The 4th Symposium on Computational Marine Hydrodynamics

The 4th CMHL Symposium 2021

第四届 CMHL 船舶与海洋工程计算水动力学研讨会

Jan. 14, 2021, online (virtual meeting)

https://dcwan.sjtu.edu.cn/News/showDetail.aspx?tid=868

Organized by

Computational Marine Hydrodynamics Lab (CMHL),
Shanghai Jiao Tong University (SJTU)

Co-organized by

Journal of Hydrodynamics (JHD)

Ocean College, Zhejiang University

Preface

Welcome to the 4th CMHL Symposium online virtual meeting!

SJTU Computational Marine Hydrodynamics Lab (CMHL) was founded 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 and Prof. Changhong Hu from Kyushu University was invited to deliver one hour plenary lecture. The 2nd CMHL Symposium was held on May 7-8, 2019 and Prof. Frederick Stern from University of Iowa was invited to deliver one hour plenary lecture. The 3rd CMHL Symposium was held on Dec. 12-13, 2019 and Prof. Atilla Incecik and Prof. Changhong Hu were invited to deliver one hour plenary lecture.

The coming 4th CMHL Symposium 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. 14, 2021. Prof. Frederick Stern and Prof. Michel Visonneau are invited to deliver one hour plenary lectures, and other 9 outstanding researchers are invited to give 45 minute keynote presentation. Several papers based on the invited plenary lectures and keynote presentations will be published in JHD as a special column for the 4th CMHL Symposium.

Zoom Meeting Information

If you have not Zoom before then need to install Zoom, you can download and install Zoom here: https://zoom.us/download.

Two Zoom meeting rooms have been set up for online virtual meeting of the 4th CMHL Symposium. The first Zoom meeting room is for time 09:00-17:00 (GMT+8, Beijing time) of Jan. 14, and the second Zoom meeting room for time 17:00-19:20 (GMT+8, Beijing time) of Jan. 14, as follows:

First Zoom meeting room:

Time	09:00-17:00, Beijing Time (GMT+8), Jan. 14, 2021
Zoom ID	692 8221 8756
Password	796846
One-click join link	https://zoom.com.cn/j/69282218756

Second Zoom meeting room:

Time	17:00-19:20, Beijing Time (GMT+8), Jan. 14, 2021
Zoom ID	968 1862 8364
Password	581440
One-click join link	https://zoom.com.cn/j/96818628364

You can join in above Zoom meeting rooms 30 minutes early as planned. We also prepare a live broadcast of the 4th CMHL Symposium on the Bilibili website. In case the above Zoom meeting rooms are full and you cannot join in, you can watch the live stream online at: http://live.bilibili.com/13140777.

Instruction for 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 speakers are suggested to join in above Zoom meeting rooms 30 minutes before your scheduled plenary lecture or keynote presentation.

Secretariat of the 4th CMHL Symposium

Dr. Weiwen Zhao, CMHL, Shanghai Jiao Tong University, Email: weiwen.zhao@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 4th CMHL Symposium

Beijing Time 09:00-19:20, Thursday, Jan. 14, 2021, Online virtual meeting

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

09:05-10:05 Plenary Lecture 1 (Jan. 13, 19:05-20:05 CST)

Experimental and CFD Study of KCS Hull-Propeller-Rudder Interaction for Self-Propulsion and Port and Starboard Turning Circles

Prof. Frederick Stern, IIHR, The University of Iowa, USA

10:05-10:50 Keynote Presentation 1 (Jan. 13, 21:05-21:50 EST)

Nonlinear Seakeeping Solution near the Critical Frequency

Dr. Yuming Liu, Department of Mechanical Engineering, Massachusetts Institute of Technology (MIT), USA

10:50-11:35 Keynote Presentation 2 (Jan. 13, 20:50-21:35 CST)

Stochastic Modelling of Trajectories of Dropped Cylindrical Objects in Offshore Operations

Dr. Xiaochuan Yu, School of Naval Architecture and Marine Engineering, University of New Orleans (UNO), USA

11:35-12:20 Keynote Presentation 3 (Jan. 13, 21:35-22:20 CST)

CFD Applications in Marine and Offshore Industry

Dr. Zhirong Shen, Global Engineering and Technology, ABS, USA

12:20-13:05 Keynote Presentation 4 (Jan. 13, 22:20-23:05 CST)

Air Entrainment and Bubble Size Distribution in Strong Free-surface Turbulent Flow

Dr. Xiangming Yu, Department of Mechanical Engineering, Massachusetts Institute of Technology (MIT), USA

13:05-13:50 Keynote Presentation 5

Wake Signature of a Submarine in Density Stratified Fluid

Dr. Liushuai Cao, CMHL, Shanghai Jiao Tong University, China

13:50-15:00 Break for Rest

15:00-16:00 Plenary Lecture 2 (Jan. 14, 08:00-09:00 CET)

Local Flow around a Surface Combatant at Various Static Drift Conditions: A Detailed Study of the Longitudinal Vortex Structure

Prof. Michel Visonneau, CNRS, Centrale Nantes, France

16:00-16:45 Keynote Presentation 6 (Jan. 14, 17:00-17:45 JST)

Experimental and Numerical Study of Dam Break Flow Impact on a Vertical Cylinder

Prof. Changhong Hu, RIAM, Kyushu University, Japan

16:45-17:00 Break for switch to the second Zoom meeting room

17:00-17:45 Keynote Presentation 7 (Jan. 14, 09:00-09:45 GMT)

Introduction of a Potential Flow Solver MHydro and its Applications in Ship Hydrodynamics in Confined Waterways

Dr. Zhiming Yuan, Department of Naval Architecture, Ocean & Marine Engineering, University of Strathclyde, UK

17:45-18:30 Keynote Presentation 8 (Jan. 14, 09:45-10:30 GMT)

Software-in-the-Loop Combined Artificial Intelligence Method for Dynamic Response Prediction of Floating Wind Turbines

Prof. Zhiqiang Hu, School of Engineering, Newcastle University, UK

18:30-19:15 Keynote Presentation 9 (Jan. 14, 11:30-12:15 CET)

An Efficient fully Lagrangian Solver for Modeling Wave Interaction with Oscillating Wave Energy Converter

Dr. Chi Zhang, Technical University of Munich (TUM), Germany

19:15-19:20 Closing Speech **Prof. Decheng Wan**

Invited Speakers

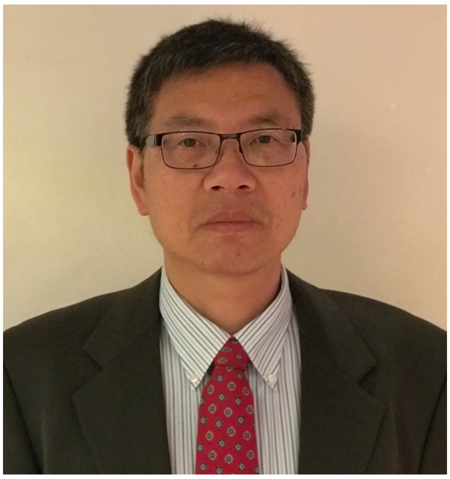
Prof. Frederick Stern

Prof. Frederick Stern is internationally recognized expert in ship hydrodynamics: computational methods, modeling, wave basin, towing-tank and flume experiments; experimental/ computational uncertainty analysis/ quantification; and deterministic/ stochastic shape optimization. He has authored, co-authored, or edited: 7 international conference proceedings/ books; 6 book chapters; 5 committee reports and 12 Quality Manual Procedures for the 21st–25th International Towing Tank Conference; 22 NATO AVT final report chapters; 178 journal articles; 4 moderate review journal articles; 2 online archive articles; 249 conference proceeding papers, and 56 reports. Prof. Frederick Stern is chair of the Steering Committee of the International Workshop on CFD in Ship Hydrodynamics since 2015 and is also a permanent member of the SNH-ONR Paper Selection Committee since 2005.

Plenary Lecture 1:
Experimental and CFD Study of KCS Hull-Propeller-Rudder Interaction for Self-Propulsion and Port and Starboard Turning Circles

Experiments and CFD are combined to explain the physics of the KCS hull-propeller-rudder interaction for turning circles and the reason for differences between port and starboard turning. The t' correlation is promising for scaling model size and both course keeping and maneuvering in clam water and waves. The MMG rudder model is useful but needs more study for general applicability. The X, Y, N force and moment balance helps to explain maneuvering and differences port vs. starboard maneuvers. The CFD shows the same X, Y, N force and moment balance as the experiments and completes the explanation of the details of the hull-propeller-rudder interaction for port vs. starboard maneuvering. The propeller inflow is different from the self-propulsion condition due to drift-angle induced hull vortices with similar trends for both port and starboard, but larger magnitudes for port. The propeller is more heavily loaded for turning compared with self-propulsion, especially for port turning. The loss in propeller efficiency as per t' correlation induces speed loss. The experimental and CFD circular motion equations (5) and (7) can be interpreted as follows. The primary state variables are the u and v velocity components since the yaw angular velocity $r = \sqrt{u^2 + v^2}/R$. The force balance can be explained in the non-inertial frame, where the centrifugal force mU^2/R due to the inertia is balanced by the hydrodynamic ship hull, propeller and rudder forces resulting in the steady state equilibrium condition with the key outcome of drift angle $\beta = tan^{-1}(v/u)$, which induces the hull vortices, propeller inflow and loss in efficiency and speed loss. The key physical mechanisms are the centrifugal acceleration, and hull-propeller interaction, whereas the propeller side force and yaw moment and rudder play largely passive role notwithstanding the rudder inducing the entire event. It is hypothesized the similar physics albeit subject to transient effects and alternating semi-circular events are exhibited for zig-zag maneuvers.

Dr. Yuming Liu



Dr. Yuming Liu is a Senior Research Scientist in the Department of Mechanical Engineering at the Massachusetts Institute of Technology (MIT). He received his S.B. degree in Harbor and Navigation Engineering from Hohai University, China, in 1985, S.M. degree in Coastal Engineering from the University of Florida in 1988, and Ph.D. degree in Marine Hydrodynamics from MIT in 1994. His research interest is in theoretical and computational marine hydrodynamics and acoustics, with special applications to ocean wave prediction, motions and sea loads of ships and offshore structures, marine energy conversion, transport of multiphase flows in pipelines, and sensing of underwater ocean environments. He has made fundamental advances in nonlinear wave-body interaction, seakeeping, ship wake prediction, complex wave prediction, and multiphase slug pipe flows by developing fully-nonlinear methods to accurately solve the strongly nonlinear fluid mechanics problems. He has also applied fluid mechanics to make fundamental advances in acoustics including the development of the acoustic conductance theory and complete second-order sound-object interaction theory. He has published over one hundred journal articles and refereed conference papers.

Keynote Presentation 1:
Nonlinear Seakeeping Solution near the Critical Frequency

We theoretically and numerically study the seakeeping problem of a submerged or floating body translating with constant forward speed U parallel to the undisturbed free surface while at the same time undergoing a small oscillatory motion and/or encountering small-amplitude waves at frequency ω. Understanding and prediction of the seakeeping solution is of fundamental importance in marine hydrodynamics. It is known that at the critical frequency corresponding to $\tau=U\omega/g=1/4$, where g is the gravitational acceleration, the classical linear solution is unbounded for a single point source, and the inclusion of third-order free-surface nonlinearity due to cubic self-interactions of waves is necessary to remove the associated singularity. We theoretically show that the linear solution is in fact bounded for a body with full geometry rather than a point source. We further show that for a body, the nonlinear correction to the (finite) linear solution due to cubic self-interactions of resonant waves in the neighbourhood of τ=1/4 is of first order in the wave steepness (or body motion amplitude), which is the same order as the linear solution. With the inclusion of nonlinear effects in the dispersion relation, the wavenumbers of resonant waves become complex-valued and the resonant waves become evanescent, with their amplitudes vanishing with the distance away from the body. To assist in understanding the theory, we derive the analytic linear and nonlinear solution for the case of a submerged two-dimensional circular cylinder in the neighbourhood of τ=1/4. The theory is confirmed by independent direct numerical simulations.

Dr. Xiaochuan Yu

Dr. "Vincent" Xiaochuan Yu is currently an assistant professor at the School of Naval Architecture and Marine Engineering, University of New Orleans (UNO). His educational background includes BS and MS degree in Naval Architecture and Ocean Engineering from Shanghai Jiao Tong University (SJTU), MS degree in Civil Engineering from University of Hawaii at Manoa (UHM), and PhD degree in Ocean Engineering from Texas A&M University (TAMU). After his PhD study, Dr. Yu has worked for various offshore engineering companies, such as FloaTEC LLC, IntecSea and SBM Offshore. He started to work for UNO from January 2015. His extensive research interests include numerical modeling and model tests of dropped objects in offshore operations, mooring system analysis and design under squalls, additive manufacturing and tests of maritime structures, analysis and design of various types of offshore platforms, smart structures, etc. He has authored more than 40 peer-reviewed journal articles and conference paper.

Keynote Presentation 2:
Stochastic Modelling of Trajectories of Dropped Cylindrical Objects in Offshore Operations

The trajectories of dropped cylindrical objects show various fall-patterns based on the previous experimental tests and numerical simulations (Aanesland, 1987; DNV, 2010). It is found that such a falling process is sensitive to parameters such as initial water-entry angle, drag coefficients, center of gravity, etc. Therefore, minor changes in these parameters can easily cause quite different trajectories, which show significantly stochastic characteristics and are difficult to predict. Monte Carlo (MC) method is one of the most prevailing methods to solve stochastic problems. However, as a method of using random samples to calculate statistics, its accuracy heavily depends on the size of sample. A systematic formulation the cylinder's three-degree-of-freedom (surge, heave, and pitch) of motions is presented as a state space model. It is assumed that the small disturbances at its initial state i.e., the velocity and the drop angle of each degree, follow a Gaussian distribution. This non-linear prediction problem can be solved by using the MC method and unscented transformation (UT). By comparing the statistical results of these two methods, we conclude that the UT method may be comparatively better when considering both accuracy and calculation requirements. Overall, such a stochastic model can predict the drop process more reasonably and accurately if compared with a deterministic model.

Dr. Zhirong Shen

Dr. Zhirong Shen is Senior Engineer in Global Engineering and Technology at ABS, specializing in CFD research and development, CFD applications, software development, and high-performance computing. He is responsible for the CFD software development and application to marine and offshore engineering. He is also working with industry partners through various joint industry projects and engineering service projects. Dr. Shen obtained his B.S. and Ph.D. degrees in Naval Architecture and Ocean Engineering at Shanghai Jiao Tong University.

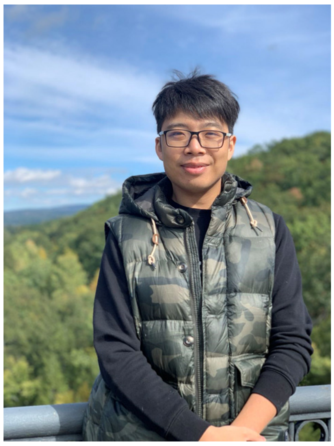
Keynote Presentation 3:
CFD Applications in Marine and Offshore Industry

With significant advancements in software engineering and the power of high-performance computing, Computational Fluid Dynamics (CFD) technologies are increasingly applied in various industries from aerospace to automobile. In the marine and offshore industry, CFD technologies are becoming powerful design and assessment tools that enable high-fidelity simulations to accurately predict hydrodynamic and aerodynamic performances of marine vessels and offshore structures. One challenge still faced by CFD technologies is to tackle its application to practical engineering problems that often require a fine balance between practicality and analytical rigors.
ABS has been actively expanding the application of CFD in marine and offshore engineering and has accumulated significant experience through numerous projects. Examples of established applications that have already passed the rigorous verification and validation process include:

- Hull-form optimization for energy efficiency
- Assessment of Energy Saving Device (ESD)
- Propeller shaft forces prediction
- Slamming loads and green water impact on marine vessels
- Airgap prediction for offshore structures

- Wind load estimation for floating offshore structures
- Simulation of drillship moonpool/FLNG turret moonpool hydrodynamics
- Gas dispersion of LNG fueled vessels

This presentation will provide the challenges and solutions of CFD applications for the marine and offshore industry, example applications, current development projects, and a vision of future implementations.



Dr. Xiangming Yu

Dr. Xiangming Yu received his Ph.D. in mechanical engineering from MIT in 2019, under the supervision of Prof. Dick K.P. Yue. His research interests focuses on free-surface turbulence, air entrainment, and multiphase flows. He received his Master and Bachelor from UT Austin in 2012 and SJTU in 2010.

Keynote Presentation 4:
Air Entrainment and Bubble Size Distribution in Strong Free-surface Turbulent Flow

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 turbulence (SFST) are performed. We first investigate the entrainment bubble-size spectrum per unit interface area, $N(r)$, with dimension length^{-3}, and develop a physical/mechanistic model for $N(r)$ through energy arguments. The model obtains two distinct regions of $N(r)$, separated by the bubble-size scale r_0. For bubble radius larger than r_0, the effects of gravity g dominate those of the surface tension force σ/ρ, and $N(r) \propto g^{-1}\epsilon^{2/3}r^{-10/3}$, where ϵ is the turbulence dissipation rate. For bubble radius smaller than r_0, surface tension is more important and $N(r) \propto (\sigma/\rho)^{-1}\epsilon^{2/3}r^{-4/3}$. The bubble-size scale is $r_0 \approx r_c = 1/2\sqrt{\sigma/\rho g}$, the capillary length scale, and not the generally assumed Hinze scale. For an air-water interface and Earth gravity, $r_c \approx 1.5mm$. The model further provides an $\epsilon - r$ entrainment regime map that identifies a critical dissipation rate (consistent for given g and σ/ρ) above which there is appreciable air entrainment, thus separating strong and weak free-surface turbulence. The theoretical model and its predictions have been confirmed by performing two-phase, high-fidelity direct numerical simulations of a canonical free-surface turbulent flow. In addition, this study characterizes isotropy of strong free-surface turbulence and investigate air entrainment events induced by near-surface coherent vortex structures.

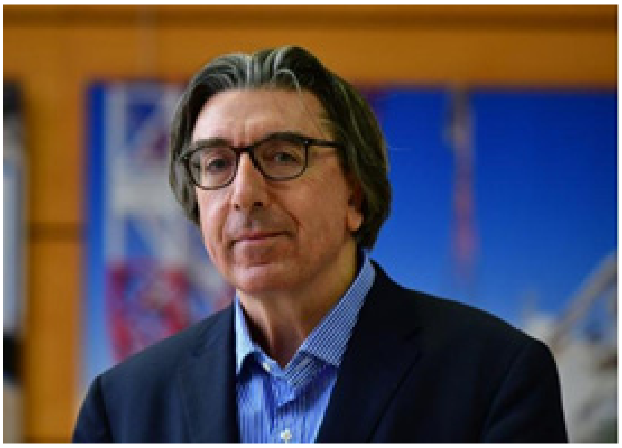
Dr. Liushuai Cao

Dr. Liushuai Cao joined the Computational Marine Hydrodynamics Laboratory (CMHL) as a research associate in August 2019. He received a BSc in Naval Architecture and Ocean Engineering from Harbin Engineering University and a PhD from Naval Engineering University. After that, he served as an engineer in the navy for four years. His research interests mainly focus on the development of numerical methods for practical applications, such as maneuvering and propulsion of submarines and complex-shaped underwater vehicles in the homogenous and density stratified fluid, cavitation and erosion prediction of propellers and diesel injectors.

Keynote Presentation 5:
Wake Signature of a Submarine in Density Stratified Fluid

The turbulent wake and surface signature characteristics of a submarine in density stratified fluid are of great importance. However, this complex question has not yet fully understood. In CMHL, we proposed a methodology to predict the hydrodynamic performance and turbulent wake of a submarine moving in the density stratified fluid. This presentation will introduce the background information of ocean pycnocline firstly, and then derive a numerical method to simulate the stratified fluid based on Boussinesq approximation. Secondly, we will show several calculated cases consisting of the bare hull, the fully appended hull without/with a propeller moving in the linearly stratified fluid. Finally, we will analyze the free surface signature, internal gravity waves, and the evolution of wake turbulence to investigate the buoyancy effects induced by density stratification. Results show that a stratified wake always differs from its unstratified counterpart at a sufficiently downstream distance.

Prof. Michel Visonneau



Prof. Michel Visonneau, born in 1957, received in 1980 an Engineer's diploma and the diploma of Advanced Naval Architecture in 1981 from Centrale Nantes, France. In 1985, he defended his PhD in Fluid Dynamics and Heat Transfer at Centrale Nantes/University of Nantes under the supervision of Prof. Jean Piquet. He joined the National Center for Scientific Research (CNRS) as Research Associate in 1985 as full-time researcher in charge of the development of Computational Fluid Dynamics for marine applications. He was the head of the CFD department of the Fluid Mechanics Laboratory (Centrale Nantes) from 1995 to 2012. He is now Research Director at CNRS since 2006 and gives lectures in several academic institutions like Centrale Nantes, the University of Nantes, the Aalto University of Helsinki and the Lebanese University. He has supervised 23 PhD thesis and is currently supervising 4 PhD thesis dealing with various topics like cavitation, fluid-structure interaction, high-order accuracy on unstructured grids, hybrid RANSE-LES and laminar-to-turbulence transition modeling. Prof. Michel Visonneau is the author of 93 publications in archival scientific journals, 223 publications in conference proceedings with editorial committee and gave 72 invited conferences. In 2006, he built a partnership with NUMECA Int. which led to the creation of FINETM/Marine, a software worldwide leader in CFD for Marine Engineering and is in charge of the scientific development of this software. Since 2009, he is involved in many collaborative Applied Vehicle Technology (AVT) research projects under the umbrella of the NATO Science and Technology Organization (STO). Prof. Michel Visonneau is a member of the Steering Committee of the International Workshop on CFD in Ship Hydrodynamics representing Europe since 2005 and a permanent member of the scientific committee of ECCOMAS/Marine. He is also a permanent member of the SNH-ONR Paper Selection Committee representing Europe since 2017. He was awarded the 2nd Cray prize for CFD in 1991, the 30th Georg Weinblum Memorial Lecture in 2007 and the NATO Science and Technology Organization - Applied Vehicle Technology Panel Excellence Award in 2020.

Plenary Lecture 2:
Local Flow around a Surface Combatant at Various Static Drift Conditions: A detailed Study of the Longitudinal Vortex Structure

This presentation is devoted to a local and global computational study of the flow around the US Navy frigate DTMB 5415 at 10° and 20° static drift conditions. A thorough validation study comparing isotropic k-omega SST, Reynolds-Stress Transport SSG/LRR turbulence statistical closures and the unsteady hybrid RANS-LES DDES-SST model is conducted for the two static drift configurations. This validation study includes global forces/moment comparison and detailed local comparisons with IIHR TPIV experiments. All these validations are conducted on very fine unstructured grids comprised of 163M to 184M cells with local boxes of refinement in the cores of the main vortices, which makes possible a very detailed study of the flow and turbulence structures of the longitudinal vortex cores from the onset and during their progression.

Prof. Changhong Hu

Changhong Hu is Professor of Ocean Engineering at Research Institute for Applied Mechanics (RIAM), Kyushu University, Japan. He received his B.S. in 1987, M.S. in 1990 and PhD in Marine Hydrodynamics in 1995 from Shanghai Jiao Tong University (SJTU). From 1995 he served as assistant, associate, and full professor at Kyushu University. In the academic year 2004 he worked in Norwegian University of Science and Technology (NTNU) as a visiting associate professor. From 2016 he became a guest professor of SJTU. His current research interests include next generation computational fluid dynamics methods for marine hydrodynamics, ocean renewable energy technologies, and hydrodynamics problems in ship and ocean engineering. Recently he has been involved in several national research projects of Japan on offshore wind and tidal current energy developments.

Keynote Presentation 6:
Experimental and Numerical Study of Dam Break Flow Impact on a Vertical Cylinder

In this research an experimental measurement of the dam break impact on a vertical cylinder placed over a dry horizontal bed has been carried to provide accurate and comprehensive data for the validation of computational fluid dynamics (CFD) codes. The experimental setup is based on a series of experiments conducted in Research Institute for Applied Mechanics (RIAM) between the years 2010 and 2017. The gate motion is thoroughly studied and a novel gate motion formula is proposed based on the recent experimental data. The effect of gate obstruction on the experimental measurements (i.e. time of impact with the cylindrical obstacle) is investigated. The effect of the cross-section of the cylinder is studied by examining the pressure signal on the cylinder as well as the vertical wall downstream of the cylinder. Two newly developed numerical methods, the unstructured FVM code in which UMTHINC is used for interface capturing, and the lattice Boltzmann method (LBM) code with a cumulant collision model, have been used for numerical simulation of the dambreak experiment. Their performance on wave impact loads prediction are investigated and compared.

Dr Zhiming Yuan

Dr Zhiming Yuan is a Senior Lecturer in the Department of Naval Architecture, Ocean & Marine Engineering (NAOME) at Strathclyde. In 2011, Dr Yuan got the sponsorship from Lloyd's Register to carry out the research on offshore offloading problem under the supervision of Prof. Atilla Incecik in Strathclyde. After obtaining his Ph.D in 2014, he worked as a Research Associate on an EU project SHOPERA, investigating the ship hydrodynamics in confined waterways. In 2015, he became a lecturer at the University of Strathclyde. He is currently leading the Hydrodynamics & Ocean Renewable Energy Laboratory @ Strathclyde (HOREL), which consists of 8 PhD researchers. He was invited by Prof. Ronald Yeung to visit and carry out joint research in UC Berkeley (03/2017 – 09/2017) under Sir David Anderson Bequest Award. Dr Yuan's research activities are mainly focused on marine hydrodynamics and offshore renewable energy systems. Over the last 5 years, Dr Yuan has published more than 40 peer-reviewed journal articles and 25 refereed conference proceedings on marine hydrodynamics. He is a member of the Royal Institution of Naval Architects (RINA), and committee member of the International Towing Tank Conference (ITTC).

Keynote Presentation 7:
Introduction of a Potential Flow Solver MHydro and its Applications in Ship Hydrodynamics in Confined Waterways

Over the past ten years, the speaker and his research group – Hydrodynamics & Ocean Renewable Energy Laboratory @ Strathclyde (HOREL) – have been continuously worked on the development of a potential flow solver MHydro and explored its applications in various areas. MHydro is based on the Rankine Source Panel Method, and it was originally developed to investigate the multi-body hydrodynamics involved in ship-to-ship operations in waves. In this presentation, the speaker will introduce its application in ship hydrodynamics in confined waterways, including ship-bank, ship-bottom, ship-ship and ship-lock interactions. Both steady and unsteady interaction problems will be presented. The discussions on the benefits and limitations of the potential flow solver will be highlighted.

Prof. Zhiqiang Hu

Zhiqiang Hu is Lloyds Professor of Offshore Engineering in School of Engineering, Newcastle University, UK. He received his doctoral degree in Shanghai Jiao Tong University with major in Naval Architecture and Ocean Engineering. He is the Deputy Editor of journal Ocean Engineering; Committee member of International Ship Structures Congress (ISSC) V.6 Ocean Space Utilization; Committee member of International Conference on Collision and Grounding of Ships and Offshore Structures (ICCGS); and Member of Royal Institute of Naval Architects (RINA). Zhiqiang Hu's research focuses on the aspects including Dynamic response analysis of floating wind turbine, Offshore hydrodynamics, Basin model experiment, Collision and grounding of ships and offshore structures; and Ship-iceberg interaction analysis.

Keynote Presentation 8:
Software-in-the-Loop Combined Artificial Intelligence Method for Dynamic Response Prediction of Floating Wind Turbines

Offshore wind industry is facing a critical challenge for predicting dynamic responses during the design phase for floating offshore wind turbines (FOWTs), due to its multidisciplinary characteristics, the high nonlinear coupling effect, and the challenge of using conventional basin experiment technology for direct results transferring. Artificial Intelligence (AI) brings a new solution to overcome this challenge with intelligent strategies. This presentation introduces a new AI technology-based method, named SADA, for the prediction of dynamic responses of FOWTs. The AI module in SADA was incorporated into a coupled aero-hydro-servo-elastic in-house program DARwind. Some other important concepts in SADA are introduced, including the selection of Key Disciplinary Parameters (KDPs), and deep deterministic policy gradient (DDPG), the policy decision by machine learning algorithms. A set of basin experimental results of a Hywind Spar-type FOWT are employed to train the AI module. SADA weights KDPs by DDPG algorithms' network and changes their values according to the training feedback of 6DOF motions of Hywind platform. Many other dynamic responses that can't be measured in basin experiment therefore are able to be predicted in higher accuracy with the intelligent DARwind. The proposed SADA method brings a new and promising solution for overcoming the handicap impeding direct use of conventional basin experimental technology in FOWTs design.

Dr. Chi Zhang

Dr. Chi ZHANG completed his doctorate at Technical University of Munich (TUM) with a dissertation about smoothed particle hydrodynamics for fluid and solid dynamics. Since then he worked as post-doctor in TUM where the focus of his research is the development of an open-source multi-physics and multi-resolution library based on SPH method and its application in fluid dynamics, solid mechanics, fluid-structure interactions and cardiac modeling. He has published more than 30 journal and international conference papers cover various aspects of computational fluid dynamics, SPH and biomechanics.

Keynote Presentation 9:
An Efficient fully Lagrangian Solver for Modeling Wave Interaction with Oscillating Wave Energy Converter

Renewable ocean wave energy has received tremendous worldwide attention thanks to its abundant and dense energy form and, in particular, low environmental impact nature. In this talk, an efficient, accurate and fully Lagrangian numerical solver for modeling wave interaction with oscillating wave energy converter (OWSC) is presented. More precisely, the wave dynamics and its interaction with OWSC is resolved by a Riemann-based weakly-compressible SPH method in SPHinXsys, an open-source multi-physics library in unified smoothed particle hydrodynamic (SPH) framework, and the solid-body kinematics is computed by Simbody library which presents an object-oriented Application Programming Interface (API) for multi-body dynamics. Numerical experiments demonstrate that the proposed solver can accurately predict the wave elevations, flap rotation and wave loading on the flap in comparison with laboratory experiment. In particularly, the new solver shows optimized computational performance through CPU cost analysis and comparison with commercial software package ANSYS FLUENT and other SPH-based solvers in literature. Furthermore, a linear damper is applied for imitating the power take-off (PTO) system to study its effects on the hydrodynamics properties of OWSC and efficiency of energy harvesting. In addition, the present solver is used to model extreme wave condition using the focused wave approach to investigate the extreme loads and motions of OWSC under such extreme wave conditions. It worth noting that though the model validation used herein is a bottom hinged oscillating Wave Energy Converter (WEC), the obtained numerical results show promising potential of the proposed solver to future applications in the design of high-performance WECs.