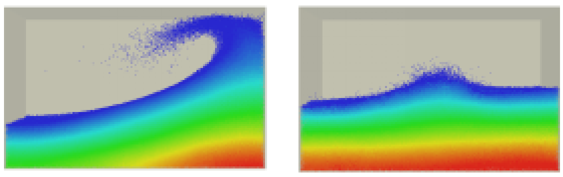


Figure 2: The flow fields of GPU

References

[1] JR Shao, HQ Li, GR Liu, and MB Liu. An improved sph method for modeling liquid sloshing dynamics. *Computers & Structures*, 100:18–26, 2012.

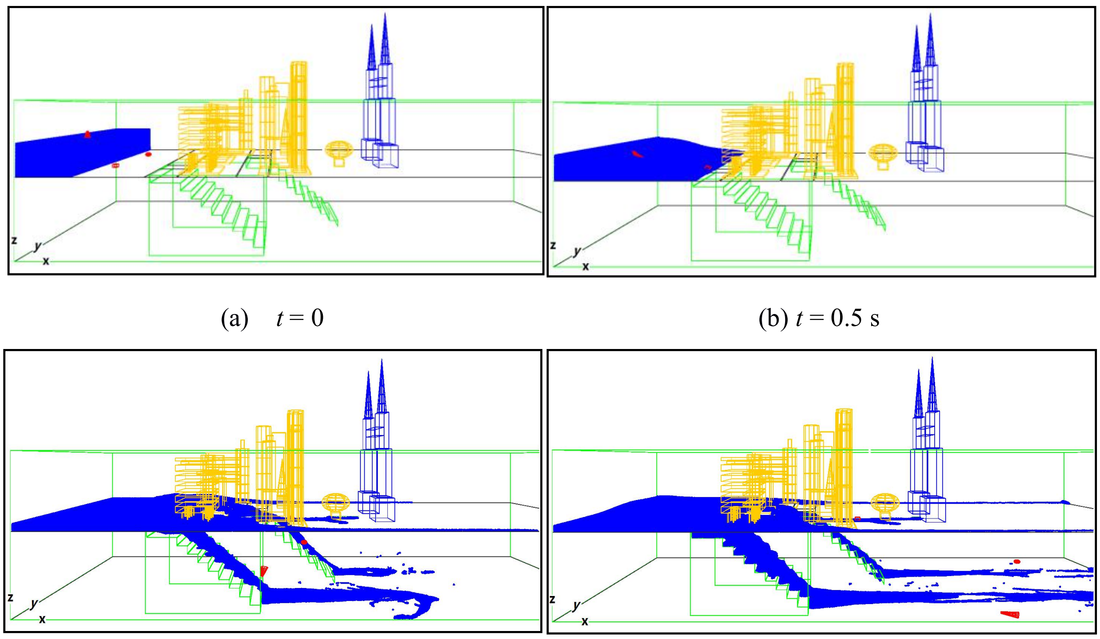
[2] Alejandro C Crespo, Jose M Dominguez, Anxo Barreiro, Moncho Gómez-Gesteira, and Benedict D Rogers. Gpus, a new tool of acceleration in cfd: efficiency and reliability on smoothed particle hydrodynamics methods. *PloS one*, 6(6):e20685, 2011.

GPU-based SPH modeling of flood with floating bodies in urban layouts including underground spaces

Jiansong Wu†[1], Na Li[1], Wenyu Liu[1], and Hui Zhang[2]
[1]School of Resources and Safety Engineering, China University of Mining & Technology, Beijing 100083, China
[2]Institute for Public Safety Research, Tsinghua University, Beijing 10083, China
jiansongwu@hotmail.com†

Abstract: Due to the "innate" low-lying weakness, underground facilities in urban areas are prone to flooding caused by the breaking of a dam or a levee, or a flash flood after an exceptional rainfall. The underground inundation may cause loss of life and very serious damage to properties. Therefore, it is very important to study the urban underground inundation in terms of hydraulic analysis and disaster prevention. An important way of clarifying the physical phenomena and determining the risk of the underground flooding flow is to numerically simulate the inundation process.

In this paper, the mesh-free Smoothed Particle Hydrodynamics (SPH) method with the Graphic Processing Unit (GPU) parallel computing technique employed (GPUSPH) is employed to investigate dam-break flooding in complex urban underground spaces. SPH explicitly conserves mass and linear momentum, and does not require explicit interface tracking treatment so that geometrically complex and moving boundaries can be handled without undue difficulty. Taking advantage of powerful GPU parallel computing ability, the following two urban flood cases involving in millions of particles (computational nodes) are simulated: i) dam-break flood with floating bodies through an intricate surface city layouts, ii) dam-break flooding immersed three floating objects into an underground facility through staircases. Numerical results fairly reproduce the complex inundation process, and reasonably present complicated underground flooding flow features involving in interactions of flooding flow with floating objects and staircases, which indicates that SPH method is an alternative tool for the modeling of flooding in complex urban underground spaces.



(a) $t = 0$ (b) $t = 0.5$ s

(c) $t = 2.0$ s (d) $t = 3.5$ s

Figure 1: Displacement of underground flooding flow with floating objects

References

[1] Eugenio Rustico, Giuseppe Bilotta, Alexis Herault, Ciro Del Negro, and Giovanni Gallo. Advances in multi-gpu smoothed particle hydrodynamics simulations. *IEEE Transactions on Parallel and Distributed Systems*, 25(1):43–52, 2014.

SPH energy balance during the generation and propagation of gravity waves

Domenico Davide MERINGOLO†[1], Yong LIU[1], and Andrea COLAGROSSI[2]
[1]Ocean University of China, Qingdao, 266100
[2]CNR-INSEAN, Rome, 00128, Italy
davide.m86@gmail.com†

Abstract: Recently, different works regarding the energy conservation properties of the SPH model have been presented. For example, Antuono et al. [1] analyzed the energy balance in the SPH scheme when a diffusive correction is used in the continuity equation for improving the stability of the scheme. In Cercos-Pita et al. [2] the energy conservation analysis has been extended to the case of moving solid boundaries when the flow is dominated by the viscous effects. In the present work, a further investigation on the energy conservation is discussed for problems where moving solid boundaries interact with an inviscid flow. The solid boundary techniques adopted are: the ghost and the fixed ghost particles. A first problem in which the dynamic is generated by a water patch falling into a still water tank is analyzed. In this case, the dissipation processes due to the diffusive and viscous numerical corrections are investigated. Then, a problem of relevant interest in coastal engineering field, concerning the wave generation and propagation in a wave flume, is studied. Two limit cases are analyzed, the first one is a wave flume with a flat bottom and a vertical wall, representative of a wave reflective condition (Fig. 1a), while the second one is a sloping bottom flume, representative of a wave absorbing condition (Fig. 1b).

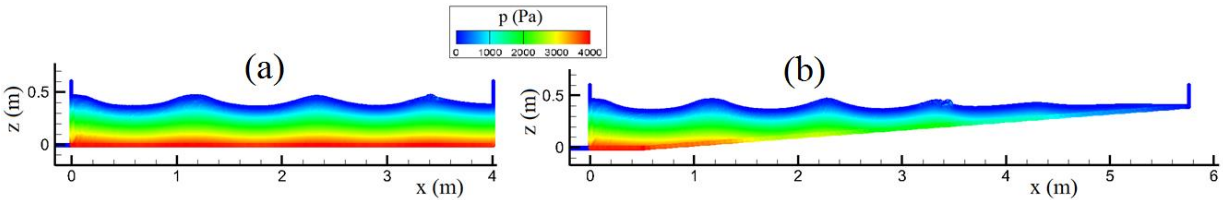


Figure 1: Waves propagation with pressure field for the analyzed cases: in (a) is the wave reflective condition, while in (b) is the wave absorbing one.

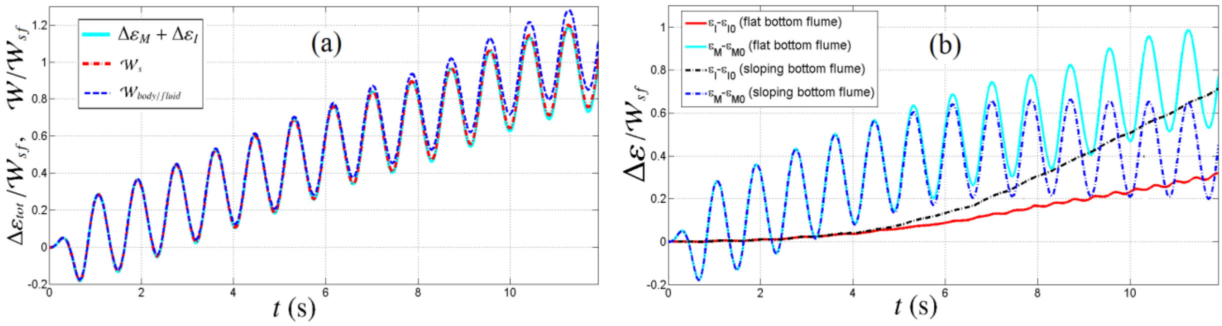


Figure 2: (a) Nominal work, $W_{\text{body/fluid}}$, effective work, W_S, made by the solid boundaries in comparison with the total energy evaluated inside the fluid domain, $\Delta\varepsilon_M + \Delta\varepsilon_I$. (b) Comparison of mechanical and internal energies variations for the flat and sloping bottom flume.

The wave generation is studied considering the work made by the solid boundary on the fluid mass. The nominal work, $W_{\text{solid/fluid}}$, and actual work, W S , are analyzed through a convergence analysis, showing that when the spatial resolution is increased $W_S \to W_{\text{solid/fluid}}$. These results are presented in Fig. 2a, in which it is possible to see that, according with the second law of thermodynamics, W_S equals the variation of total energy (mechanical+internal) evaluated inside the fluid domain $\Delta\varepsilon_{tot} = \Delta\varepsilon_M + \Delta\varepsilon_I$ (being $\Delta\varepsilon = \varepsilon - \varepsilon_0$ with ε_0 the initial energy), while at the same time it underestimates the nominal work performed by the wave-maker. Successively, the dynamics of the wave propagation is analysed from an energy viewpoint, by inspecting the single energy components, namely the potential, kinetic, compressible and numerical-dissipated energies, evaluated inside the fluid domain. Fig. 2b presents the comparison of the time evolution of mechanical, $\Delta\varepsilon_M$, and internal energies, $\Delta\varepsilon_I$, for the reflective and non-reflective cases. Because of the development of breaking waves, in the latter case the mechanical energy reaches quickly a stable condition with oscillations around a constant value.

References

[1] Matteo Antuono, S Marrone, A Colagrossi, and B Bouscasse. Energy balance in the δ-sph scheme. *Computer Methods in Applied Mechanics and Engineering*, 289:209–226, 2015.

[2] JL Cercos-Pita, M Antuono, A Colagrossi, and A Souto-Iglesias. Sph energy conservation for fluid–solid interactions. *Computer Methods in Applied Mechanics and Engineering*, 317:771–791, 2017.

Water Hammer Analysis Using SPH in Density Summation Form

Darcy Q. HOU†[1], Chunying HUANG[2], Manli WANG[3], and Huanfeng DUAN[4]
[1]School of Computer Science and Technology, Tianjin University, China
[2]School of Computer Software, Tianjin University, China
[3]School of Municipal and Environmental Engineering, Shandong Jianzhu University, China
[4]Department of Civil and Environmental Engineering, The Hong Kong Polytechnic University, Hong Kong
qhou@tju.edu.cn†

Abstract: Water hammer in fluid piping systems can induce serious damages to hydraulic machinery, piping and supports, and it should be prevented in practice. Therefore, numerical analysis of this phenomenon plays an important role in hydraulic engineering. The discontinuity of the solution, however, results in various difficulties for numerical methods.

We had successfully simulated the water hammer problem with both the original smoothed particle hydrodynamics (SPH) method [1] and its correction, the corrective smoothed particle method (CSPM) [2]. Comparing with traditional SPH, boundary conditions can be efficiently imposed in CSPM without using any fictitious or ghost particles. In [1, 2] together with momentum equation, time dependent continuity equation is solved.

Based on the fact that the density evolution in summation form works much better than the continuity equation in shock tube problems, we explore here the possibility of simulating water hammer problem by solving the SPH density summation equation and momentum equation. Numerical results shown in Fig. 1 indicate that promising results can be obtained if the equation of state is properly treated. In addition, as long as the CFL condition is satisfied, time stepping and variable smoothing length have almost no effect on the numerical results (see Fig. 1).

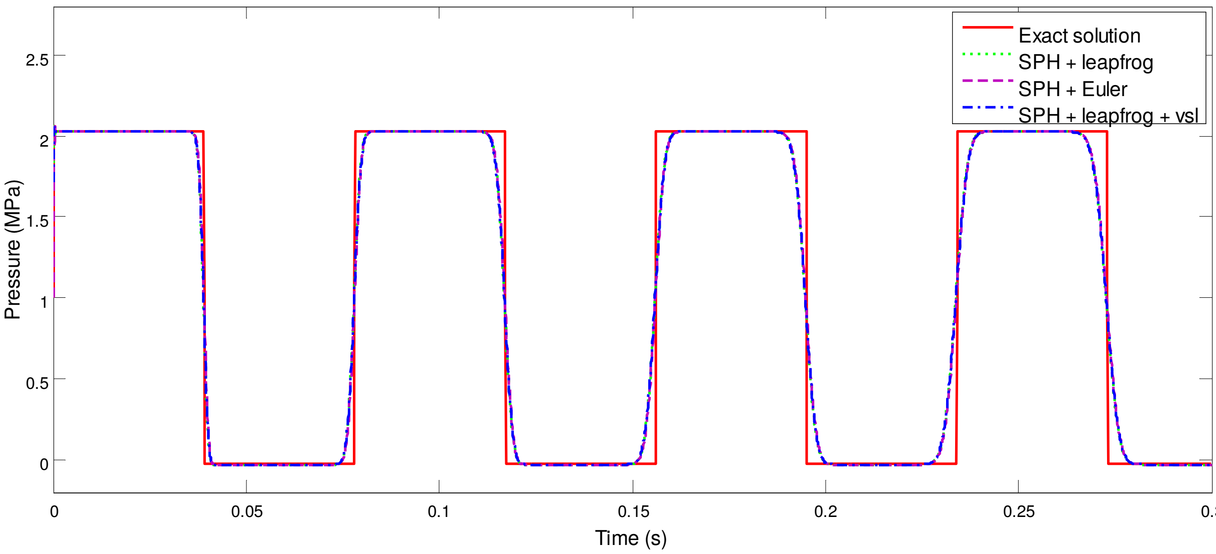


Figure 1: Pressure traces at downstream valve solved by SPH in density summation form with different time stepping and variable smoothing length.

References

[1] QZ Hou, CY Huang, JU Wei, L Zhou, and HF Duan. Meshless particle methods for solving the water hammer equations. *The Second Conference of Global Chinese Scholars on Hydrodynamics (CCSH'2016) Wuxi, Jiangsu, China*, 2016.

[2] Q Hou, ACH Kruisbrink, AS Tijsseling, and A Keramat. Simulating transient pipe flow with corrective smoothed particle method. In *Proceedings of 11th International Conference on Pressure Surges, BHR Group, Lisbon, Portugal*, pages 171–188, 2012.

Particle Trajectory Calculation in SPH

Jiayuan SHEN†[1], Wenhuan LU[1], Darcy Q. HOU[2], and Arris S. TIJSSELING[3]
[1]School of Computer Software, Tianjin University, China
[2]School of Computer Science and Technology, Tianjin University, China
[3]Department of Mathematics and Computer Science, Eindhoven University of Technology
shenjiayuan@tju.edu.cn†

Abstract: Smoothed particle hydrodynamics (SPH) method is a Lagrangian meshless approach for modelling fluid dynamics problems [1, 2]. Due to the Lagrangian property, particles change their positions at every time step and hence the calculation of particle trajectories plays an important role in SPH and may largely affect the overall accuracy. Many explicit time integration algorithms have been used in SPH, but the effect of these timestepping schemes on the calculation of particle trajectories has not been well studied. With specially designed 1D and 2D examples, this paper compares the performance of six commonly used timestepping schemes for particle trajectory calculation, including the Euler forward method, modified Euler, second-order Runge-Kutta (RK2), fourth-order Runge-Kutta (RK4), velocity Verlet and leap-frog. Numerical analysis indicates that for problems with (nearly) uniform velocity fields, all schemes can give solutions with adequate accuracy, but the Euler method, modified Euler and RK2 may not predict accurate particle trajectories in certain highly non-uniform velocity fields (see Fig. 1b-d). These methods may also introduce varying degrees of artificial dispersion that could lead to overestimation of spill area of particle clouds. On the other hand, the algorithms of RK4, velocity Verlet and leap frog can accurately calculate the particle trajectories without artificial dispersion (see Fig. 1e-f). In order to find the most efficient scheme with acceptable accuracy and cost, the computational costs of these schemes have also been compared.

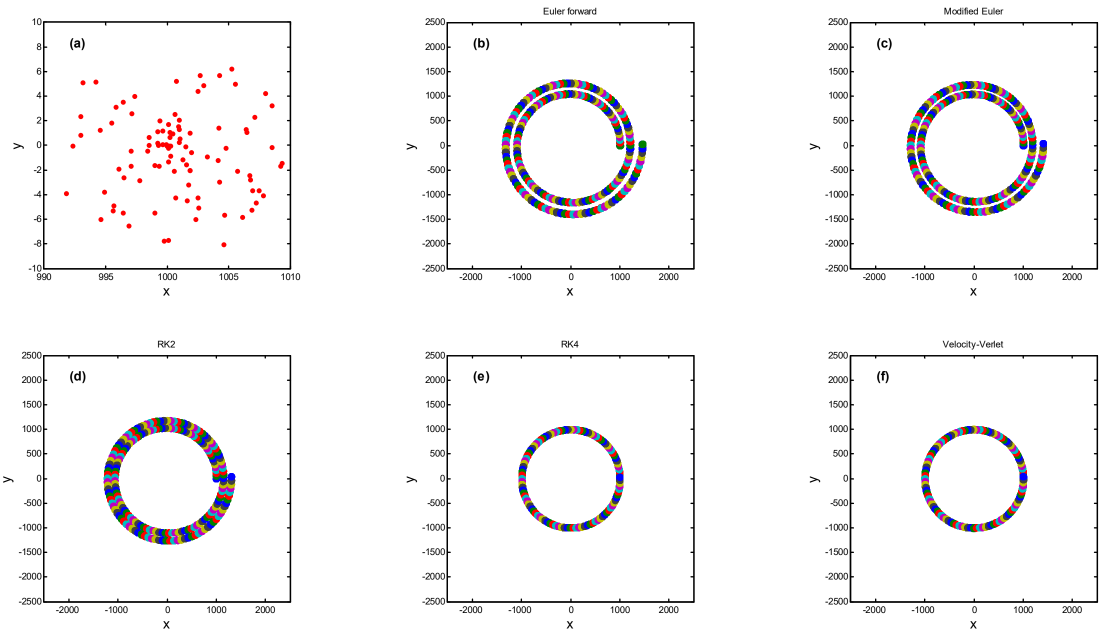


Figure 1: Trajectories of (a) 100 random particles calculated by (b) Euler forward, (c) modified Euler, (d) RK2, (e) RK4 and (f) velocity Verlet.

References

[1] Gui-Rong Liu and Moubin B Liu. *Smoothed particle hydrodynamics: a meshfree particle method.* World Scientific, 2003.

[2] Damien Violeau. *Fluid mechanics and the SPH method: theory and applications.* Oxford University Press, 2012.

Simulating shock waves with corrective smoothed particle method (CSPM)

Chunying HUANG†1, Jie DENG[1], Darcy Q.HOU[2], and Arris S. TIJSSELING[3]
[1]School of Computer Software, Tianjin University, China
[2]School of Computer Science and Technology, Tianjin University, China
[3] Department of Mathematics and Computer Science, Eindhoven University of Technology, The Netherlands
cyhuang416@163.com†

Abstract: Numerically solving the compressible Euler equations plays a vital role in many science and engineering problems, among which the shock wave is one of the benchmark tests providing a valuable evaluation for numerical methods. To identify prototypical meshless particle methods for simulating shock waves, the current state-of-the-art Smoothed Particle Hydrodynamics (SPH) schemes with kernel corrections are reviewed.

Among others, the corrective smoothed particle method (CSPM) is an early version of the traditional SPH with corrected kernel and has achieved many successes in both solid [1, 2] and fluid dynamics problems [3]. It solves the boundary deficiency and tensile instability problems in traditional SPH. The meshless finite particle method (FPM) of Liu et al. [4] shares the same merit as CSPM. However, as shown in the well-known textbook on SPH [5], CSPM failed to capture the shock physics (see Figs. 5.7-5.10 in [5]). Based on Taylor series expansion borrowed from CSPM, Liu and Liu [5] developed a discontinuous SPH (DSPH) for shock waves. However, an effective discontinuity detection algorithm has to be used, which can be rather challenging in multi-dimensional problems. In this paper, together with the SPH summation form of the continuity equation, the CSPM is applied to simulate shock waves and good results were obtained as shown in Fig. 1. To enhance the performance, a variable smoothing length was also employed, together with virtual particle method for boundaries.

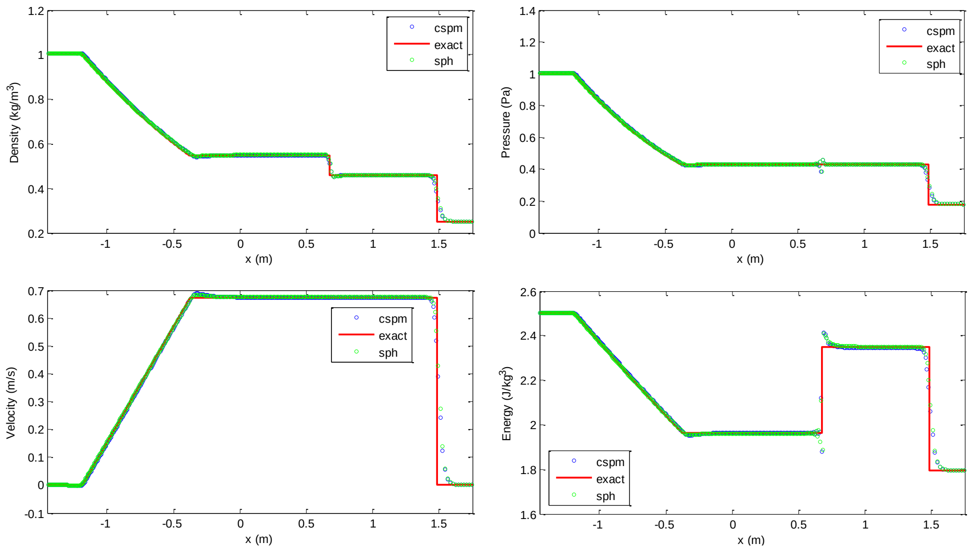


Figure 1: Numerical results for the shock tube problem obtained with different versions of SPH formulation.

References

[1] JK Chen, JE Beraun, and TC Carney. A corrective smoothed particle method for boundary value problems in heat conduction. *International Journal for Numerical Methods in Engineering*, 46(2):231–252, 1999.

[2] JK Chen, JE Beraun, and CJ Jih. A corrective smoothed particle method for transient elastoplastic dynamics. *Computational Mechanics*, 27(3):177–187, 2001.

[3] Jiannong Fang, Aurèle Parriaux, Martin Rentschler, and Christophe Ancey. Improved sph methods for simulating free surface flows of viscous fluids. *Applied Numerical Mathematics*, 59(2):251–271, 2009.

[4] MB Liu, WP Xie, and GR Liu. Modeling incompressible flows using a finite particle method. *Applied mathematical modelling*, 29(12):1252–1270, 2005.

[5] Gui-Rong Liu and Moubin B Liu. *Smoothed particle hydrodynamics: a meshfree particle method.* World Scientific, 2003.

SPH modeling of fluid-structure interaction (FSI)

Luhui Han† and Xiangyu Hu*
Department of Mechanical Engineering, Technical University of Munich, 85747 Garching, Germany
Luhui.han@tum.de†, Xiangyu.hu*

Abstract: This work concerns a numerical modeling of fluid-structure interaction (FSI) in a uniform SPH framework. It combines a transport-velocity SPH scheme [1] advancing fluid motions with a total Lagrangian SPH formulation [2] dealing with the structure deformations. To remedy the incompleteness of the kernel support at structure boundaries when evaluating strains and inter-particle forces between solid particles, a correction matrix [3] is employed to restore first order consistency and rotational invariance of Green strain tensor. Since both fluid and solid governing equations are solved in SPH framework, coupling becomes straightforward and meanwhile the momentum of an FSI system is strictly conservative. Several FSI benchmark test cases [4] have been performed to validate the modeling and demonstrate its potential.

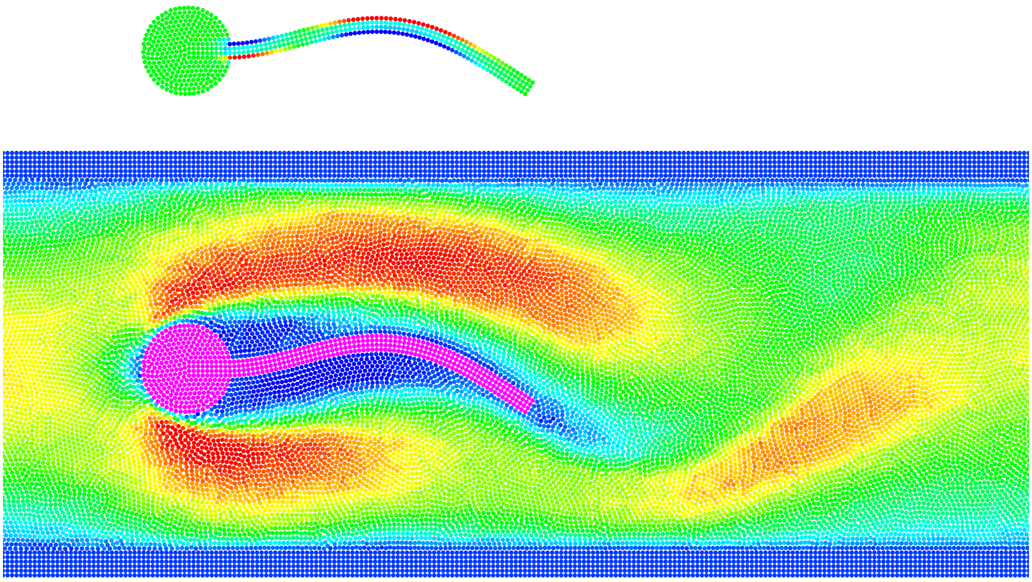


Figure 1: Deformation of beam for benchmark case FSI2 [4] ($\rho_s/\rho_f = 10$ and Re = 100) with solid particles colored by contours of von Mises stress (top). Distribution of axial velocity component u_x (bottom).

References

[1] S Adami, XY Hu, and Nikolaus A Adams. A transport-velocity formulation for smoothed particle hydrodynamics. *Journal of Computational Physics*, 241:292–307, 2013.

[2] Rade Vignjevic, Juan R Reveles, and James Campbell. Sph in a total lagrangian formalism. *CMC-TECH SCIENCE PRESS-*, 4(3):181, 2006.

[3] Javier Bonet and Sivakumar Kulasegaram. A simplified approach to enhance the performance of smooth particle hydrodynamics methods. *Applied Mathematics and Computation*, 126(2):133–155, 2002.

[4] Hans-Joachim Bungartz and Michael Schäfer. *Fluid-structure interaction: modelling, simulation, optimisation*, volume 53. Springer Science & Business Media, 2006.

Numerical modeling of 2D complex movement patterns to FSI problems using Smoothed Particle Hydrodynamics

Chen Zhuang[1,2], Dean Hu[†1], Ting Long[1,2], and Gang Yang[1,2]

[1]State Key Laboratory of Advanced Design and Manufacturing for Vehicle Body, Hunan University, Changsha 410082, P. R. China

[2]Key Laboratory of Advanced Design and Simulation Techniques for Special Equipment, Ministry of Education, Hunan University, Changsha 410082, P. R. China

hudean@hnu.edu.cn[†]

Abstract: Understanding dynamic behaviors of natural locomotion, such as aquatic animal swimming and aerial animal flight, is very important for researchers and engineers who wish to develop robotics with good locomotion capability. Aquatic and aerial locomotion generally have high efficiency, high maneuverability and low noise with which no man-made robotics can compare. However, modeling and reproducing natural locomotion using mesh-based numerical schemes, such as finite element method (FEM), finite volume method (FVM) and finite difference method (FDM), are extremely challenged due to the highly mixed patterns including translation, rotation, flexible motion and etc.

This paper presents a study based on weakly compressible smoothed particles hydrodynamics (WCSPH) method, aiming at an available numerical modeling of two-dimensional (2D) complex movement patterns to fluid-solid interaction (FSI) problems. The SPH scheme is first briefly recalled and discussed through its formulations. Then a new technic based on dynamic boundary condition and designed to mimic the arbitrary shapes with complex movement patterns is introduced. Two distinct test cases, including wedge water entry and angularly reciprocating plate, are presented in order to validate this method. It has been found that there is a general agreement between the simulation results and experimental data. Moreover, two novel cases inspired by basilisk lizard locomotion [1] and anguilliform swimming are conducted to show the capacity of the computational model developed here for modeling the complex movement patterns.

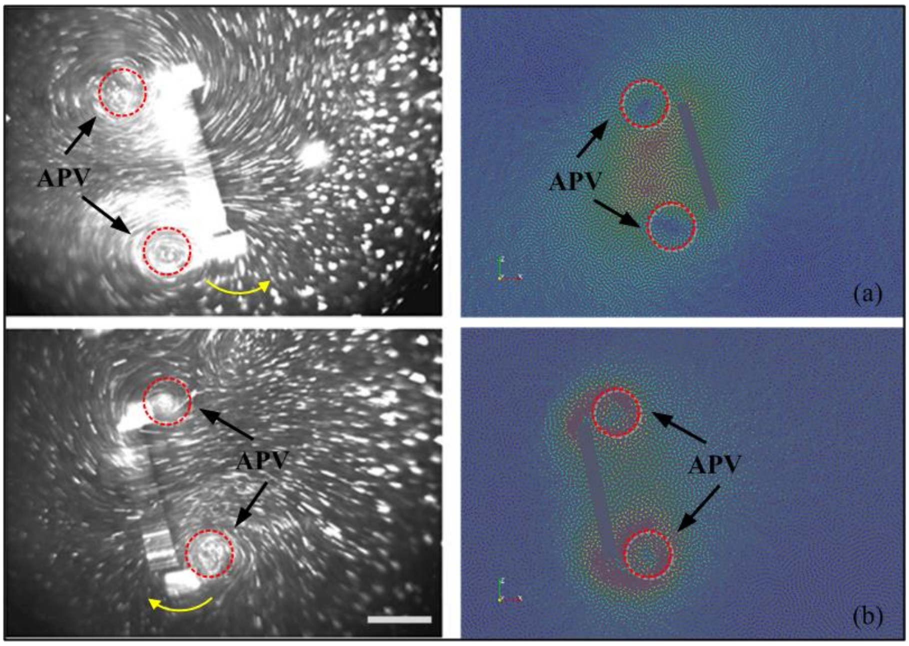


Figure 1: Comparison of flow structure around the angularly reciprocating plate between experimental image by Lee [2] (left) and SPH (right). (a) APV at $\theta = 10°$; (b) APV at $\theta = 5°$. The round arrow indicates the instantaneous direction of flat-plate rotation.

References

[1] JW Glasheen and TA McMahon. A hydrodynamic model of locomotion in the basilisk lizard. *Nature*, 380(6572):340, 1996.

[2] Jeongsu Lee, Yong-Jai Park, Useok Jeong, Kyu-Jin Cho, and Ho-Young Kim. Wake and thrust of an angularly reciprocating plate. *Journal of Fluid Mechanics*, 720:545–557, 2013.

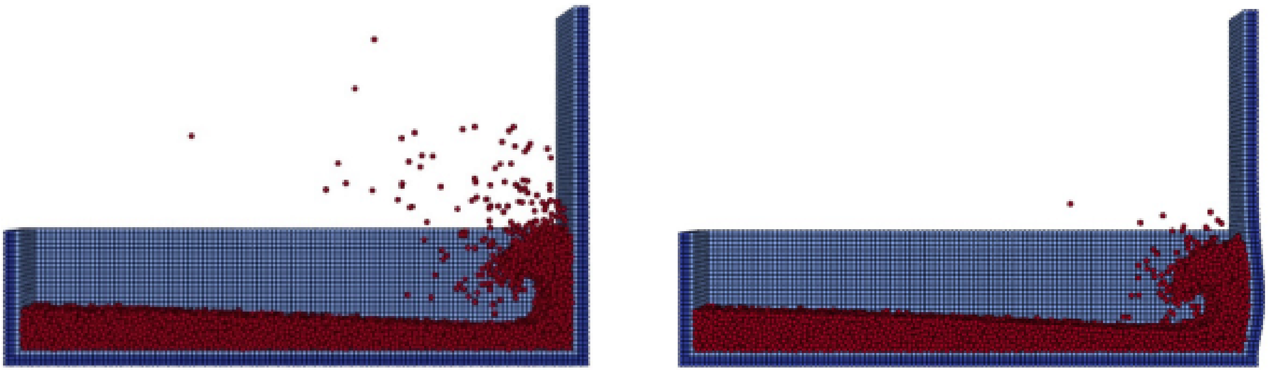
Implement of the MPS-FEM coupled method for the FSI simulation of the 3-D dam-break problem

Youlin Zhang and Decheng Wan†
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai 200240, China
dcwan@sjtu.edu.cn†

Abstract: The fluid structure interaction (FSI) problems with violent free surface have gained great attentions since they are often encountered in many engineering applications, such as the liquid sloshing in an oil tanker, very large floating structure interacting with waves, flexible structures experiencing dam-break flows, etc. In the past decades, the grid-based methods are much popular in the contributions regarding the simulation of FSI problems. However, it's quite a challenge for such methods to model phenomena that involve complex free surface flows, large deformations of flexible structures. By contrast, the mesh-less methods are free from these difficulties. Therefore, the mesh-less methods, cooperating with the finite element method (FEM), are promising for the FSI problems involving flexible structures and free surface flows.

The current article presents a partitioned approach for the 3-D FSI problems in which the moving particle semi-implicit (MPS) method is coupled with the FEM method. Herein, the MPS method is employed for the simulation of fluid domain while the FEM approach is used for the analysis of structural domain. For the implement of the coupling approach, we proposed a mapping algorithm to transfer quantity values between the particles of flow field and the elements of structural field. In this mapping algorithm, the nonmatching refinement levels of both domains are permitted, which implies that the much larger size of element can be used in the FSI simulation and the computational efficiency can be improved.

With the benefit of the proposed MPS-FEM coupled method, the 3-D FSI problem of dam-break flow impacting onto the flexible wall is numerically investigated. The evolutions of free surface and the impacting loads on the wall are comparative against those regarding rigid tank, as shown in figure 1. In addition, the deformation and the strength behaviors of the flexible wall are exhibited.



(a) rigid tank (b) elastic tank

Figure 1: Comparison of free surface between rigid and elastic tanks

References

[1] YL Zhang, ZY Tang, and DC Wan. Mps-fem coupled method for interaction between sloshing flow and elastic structure in rolling tanks. In *Proceedings of the 7th International Conference on Computational Methods*, pages 1–4, 2016.

A new numerical method for SPH fluid-solid coupling simulation and its preliminary verification

MA Xiao-jing†, Mamtimin Geni, and JIN A-fang
College of Electrical Engineering, Xinjiang University, Urumqi, China
maxiaojing1983@163.com†

Abstract: It is difficult to process the material interface in conventional SPH method. In this study, two-way interaction effect between fluid and solid is realized by involving different material particles in the calculation of the conservation equation. Although it does not need to add the additional coupling term, it causes numerical disturbance and calculation deviation to the stress field and velocity field near the material interface due to the big difference of physical properties. In order to overcome the solving error caused by the particle inconsistency, a simple and feasible coupling algorithm for fluid-solid interface is proposed to deal with calculation near the material interface in this work. The judgment is made that whether particles are involved in the governing equations calculation according to the motion direction and the stress of particles.

The modified SPH method is applied to simulate the fluid-solid interaction problems in impacting process, such as the drainage impacting on elastic baffle and dam-breaking impacting on elastic baffle. Simulation results are truly reproduced the change process of fluid flow field and the dynamic deformation process of elastic baffle in the drainage process. Through the comparative analysis of the experimental results under the same condition, it verifies that the proposed SPH fluid-solid coupling algorithm is capable of effectively and accurately simulating the deformation of fluid with free surface and the elastic solid as well as the process of rebound during the fluid-solid impacting. This provides a foundation for the study of the complex fluid-solid interaction problems.

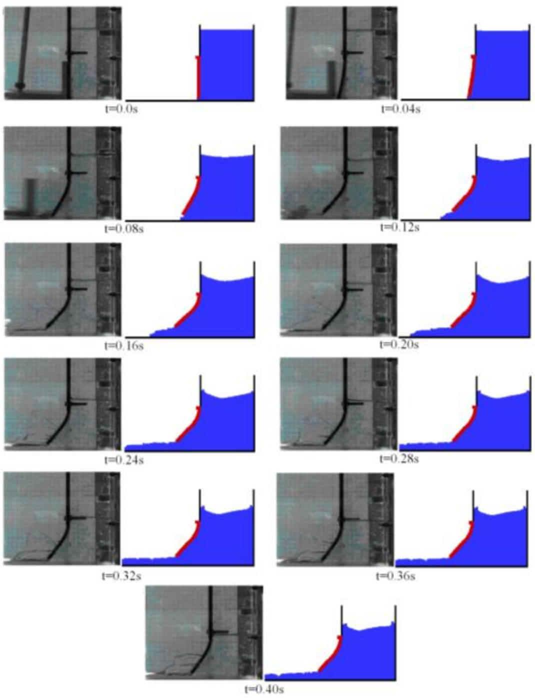


Figure 1: Comparison of experimental results (left) with SPH simulation (right) at different time

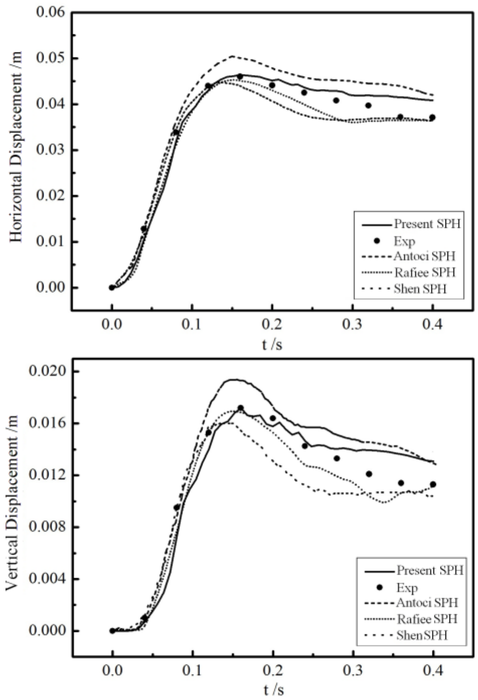


Figure 2: The horizontal and vertical displacement of the free end of the baffle

References

[1] Carla Antoci, Mario Gallati, and Stefano Sibilla. Numerical simulation of fluid–structure interaction by sph. *Computers & Structures*, 85(11):879–890, 2007.

[2] Ashkan Rafiee and Krish P Thiagarajan. An sph projection method for simulating fluid-hypoelastic structure interaction. *Computer Methods in Applied Mechanics and Engineering*, 198(33):2785–2795, 2009.

[3] YM Shen, K He, JQ Chen, and XX Yuan. Numerical simulation of free surface flow impacting elastic structure with sph uniform method. *Zhendong yu Chongji/Journal of Vibration and Shock*, 34(16):60–65, 2015.

An Enhanced ISPH-SPH Coupled Method for Incompressible Fluid-Elastic Structure Interactions

Abbas Khayyer†[1], Hitoshi Gotoh*[1], Yuma Shimizu[1], and Hosein Falahaty[1]
[1]Department of Civil and Earth Resources Engineering, Kyoto University, Kyoto, Japan
[1]khayyer@particle.kuciv.kyoto-u.ac.jp†, gotoh@particle.kuciv.kyoto-u.ac.jp*

Abstract: Precise evaluation of highly interactive fluid-structure systems (e.g. hydrodynamic slammings of marine vessels, tsunami/storm surge impact on onshore structures) has been a substantial challenge for reliable design of coastal/ocean structures. In the view of the intrinsic difficulties, usually encountered in numerical modeling of FSI (Fluid-Structure Interaction) problems associated with coastal/ocean engineering (e.g. existence of violent free-surface flows as well as large/abrupt hydrodynamic loads and consequently large structural deformations), Lagrangian meshfree methods including Smoothed Particle Hydrodynamics, SPH, are potentially robust and appropriate candidates.

In this study, a projection-based particle method, namely, Incompressible SPH (ISPH), is coupled with a SPH-based structure model in a mathematically-physically consistent manner via a careful attention to the mathematical concept of ISPH, namely, Helmholtz-Leray decomposition. The ISPH-based fluid model is founded on Navier-Stokes and continuity equations, while the SPH-based structure model is based on conservation laws for linear and angular momenta corresponding to an elastic solid. A set of previously developed enhanced schemes [1] are incorporated in the ISPH fluid model. Hence, the developed coupled method is referred to as Enhanced ISPH-SPH. **To the best of our knowledge, this study presents the first ISPH-SPH FSI solver for computational modeling of incompressible fluid flows interacting with deformable elastic structures**.

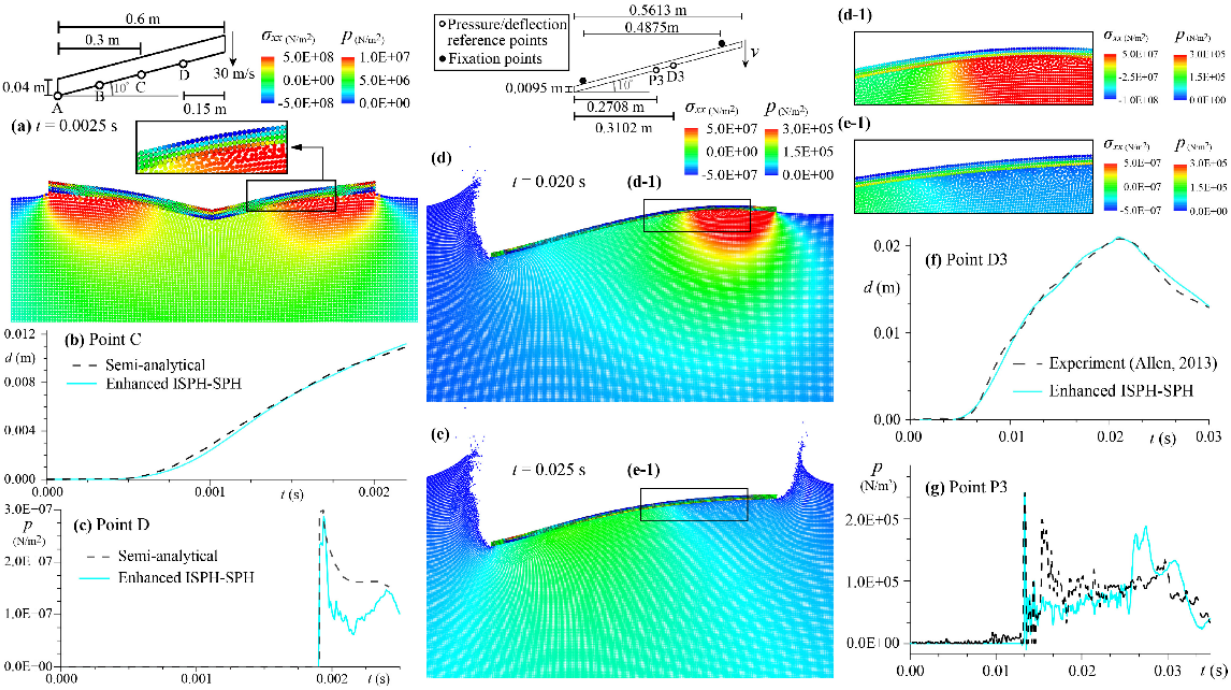


Figure 1: Representative results related to (a-c) high velocity impact of an elastic aluminum beam [2] and (d-g) hydroelastic slammings of a marine panel [2, 3], illustrating the performance of the developed Enhanced ISPH-SPH FSI solver

Followed by validation of the SPH structure model, the performance of Enhanced ISPH-SPH FSI solver is verified through a set of benchmark tests, namely, high velocity impact of an elastic aluminum beam [4] and hydroelastic slammings corresponding to a marine panel [2, 3]. Fig. 1(a) presents a snapshot corresponding to high velocity impact of an aluminum beam, illustrating smooth and qualitatively acceptable pressure/stress fields. Fig. 1(b-c) present a quantitative validation of the accuracy of proposed FSI solver by considering the deflection and pressure time histories at two reference points. Fig. 1(d-e) portray typical snapshots related to hydroelastic slammings of a marine panel (for $v = 4$ m/s), illustrating qualitatively accurate pressure/stress fields. Fig. 1(f-g) present quantitative validations by considering time histories of deflection and pressure at reference points D3 (for $v = 4$ m/s) and P3 (for $v = 3$ m/s), respectively. From the figures, the Enhanced ISPH-SPH FSI solver has provided quite accurate results corresponding to a hydroelastic slamming phenomenon.

References

[1] Abbas Khayyer, Hitoshi Gotoh, Yuma Shimizu, and Kohji Gotoh. On enhancement of energy conservation properties of projection-based particle methods. *European Journal of Mechanics-B/Fluids*, 2017.

[2] Thomas Allen. *Mechanics of flexible composite hull panels subjected to water impacts*. PhD thesis, University of Auckland, 2013.

[3] Ivan Stenius, Anders Rosén, M Battley, and T Allen. Experimental hydroelastic characterization of slamming loaded marine panels. *Ocean Engineering*, 74:1–15, 2013.

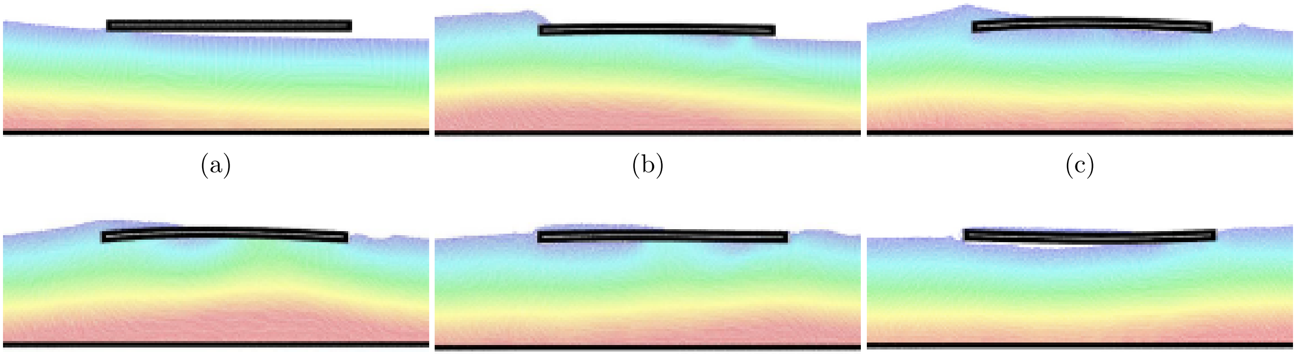
[4] Guillaume Oger, P.M Guilcher, E Jacquin, L Brosset, J.B Deuff, and David Le Touzé. Simulations of hydro-elastic impacts using a parallel sph model. *International Journal of Offshore and Polar Engineering*, 30(3):181–189, 2010.

Interaction between Solitary Wave and Flexible Plate based on MPS-FEM coupled Method

Chengping Rao and Decheng Wan[†]
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai 200240, China
dcwan@sjtu.edu.cn[†]

Abstract: The model of wave interacting with the plate is commonly seen in the offshore and coastal engineering. For example, a very large floating structure (VLFS) with its horizontal size much greater than the vertical size is usually treated as thin plate floating in the ocean. While encountering severe wave, it could produce considerable deformation which will exert a great influence on the flow field nearby, making the problem more complex. To conduct the FSI analysis of the wave-plate interaction problem, the Moving Particle Semi-Implicit and finite element coupled method (MPS-FEM) is proposed. The MPS method is adopted to calculate the fluid domain while the structural domain is solved through FEM method.

The solitary wave is first generated in a numerical wave tank and then be compared with the theoretical wave profile. The convergence study with regard to particle resolution is conducted to find the appropriate particle spacing employed in the following simulations. Then the simulations of various solitary wave impacting onto flexible plate are conducted. To validate the accuracy of the current FSI solver, the wave-induced force on the plate is compared with the existent experimental result, which shows a good agreement. The effects of the structural deformation on the flow is investigated. It turns out that the vibration of the flexible plate may intensify the impact with the free surface. In addition, the maximum displacement on the middle-point of the plate under wave with various amplitude is collected to examine quantitatively the structural response to the impact.



(a) (b) (c)

(d) (e) (f)

Figure 1: Snapshots of the wave-plate interaction

Modeling of single film bubble and numerical study of the Plateau structure in foam system

Zhongguo SUN†, Ni NI, Yijie SUN, and Guang XI
School of Energy and Power Engineering, Xian Jiaotong University, China
sun.zg@xjtu.edu.cn†

Abstract: The single-film bubble has a special geometry that a certain amount of gas is shrouded by a layer of liquid film under the surface tension, which acts both on the inside and outside surfaces of the bubble. Based on the mesh-less Moving Particle Semi-implicit method (MPS), a new surface tension model was established for the single-film bubble which has double gas-liquid interfaces. Then the complex interface movement in the oscillation process of the single-film bubble was captured.

Typical flow phenomena and deformation characteristics of the liquid film were obtained by simulating and analyzing the coalescence (Fig. 1) and connection (Fig. 2) process of two single-film bubbles. In addition, a concave tangent method was proposed to calculate the angle of the liquid film of the connected bubbles, which could help describe the shape quantitatively.

Furthermore, the classic Plateau structure in foam system was simulated and numerically proved to be the steady statues for multi-bubble connections.

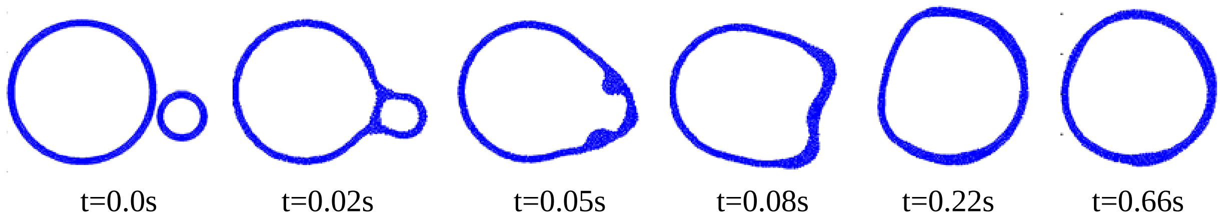


Figure 1: Coalescence of two single film bubbles with different sizes.

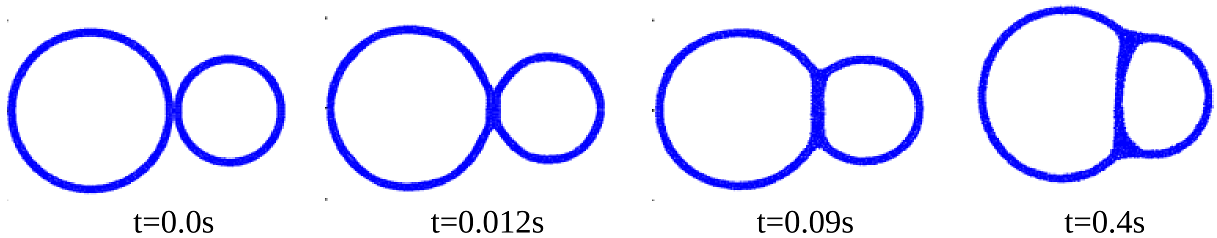


Figure 2: Connection of two single film bubbles with different sizes.

References

[1] Ni Ni, Zhong-guo SUN, Xiao Chen, and Guang Xi. Numerical simulation of coalescence process of single film bubbles using mps method. *Conference of Chinese engineering thermal physics, Wuxi (China)*, 7:22–25, 2016.

[2] Xiao Chen, Zhong-guo SUN, and Guang Xi. Improvement of surface tension model and numerical investigation on droplet wetting effect [j]. *Journal of Xi'an Jiaotong University*, 7:022, 2012.

Numerical Simulation of Rayleigh-Taylor Instability by MPS Multiphase Method

Xiao Wen and Decheng Wan†
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai 200240, China
dcwan@sjtu.edu.cn†

Abstract: Compared with grid-based methods, the particle method is advantageous for solving multiphase problems with large deformation of the interface between different phases. However, stable multiphase simulations is difficult to be obtained due to the discontinuity of pressure gradient field in the interface and the phenomenon of pressure fluctuation commonly existing in particle method. Therefore, an accurate and stable multiphase model is necessary to be studied.

The paper proposed a multiphase model based on the moving particle semi-implicit method (MPS) for multiphase flows. In this model, the multi-fluids system is treated by a single set of equation and the conservation of volume of both fluids is implicitly satisfied by solving a Poisson pressure equation. A transition region with application of density smoothing technique is introduced to deal with the mathematical discontinuity of density and viscosity in the interface, and the discontinuity of pressure gradient field, which is the main cause of instability in multiphase flows, can be avoided.

To validate this model, one of the classic hydrodynamic instability cases in many natural scenarios and industrial applications, Rayleigh-Taylor instability, is numerical simulated in this paper. At first, different smoothing scheme is adapted in numerical simulation, to test their effects on numerical instability and interface keeping. Then the results are compared with analytical solutions and other numerical methods, proving the stability of this model and the ability of MPS Multiphase method to capture the evolution of the interface between different phases.

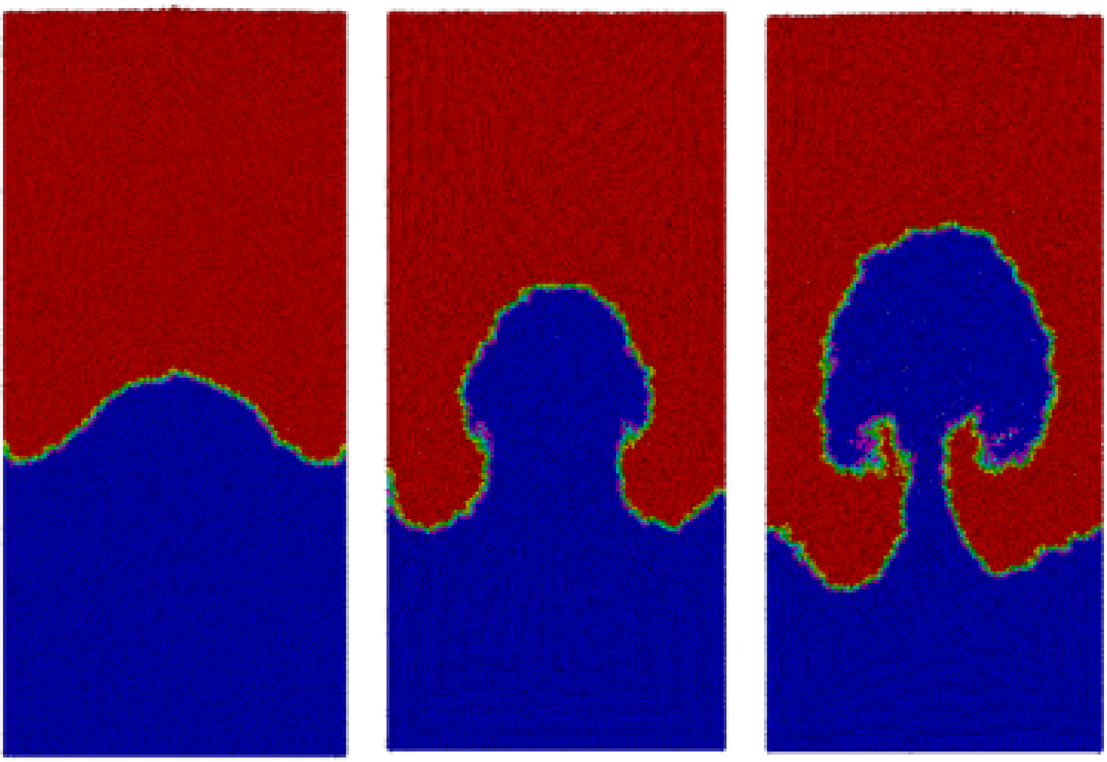


Figure 1: The evolution of interface in Rayleigh-Taylor instability simulated by MPS multiphase method: earlier stage (left), middle stage (middle), later stage (right).

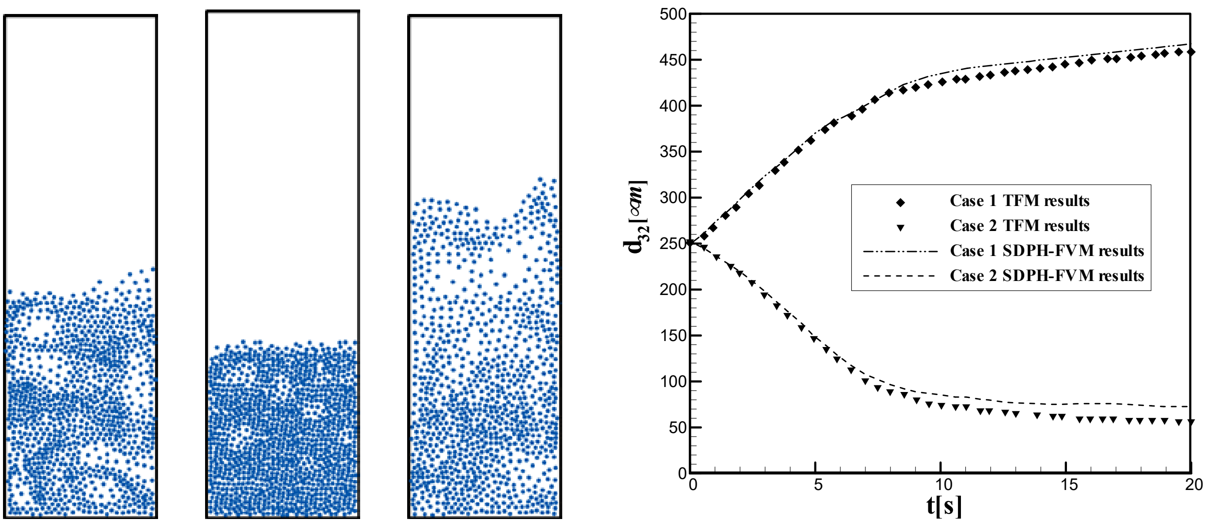
Numerical Simulation of Particle Collision and Breakup Behavior by SDPH-FVM Coupling Method

Hongfu Qiang and Fuzhen Chen†
Department of Power Engineering, Xi'an Hi-Tech Institute, China
chen_fu_zhen@163.com†

Abstract: The collision of particles and the breakup of single particles are widespread in nature and industry, which are critical to the prediction and control of them. Smoothed discrete particle hydrodynamics (SDPH) and finite volume (FVM) coupling method, as a new approach to solve the gas-particle two-phase flow, has been successfully applied to simulate aeolian sand transport, bubbling fluidized bed, spouted bed and gas-particle two-phase heat transfer processes.

The present paper aims to simulate the microscopic behavior such as particle collision polymerization and breakup in the isotropic system by the population balance model (PBM) from crystallization kinetics. The direct quadrature method of moment is applied to solve the PBM from the perspective of macroscopic system based on the droplets collision and breaking mechanism. The relation between the population distribution moment and the particle size distribution parameters is built. And then they are imported into SDPH-FVM algorithm to realize the coupling of population balance model and SDPH-FVM coupled method. The numerical simulation of the fluidization and size of the particles in the gasification fluidized bed reactor was carried out, and the accuracy and practicability of the new method were verified by comparing with the traditional TFM calculation results.

Two cases are calculated by using the [Fan, Marchisio and Fox (2004)] equation to describe the particle aggregation and fragmentation. In case 1, the polymerization process is the most important, and in case 2, the breakup process is the main one. In case 1, the polymerization parameters and breakup parameters are set to be 0.001 and 0.0001 respectively. In case 2, the polymerization parameters and the crushing parameters are set to be 0.001. Figure 1 is the comparison of the spatial distribution of the particles obtained at the time 6sof case 1 and 2 with no aggregation and fragmentation. The numerical results show that the new SDPH-FVM method has captured the particle flow in detail, the population balance model solving by the moment method to deal with the coalescence and breakup micro behaviors is accurate and reliable, which can be used in engineering practice.



(a) (b) (c) (d)

Figure 1: The spatial distribution of particles under different conditions. (a) No polymerization and breakup; (b) Case 1; (c) Case 2; (d) Curves of sauter diameter d_{32} over time in case 1 and 2.

References

[1] Fuzhen Chen, Hongfu Qiang, Han Zhang, and Weiran Gao. A coupled sdph–fvm method for gas-particle multiphase flow: Methodology. *International Journal for Numerical Methods in Engineering*, 109(1):73–101, 2017.

[2] Fuzhen Chen, Hongfu Qiang, and Weiran Gao. Coupling of smoothed particle hydrodynamics and finite volume method for two-dimensional spouted beds. *Computers & Chemical Engineering*, 77:135–146, 2015.

A Physics Evoked Meshfree Method

MA Zhi-bo[1] and ZHAO Ya-zhou[†2]
[1]Institute of Applied Physics and Computational Mathematics, Beijing 100094, China
[2]Institute of Geology and Geophysics, Chinese Academy of Science, Beijing 100029, China
asiabuaasa@163.com[†]

Abstract: In mesh free methods, discrete equations are built according to physics information of micro-bodies arbitrarily spread in vicinal space. As the requirements about topology of micro-bodies are reduced, simulations with Lagrangian approach may be easier even with large distortions. Owing to the insufficiency of topological information, there is a challenge for mesh-free method to reflect physics especially as discontinuities exist. Based on the physical laws and developing trend of numerical simulation, a new mesh free systematic method PECM (Physics Evoked Cloud Method) which has excellent applicability is shown. High fidelity to physics of the method is demonstrated through five 1-dimentional challenging problems in which strong discontinuities exist.

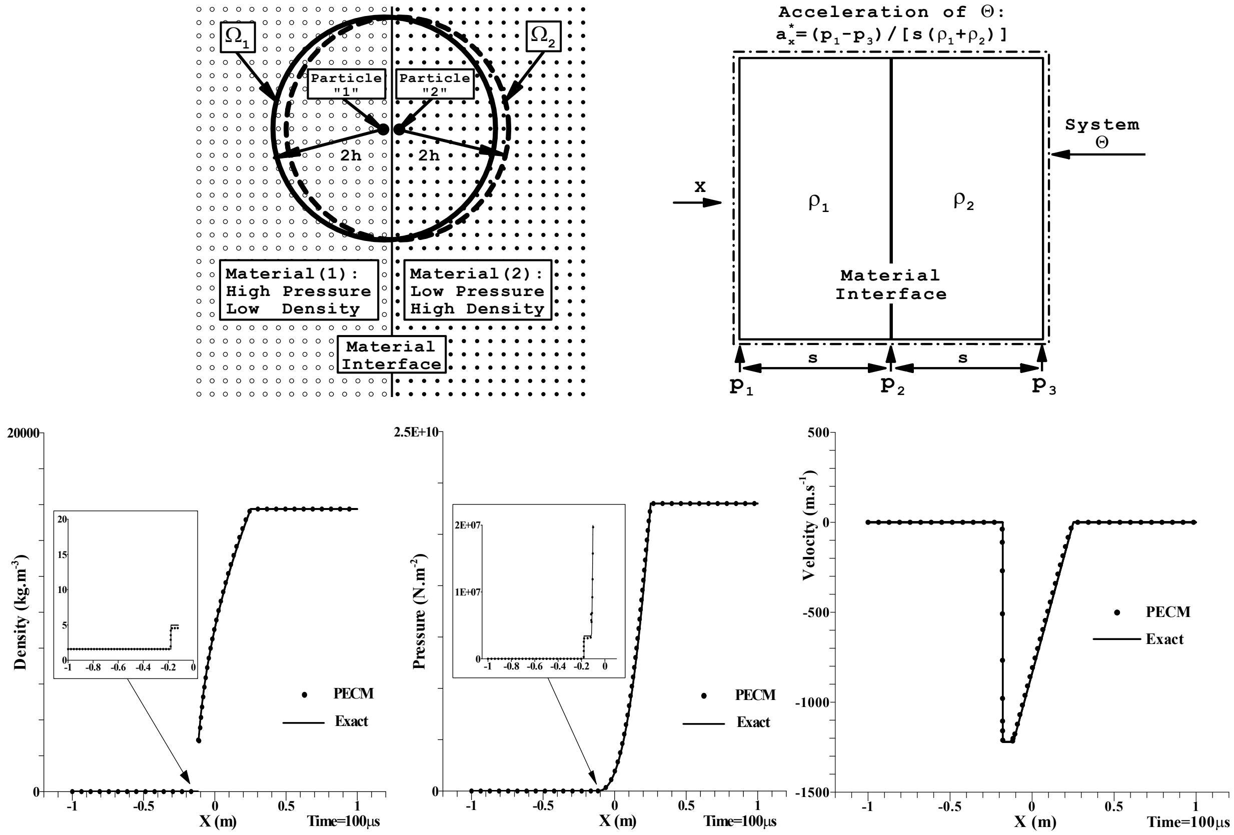


Figure 1: Kernel Modification in PECM (top) & Validation of PECM(down): Numerical results about Case 5

References

[1] RA Gingold and JJ Monaghan. Kernel estimates as a basis for general particle methods in hydrodynamics. *Journal of Computational Physics*, 46(3):429–453, 1982.

[2] Leonardo Di G Sigalotti, Hender López, Arnaldo Donoso, Eloy Sira, and Jaime Klapp. A shock-capturing sph scheme based on adaptive kernel estimation. *Journal of Computational Physics*, 212(1):124–149, 2006.

Suppression of non-physical voids in the finite volume particle method

Mohsen H. Moghimi† and Nathan J. Quinlan*
National University of Ireland Galway, Ireland
m.hassanzadehmoghimi1@nuigalway.ie†,nathan.quinlan@nuigalway.ie*

Abstract: In this work, we present a simple algorithm to differentiate non-physical voids from physical free surfaces in the Finite Volume Particle Method (FVPM), which facilitates the suppression of spurious voids. FVPM is a meshless method in which particles behave like cells in the classical-finite volume method, but are allowed to overlap each other and move arbitrarily. Like Smoothed Particle Hydrodynamics (SPH), FVPM may suffer from poor particle distribution as a result of fully Lagrangian particle motion, and the formation of unphysical voids in regions of negative pressure. This is closely related to tensile instability in SPH. In order to maintain good particle distribution, a small correction added to the Lagrangian particle transport velocity (similar to particle shifting), taking advantage of the Arbitrary Lagrangian-Eulerian (ALE) nature of FVPM [1].

FVPM particles interact through pairwise interparticle area vectors, analogous to face area in finite volume cells. In a fully covered particle neighbourhood, a particle's area vectors sum to zero. However, on a (physical or non-physical) free surface, the sum is non-zero. Thus, a free surface is easily detected. In the new method, the physical free surface is identified at initialization. To prevent formation of a new free surface in the interior of the fluid, particles are confirmed as physical free-surface particles only if they neighboured a physical free-surface particle on the previous time-step. To suppress spurious voids, an atmospheric pressure is applied only at the physical free surface, maintaining positive absolute pressure everywhere. This method is used in conjunction with the particle transport correction.

The first test case is a simple tank oscillating vertically with displacement , where , , and are amplitude, frequency, and time, respectively. The oscillation results in negative pressure, which causes the growth of regions without particles (Figure 1). The voids can be remedied if correct free-surface particle detection is implemented by the new method, because zero absolute pressure is applied on non-physical free surfaces, and no voids grow (Figure 2); but still there are some small voids which can be treated by applying the particle transport correction on the flow domain (Figure 3). These results were obtained for Reynolds number 2011, = 200, and = 26, where L and T are fluid depth and period of oscillation respectively.

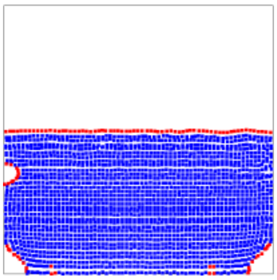


Figure 1: Particle distribution with basic Lagrangian FVPM (Red points are free surface).

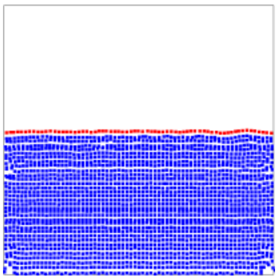


Figure 2: Particle distribution with Lagrangian particle motion and free surface detection (Red points are free surface).

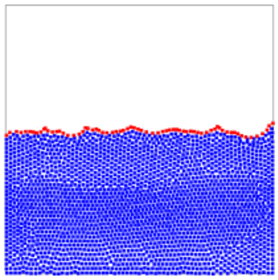


Figure 3: Particle distribution with particle transport correction and free surface detection (Red points are free surface).

The second test case is a translating rotating square in which the regions without particles are seen in Figures 4 to 9. The voids can be treated if correct free-surface particle detection is implemented by the new method (Figures 10 to 12).

References

[1] Ruairi M Nestor, Mihai Basa, Martin Lastiwka, and Nathan J Quinlan. Extension of the finite volume particle method to viscous flow. *Journal of Computational Physics*, 228(5):1733–1749, 2009.

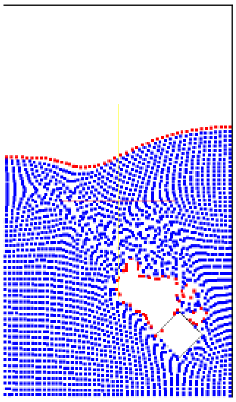


Figure 4: Particle distribution with basic Lagrangian FVPM, $t = 0.52313$ s (Red points are free surface).

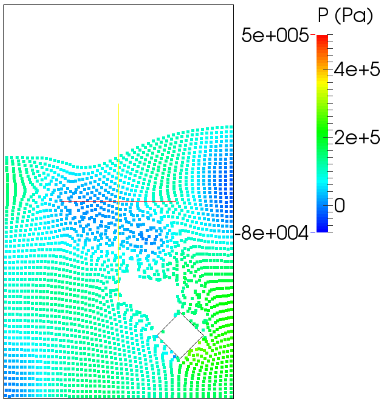


Figure 5: Pressure distribution with basic Lagrangian FVPM, $t = 0.52313$ s.

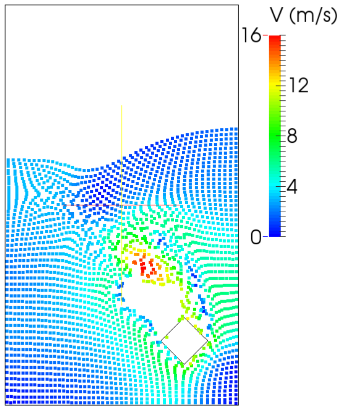


Figure 6: Velocity field with basic Lagrangian FVPM, $t = 0.52313$ s.

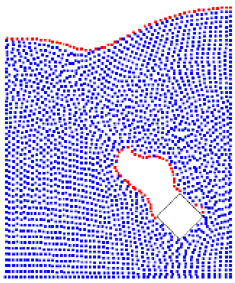


Figure 7: Particle distribution with particle transport correction, $t = 0.52313$ s (Red points are free surface).

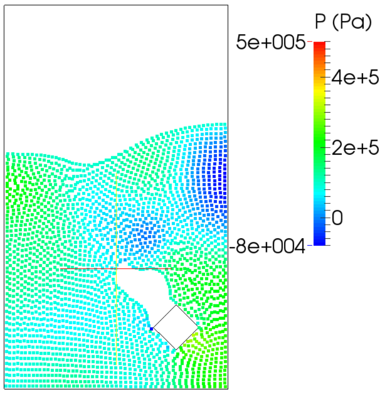


Figure 8: Pressure distribution with particle transport correction, $t = 0.52313$ s.

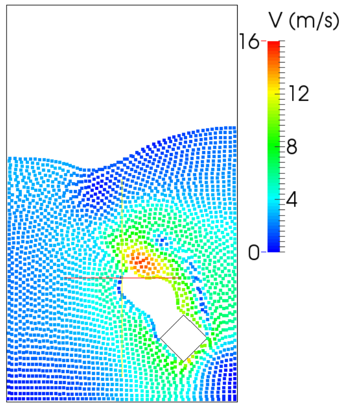


Figure 9: Velocity field with particle transport correction, $t = 0.52313$ s.

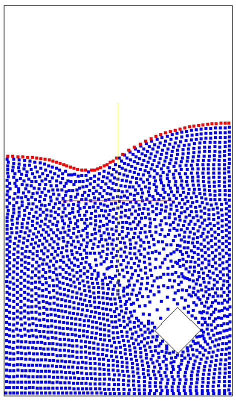


Figure 10: Particle distribution with particle transport correction and free surface detection, $t = 0.52313$ s (Red points are free surface).

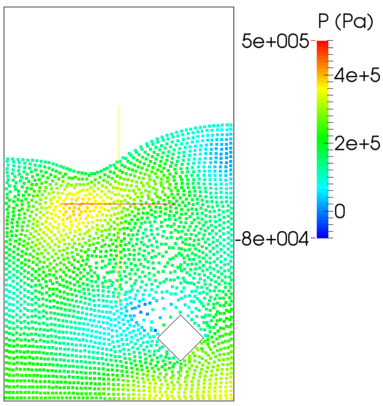


Figure 11: Pressure distribution with particle transport correction and free surface detection, $t = 0.52313$ s.

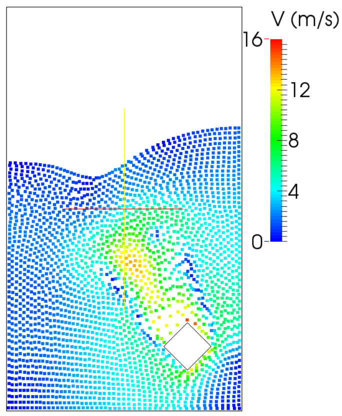


Figure 12: Velocity field with particle transport correction and free surface detection, $t = 0.52313$ s.

The Hermit-type RRKPM for piezoelectric materials

Jichao MA and Gaofeng WEI†
Qilu University of Technology, China
wgf@spu.edu.cn†

Abstract: In this paper, the radial basis function (RBF) and its normal derivative are introduced into the reproducing kernel particle method (RKPM), and the Hermit-type radial reproducing kernel particle method (Hermit-type RRKPM) is proposed. The method can reduce the adverse effect of the kernel function on the calculation precision. The errors can be decreased on the boundary, and the accuracy and stability of the method are improved. Then the proposed method is applied to the numerical simulation of piezoelectric materials. The numerical results show that the Hermit-type RRKPM is more stable and accurate than the RKPM.

Take a piezoelectric strip as the example to analyze the bending deformation. It is subjected to a linear stress in the x-direction and an applied voltage in the z-direction. (i) Analytical solutions and numerical solutions are analyzed in the x-direction. (ii) Analytical solutions and numerical solutions are analyzed in the z-direction. The results are pretty close.

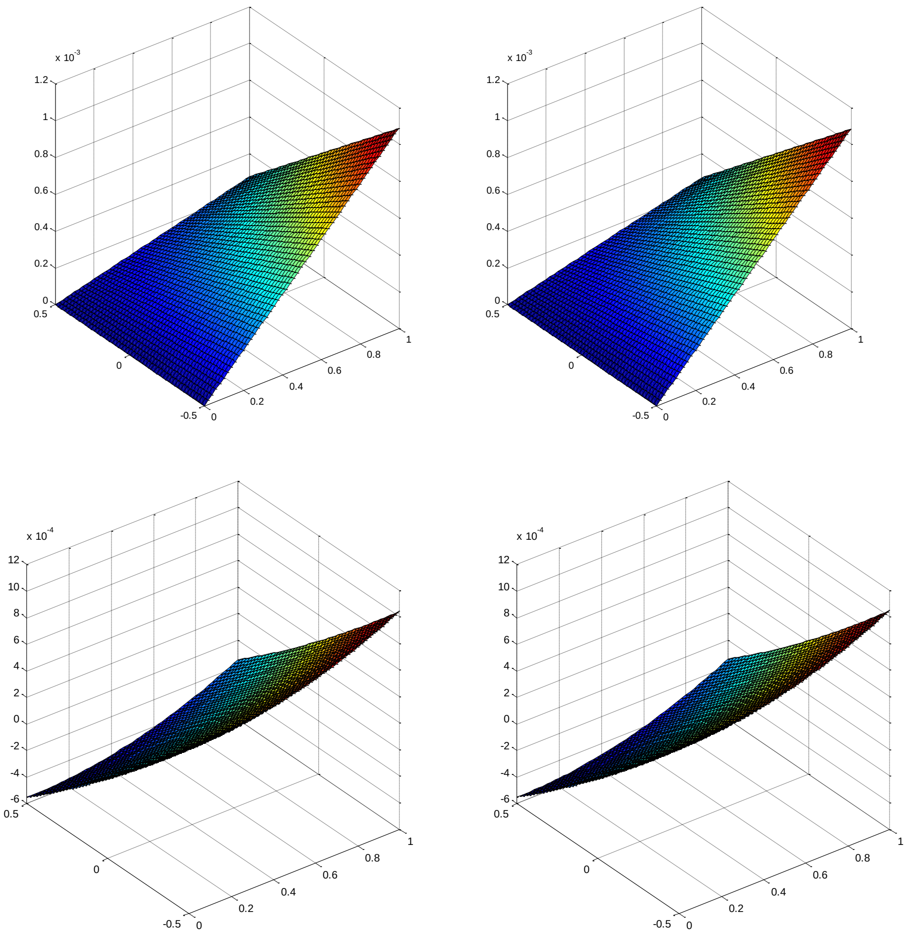


Figure 1: The comparison between the analytical solutions and numerical solutions

References

[1] MB Liu and GR Liu. Smoothed particle hydrodynamics (sph): an overview and recent developments. *Archives of computational methods in engineering*, 17(1):25–76, 2010.

A development of a SPH model for simulation of abrasive-water-jet impacting on a metallic surface

Xiangwei Dong†[1,2], Zengliang Li[1], and Jianlin Liu[2]
[1]College of mechanical and electronic engineering, China University of Petroleum (East China), 66 Changjiang Rd, Huangdao District, Qingdao, China
[2]College of Pipeline and Civil Engineering, China University of Petroleum (East China), 66 Changjiang Rd, Huangdao District, Qingdao, China
dongxw139@163.com†

Abstract:

A lagrangian model for the numerical simulation of the impact of abrasive water-jet on metallic surface is proposed in this paper. In the method both fluid and solid phases are described by smoothing particle hydrodynamics: water is modeled as a viscous fluid with weak compressibility, while metallic plate is modeled as an elastic-plastic material. Abrasive particles are modeled as rigid bodies with specified geometries. The interactions between fluid and solid, fluid and particles, particles and solid are realized by suitable terms, which are commonly used in the SPH method. Simulation tests of surface erosion by a water-jet containing single and multiple particles are carried out as challenging examples to verify the applicability of the SPH model. This supports the attractiveness of this new approach in relevant applications, such as solid particle erosion, abrasive water-jet machining, etc. Advantages of the method are robustness, conceptual simplicity and relative ease of incorporating new physics.

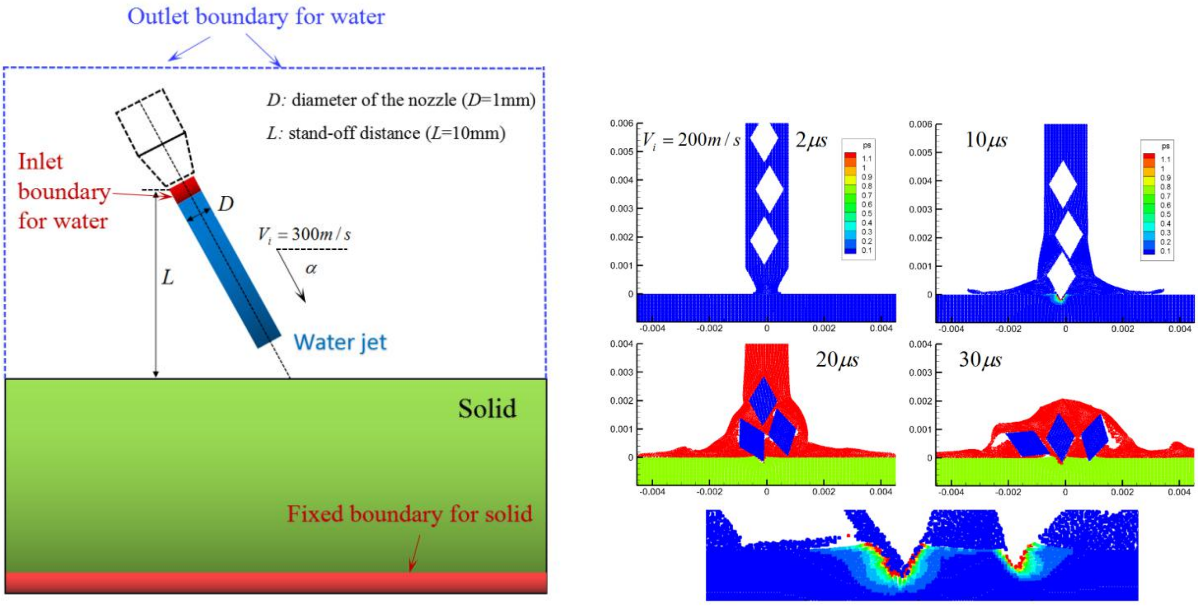


Figure 1: Graphic abstract

SPH Simulation of Couette Flow with Sinusoidally Moving Solid Boundary

Haiqiao LI†, Hantao LIU, and Jianzhong CHANG
Laboratory of Energy and Environment and Computational Fluid Dynamics, North University of China, Taiyuan, China
Lihaiqiao8@sina.com†

Abstract: The transport of lubricating oil between the cylinder liner and the piston provides a prerequisite for the internal combustion engine normal and long-term operation. Dongfang [1] pointed out that oil transport at the skirt-liner interface can be simplified as two simple models: one is the Couette flow (shear flow) which is driven by the viscous drag force and the inertia acting on the fluid due to the relative movement of the surfaces. The other is the Poiseuille flow (pressure flow), which is driven by the pressure gradient. However, the oil transport between the piston and the piston skirt is closer to the former. In this paper, the motion process of the piston is simplified to the sinusoidal periodic motion. The SPH method [2] is used to simulate the Couette flow under the sinusoidal period dragged speed for the first time. The velocity distribution of the flow field and the speed variety of different positions are studied under different dynamic viscosity coefficients of oils, different amplitudes of velocity and different frequencies of speed variety. This paper aims to provide a feasible approach to further study the lubrication between the cylinder liner and the piston in the internal combustion engine.

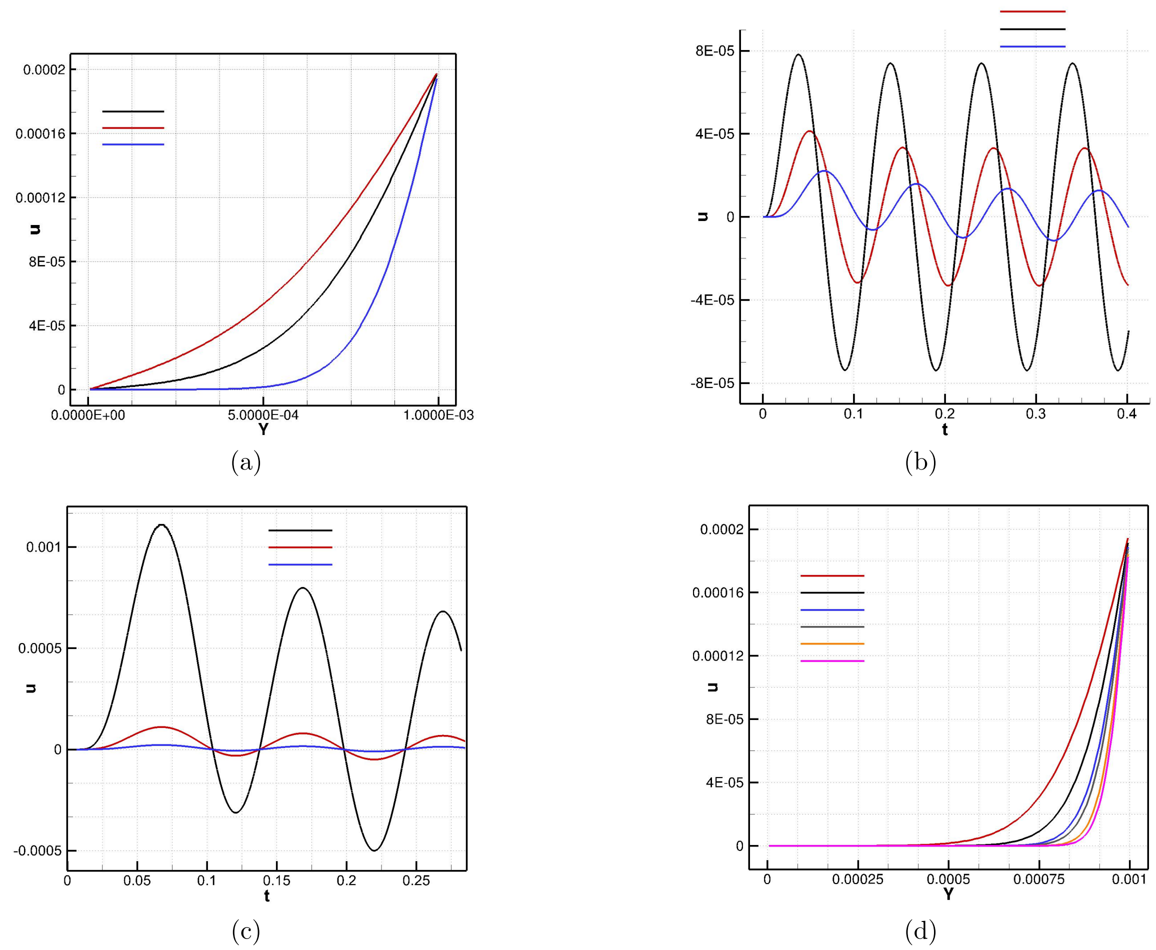


Figure 1: The velocity distribution and the speed variety curves. (a. The velocity distribution along the y-direction of calculation area at T/4 under different dynamic viscous coefficients, $u_0 = 2 \times 10^{-4}$ m/s and $f = 10$. b. The velocity curve in the x direction over time at $y = 5.05 \times 10^{-4}$ m under different dynamic viscous coefficients, $u_0 = 2 \times 10^{-4}$ and $f = 10$. c. The velocity curve in the x direction over time at $y = 5.05 \times 10^{-4}$ m under different velocity amplitude, $\mu = 1.0 \times 10^{-6}$ N $\cdot$ s/m^2 and $f = 10$. d. The velocity distribution along the y-direction of calculation area at $T/4$ under different velocity change frequency, $\mu = 1.0 \times 10^{-6}$ N $\cdot$ s/m^2 and $u_0 = 2 \times 10^{-4}$ m/s).

In this paper, the boundary drag speed is $u = u_0 \sin(2\pi f t)$, where u_0 is the amplitude of the velocity, f is the frequency of velocity evolution, t is time, and the period of velocity change is T. The calculation area is a rectangular region, which along x is 5×10^{-4} m and y is 1×10^{-3} m. The results show that: 1) When the velocity amplitude is small and the velocity change frequency is constant, the viscous drag force plays a leading role in the oil transport process, while the velocity is stabilized after two cycles. 2) When the velocity variation frequency and the dynamic viscosity coefficient keep unchanged, the inertial force become more and more obvious with the increase of the velocity amplitude, while the speed is stable with more cycles. 3) When the speed amplitude and the dynamic viscosity coefficient maintain constant, the inertial force become more and more obvious with the increase of the velocity variation frequency.

References

[1] Dongfang Bai et al. *Modeling piston skirt lubrication in internal combustion engines.* PhD thesis, Massachusetts Institute of Technology, 2012.

[2] Gui-Rong Liu and Moubin B Liu. *Smoothed particle hydrodynamics: a meshfree particle method.* World Scientific, 2003.

Application of particle-based computational acoustics to sound propagation and scattering

Yong Ou ZHANG†
School of Transportation, Wuhan University of Technology, China
zhangyo@whut.edu.cn†

Abstract: The Lagrangian meshfree method with interacting particles is a powerful and natural approach for simulating physical systems with complicated domain topologies, moving boundaries, and multiphase media. Particle-based computational acoustics (PCA), as a noval branch of computational acoustics, aims to simulate acoustic phenomenon by Lagrangian meshfree particle methods. The ability of different particle methods to simulate flow-acoustic and flow-structure-acoustic interaction problems is evaluated. To separate the acoustic perturbation from the particle motion, Lagrangian acoustic perturbation equations (LAPE) including two sets of governing equations are used. Smoothed particle hydrodynamics (SPH), corrective smoothed particle method (CSPM), and finite difference particle method (FDPM) are selected for a comparison. Several checks on the accuracy and convergence of the Lagrangian meshfree PCA method are discussed. Numerical results are obtained for various sound propagation and scattering problems, and different acoustic boundary conditions including perfectly matched layers are examined.

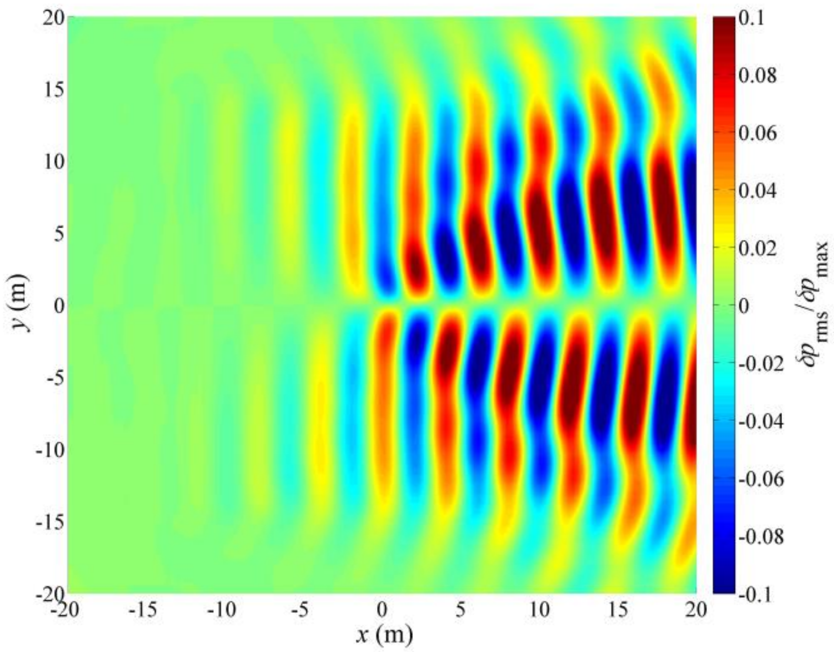


Figure 1: Non-dimensional sound pressure of vortex scattering at Ma = 0.2

Image processing with the SPH method

Chunying HUANG†[1], Wenhuan LU[1], Darcy Q. HOU[2], and Xin CHENG[2]
[1]School of Computer Software, Tianjin University, China
[2]School of Computer Science and Technology, Tianjin University, China
cyhuang416@163.com†

Abstract: One of the most significant research fields in computer graphics is the digital image processing. Scaling, rotating and repairing are the fundamental components in image processing which are all based on interpolation. There are many grid-based interpolation algorithms for image processing such as the nearest neighbor interpolation, bilinear interpolation, polynomial interpolation, B-spline interpolation and cubic convolution interpolation [1, 2]. Although these algorithms have achieved great success, their dependence on the grids might introduce difficulties and disadvantages for advanced image processing. On the other hand, the meshless methods only use the image information in the support domain to compensate the missing parts without the limit of grids. In this paper, the meshless smoothed particle hydrodynamics (SPH) method and the corrective smoothed particle method (CSPM) are used to deal with scaling, rotating and repairing of three typical images including Lenna, Pepper and Stanford Dragon as shown in Fig. 1. The numerical results indicate that the meshless methods can obtain better results according to the Peak Signal to Noise Ratio (PSNR) as shown in Table 1. Moreover, dissipation of moving images has also been successfully achieved by modelling it with a convection-diffusion process.

(a) (b) (c)

Figure 1: Typical images for processing

Table 1: PSNRs in scaling for different image interpolation methods

Case name	Interpolation methods			
	Nearest neighbor	Bilinear	SPH	CSPM
Lenna	28.1987	28.1984	29.7391	31.1865
Pepper	27.1608	28.3115	30.7758	28.9502
Standford Dragon	28.5788	30.5884	31.7239	31.8727

References

[1] J Anthony Parker, Robert V Kenyon, and Donald E Troxel. Comparison of interpolating methods for image resampling. *IEEE Transactions on medical imaging*, 2(1):31–39, 1983.

[2] W Knox Carey, Daniel B Chuang, and Sheila S Hemami. Regularity-preserving image interpolation. *IEEE transactions on image processing*, 8(9):1293–1297, 1999.

Analysis of the hydrological safety of dams using numerical tools: Iber and DualSPHysics

J. González-Cao†, O. García-Feal, A.J.C. Crespo, J.M. Domínguez, and M. Gómez-Gesteira
Enviromental Physics Laboratory (EPHYSLAB), Universidade de Vigo, Campus As Lagoas, Orense, Spain
jgcao@uvigo.es†

Abstract: The probability of performance failures of the exceedance structures of a dam defines its hydrological safety. These failures are related to water excess in the impoundment of the dam. Recently the failure of the spillways of the Oroville dam (California, USA) forced more than 100000 people to leave their houses. In this work the hydrological safety of Belesar dam is analysed by means of two numerical codes: Iber [1] and DualSPHysics [2]. Iber is a meshbased code that uses the finite volume method to solve the shallow water equations while DualSPHysics is a meshfree code based on SPH methodology that solves the Navier-Stokes equations. The dam (Figure 1) was built in 1962 and it is 127 meters height with a crest 500 meters long. Its impoundment (∼50 km long) is supplied by the hydrological network of the Miño river. The main exceedance structures of the dam are two spillways and four low level outlets. According to the technical specifications, the maximum level of the pool is equal to 330 m and the maximum expected flow of the Miño river is 4000 m^3/s.

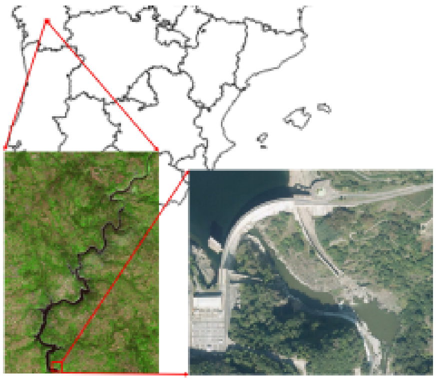


Figure 1: Location of "Belesar" dam, impoundment associated to the dam and aerial image of the dam.

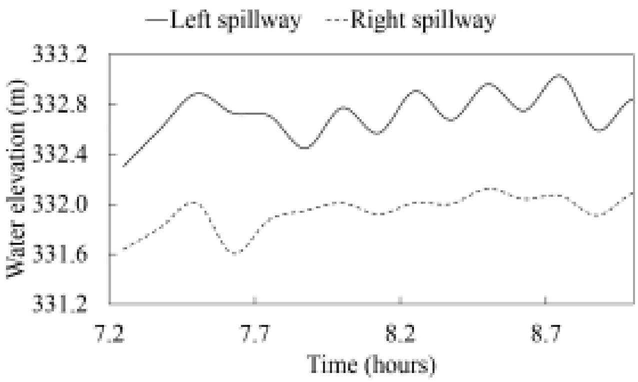


Figure 2: Water elevation close to the dam obtained using Iber.

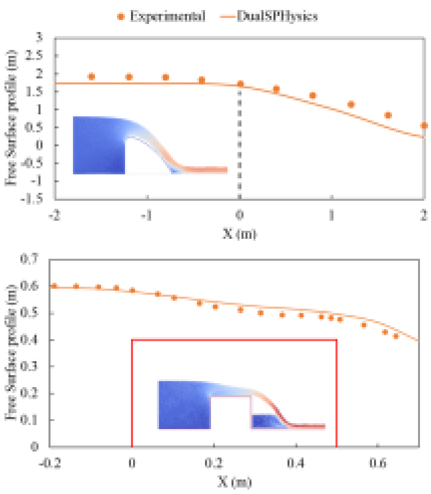


Figure 3: Free surface profile of validation spillways cases obtained with DualSPHysics: ogee spillway (top) and broad crested weir (bottom).

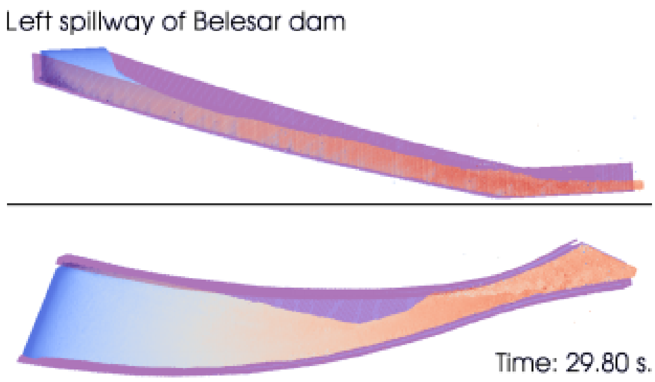


Figure 4: Velocity field of the left spillway of Belesar dam obtained with DualSPHysics.

First the water elevation (Figure 2) and the outflow of the spillways associated with the maximum flow of the Miño river were obtained using the numerical code Iber. The numerical domain defined for this simulation uses the real geometry of the dam and its impoundment (obtained from raster files).

Once the water elevation and the outflow near the spillways were computed, the behaviour of the left spillway of the dam was analysed using the numerical code DualSPHysics. First, two validation cases were considered (Figure 3): an ogee spillway and a broad crested weir. Figure 4 shows the velocity field of the left spillway obtained with DualSPHysics.

The main achievement of the present work is the combination of the strengths of Iber and DualSPHysics. Spillways can be simulated starting from the real topography of the impoundment. Iber provides an accurate description of the reservoir that is combined with the capabilities of DualSPHysics to describe extreme flows.

References

[1] Alejandro JC Crespo, José M Domínguez, Benedict D Rogers, Moncho Gómez-Gesteira, S Longshaw, R Canelas, Renato Vacondio, A Barreiro, and O García-Feal. Dualsphysics: Open-source parallel cfd solver based on smoothed particle hydrodynamics (sph). *Computer Physics Communications*, 187:204–216, 2015.

[2] E Bladé, L Cea, G Corestein, E Escolano, J Puertas, E Vázquez-Cendón, J Dolz, and A Coll. Iber: herramienta de simulación numérica del flujo en ríos. *Revista Internacional de Métodos Numéricos para Cálculo y Diseño en Ingeniería*, 30(1):1–10, 2014.

Construction of Two-dimensional SPH Numerical Wave Tank

Jingyu Wang, Fei Xu†, and Yang Yang
School of Aeronautics, Northwestern Polytechnical University, 710072, Xi'an, P.R.China
xufei@nwpu.edu.cn†

Abstract: The numerical wave tank is one of the effective tools to study the wave and its effect on the floating structure. Compared with the traditional methodology of CFD to construct the tank, the Smoothed Particle Hydrodynamics (SPH) method is superior in dealing with problems concerning complex boundary and large deformation.

In this paper, a numerical wave tank is established and verified based on the SPH method. The main contents of the paper are as follows. Firstly, the mathematical derivation of discrete calculation of Navier-Stokes control equations is completed, based on the fundamental concepts of SPH method and accompanied with relevant numerical processing techniques. The repulsive function is employed to simulate the interaction between the swinging plate and the water. Secondly, adopted the linear wave-maker theory, the calculation process of numerical simulation of swinging plate is given, in which the amplitude and circular frequency of the swinging plate can be obtained by solving the continuous equation. In the construction of the numerical tank, apart from the swinging plate at the upstream that excites the motion of water to generate the required waves, an artificial viscosity sponge layer is introduced at the downstream to remove the unwanted wave reflections from the boundaries, which ensures the wave characteristic of the workspace.

During the construction of the numerical pool, the following work is carried out: (i) The technical parameters in artificial viscosity, boundary treatment and free surface recognition in the SPH method are thoroughly investigated. (ii) Wave generation and propagation in the numerical wave tank is tested and confirmed with analytical results. (iii) The influence of swinging speed, the immersion of the swinging plate and the depth of water on the generation of various wave are discussed.

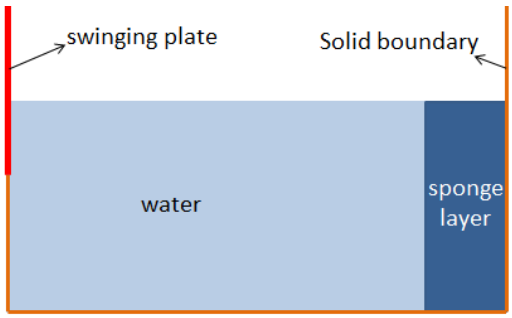


Figure 1: Model of two-dimensional wave tank

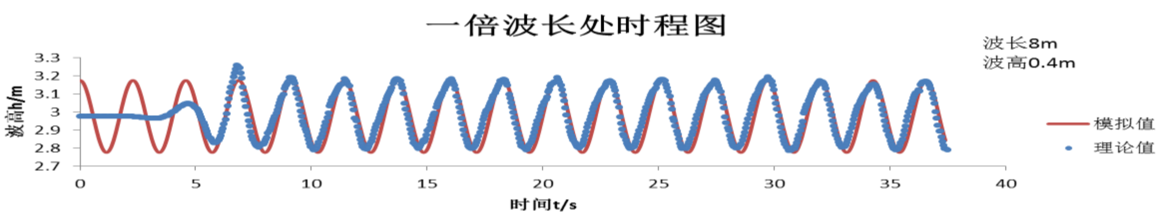


Figure 2: Results of linear regular wave generating

References

[1] Matteo Antuono, Andrea Colagrossi, Salvatore Marrone, and C Lugni. Propagation of gravity waves through an sph scheme with numerical diffusive terms. *Computer Physics Communications*, 182(4):866–877, 2011.

An SPH numerical wave-current tank

Ming HE, Hongshu WANG, Xifeng GAO†, Wanhai XU, and Yang SHI
The State Key Laboratory of Hydraulic Engineering Simulation and Safety, Tianjin University, Tianjin, China
gaoxifeng@tju.edu.cn†

Abstract: Numerical simulation of wave-current-structure interaction is of great value to coastal and ocean engineering, and is heavily dependent on the state of the art of the numerical wave-current tank (hereinafter denoted as "NWCT"). Up to now, NWCT has been successfully established based on conventional Eulerian methods. But for Lagrangian method, it still lies in the stage of wave-structure interaction or current-structure interaction.

In this paper, the SPH method is extended to build a NWCT for the first time. The propagation wave is generated by a piston-type wave-maker set at the upstream end of the tank. At the other end, an artificial damping layer is arranged to absorb the outgoing wave. An inflow region and a symmetrical outflow region are set below the bottom of the tank, lying on the opposite sides but between the wave-maker and damping layer. By imposing a constant velocity and the hydrostatic pressure in these two regions together with a cyclic boundary condition, the desired uniform current is implemented. In addition, two approaches are used to accelerate the stabilization of the flow field. One is applying a ramp function to the particle velocity in the inflow and outflow regions. The other is employing a temporary "rigid-lid treatment" to the fluid particles close to the water surface. In other words, the vertical positions and velocities of these near-surface particles are restricted, while their horizontal components are free. The former approach buffers the jet flow at the inlet and accordingly weakens the unnecessary sound wave. The latter approach reduces the quasi "U"-tube effect caused by the variations of surface elevation above the inlet and outlet. Thus, the simulation duration is shortened at least fivefold.

To validate the proposed model, a test case of surface wave following a current in a constant water depth is conducted. The calculated water surface profile and velocity distribution covering the whole water depth are compared with the experimental data of Umeyama [1] in Figure 1. The favourable agreement demonstrates that the established NWCT based on SPH method is capable of accurately reproducing wave-current coupling.

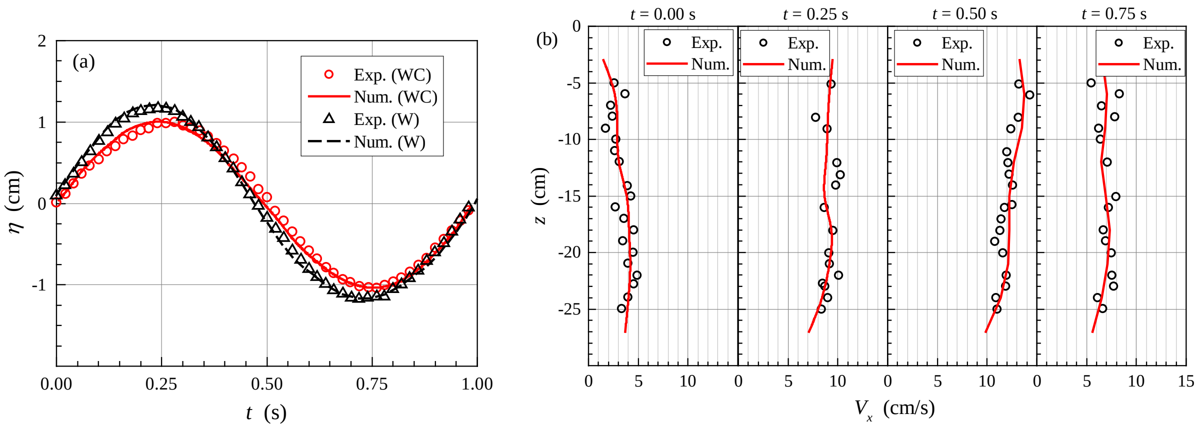


Figure 1: Comparison of numerical and experimental (a) wave surface profile and (b) horizontal-velocity distribution

References

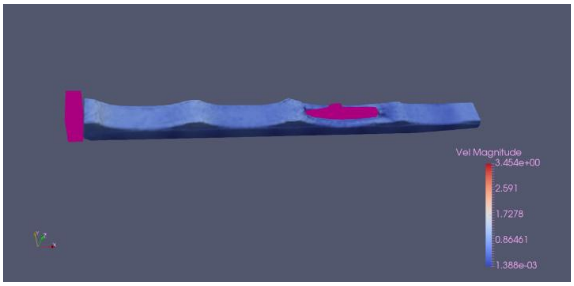
[1] Motohiko Umeyama. Coupled piv and ptv measurements of particle velocities and trajectories for surface waves following a steady current. *Journal of Waterway, Port, Coastal, and Ocean Engineering*, 137(2):85–94, 2010.

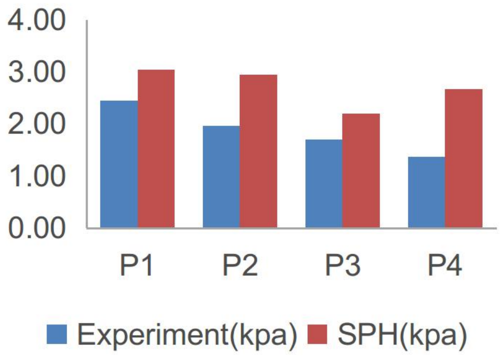
Numerical simulation of green water using SPH method

Liang-Jun WEN and Qiang-dong FENG
China Ship Scientific Research Center, Wuxi 214082,China

Abstract: When the ship is navigating in the sea, it is inevitable to meet the rough sea condition. In this case, the ship can suffer great danger from green water, and severe green water can cause structure damages. The flows tend to be highly dynamic, with large amounts of free surface deformation. A numerical method to simulate the phenomenon of the green water on deck is established by taking advantage of SPH method.

The paper aims to extend this method to deal with green water. The ship motion is given by potential flow theory. The ship model is regard as rigid body. Numerical results of water flow on deck and green water loads on the deck structure are compared with the corresponding experimental data. It is shown that the SPH method can be applied to describe and analyze green water.





References

[1] Ricardo B Canelas, Jose M Domínguez, Alejandro JC Crespo, Moncho Gómez-Gesteira, and Rui ML Ferreira. A smooth particle hydrodynamics discretization for the modelling of free surface flows and rigid body dynamics. *International Journal for Numerical Methods in Fluids*, 78(9):581–593, 2015.

LIST OF PARTICIPANTS

A. Colagrossi	Marine Tech. Research Inst., Italy
Aman Zhang	Harbin Engineering University, China
Benedict D. Rogers	University of Manchester, UK
Can Huang	Zhejiang University, China
Chao Shi	Xi'an Hi - Tech Institute, China
Chen Zhuang	Hunan University, China
Chengping Rao	Shanghai Jiao Tong University, China
Chunying Huang	Tianjin University, China
D. Le Touze	Ecole Centrale Nantes, France
Darcy Q. Hou	Tianjin University, China
Domenico Davide Meringolo	Ocean University of China, China
Feng Zhang	Xiamen University, China
Furen Ming	Harbin Engineering University, China
Fuzhen Chen	Xi'an Hi - Tech Institute, China
Gang Yang	Hunan University, China
Gaofeng Wei	Qilu University of Technology, China
Guoxing Zhang	Xi'an Hi - Tech Institute, China
Haiqiao Li	North University of China, China
Haisheng Fang	Huazhong University of Science & Technology, China
Han Cheng	Harbin Engineering University, China
Hongfu Qiang	Xi'an Hi - Tech Institute, China
Hongshu Wang	Tianjin University, China
Huabin Shi	Tsinghua University, China
Hualin Zheng	Xi'an Hi - Tech Institute, China
J. González-Cao	Universidade de Vigo, Spain
J. S. Chen	University of California San Diego, USA
J. M. Domínguez	Universidade de Vigo, Spain
Jianqiang Chen	China Aerodynamics Research and Development Center, China
Jiansong Wu	China University of Mining & Technology, China
Jiaru Shao	Chongqing University of Technology, China
Jiayuan Shen	Tianjin University, China
Jichao Ma	Qilu University of Technology, China
Jie Deng	Tianjin University, China
Jie Zhao	Xinjiang University, China

Jingjun Li	China Agricultural University, China
Jingyu Wang	Northwestern Polytechnical University, China
Li Zhou	Dalian University of Technology, China
Liangjun Wen	China Ship Scientific Research Center, China
Liuchao Qiu	China Agricultural University, China
Lu Wang	Northwestern Polytechnical University, China
M. De Leffe	Nextflow Software Nantes, France
Martin Rentschler	ANDRITZ Hydro R&D, Switzerland
Matthias Mimault	James Hutton Institute, UK
Ming He	Tianjin University, China
Mingyu Zhang	Institute of Applied Physics and Computational Mathematics, China
Moubin Liu	Peking University, China
Na Li	China University of Mining & Technology, China
Nan Qiao	Beijing Paratera, China
Nathan J. Quinlan	National University of Ireland Galway, Ireland
Pengnan Sun	Harbin Engineering University, China
Qing Fan	Sun Yat-sen University, China
S. Marrone	Marine Tech. Research Inst., Italy
Sam Raymond	Massachusetts Institute of Technology (MIT), USA
Shaohan Zhao	Monash University, Australia
Stefano Sibilla	University of Pavia, Italy
Ting Long	Hunan University, China
Weizhen Kong	Beijing Institute of Technology, China
Wenkui Shi	China Aerodynamics Research and Development Center, China
Wenqing Hu	Sun Yat-sen University, China
Wentao Zhang	Institute of Mechanics, Chinese Academy of Sciences, China
Xiang Chen	Shanghai Jiao Tong University, China
Xiangwei Dong	China University of Petroleum, China
Xiangyu Hu	Technical University of Munich, Germany
Xiao Wen	Shanghai Jiao Tong University, China
Xiaohu Guo	Science and Technology Facilities Council (STFC) , UK
Xiaojing Ma	Xinjiang University, China
Xing Zheng	Harbin Engineering University, China
Xinya Sun	Xi'an Hi - Tech Institute, China
Xiufeng Yang	Iowa State University, USA
Yanming Shen	China Aerodynamics Research and Development Center, China
Yaxuan Xing	Tianjin University, China
Yazhou Zhao	Chinese Academy of Science, China
Yi An	Institute of Mechanics, Chinese Academy of Sciences, China
Yingnan Wang	Monash University, Australia
Yingxuan Shi	Zhejiang University, China
Yongkang Hu	Triangle Tire R&D, China
Yongou Zhang	Wuhan University of Technology, China
Youlin Zhang	Shanghai Jiao Tong University, China

Yuantong Gu	Queensland University of Technology, Australia
Yuma Shimizu	Kyoto University, Japan
Zhi Zong	Dalian University of Technology, China
Zhibo Ma	Institute of Applied Physics and Computational Mathematics, China
Zhilang Zhang	Peking University, China
Zhiwen Cai	China Ship Scientific Research Center, China
Zhongguo Sun	Xian Jiaotong University, China
Zhongyi Liu	Huazhong University of Science & Technology, China