The 5th Symposium on Computational Marine Hydrodynamics

The 5th CMHL Symposium 2022

第五届 CMHL 船舶与海洋工程计算水动力学研讨会

Jan. 18, 2022, online virtual meeting

Organized by

Computational Marine Hydrodynamics Lab (CMHL)

Co-Organized by

Journal of Hydrodynamics (JHD)

Ocean College, Zhejiang University

Preface

Welcome to the 5th CMHL Symposium 2022 online virtual meeting!

Computational Marine Hydrodynamics Lab (CMHL) was founded by Prof. Decheng Wan in 2006. To meet the requirements of marine structures design for digitization, refinement, intelligence and system synthesis, CMHL has long been devoted to the researches of advanced CFD methods for marine hydrodynamics, developments of CAE software and platform, as well as applications of CAE software for complex flows in the fields of integrated ship, marine structures, underwater vehicles, offshore renewable energy devices, etc.

CMHL Symposium is named after “CMHL” and held every year to provide a forum for promoting scientific advancement, technological progress, information exchange, and innovative cooperation among scientists, researchers, engineers, developers, modellers and users of CAE software for solutions of marine hydrodynamics and other related fields. It is an attractive event opening to scientists, scholars, engineers, students, developers and users from universities, institutes as well as industries to exchange ideas and share recent advances on computational marine hydrodynamics and applications of CFD simulations for naval architecture and ocean engineering. The first CMHL Symposium was held on Dec. 27-28, 2018. The 2nd CMHL Symposium was held on May 7-8, 2019. The 3rd CMHL Symposium was held on Dec. 12-13, 2019. The 4th CMHL Symposium was held online (virtual meeting) on Jan. 14, 2021.

The coming 5th CMHL Symposium 2022 organized by CMHL and co-organized with *Journal of Hydrodynamics (JHD)* and *Ocean College of Zhejiang University* will be taken place online (virtual meeting) on Jan. 18, 2022. Prof. Chaoqun Liu from University of Texas at Arlington, USA, Prof. Rickard Bensow from Chalmers University of Technology, Sweden, and Prof. Hrvoje Jasak from University of Cambridge, UK, are invited to deliver one hour plenary lectures, as well as 10 outstanding researchers are invited to give 45 minute keynote presentations. Several papers based on the invited plenary lectures and keynote presentations will be published in *Journal of Hydrodynamics* as a special column for the 5th CMHL Symposium 2022.

Online Virtual Meeting Information

Microsoft Teams meeting room has been set up for the online virtual meeting of the 5th CMHL Symposium 2022 at 09:00-20:00 (GMT+8, Beijing time) of Jan. 18, 2022. You can scan the following QR code or click the following link to join in the Teams meeting rooms 30 minutes early as planned.

https://teams.microsoft.com/l/meetup-join/19%3ameeting_N2EyNWVhM2ItOGQ4NC00Y2RjLWI4NmQtMmVjZWI5NzIzNjIx%40thread.v2/0?context=%7b%22Tid%22%3a%22631e0763-1533-47eb-a5cd-0457bee5944e%22%2c%22Oid%22%3a%227b5ad9ee-3a3e-49a9-ab6b-c6f0442c022e%22%7d

We also prepare a live broadcast of the 5th CMHL Symposium 2022 on the Bilibili website. In case the above Teams meeting rooms are full and you cannot join in, you can watch the live stream online via the following QR code or link:

http://live.bilibili.com/24017914

Instruction for Invited Speakers

Each plenary lecture is allocated 60 minutes (50-min presentation + 10-min discussion), and each keynote presentation is allocated 45 minutes (40-min presentation + 5-min discussion). The invited speakers are suggested to join in above Teams meeting rooms 30 minutes before your scheduled plenary lecture or keynote presentation.

Secretariat of the 5th CMHL Symposium 2022

Dr. Weiwen Zhao, CMHL, Shanghai Jiao Tong University, Email: weiwen.zhao@sjtu.edu.cn
Dr. Liushuai Cao, CMHL, Shanghai Jiao Tong University, Email: liushuaicao@sjtu.edu.cn
Dr. Zhen Ma, Editorial Board, Journal of Hydrodynamics (JHD), Email: mazh8888@sina.com
Dr. Yangyang Gao, Ocean College, Zhejiang University, Email: yygao@zju.edu.cn

Programm of the 5th CMHL Symposium 2022

Beijing Time 09:00-19:40, Tuesday, Jan. 18, 2022, Online virtual meeting

09:00-09:05 Opening Speech and Chair Prof. Decheng Wan

09:05-10:05 Plenary Lecture 1 (Jan. 17, 19:05-20:05 CST)

Liutex-Based New Fluid Kinematics and New Fluid Dynamics

Prof. Chaoqun Liu, Director of Center for Numerical Simulation and Modeling at University of Texas at Arlington, USA

10:05-10:50 Keynote Presentation 1 (Jan. 17, 20:05-20:50 CST)

Intelligent Fluid Mechanics for Next-Generation Aerial/Marine Exploration and Exploitation: A confession by a "lazy" fluid mechanist

Dr. Dixia Fan, Assistant Professor at Queens University, a Research Scientist at MIT Sea Grant Initiative and ABS Research Fellow, USA

10:50-11:35 Keynote Presentation 2 (Jan. 17, 20:50-21:35 CST)

Integrating Physics into Deep Learning for Modeling Scientific Problems

Dr. Chengping Rao, Applied Scientist in Amazon, USA

11:35-12:20 Keynote Presentation 3 (Jan. 18, 12:35-13:20 Japan Time)

Lattice Boltzmann Simulation Using Actuator Line Model for Tidal Current Turbines

Dr. Seiya Watanabe, Assistant Professor at Research Institute for Applied Mechanics (RIAM), Kyushu University, Japan

12:20-13:05 Keynote Presentation 4

Numerical Improvements of Turbulent Two-Phase Flow Past Surface-Piecing Vertical Cylinder

Dr. Weiwen Zhao, CMHL, Shanghai Jiao Tong University, China

13:05-13:50 Keynote Presentation 5

Three-Dimensional Numerical Investigation on Flow Past Two Side-by-Side Curved Cylinders

Dr. Yangyang Gao, Associate Professor of Ocean College, Zhejiang University, China

13:50-14:35 Keynote Presentation 6 (Jan. 18, 6:50-7:35 France Time)

Ship Maneuvering Simulation with Simplified Propeller Model

Dr. Ganbo Deng, LHEEA lab, Ecole Centrale de Nantes, France

14:35-15:20 Keynote Presentation 7

High-Pressure Gas Bubble Dynamics and Its Applications

Prof. A-Man Zhang, Distinguished Professor of Changjiang Scholars, Harbin Engineering University, China

15:20-16:20 Plenary Lecture 2 (Jan. 18, 08:20-09:20 Sweden Time)

Numerical Assessment of Cavitation Nuisance in Marine Propulsion Systems

Prof. Rickard Bensow, Department of Mechanics and Maritime Sciences, Chalmers University of Technology, Sweden

16:20-17:20 Plenary Lecture 3 (Jan. 18, 08:20-09:20 London Time)

Non-Linearity in Irregular Sea States Coupled CFD-HOS Models

Prof. Hrvoje Jasak, The Cavendish Laboratory, University of Cambridge, UK

17:20-18:05 Keynote Presentation 8 (Jan. 18, 09:20-10:05 London Time)

Unraveling the Fluid-Structure-Interaction Mystery of Fish Swimming

Prof. Qing Xiao, Department of Naval Architecture, Ocean and Marine Engineering, Strathclyde University, UK

18:05-18:50 Keynote Presentation 9 (Jan. 18, 10:05-10:50 London Time)

Development of Advanced CFD Tools for Turbulent Multiphase Flows and Wave-Structure Interaction

Dr. Zhihua Xie, Senior Lecturer in the School of Engineering, Cardiff University, UK

18:50-19:35 Keynote Presentation 10 (Jan. 18, 19:50-20:35 Japan Time)

Entirely Lagrangian Meshfree Methods for Hydroelastic Fluid-Structure Interactions - Recent Advances and Future Perspectives

Dr. Abbas Khayyer, Associate Professor at Applied Mechanics Laboratory in Department of Civil and Earth Resources Engineering, Kyoto University, Japan

19:35-19:40 Closing Speech Prof. Decheng Wan

Introduction of Invited Speakers

Prof. Chaoqun Liu

Dr. Chaoqun Liu received both BS (1968) and MS (1981) from Tsinghua University, Beijing, China and PhD (1989) from University of Colorado at Denver, USA. He is currently the Tenured and Distinguished Professor and the Director of Center for Numerical Simulation and Modeling at University of Texas at Arlington, Arlington, Texas, USA. He has worked on high order direct numerical simulation (DNS) and large eddy simulation (LES) for flow transition and turbulence for almost 30 years since 1990. He was the Chairman of the First and Third AFOSR International Conference on DNS/LES. As PI, he has been awarded by NASA, US Air Force and US Navy with 50 federal research grants of over 5.7 million US dollars in the United States. He has published 14 professional books, 128 journal papers and 152 conference papers. He is the founder and major contributor of Liutex and the third generation of vortex definition and identification methods including the Omega, Liutex/Rortex, Modified Liutex-Omega, Liutex-Core-Line methods, RS vorticity decomposition and UTA R-NR decomposition, Principal Coordinate, and Principal Decomposition of velocity gradient tensor. He is also the founder of new fluid kinematics.

Plenary Lecture 1:
Liutex-Based New Fluid Kinematics and New Fluid Dynamics

Liutex is a physical quantity like velocity, vorticity, pressure, temperature, etc. describing local fluid rotation or vortex, which was ignored for centuries. Liutex was defined by the UTA Team in 2018 as a vector for vortex. Its direction is local rotation axis and magnitude is twice local angular rotation speed. As the third generation of vortex definition and identification, Liutex has been widely applied for visualization of vortex structure to replace the first generation or vorticity which cannot distinguish shear from rotation and the second generation such as Q, Δ, $\lambda 2$ and λci methods, which are all scalar without rotation axis, dependent on threshold and contaminated by shear and stretching. A number of new vortex identification methods have been developed especially Modified Liutex-Omega method which is threshold insensitive and Liutex-Core-Line method which is unique and threshold-free. According to Liutex vector, a unique coordinate system called Principal Coordinate can be set up and consequent Principal Decomposition of velocity gradient tensor can be made. Being different from classical fluid kinematics, the Liutex-based new fluid kinematics decomposes the fluid motion to a rotational part and non-rotational part (UTA R-NR decomposition). The non-rotational part can be further decomposed to stretching and shear including symmetric shear and anti-symmetric shear in contrast with the classical fluid kinematics which decomposes fluid motion to deformation and vorticity which was misunderstood as rotation. As fluid stress is determined by fluid strain, the new fluid kinematics will determine the viscous force. Traditional Navier-Stokes (NS) equations define the stress based on Stokes's assumptions that the stress is supposed proportional to strain, and both strain and stress tensors are symmetric. There are several questions with NS, which include: 1. Both symmetric shear terms and stretching terms in strain and stress are coordinate-dependent and thus not Galilean invariant; 2. The physical meaning of both diagonal and off-diagonal elements are not clear, which is coordinate-dependent; 3. It is hard to measure the strain and stress quantitatively, and viscosity is really measured by vorticity not by symmetric strain; 4. There is no vorticity terms in NS, which plays important role in fluid flow especially for turbulent flow. The new proposed governing equations for fluid dynamics use vorticity tensor only, which is anti-symmetric. The advantages include: 1. Both shear and stress are anti-symmetric, which are Galilean invariant and independent of coordinate rotation; 2. The physical meaning of off diagonal elements is clear, which is anti-symmetric shear stress, 3. Viscosity coefficients are obtained by experiment which uses vorticity, 4. The vorticity term can be further decomposed to rigid rotation and anti-symmetric shear, which are important to turbulence research, 5. The computation cost for viscous term is reduced to half as the diagonal terms are all zero as six elements are reduced to three. Several computational results are made, which clearly demonstrate both NS and new governing equations have exactly same results. In fact, the new governing equation is identical to NS in mathematics, but its physical assumptions are just the opposite of NS. It is recommended to use the new governing equations to replace Navier-Stokes equations. New fluid dynamics by considering high order terms for turbulent flow is still under development.

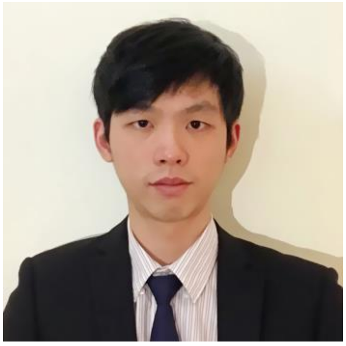
Dr. Dixia Fan

Dr. Dixia Fan obtained his Ph.D. (2019) and MSc (2016) from MIT Mechanical Engineering and BSc (2013) from Shanghai Jiaotong University Naval Architecture, Ocean and Civil Engineering. He is currently an assistant professor at Queens University, a research scientist at MIT Sea Grant Initiative and the American Bureau of Shipping (ABS) research fellow. In 2022, He will join Westlake University as an assistant professor. He founded the MIT "lab of pink coach" in 2019, featuring the world's first intelligent towing tank (ITT) and is in charge of the i4-FSI lab at Queen's University (intelligent, informational, integrative, and interdisciplinary fluid-structure interaction). His research interests focus on physics-informed (and -informative) machine learning and bio-inspired design of vortical flow control and sensing for marine and aerospace applications. Recently he has been awarded the de Florez prize from MIT and the Nico van Wingen prize from Society of Petroleum Engineers (SPE), and his work has been featured in multiple media, including the cover page of the 40 years anniversary of the Discover Magazine.

Keynote Presentation 1:
Intelligent Fluid Mechanics for Next-Generation Aerial/Marine Exploration and Exploitation: A confession by a "lazy" fluid mechanist

Fluid motions can be seen everywhere in our environmental and industrial processes and account for a big part of the human energy consumption that, for example, propels aircraft and ships. However, our understanding and control of the flow motion are still limited due to the inherent spatial and temporal non-linearity and multiscality of fluid-related problems. In this talk, I will discuss how automated science and intelligent bio-inspired design may provide a potential paradigm shift in fluid-structure interaction (FSI) research and can help create next-generation marine and aerospace applications to explore and exploit our resource safely, efficiently and environmental-friendly. I will provide two examples: 1) the development of the world's first intelligent towing tank as part of an effort to construct a digital twin for marine riser vortex-induced vibration monitoring, and 2) the vortical flow control and sensing of flapping wings for novel dual aerial/aquatic vehicles. Finally, I will close my talk with my humble vision of creating future intelligent fluid mechanics research.

Dr. Chengping Rao



Dr. Chengping Rao is an Applied Scientist in Amazon. He received his Ph.D. in Mechanical Engineering from Northeastern University in 2021. His research lies on the interdisciplinary area of the computational physics and artificial intelligence, including the physics-informed neural network (PINN), explainable and interpretable DL for data-driven modeling of complex systems and governing partial derivative equation (PDE) discovery. He is also interested in the applications of Bayesian optimization and reinforcement learning to scientific problems. Before his Ph.D. study, he obtained B.S. and M.S. degrees in Naval Architecture and Ocean Engineering at Huazhong University of Science and Technology (2015) and Shanghai Jiao Tong University (2018) respectively.

Keynote Presentation 2:
Integrating Physics into Deep Learning for Modeling Scientific Problems

In recent years, successful applications of deep learning (DL) have inspired scientists to explore the possibilities of applying DL approaches to modeling scientific problems. Existing studies have revealed that to leverage the physics into the DL makes a good supplement to the traditional numerical methods (e.g., finite element, finite volume method) which primarily rely on partial differential equations (PDEs). While DL models are usually trained in a purely data-driven manner, integrating physics with them for simulating scientific problems could bring several benefits such as (i) physics constraints could regularize the over-parameterized model and hence mitigate the overfitting issue commonly seen in DL; (ii) physics information could also effectively reduce the amount of data needed for training the model; (iii) the resultant physics-informed DL models feature better interpretability and generalizability compared with the conventional black-box model. In this presentation, I will specifically discuss the physics-informed neural network (PINN) – one of the most prominent approaches in this area. Numerical examples would be provided to demonstrate the effectiveness of PINN on simulating a variety of physical systems, such as the laminar flow, vortex shedding and seismic wave propagation. Several applications including the data-driven simulations and solving inverse problems are presented to further exemplify the advantages of PINN over traditional numerical methods.

Dr. Seiya Watanabe

Dr. Seiya Watanabe is an assistant professor at Research Institute for Applied Mechanics (RIAM), Kyushu University, Japan. He received his B.S. in mechanical engineering from Gunma National College of Technology, M.S. in energy science from Tokyo Institute of Technology, and Ph.D. degree in mechanical engineering from Tokyo Institute of Technology in Japan. After his Ph.D. study, he worked as an appointment researcher at Global Scientific Information and Computing Center in Tokyo Institute of Technology. He started to work for RIAM from January 2020. His current research interests include ocean renewable energy technologies, computational fluid dynamics by lattice Boltzmann method, and high-performance computing.

Keynote Presentation 3:
Lattice Boltzmann Simulation Using Actuator Line Model for Tidal Current Turbines

In numerical simulations of tidal current power generation farms, large-scale CFD simulations with a high-resolution grid are required to calculate the interactions between tidal turbine wakes. In this study, we develop a numerical simulation method for tidal current turbines using the lattice Boltzmann method (LBM), which is suitable for large-scale CFD simulations. Turbines are modeled using the actuator line model (ACL), which represents each blade as a point cloud. In order to validate our LBM-ACL model, we simulate the NREL-5MW turbine, which has been analyzed and tested in many previous studies, and LBM-ACL simulation results are in good agreement with an NS-ACL simulation. We conducted a water tank experiment with two turbines and confirmed that LBM simulations could reproduce the experiment in terms of wake interaction and wave effects on turbine performance. Our LBM-ACL code can run a large-scale LES with 1.48 billion grid points for a tidal current power farm using the multi-GPU system ITO subsystem B at Kyushu University.

Dr. Weiwen Zhao

Dr. Weiwen Zhao is currently a research associate in Computational Marine Hydrodynamics Laboratory (CMHL). He received a BSc and MSc in Naval Architecture and Ocean Engineering from Huazhong University of Science and Technology, and a PhD from Shanghai Jiao Tong University. He is currently a member of the 30th specialist committee on Ocean Renewable Energy in the

International Towing Tank Conference (ITTC). He is also an editorial board member of the Journal of Hydrodynamics. His research interest mainly focus on the development and application of high fidelity numerical methods for marine dynamics, such as the Ghost Fluid Methods of incompressible two-phase flow, hybrid RANS/LES methods, dynamic overset grid methods, wave-current interaction, vortex identification and visualization.

Keynote Presentation 4:
Numerical improvements of turbulent two-phase flow past surface-piecing vertical cylinder

Strong turbulence near an air-water interface, characterized by large Froude (Fr) and Weber number (We), leads to significant interactions and exchanges between gas and liquid, resulting in measurable air entrainment. Air entrainment influences a number of physical processes in the nature, including air-sea gas transfer, production of the sea-salt aerosol and scavenging of biological surfactant. The key factor in controlling these processes is the size distribution of entrained bubbles. In this study, both theoretical and numerical studies on air entrainment and bubble size distribution in strong free-surface tu apturing, an improved turbulent model for two-phase flow to avoid excessive turbulence damping near free surface. These numerical improvements are applied to two-phase flow past fixed and mounted surface-piercing circular cylinder for validation.

Dr. Yangyang Gao

Dr. Yangyang Gao is Associate Professor of Ocean College, Zhejiang University. Dr. Gao obtained the PhD degree in Harbour, Coastal and Offshore Engineering from Ocean University of China in 2011, and worked as a research fellow in Nanyang Technological University, Singapore in 2011-2013. The research interest of Dr. Gao is hydrodynamics, fluid-structure interaction, vortex-induced vibration of risers and pipelines, offshore wind turbine foundations. Dr. Gao has published more than 40 journal papers, including Ocean Engineering, Applied Ocean Research, Journal of Fluids Engineering(ASME) etc.

Keynote Presentation 5:
Three-dimensional numerical investigation on flow past two side-by-side curved cylinders

Three-dimensional numerical simulations of flow past two side-by-side convex curved cylinders are performed for various spacing ratios ($1.25 \leqslant L/D \leqslant 5$) and Reynolds numbers ($100 \leqslant Re \leqslant 500$). A comprehensive investigation of the effects of spacing ratio and Reynolds number on the wake flow features, pressure coefficients and axial flow velocity is conducted. Four flow patterns: a single bluff body pattern, biased flow pattern, coupling vortex shedding pattern and co-shedding pattern are identified for the vertical straight sections. The flows along the curved sections of two cylinders are classified into five flow regimes: normal shedding, vortex dislocation, oblique shedding, non-shedding and instability of shear layer regimes. It is revealed that at L/D=1.5, the switchover of the gap flow deflection along the curved spans leads to the oblique vortex shedding pattern for Re = 100 and 300. The Reynolds stress intensity, vortex strength, and the mean pressure coefficient are found to be reduced significantly from the vertical to the horizontal sections of the cylinders. With the increase of the spacing ratio, the axial flow velocity increases along the curved span of two cylinder, whereas the absolute value of base coefficient decreases.

Dr. Ganbo Deng

Dr. Ganbo DENG was graduated at Tsinghua in 1984. He obtained his PHD at Ecole Centrale de Nantes (France) in 1989. Since then, he has been working in the same institute as a research scientist. His research topics cover different aspects concerning CFD simulation for incompressible flow for marine applications such as discretization scheme with finite volume as well as high order discretization scheme such as DG, HDG and spectral volume approach, fully coupled resolution for incompressible Navier-Stokes equation, fast linear solver, overset algorithm, turbulence modelization, RANSE simulation with simplified propeller model, etc. As one of the developers of a commercial flow solver, he also has an excellent expertise in RANSE simulation for marine engineering applications.

Keynote Presentation 6:
Ship maneuvering simulation with simplified propeller model

Ship maneuvering prediction with CFD has been performed with a simplified propeller model based on a open water performance curve in order to reduce CPU cost. To assess the accuracy of such prediction, comparison has been made with CFD prediction using actual propeller approach with which the rotating propeller is directly simulated, as well as with the measurement data. Both zigzag motion and turning circle motion have been simulated for two different configurations, namely the KCS and the ONRT test cases. For the ONRT test case equipped with a twin-screw propeller, as the inflow velocity to the propeller is more uniform, the simplified propeller model can simulate the action of the propeller with good accuracy. Although the side force is not taken into account in the simplified propeller model, not too much deterioration in the predicted ship motion is found due to error cancellation. For the single screw propeller configuration where the propeller operates in a non-uniform flow in the middle of the ship in the wake (the KCS test case), propeller thrust predicted by the simplified propeller model is lower. The comparison with the measurement data reveals that in addition to propeller model, one of the main sources of error in CFD prediction for ship maneuvering application comes from the prediction of rudder force. For half spade rudder, flow around the rudder is forced to separate from the gaps between the mobile part and the fixed part of the propeller due to the pressure. CFD simulation with the SST turbulence model under predicts both the axial force and the side force when flow separate around the rudder, especially when the rudder angle is high. More accurate prediction is obtained for the ONRT test case where a spade propeller is used.

Prof. A-Man Zhang

Porf. A-Man ZHANG is a Distinguished Professor of Changjiang Scholars at Harbin Engineering University. He was funded by the National Science Foundation for Distinguished Young Scholars and was selected as the Science and Technology Innovation Leading Talent by the National "Ten Thousand Talents Program". The main research interest of him includes bubble dynamics and fluid-structure interaction. In this area of research, he has managed over 40 research projects, including the Key Projects sponsored by the National Key R&D Program of China, the National Natural Science Foundation of China and so on. He has published over 200 articles in journals including JFM, JCP, PRF and POF, etc and serves as the Associate Editor for the journal CMES and as a member in the editor board of APOR, JHD, etc. With more than 6000 citations, he was selected as one of the most highly-cited researchers in China. Due to his contribution in the research area, he was rewarded with the National Innovation & Competition Award, the Xplorer Prize, the Second-Class Prize of the National Award for Technological Invention, the National Prize for Progress in Science and Technology, and so on.

Keynote Presentation 7:
High-pressure gas bubble dynamics and its applications

High-pressure bubbles have wide and important applications in shipbuilding, ocean engineering and other fields. However, due to the complexity of bubble dynamics, many difficult mechanical problems still remain unsolved. For this reason, targeted at problems in the field of shipbuilding and ocean engineering such as underwater explosion bubble and structure damage, exploration of deep-sea resources with high-pressure air-gun bubbles, ice breaking with bubbles, and high-speed object cross-media water entry, we established models and methods of near-boundary bubble dynamics combining theoretical analysis, numerical calculation and model tests, revealed dynamical behavior and load properties of high-pressure bubbles near boundaries, unveiled mechanical laws and mechanisms of structure damage subject to bubbles, and presented corresponding engineering applications, with an aim to provide references for related researches in bubble dynamics.

Prof. Rickard Bensow

Prof. Rickard Bensow is full professor in hydrodynamics since 2011 at the Department of Mechanics and Maritime Sciences at Chalmers University of Technology, Sweden. He is heading Chalmers research on ship resistance and propulsion with specialisation on simulation of cavitation and cavitation erosion, as well as developing scale resolving simulation methodologies, such as PANS and WMLES, for ship hydrodynamics. He is the director of the Kongsberg University Technology Centre in Computational Hydrodynamics since 2010 (previously within the Rolls-Royce group). He has currently almost 200 papers listed in Google scholar and about 2000 citations.

Plenary Lecture 2:
Numerical assessment of cavitation nuisance in marine propulsion systems

Allowing propeller cavitation is necessary to achieve acceptable efficiency which makes it crucial to be able to control its negative effects, primarily related to induced pressure pulses, radiated noise, and erosion. In this talk, I will review our latest development and results related to predicting these effects in CFD. This includes a discussion on the physical mechanisms behind these nuisances that needs to be captured in CFD and thereby what the requirements will be on resolution and modelling to make this feasible.

Prof. Hrvoje Jasak

Prof. Hrvoje Jasak is a lecturer at the Department of Physics, University of Cambridge and director of Wikki Ltd (UK). His research group on fundamental developments of CFD methodology in complex coupled systems, with applications in turbo-machinery, naval hydrodynamics, non-linear solid mechanics. The work includes numerical modelling and linear solver technology for High-Performance Computing method development. Hrvoje is a practical programmer and a member of OpenFOAM Governance structure. Hrvoje Jasak graduated mechanical engineering at the University of Zagreb in 1992. He completed his PhD in Computational Fluid Dynamics in prof. Gosman's group at Imperial College in 1996. He is one of two original authors of OpenFOAM, a leading Open Source CFD package today. His research interests are focused on numerical simulation in Continuum Mechanics, specifically on the Finite Volume discretisation and OpenFOAM.

Plenary Lecture 3:
Non-Linearity in Irregular Sea States Coupled CFD-HOS Models

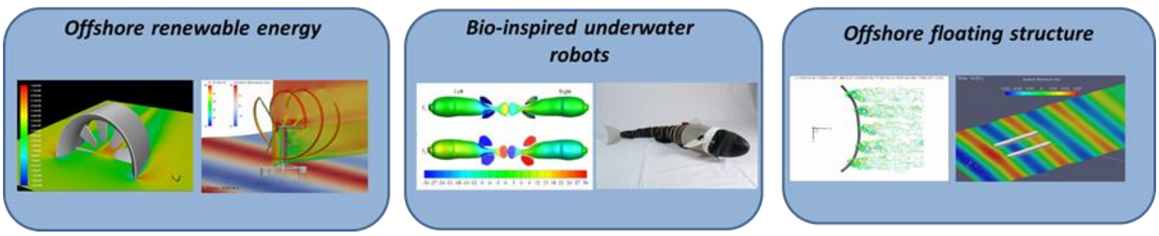
Over the last 15 years, maturity of Computational Fluid Dynamics (CFD) simulation tools for naval hydrodynamics applications has developed considerably and is routinely deployed in design and optimisation of ship hulls and off-shore structures: this is a clear success. Current points of research and validation of naval hydrodynamics CFD are shifting towards simulation of full-scale ships under self-propulsion and active steering. Further, off-design loading conditions such as slamming, hydro-elasticity, green water and freak wave loads can be modelled with some confidence. Aspects of turbulence modelling in free surface flows, presence of captured air bubbles in water, cavitation on propulsors and other similar enhancements of the flow model can be used with a good level of confidence. However, questions arise in terms of applicability and fidelity of simulations with reference to practical use cases. Are the simulations – in addition to being accurate – actually relevant for realistic load conditions in operation? Are we modelling the stochastic nature of wave loading in an appropriate manner? While, for example, external aerodynamics CFD for cars does not account for wind direction, such uncertainties in loading conditions in naval hydrodynamics are order of magnitude larger. Optimising ship resistance on calm seas or calculating sea-keeping in regular waves does not give a fair representation of actual load conditions. In this talk, we shall address the problems of realistic representation of load condi- tions and compare them with routine naval hydrodynamics CFD today. The presenta- tion includes various examples of non-linearity and coupling across the scales needed for practical CFD. Ideas for future direction of simulations and examples of High Order Spectrum (HOS) non-linear potential flow solvers to practical CFD shall be presented.

Prof. Qing Xiao

Dr. Qing Xiao is Professor of Marine Hydrodynamics in the Department of Naval Architecture, Ocean and Marine Engineering (NAOME) at Strathclyde University, United Kingdom. She is leading a Computational Fluid Dynamics & Computational Structural Dynamics research group at NAOME. Her major research interests are computational fluid dynamics with particular interests in bioinspired hydrodynamics, marine renewable energy devices and offshore fluid-structure-interaction problems. Her current research projects are funded by Royal Society, Engineering and Physical Sciences Research Council (EPSRC), Royal Academy Engineering and industry companies in UK, France and USA. Professor Xiao has published over 140 original research papers at peer reviewed journals. She is a senior Member of the AIAA, a Member of ASME. Professor Xiao is editorial member of Ocean Engineering Journal, associated editor of Journal of Offshore Mechanics and Arctic Engineering (JOMAE), International Towing Tank Committee (ITTC) Ocean Engineering Committee member.

Keynote Presentation 8:
Unraveling the fluid-structure-interaction mystery of fish swimming

In this talk, the research work currently being performed in a CFD research group at NAOME department in the Strathclyde University UK will be introduced. We will focus our investigation on using numerical modelling methods to tackle the challenges around the understanding of fish swimming for bio-inspired aquatic locomotion systems. Other CFD and Computational Structure Dynamics (CSD) studies in ocean renewable energy will be described briefly.



Dr. Zhihua Xie

Dr. Zhihua Xie is a Senior Lecturer in the School of Engineering at Cardiff University, where he was appointed to a lectureship in 2017. He obtained his PhD in Computational Fluid Dynamics (CFD) at the University of Leeds in 2010, funded by the Marie Curie EST Fellowship. After that, he worked as a research associate at Cardiff University from 2010 to 2012 and at Imperial College London between 2013 and 2016. His research interests span a broad range of topics in the development of novel CFD methods, multiphase flows, turbulence modelling and simulation, fluid-structure interaction, and high-performance computing with applications in hydraulic, coastal, and ocean engineering. He has published over 60 peer-reviewed journal papers spanning from reputable numerical method and fluid mechanics journals (JCP, JFM etc). His research is funded by EPSRC, Royal Society and British Council. He is a member of the EPSRC Peer Review College, UK Turbulence Consortium, Leadership Team of IAHR Technical Committee on Coastal and Maritime Hydraulics, Technical Program Committee for ISOPE, and an active reviewer for over 40 international leading journals.

Keynote Presentation 9:
Development of advanced CFD tools for turbulent multiphase flows and wave-structure interaction

Turbulent multiphase flows where two or more fluids have interfacial surfaces are often found in environmental and industrial engineering applications. The objective of this study is to investigate the fluid dynamics of three-dimensional (3D) two- and three-phase multiphase flow problems, such as bubbles, droplets, liquid jet, falling liquid films, breaking waves and their interaction with fixed or moving complex geometries. In this talk, three numerical codes developed (one for finite volume method, one for control-volume finite-element method, and the other for finite difference method) for solving 3D Navier-Stokes equations with interface capturing are briefly introduced first, with different techniques to deal with the solid and air-water boundaries. And then the FVM code Xdolphin3D will be presented in detail, with a focus on the 3D wave-structure interaction. The large-eddy simulation approach has been adopted for turbulent multiphase flows, for which the spatially filtered Navier–Stokes equations are solved and the dynamic Smagorinsky sub-grid scale model is employed to compute the unresolved scales of turbulence. Both algebraic and geometric volume-of-fluid methods together with the continuum surface force model for the surface tension have been implemented to capture the air-water interface, which can accurately simulate a range of free-surface flow problems, including the turbulence statistics and coherent structures. For the simulation of complex geometries in a fixed Cartesian grid, a novel Cartesian Cut-cell method has been developed, which allows the representation of fixed or moving bodies in the flow. The code has been parallelised using MPI and a domain decomposition technique. Some numerical examples are shown to provide some insight into physics and also to demonstrate the capability of the numerical tool developed.

Dr. Abbas Khayyer

Dr. Abbas Khayyer is an Associate Professor at Applied Mechanics Laboratory in Department of Civil and Earth Resources Engineering at Kyoto University. He holds a BSc in Civil Engineering (2002), MSc in Hydraulic Structures (2005) and PhD in Civil/Coastal Engineering (2008). After obtaining his PhD in 2008, Abbas continued his research as a postdoctoral research fellow at Kyoto University for one year. In 2009, he was appointed as a Lecturer and then in April 2013 he was promoted to a tenured Associate Professor at Applied Mechanics Laboratory at the same department and university. Since then, Abbas has been a co-chair of Applied Mechanics Laboratory. The research interests of Abbas mainly include Computational Fluid and Structure Dynamics, Particle Methods and Fluid-Structure Interactions. He has published more than 80 journal papers and 50 international conference articles and currently has an h-index of 30 in Google Scholar and 28 in Scopus. Abbas has been listed in world's top 2% scientists lists by Stanford University in 2020 and 2021 in both yearly and full career lists. Abbas received the international prestigious C.H. Kim award by ISOPE in 2018 for his outstanding contribution to CFD in ocean engineering. He is an associate editor of Applied Ocean Research, one of the editors of Coastal Engineering Journal (CEJ), an associate editor of international journal of offshore and polar engineering, and an editorial board member for several international journals including ocean engineering and European journal of mechanics B/Fluids. Abbas is also a steering committee member of SPHERIC as the leading international community on particle methods.

Keynote Presentation 10:
Entirely Lagrangian Meshfree Methods for Hydroelastic Fluid-Structure Interactions - Recent Advances and Future Perspectives

This talk summarizes the latest developments corresponding to hydroelastic Fluid-Structure Interaction (FSI) solvers established within the context of Lagrangian meshfree or particle methods. In specific, the developments made for establishment of entirely Lagrangian meshfree FSI solvers comprising of projection-based Newtonian fluid models and Newtonian/Hamiltonian structure models will be discussed for FSI encountered in marine engineering including hydroelastic slamming. The achieved advances can be viewed with respect to three aspects of reliability, adaptivity and generality. Regarding reliability, the importance of rigorous achievements of stability, accuracy, consistency, conservation and convergence would be outlined. The emphasis will be on coherent and scrupulous validations through consideration of reliable analytical and experimental reference solutions. As for adaptivity, achievement of adaptive and consistent solvers will be presented within the context of two well-known particle methods, namely, SPH and MPS. Regarding generality, advances corresponding to composite structures and material anisotropy would be discussed. In specific, the advantageous features of variationally consistent Hamiltonian structure models for reliable simulations of composite structures would be highlighted. Finally, the future perspectives for continuous development of entirely Lagrangian meshfree FSI solvers will be discussed.