WELCOME TO IWSH 2017

On behalf of the local organizing committee, I would like to welcome all delegates to Taiwan to participate in the 10th International Workshop on Ship and Marine Hydrodynamics (10th IWSH). It is our honor to host the Conference at Keelung.

The challenge of marine hydrodynamics has been widely recognized since the past century. In fact, marine hydrodynamics usually presents the most complicated and challenging phenomena in classical physical science and modern engineering applications. Cavitation, wave impact on marine structures, sloshing, and fluid-structure interactions are just a few classical examples. Furthermore, new challenges also eagerly step in as new applications emerge. Floating wind power, marine energy, and energy saving and green marine vehicles are some of the instances.

Since the first IWSH held at Wuhan, China in 1999, the Conference has documented the development trajectory of marine hydrodynamics for 20 years. The conference series provide a forum to promote scientific advancement, technological progress, information exchange, and cooperation among engineers and researchers in ship and ocean engineering and other related fields. It is no doubt that the achievements obtained in the past 20 years are admirable.

The National Museum of Marine Science and Technology is an exceptional location for the Conference, with the sparkling ocean and coast hills providing a unique and spectacular setting. Meanwhile, just 15 minutes away, the National Northeast Scenic Coast invites you into a world of emerald seas and mountains and blue skies, a winding coastline of capes and bays, and fine sandy beaches. In addition, Keelung is also a place with rich local ocean culture. You may spend a day visiting the National Palace Museum and linger for another day along old streets and Gold Ecological Park of Jioufeng, a small mountain village and Taiwan’s gold mining center in the past, for a deep traditional touch of Taiwan.

I do wish you a rewarding experience in participating in IWSH 2017. Enjoy your stay in Taiwan.

Jiahn-Horng Chen
Conference Chair
Table of Contents

WELCOME TO IWSH 2017........................................................................................................... 1

TABLE OF CONTENTS.................................................................................................................. 3

ORGANIZATIONS.......................................................................................................................... 4

  International standing committee ................................................................................................. 4
  International scientific committee ................................................................................................. 4
  Local organizing committee ......................................................................................................... 5
  Organizers .................................................................................................................................. 5
  Co-organizers .............................................................................................................................. 5
  Sponsors ...................................................................................................................................... 5
  Contact us .................................................................................................................................. 5

PREVIOUS IWSH............................................................................................................................ 6

GENERAL INFORMATION.............................................................................................................. 7

CONFERENCE VENUES.................................................................................................................. 9

IWSH 2017 PROGRAM AT A GLANCE........................................................................................ 12

IWSH 2017 PROGRAM.................................................................................................................... 14

ABSTRACTS OF IWSH 2017......................................................................................................... 23

  Abstracts of keynote speeches ..................................................................................................... 24
  Abstracts of invited speeches ....................................................................................................... 27
  Abstracts of contributed papers ................................................................................................... 29
ORGANIZATIONS

International Standing Committee

Moustafa Abdel-Maksoud (Hamburg University of Technology, Germany)
Jiahn-Horng Chen (National Taiwan Ocean University, Taiwan)
Atilla Incecik (Strathclyde University, UK)
Masashi Kashiwagi (Osaka University, Japan)
Yonghwan Kim (Seoul National University, Korea)
Decheng Wan (Shanghai Jiaotong University, China)

International Scientific Committee

Moustafa Abdel-Maksoud (Hamburg University of Technology, Germany)
Neil Bose (University of Tasmania, Australia)
Jiahn-Horng Chen (National Taiwan Ocean University, Taiwan)
Fai Cheng (Head of Strategic Research & Technology Policy Group, Lloyd’s Register, UK)
Sören Ehlers (Hamburg University of Technology, Germany)
Pierre Ferrant (Ecole Centrale Nantes, France)
Nuno Fonseca (Technical University of Lisbon, Portugal)
Alberto Francescutto (University of Trieste, Italy)
Ismail Hakki Helvacioglu (Istanbul Technical University, Turkey)
Tomoki Ikoma (Nihon University, Japan)
Atilla Incecik (University of Strathclyde, UK)
Masashi Kashiwagi (Osaka University, Japan)
Yonghwan Kim (Seoul National University, Korea)
Yougang Tang (Tianjin University, China)
Bin Teng (Dalian University of Technology, China)
Stephen Turnock (University of Southampton, UK)
Marc Vantorre (Ghent University, Belgium)
Decheng Wan (Shanghai Jiaotong University, China)
Ray-Yeng Yang (National Cheng Kung University, Taiwan)
Liandi Zhou (Editorial Board of Journal of Hydrodynamics, China)
Zaojian Zou (Shanghai Jiaotong University, China)
Local organizing committee

Wen-Lon Cheng (Honorary Chair, CSBC)
Jiahn-Horng Chen (Chair, National Taiwan Ocean University)
Yi-Chih Chow (Technical Chair, National Taiwan Ocean University)
Yaw-Huei Lee (Program Chair, National Taiwan Ocean University)
Forng-Chen Chiu (National Taiwan University)
Chih-Chung Fang (National Taiwan Ocean University)
Ming-Chung Fang (National Cheng-Kung University)
Ching-Yeh Hsin (National Taiwan Ocean University)
Tai-Wen Hsu (National Taiwan Ocean University)
Ching-Jer Huang (National Cheng-Kung University)
Hwung-Hweng Hwung (National Cheng-Kung University)
Syue-Sinn Leu (National Kaohsiung Marine University)
Cheng Lin (National Chung Hsing University)

Organizers

National Taiwan Ocean University

Co-organizers

Ministry of Science and Technology
National Museum of Marine Science and Technology
Taiwan Society of Naval Architects and Marine Engineers
Editorial Board of Journal of Hydrodynamics

Sponsors

ONR Global
Chungyu University of Film and Arts
CSBC Corporation, Taiwan
CR Classification Society
Ship and Ocean Industries R&D Center

Contact us

contact email: iws2017.in.taiwan@gmail.com
PREVIOUS IWSH

1st IWSH  Wuhan, China, 1999
2nd IWSH  Wuhan, China, 2001
3rd IWSH  Wuhan, China, 2003
4th IWSH  Shanghai, China, 2005
5th IWSH  Zhenjiang, China, 2007
6th IWSH  Harbin, China, 2010
7th IWSH  Shanghai, China, 2011
8th IWSH  Seoul, Korea, 2013
9th IWSH  Glasgow, UK, 2015
10th IWSH  Keelung, Taiwan, 2017
GENERAL INFORMATION

Registration Package
Each registered delegate will receive a conference bag, a name badge, an IWSH mug, a Conference Program and Book of Abstracts, a receipt of the registration fee, a pen, a notebook, an USB flash drive, the password of free Wi-Fi internet, and some tourist maps. All participants are requested to wear their conference badges throughout the conference. Refreshments will be served only to identifiable conference participants during the morning and afternoon breaks.

Oral Presentations
The time allotted to each speaker for presentation and discussion is as follows:
Keynote speakers – 40 minutes
Invited speakers – 30 minutes
Other speakers – 20 minutes
Please respect each speaker’s rights and adhere strictly to the time scheduled in the program. Before the start of their session, presenters are requested to contact their session chair and load their presentation files onto the computer. Standard presentation applications (Word, PowerPoint, Acrobat, RealPlayer, Media Player, etc) are installed on these computers.

Internet
Participants have free access to Wi-Fi for wireless connection of their computers to the internet at the conference venue. The password is in the registration material package.

Messages
Participants may post messages on a notice board located near the registration desk. The organizer may also post updated information on the board. Please check the board frequently.

Reception
Reception is held at Peng's Gourmet & Banquet on the 5th Floor of Evergreen Laurel Hotel, Keelung, on the evening of November 5 (4:30- 6:30pm). All registered delegates are welcome. All participants are requested to pick up the registration package at the registration desk which is open at 3:30pm and located the entrance of Peng's Gourmet & Banquet on the 5th Floor of the Hotel. All delegates are requested to wear their conference badges before attending the reception.

Reception Dinner
The reception dinner is held at Café Laurel on the 18th Floor of Evergreen Laurel Hotel, Keelung, on the evening of November 6 after all afternoon technical sessions are closed (6:30-8:30pm). Shuttle buses are available at the venue. The departure time will be 5:50pm. Please wear the conference badge when attending the reception.

Banquet
The banquet is held at the Peng's Gourmet and Banquet on the 3rd Floor of Evergreen Laurel Hotel, Keelung, on the evening of November 7 after all afternoon technical sessions are closed (6:30-9:00pm). Shuttle buses are available at the venue. The departure time will be 5:50pm. Please wear the conference badge when attending the banquet. The next IWSH and the winners of best
student paper will be announced in the banquet.

**Lunches**

Lunches for registered participants and accompanying persons are served on November 6, 7, and 8.

**Shuttle Bus**

Free shuttle buses are available in the morning and evening for registered delegates to travel between the conference-designated hotels and the venues. The bus will depart at 8:20am on November 6, 7 and 8.
CONFERENCE VENUES

National Museum of Marine Science and Technology (NMMST)

No. 367, Beining Rd., Zhongzheng Dist., Keelung City 202, Taiwan, R.O.C.
Website: http://www.nmmst.gov.tw/enhtml/index
Conference Center (Education center)

International Lecture Hall (Room A) and science education classrooms (Room B & C)
Traffic routes from airport to the hotel
IWSH 2017 PROGRAM AT A GLANCE

November 5 (Sunday)

15:30-18:30  Registration desk open at the lobby of Evergreen Laurel Hotel, Keelung
16:30-18:30  Reception at Evergreen Laurel Hotel, Keelung
18:30-20:30  Standing Committee meeting

November 6 (Monday)

08:00-17:00  Registration desk open at the venue
09:20-09:35  Opening ceremony
             Welcome Address 1: Dr. J.H. Chen (Conference Chair)
             Welcome Address 2: Dr. W.L. Cheng (Chairman of CSBC)
             Welcome Address 3: Dr. J.L. Wu (Director General of NMMST)
09:35-10:55  Opening keynote speeches
             09:35-10:15  Prof. You-Sheng Wu, China Ship Scientific Research Center
             10:15-10:55  Prof. Yasuyuki Toda, Osaka University
10:55-11:15  Coffee Break and Group Photograph
11:15-12:15  Parallel Sessions I
12:15-13:35  Lunch
13:35-14:35  Invited speeches
             13:35-14:05  President Forng-Chen Chiu, SOIC
             14:05-14:35  President Ying-Ru Liu, CR Classification Society
14:40-15:40  Parallel Sessions II
15:40-16:00  Coffee Break
16:00-17:40  Parallel Sessions III
18:30-20:30  Reception Dinner

November 7 (Tuesday)

08:30-17:00  Registration desk open at the venue
09:00-10:20  Keynote speeches
             09:00-09:40  Dr. Woei-Min Lin, Office of Naval Research
             09:40-10:20  Prof. Chi Yang, George Mason University
10:20-10:40  Coffee Break
10:40-12:00  Parallel Sessions IV
12:00-13:30  Lunch
13:30-14:30 Invited speeches
   13:30-14:00 President Sheldon S.C. Huang, Lung Teh Shipbuilding Co.
   14:00-14:30 President Robert Tseng, CSBC Corporation

14:35-15:35 Parallel Sessions V

14:00-14:30 President Robert Tseng, CSBC Corporation

15:35-15:55 Coffee Break

15:55-17:35 Parallel Sessions VI

18:30-21:00 Conference Banquet & Best Student Paper Awards announcement

---

**November 8 (Wednesday)**

08:30-14:00 Registration desk open at the venue

09:00-09:40 Keynote speech
   Prof. Atilla Incecik, University of Strathclyde

09:40-10:00 Coffee Break

10:00-12:00 Parallel Sessions VII

12:00-12:20 Closing Ceremony

12:20-14:00 Farewell Buffet Lunch
## IWSH 2017 PROGRAM

### November 5 (Sunday)

<table>
<thead>
<tr>
<th>Time</th>
<th>Activity</th>
</tr>
</thead>
<tbody>
<tr>
<td>15:30-18:30</td>
<td>Registration desk open at the Lobby of Evergreen Laurel Hotel, Keelung</td>
</tr>
<tr>
<td>16:30-18:30</td>
<td>Reception at Evergreen Laurel Hotel, Keelung <em>(Café Laurel, 19th Floor)</em></td>
</tr>
</tbody>
</table>

### November 6 (Monday)

<table>
<thead>
<tr>
<th>Time</th>
<th>Activity</th>
</tr>
</thead>
<tbody>
<tr>
<td>08:00-17:00</td>
<td>Registration desk open <em>(Education Center, National Museum of Marine Science and Technology)</em></td>
</tr>
<tr>
<td>09:20-09:35</td>
<td>Opening ceremony <em>(International Lecture Hall – Room A)</em></td>
</tr>
<tr>
<td></td>
<td>Welcome Address 1: <em>Dr. J.H. Chen (Conference Chair)</em></td>
</tr>
<tr>
<td></td>
<td>Welcome Address 2: <em>Dr. W.L. Cheng (Chairman of CSBC)</em></td>
</tr>
<tr>
<td></td>
<td>Welcome Address 3: <em>Dr. J.L. Wu (Director General of NMMST)</em></td>
</tr>
<tr>
<td>09:35-10:55</td>
<td><strong>Keynote Speech</strong> <em>(International Lecture Hall - Room A)</em></td>
</tr>
<tr>
<td></td>
<td>Chair: Prof. Wei-Hui Wang</td>
</tr>
<tr>
<td></td>
<td>Hydroelastic Responses of VLFS in Complicated Geographical Environment <em>(KS01)</em></td>
</tr>
<tr>
<td></td>
<td><em>Prof. You-Sheng Wu (China Ship Scientific Research Center, Member of Chinese Academy of Engineering, China)</em></td>
</tr>
<tr>
<td></td>
<td>Brief Introduction of Detailed Flow Field Measurement and CFD computation around Model Ship Stern in Waves in Towing Tank <em>(KS02)</em></td>
</tr>
<tr>
<td></td>
<td><em>Prof. Yasuyuki Toda (Osaka University, Japan)</em></td>
</tr>
<tr>
<td>10:55-11:15</td>
<td>Coffee break and Group Photograph</td>
</tr>
<tr>
<td>Time</td>
<td>Session 1-1 (Room A)</td>
</tr>
<tr>
<td>--------</td>
<td>--------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>11:15-</td>
<td>Hydrodynamic performance of offshore platforms, pipelines, risers and mooring systems (I)</td>
</tr>
<tr>
<td>11:15-</td>
<td>Chair: Prof. Decheng Wan</td>
</tr>
<tr>
<td>11:35-</td>
<td>Fully Coupled Effects on Waves and Barge with Single Sloshing Tank by CFD Methods (443) Yuan Zhuang, Decheng Wan</td>
</tr>
<tr>
<td>12:15-</td>
<td>Lunch</td>
</tr>
<tr>
<td>13:35-</td>
<td>Invited Speech (International Lecture Hall – Room A)</td>
</tr>
<tr>
<td>13:35-</td>
<td>Chair: Prof. Chih-Chung Fang</td>
</tr>
<tr>
<td></td>
<td>SOIC, A Cornerstone of Ship and Ocean Industries in Taiwan (IS01)</td>
</tr>
<tr>
<td></td>
<td>Dr. Forng-Chen Chiu (President, Ship and Ocean Industries R&amp;D Center)</td>
</tr>
<tr>
<td></td>
<td>Recent Development in CR Classification Society (IS02)</td>
</tr>
<tr>
<td></td>
<td>Ms. Ying-Ru Liu (President, CR Classification Society)</td>
</tr>
<tr>
<td>Time</td>
<td>Session 2-1 (Room A)</td>
</tr>
<tr>
<td>-----------------</td>
<td>--------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>14:40-15:00</td>
<td>Experimental techniques for towing tank, wave flume and water basin</td>
</tr>
<tr>
<td></td>
<td>Chair: Prof. Moustafa Abdel-Maksoud</td>
</tr>
<tr>
<td>14:40-15:00</td>
<td>Experimental Investigation of Motion Responses of Moored Twin Barge Model in Regular Wave in Square Tank (343)</td>
</tr>
<tr>
<td>15:00-15:20</td>
<td>Kinematics of Breaking Waves (436)</td>
</tr>
<tr>
<td></td>
<td>De Wang Chia, Longbin Tao, Arun Kr Dev, Xin Wang, Yali Zhang</td>
</tr>
<tr>
<td>15:40-16:00</td>
<td>Coffee break</td>
</tr>
<tr>
<td>Time</td>
<td>Session 3-1 (Room A)</td>
</tr>
<tr>
<td>------------</td>
<td>--------------------------------------------------------------------------------------</td>
</tr>
<tr>
<td>16:00-16:20</td>
<td>Resistance, propulsion, seakeeping, maneuverability (I)</td>
</tr>
<tr>
<td></td>
<td>Chair: Prof. Yonghwan Kim</td>
</tr>
<tr>
<td></td>
<td>Tian-long Mei, Zao-Jian Zou, Jing-ping Wu</td>
</tr>
<tr>
<td>16:20-16:40</td>
<td>Multi-objective Optimization of Principal Dimensions of Ship for Improvement of Operational Efficiency in Waves (378)</td>
</tr>
<tr>
<td></td>
<td>Yoo-won Jung, Yonghwan Kim</td>
</tr>
<tr>
<td></td>
<td>Stochastic Parametric Roll Motion of a Vessel under Random Wave (313)</td>
</tr>
<tr>
<td></td>
<td>Liyuan Wang, Yougang Tang, Xiaorui Zhang</td>
</tr>
<tr>
<td>16:40-17:00</td>
<td>Study on Steady Flow Effects in Numerical Computation of Added Resistance of Ship in Waves (383)</td>
</tr>
<tr>
<td></td>
<td>Jae-Hoon Lee, Beom-Soo Kim, Yonghwan Kim</td>
</tr>
<tr>
<td></td>
<td>Manoeuvering Simulations of a VLCC in Adverse Weather Conditions (332)</td>
</tr>
<tr>
<td>17:00-17:20</td>
<td>Study on Steady Flow Effects in Numerical Computation of Added Resistance of Ship in Waves (383)</td>
</tr>
<tr>
<td></td>
<td>Jae-Hoon Lee, Beom-Soo Kim, Yonghwan Kim</td>
</tr>
<tr>
<td></td>
<td>Maneuvering Simulations of a VLCC in Adverse Weather Conditions (332)</td>
</tr>
<tr>
<td></td>
<td>Mochammad Zaky, Hironori Yasukawa</td>
</tr>
</tbody>
</table>
**November 7 (Tuesday)**

<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
</tr>
</thead>
<tbody>
<tr>
<td>08:30-17:00</td>
<td>Registration desk open <em>(Education Center, National Museum of Marine Science and Technology)</em></td>
</tr>
<tr>
<td>09:00-10:20</td>
<td><strong>Keynote Speech</strong> <em>(International Lecture Hall – Room A)</em>&lt;br&gt;Chair: Prof. Omar Bin Yaakob&lt;br&gt;The New Naval Research Framework and the Hydrodynamic Science and Technology Program at the U.S. Office of Naval Research (KS03)&lt;br&gt;<em>Dr. Woei-Min Lin (Office of Naval Research, USA)</em>&lt;br&gt;Neumann-Michell Theory and Hull Form Optimization (KS04)&lt;br&gt;<em>Prof. Chi Yang (George Mason University, USA)</em></td>
</tr>
<tr>
<td>10:20-10:40</td>
<td>Coffee break</td>
</tr>
</tbody>
</table>

**Parallel Sessions IV**

<table>
<thead>
<tr>
<th>Time</th>
<th>Session 4-1 (Room A)</th>
<th>Session 4-2 (Room B)</th>
<th>Session 4-3 (Room C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>10:40-12:00</td>
<td><strong>Computational fluid dynamics, numerical wave tank and virtual displaying system (III)</strong>&lt;br&gt;<em>Chair: Prof. Zao-Jian Zou</em>&lt;br&gt;Dual BEM for Wave Diffraction of a Thin Structure with an Arbitrary Cross Section (419)&lt;br&gt;Ching-Yi Tu, Ching-Yun Yueh, Chih-Ting Chang, Shih-Hsuan Chuang</td>
<td><strong>Hydrodynamics of renewable marine energy and ocean resources (II)</strong>&lt;br&gt;<em>Chair: Prof. Ray-Yeng Yang</em>&lt;br&gt;A Fully Coupled Simulation Method for Floating Offshore Wind Turbine Dynamics using a Boundary Element Method in Time Domain (360)&lt;br&gt;Stefan Netzband, Christian W. Schulz, Moustafa Abdel-Maksoud</td>
<td><strong>Linear and nonlinear waves and current (II)</strong>&lt;br&gt;<em>Chair: Prof. Chein-Shan Liu</em>&lt;br&gt;Evolution of an ISW with Different Modes Propagating across Vertical Cylinder (413)&lt;br&gt;Ming-Hung Cheng, Chih-Min Hsieh, Robert R. Hwang</td>
</tr>
<tr>
<td>11:00-11:20</td>
<td><strong>A Computational Study on System Dynamics of an Ocean Current Turbine (320)</strong>&lt;br&gt;Jo-Ti Wu, Jiahn-Hong Chen, Ching-Yeh Hsin, Forng-Chen Chiu</td>
<td><strong>Dynamic Pressure Distribution and Wave Forces on Offshore Spar-type Wind Turbines in Diffracted Wave Field (399)</strong>&lt;br&gt;S. Kimura, Y. Nihei, S. Srinivasamurthy</td>
<td><strong>The Research and Verification of Wave-generating and Wave-absorbing in 3-D Numerical Wave Tank (381)</strong>&lt;br&gt;Jingjing Lu, Peng Wei, Zhiguo Zhang</td>
</tr>
<tr>
<td>Time</td>
<td>Session Title</td>
<td>Presenter(s)</td>
<td></td>
</tr>
<tr>
<td>--------------</td>
<td>-------------------------------------------------------------------------------</td>
<td>------------------------------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>11:20-11:40</td>
<td>Numerical Validation of Aerodynamics for Two In-line Model Wind Turbines Using Actuator Line Model and CFD technique (392)</td>
<td>Yong Ai, Ping Cheng, Decheng Wan</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>11:40-12:00</td>
<td>A CFD-based Comparative Study for Open and Ducted Propeller (318)</td>
<td>Jie Gong, Chunyu Guo, Kewei Song, Tiecheng Wu, Zhe Guo</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>12:00-13:30</td>
<td>Lunch</td>
<td></td>
<td></td>
</tr>
<tr>
<td>13:30-14:30</td>
<td>Invited Speech (International Lecture Hall – Room A)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Chair: Prof. Ching-Yeh Hsin</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>The Origin and Development of Medium and High Speed Vessels at Lung Teh Shipbuilding Co., LTD. (LTS) (IS03)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Mr. Sheldon S.C. Huang (President, Lung-Teh Shipbuilding Co.)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>An Overview of Hydrodynamic Design and Development in CSBC (IS04)</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Mr. Robert Tseng (President, CSBC Corporation)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Parallel Sessions V

<table>
<thead>
<tr>
<th>Time</th>
<th>Session Title</th>
<th>Presenter(s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>14:35-15:35</td>
<td>Session 5-1 (Room A)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Computational fluid dynamics, numerical wave tank and virtual displaying system (IV)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Chair: Dr. Sopheak Seng</td>
<td></td>
</tr>
<tr>
<td>14:35-14:55</td>
<td>Accuracy Improvement of SVR in Identification of Hydrodynamic Coefficients with Denoising (377)</td>
<td>Yan Jiang, Zao-Jian Zou, Xue-gang Wang</td>
</tr>
<tr>
<td></td>
<td>Session 5-2 (Room B)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Cavitation and cavitating flows (II)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Chair: Prof. Ping-Chen Wu</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Vortex Structure Behind the Cylinder Mounted on Flat Plate (382)</td>
<td>Jiajun Chen</td>
</tr>
<tr>
<td></td>
<td>Session 5-3 (Room C)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Hydrodynamic performance of offshore platforms, pipelines, risers and mooring Systems (II)</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Chair: Dr. Xingya Feng</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Dynamic Truncation Analysis for Mooring Lines with Multi-Objective Optimization Method (423)</td>
<td>Gang Ma, Yue Jiang, Hongwei Wang, Liping Sun</td>
</tr>
<tr>
<td>Time</td>
<td>Session</td>
<td></td>
</tr>
<tr>
<td>------------</td>
<td>-------------------------------------------------------------------------</td>
<td></td>
</tr>
</tbody>
</table>
| 14:55-15:15| **Computational Simulation of a Submarine Propeller Using Overlapping Grid Method** (380)
Peng Wei  |
|            | **The Numerical Study on the Cavitation-structure Interaction of the Elastic Cylinder Shell Exiting Water** (430)
Shi-liang Hu, Chuan-jing Lu, Jing-pu Chen  |
|            | **Neumann-Kelvin Theory versus Neumann-Michell Theory for the Ship Wave Problem by a Vertical Surface-Piecing Cylinder** (374)
Hui Liang, Xiaobo Chen  |
| 15:15-15:35| **Time-stepping Schemes for Seakeeping in OpenFOAM** (426)
C. Monroy, S. Seng  |
|            | **Numerical Simulation of Cavitating Flow Over a Projectile with Adaptive Shape** (314)
Tezhuan Du, Yiwei Wang, Chenguang Huang  |
|            | **Analysis Aero Dynamic Performances of an Offshore with Effects of Wind Attacked Angle and Hull Shape by Using CFD** (428)
N.C. Cong, B.D. The, L.T. Thai, Ph.A. Tuan, N.V. He  |
| 15:35-15:55| Coffee break                                                          |

**Parallel Sessions VI**

<table>
<thead>
<tr>
<th>Time</th>
<th>Session 6-1 (Room A)</th>
</tr>
</thead>
</table>
| 15:55-17:35| **Computational fluid dynamics, numerical wave tank and virtual displaying system (V)**
*Chair: Prof. Changhong Hu*  |
|            | **Horseshoe Vortex Suppression Due to a Strake** (328)
Jun Pei Lee, Jiahn-Horng Chen, Ching-Yeh Hsin  |
| 15:55-16:15| **Experimental and Theoretical Investigation of Liquid Sloshing in a Rotating, Laterally Oscillating Cylindrical Container** (361)
Y. Saito, T. Sawada  |
| 16:15-16:35| **A Numerical Study on Wake Structure and Hydrodynamic Characteristics of Fish-like Undulation** (347)
Zeyu Guo, Zuogang Chen  |
| 16:35-16:55| **CFD Application for Sloshing Flow in a Rectangular Tank under Forced Excitation** (376)
Jeoungkyu Lee, Jieung Kim, Yonghwan Kim  |

<table>
<thead>
<tr>
<th>Time</th>
<th>Session 6-2 (Room B)</th>
</tr>
</thead>
</table>
| 15:55-17:35| **Slamming, sloshing, green water, run-up, impact (II)**
*Chair: Prof. Tatsuo Sawada*  |
| 16:15-16:35| **Numerical Investigation on Global Wave Loads of a High Speed Monohull in Irregular Waves** (312)
Chung-Ju Tsai, Ming-Jyun Dai, Chih-Chung Fang  |

<table>
<thead>
<tr>
<th>Time</th>
<th>Session 6-3 (Room C)</th>
</tr>
</thead>
</table>
| 15:55-17:35| **Resistance, propulsion, seakeeping, maneuverability (II)**
*Chair: Prof. Touvia Miloh*  |
| 16:15-16:35| **Numerical Analysis of Ship Generated Unsteady Waves Based on a Cartesian-Grid Method** (461)
Kyung-Kyu Yang, Masashi Kashiwagi  |

<table>
<thead>
<tr>
<th>Time</th>
<th>Session 6-3 (Room C)</th>
</tr>
</thead>
</table>
| 16:35-16:55| **Mesh Properties for RANS Simulations of Aerofoil-Shape Rudder Hydrodynamics** (412)
Jialun Liu, Robert Hekkenberg, Zhonglian Jiang, Xiumin Chu  |
|            | **The Effect of the Sloshing Tank on the Ship Rolling Motion in Beam Waves** (416)
Ming-Chung Fang, Jiun-Ting Lin, Zi-Yi Lee  |

<table>
<thead>
<tr>
<th>Time</th>
<th>Session 6-3 (Room C)</th>
</tr>
</thead>
</table>
| 16:35-16:55| **Numerical Analysis of Ship Generated Unsteady Waves Based on a Cartesian-Grid Method** (461)
Kyung-Kyu Yang, Masashi Kashiwagi  |
<table>
<thead>
<tr>
<th>Time</th>
<th>Session</th>
<th>Chair</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>16:55-17:15</td>
<td>Simulation of Violent Free Surface Flow by AMR-IBM Method</td>
<td>Changhong Hu, Cheng Liu (327)</td>
<td>Room A</td>
</tr>
<tr>
<td></td>
<td>Sloshing in a Partially Filled Membrane tank with a Tank Length and Tank Breadth that are Almost Equal (321)</td>
<td>Makoto Arai, Gustavo Massaki Karuka, Yuka Akiyama, Tomomi Yoshida</td>
<td>Room B</td>
</tr>
<tr>
<td></td>
<td>Hydrodynamics of a Two-ship Encounter Problem (345)</td>
<td>Touvia Miloh, Ioannis K. Chatjigeorgiou</td>
<td>Room C</td>
</tr>
<tr>
<td>17:15-17:35</td>
<td>Design and Experiment of the Improved Parallel Correction Guidance Autopilot System (365)</td>
<td>Sin-Der Lee</td>
<td>Room D</td>
</tr>
<tr>
<td></td>
<td>Evaluation of Acceleration in Retreated Flow during Run-down Phase of Solitary Wave over Steep Slope (459)</td>
<td>Wei-Ying Wong, Chang Lin, Ming-Jer Kao</td>
<td>Room E</td>
</tr>
<tr>
<td>18:30-21:00</td>
<td>Conference Banquet &amp; Best Student Paper Awards announcement at Evergreen Laurel Hotel, Keelung (The Peng’s Gourmet and Banquet, 3rd Floor)</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**November 8 (Wednesday)**

<table>
<thead>
<tr>
<th>Time</th>
<th>Event</th>
<th>Chair</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>08:30-14:00</td>
<td>Registration desk open (Education Center, National Museum of Marine Science and Technology)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>09:00-09:40</td>
<td>Keynote Speech (International Lecture Hall – Room A)</td>
<td>Prof. Jiahn-Horng Chen</td>
<td>Room A</td>
</tr>
<tr>
<td></td>
<td>Research and Development Activities in Offshore Renewable Energy in the UK (KS05)</td>
<td>Prof. Incecik, Atilla (University of Strathclyde, United Kingdom)</td>
<td>Room B</td>
</tr>
<tr>
<td>09:40-10:00</td>
<td>Coffee break</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Parallel Sessions VII**

<table>
<thead>
<tr>
<th>Time</th>
<th>Session</th>
<th>Chair</th>
<th>Location</th>
</tr>
</thead>
<tbody>
<tr>
<td>10:00-12:00</td>
<td>Session 7-1 (Room A) Fluid-structure interaction and hydro-elasticity</td>
<td>Prof. Lian Shen</td>
<td>Room A</td>
</tr>
<tr>
<td></td>
<td>Session 7-2 (Room B) Resistance, propulsion, seakeeping, maneuverability (III)</td>
<td>Prof. Yi-Chih Chow</td>
<td>Room B</td>
</tr>
<tr>
<td></td>
<td>Session 7-3 (Room C) Computational fluid dynamics, numerical wave tank and virtual displaying system (VI)</td>
<td>Prof. Liang-Yee Cheng</td>
<td>Room C</td>
</tr>
<tr>
<td>10:00-10:20</td>
<td>Review of VIV of Cylindrical Structure in Current and Suppression Methods (465)</td>
<td>Liping Sun, Daming Wang, Shuhong Chai</td>
<td>Room D</td>
</tr>
<tr>
<td></td>
<td>Comparison of Different Added Power in Waves Prediction Methods (409)</td>
<td>Weimin Chen, Jianpeng Li, Guoxiang Dong, Lei Xing</td>
<td>Room E</td>
</tr>
<tr>
<td></td>
<td>Study on Visualization of Ship Collision Risk in Complex Intersection Waters (410)</td>
<td>Yi Liu, Youxi Hu, Jiao Zhu, Jingxian Liu</td>
<td>Room F</td>
</tr>
<tr>
<td>Time</td>
<td>Title</td>
<td>Authors</td>
<td>Location</td>
</tr>
<tr>
<td>------------</td>
<td>-----------------------------------------------------------------------</td>
<td>----------------------------------------------</td>
<td>----------------------------------------------------</td>
</tr>
<tr>
<td>10:20-10:40</td>
<td>Numerical Analysis the Dynamic Performance of Composite Propeller (386)</td>
<td>Fanchen Zhang, Dakui Feng, Peng Wei</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Study on the Flow Field between the Main- and Demi-Hull of a Trimaran (466)</td>
<td>Lianzhou Wang, Chunyu Guo, Zuotian Zhang, Yumin Su</td>
<td></td>
</tr>
<tr>
<td></td>
<td>CFD-based Hull Form Multi-objective Optimization for Better Resistance and Wake Performances (393)</td>
<td>Xinwang Liu, Aiqin Miao, Decheng Wan</td>
<td></td>
</tr>
<tr>
<td>10:40-11:00</td>
<td>Nonlinear Effects on Ship Hydroelastic Responses with Forward Speed in Large-amplitude Waves (460)</td>
<td>Ryuta Tanaka, Takuya Taniguchi, Masashi Kashiwagi</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Experimental Research on the Resistance and Motion Attitude Variation of Ship-wave-ice Interaction in Marginal Ice Zones (464)</td>
<td>Wan-zhen Luo, Chun-yu Guo, Tie-cheng Wu, Yu-min Su</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Parametric Study on the Flow Passing the Cylinder Mounted on the Edge of Backward-facing Step (442)</td>
<td>Trieu V. Nguyen, Jiahn H. Chen</td>
<td></td>
</tr>
<tr>
<td>11:00-11:20</td>
<td>Analysis of Dissipation on Gap Resonance Between side-by-side Barges (348)</td>
<td>X.Y. Feng, X.B. Chen, H. Liang, F. Dias</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Transient Diffraction Waves by Fourier-Laguerre Expansions (387)</td>
<td>R.P. Li, H. Liang, X.B. Chen, W.Y. Duan</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Nonlinear Numerical Analysis of the Flow of Fluid Trapped inside a Narrow Gap by a Particle Method (458)</td>
<td>Cezar A. Bellezi, Liang-Yee Cheng, Makoto Arai</td>
<td></td>
</tr>
<tr>
<td>11:20-11:40</td>
<td>Numerical Simulation of Interactions among Air, Water, and Rigid/Flexible Solid Bodies (337)</td>
<td>Sida He, Zixuan Yang, Lian Shen</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Analysis of the Contribution of Rim-Driven Thruster System to Energy Efficiency Design Index (334)</td>
<td>Jiafen Lan, Yan Xinping, Wu Ouyang</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Investigation of the Pre-Swirl Stator Effect to the Ship Resistance in Calm Water and in Waves (450)</td>
<td>Yao-Tang Mao, Yu-Wen Hsieh, Ling Lu, Ching-Yeh Hsin, Min-Mei Shih, Sheng-Ann Shieh</td>
<td></td>
</tr>
<tr>
<td>11:40-12:00</td>
<td>An Empirical Formula for the Fundamental Resonance Frequency in a Gap Between two 2D Barges (316)</td>
<td>B. Teng, L.F. Cong, Y. Gou</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Resistance Prediction of Viscous Flow by OpenFOAM Solver (330)</td>
<td>Tran gia Thai, Le Van Toan</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Three-dimensional Numerical Study on Green Island Wake(452)</td>
<td>Tien-Hung Hou, Shih-Chun Hsiao, Chia-Cheng Tsai, Tai-Wen Hsu</td>
<td></td>
</tr>
<tr>
<td>12:00-12:20</td>
<td>Closing Ceremony (International Lecture Hall – Room A)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12:20-14:00</td>
<td>Farewell Buffet Lunch</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
ABSTRACTS OF IWSH 2017
**IWSH 2017-KS01**  
**Hydroelastic Responses of VLFS in Complicated Geographical Environment**  
You-Sheng Wu  
China Ship Scientific Research Center, China

The requirement of sustainable development of world economy has strengthened the efforts of mankind to increase the capability of resource exploitation and space utilization in the ocean. Very Large Floating Structures (VLFS) have attracted long lasting attention in ocean utilization for several decades. The applications of different size VLFSs as floating piers, floating airports, floating hotels, floating fuel facilities and even floating cities have attracted extensive researches. Due to much larger dimensions, relatively smaller global rigidities and lower natural frequencies than an ordinary ship, a VLFS has apparent flexible structural responses rather than rigid body motions in waves. Hence hydroelastic analyses are of great importance in design and safety assessment of a VLFS. Most of the previous researches about the hydroelastic responses of a VLFS were for the body floating in open sea with constant water depth and encountering one-directional incoming waves of a specified wave spectrum. If a VLFS is deployed near islands and reefs in complicated geographical environment, the wave conditions, wave loads and the hydroelastic responses will be quite different and more complicated than in open sea. In this paper the three-dimensional hydroelasticity theory of floating bodies is combined with the shallow water wave theory, to allow for the influence of the complicated geographical environment of seabed and islands on the hydroelastic responses of a VLFS deployed near island and reefs in shallow sea. Some important technical problems regarding the design and safety of a multi-module VLFS in the complicated geographical environment are also discussed.

**IWSH 2017-KS02**  
**Brief Introduction of Detailed Flow Field Measurement and CFD computation around Model Ship Stern in Waves in Towing Tank**  
Yasuyuki Toda  
Osaka University, Japan

Investigating the propeller load fluctuation in waves is very important to consider ship propulsion performance in waves. Many studies have been done for the propulsion in waves. But the velocity distribution at propeller plane in waves has not been measured in detail due to the difficulty of experiment. On the contrary, with the rapid progress of CFD code, the computation of flow field around a moving ship in waves can be possible. But the validation data for detailed flow field is not available. Some phase-averaged flow measurements were done for various conditions, but the flow fields around a ship with motion in waves is very rare. So, the phase averaged flow measurement is conducted in this study using stereo PIV system and phase synchronizer with heave motion. In this study, phase-averaged flow field in waves was measured using SPIV system in the towing tank around the KVLCC2 model ship for full load condition and ballast load condition. Some other measurement results for hull-propeller-rudder interaction problem in waves are presented. Comparisons with CFD computation are shown a little bit in the presentation.
The New Naval Research Framework and the Hydrodynamic Science and Technology Program at the U.S. Office of Naval Research
Woei-Min Lin
Office of Naval Research, USA

In July 2017, the U.S. Office of Naval Research rolls out a new document, entitled "Naval Research: A Framework for the Navy & Marine Corps After Next" to replace the original "Naval S&T Strategy". This new approach reflects a change in thinking about the science, business and people of research to gain agility and accelerate technology innovation. In this presentation, Dr. Lin will discuss this new naval research framework and the focus of the current hydrodynamics science and technology program at the U.S. Office of Naval Research.

Neumann-Michell Theory and Hull Form Optimization
Chi Yang
George Mason University, USA

Hydrodynamic optimization is an important aspect of ship design. It has become increasingly important to both model hull forms accurately and evaluate hydrodynamic performance, especially the drag of a ship hull, efficiently during the early stage of the design process. An overview of a practical theory, called Neumann-Michell (NM) theory, for evaluating the flow around the hull surface of a ship that advances at a constant speed in calm water of large depth will be given. The theory, expounded in a series of studies, provides realistic predictions of the sinkage, the trim, the drag, wave profiles and wave patterns that are in satisfactory agreement with experimental measurements. Moreover, the computational tool based on the NM theory is very robust and highly efficient. Only surface meshes on the hull are required. The CPU time for evaluating the flow about a ship hull that is approximated by 10,000 panels is only a few seconds per Froude number using an Intel Pentium 7 Processor PC. Therefore, the NM theory based simple computational tool is very well suited for the early-stage simulation-based hydrodynamic design optimization of ship hull forms. The applications of the NM theory to the hydrodynamic optimization of both mono-hull ships and multi-hull ships and experimental validations will be presented in this talk. The future outlook of simulation-based hydrodynamic design optimization of ship hull forms will also be addressed.
Research and Development Activities in Offshore Renewable Energy in the UK
Atilla Incecik
University of Strathclyde, United Kingdom

The presentation will start with the introduction to the Industrial Doctoral Centre in Offshore Renewable Energy (IDCORE).

During the second part of the presentation offshore renewable research activity in Kelvin Hydrodynamics Laboratory at Strathclyde University, Glasgow, offshore renewable research Landscape in the UK, and the EU funded MARINET programme on Marine Renewable Infrastructure Network will be described. A summary of research requirements identified by the Offshore and Marine Renewable Committees of the International Towing Tank Conference (ITTC) and International Ship and Offshore Structure Congress will be summarised.

The presentation will be concluded by describing current offshore and marine renewable energy projects under development or planned in Scotland and elsewhere in the UK.
SOIC, A Cornerstone of Ship and Ocean Industries in Taiwan
Forng-Chen Chiu
President, Ship and Ocean Industries R&D Center

The presentation will focus on the introduction to the Ship and Ocean Industries R&D Center (SOIC), which is a non-profit agency supervised by the Ministry of Economic Affairs of Taiwan (MOEA). SOIC will be devoted to promote the development of Taiwan's marine economy, standing firmly in Taiwan to pursue technical excellence, while keeping an eye on Asia and marching towards international markets, continuing to be diligent in the field of “advanced ship and ocean technology.” At current stage, we will coordinate with Government's policies, actively participate in the role of building indigenous defense ships, supporting offshore wind farm development, starting to develop smart ship technology, and integrate industry capacities to drive the overall marine development in Taiwan.

Recent Development in CR Classification Society
Ying-Ru Liu
President, CR Classification Society

CR Classification Society, founded in 1951 as a non-governmental and non-profit technical organization, permanently serves the marine industry in respect of research, design process, construction, trials, operation and maintenance of marine vessels and structures as well as our conformity confirmation of relevant classification and statutory requirements. To keep technical excellence, CR develops advanced software for ship structure and offshore wind turbines, enhances the direct calculation applications in structure, hydrodynamics and fluid-structure interaction and builds its classification rules and guidelines. Recent developments specially focus on facilitating the national policies for the offshore wind farm, surface naval ship and indigenous defense submarine. This speech gives an idea of how CR provides technical supports to the industry with its advanced knowledge and its independent 3rd role of certification body, to prove both safety and efficiency.

The Origin and Development of Medium and High Speed Vessels at Lung Teh Shipbuilding Co., LTD. (LTS)
Sheldon S.C. Huang
President, Lung-Teh Shipbuilding Co.

Lung-Teh Shipbuilding was established in 1979. In the beginning, LTS built offshore fishing boats with FRP for the local market. In 1983, LTS won the first order for the medium speed patrol boat made of FRP with a 17-knot speed. Since then, LTS has delivered more than 450 medium and high speed vessels for the world market. Now, LTS can build high speed vessels made of FRP,
steel, and aluminum alloy, with a maximum of 60-knot speed, and the maximum hull length is up to 70 meters, including mono hull and multi hull. In this talk, I will briefly introduce how we began and what we have done to build medium and high speed vessels.

**An Overview of Hydrodynamic Design and Development in CSBC**

Robert Tseng
President of CSBC Corporation, Taiwan

Before 1990, the key design of CSBC built vessels were mostly outsourcing. From the project of 149,000 DWT Bulker, CSBC start to develop its own design ability and tried to verify the design of 260,000 DWT Oil Tanker and develop own design. In that stage, due to the lack of knowledge and tools of hydrodynamic, even the hull lines were hand-made and fairing, then analysed by with panel code. Propeller design was based on charts of MAU series. At the meantime, validation of the performance was mostly relied on model test in HSVA, Germany. Fortunately, the 165,000 DWT Bulker became the first self-developed product and delivered smoothly, lead to the success of followed Bulker design. In 1997, CSBC built up its reputation and the name of the professional container vessel shipyard, by the project of 2,200 TEU C/V, owned by Rickmers, Germany. In that project, more than 9 versions of hull form design were tested by model test, so that the EFD ability was grown up substantially. Nowadays, almost full sizes of C/V were well developed and operated over the world. Furthermore, CSBC had designed and delivered two famous Semi-Submersible Heavy Lift Deck Cargo Carrier, Black Marine and Blue Marlin. A well-known shipyard was established no doubtfully.

Since 2006, CSBC had entered a new era, not only inducing the new analysis tool and design concept, such as RANS CFD and theory propeller design, but also taking part in and keep its pace with the design trend of the world, especially the energy saving technology. CSBC deeply devotes to the research of hydrodynamics, by cooperating with academies and international resources. The significant milestone is so-called ES-10 project, which made achievements of 10% energy saving from hydrodynamics and consequently brought up the inventions. Now, CSBC’s new designs are all under the core design philosophy of SODO, seaway optimum design and operation. Applications could cover from commercial to special purpose ships, even navy ships, including the EMS concept (Easy Manoeuvring Ship), providing better manoeuvrability for operation flexibility.

To take a step further, under the trend of IoT and industry 4.0, CSBC extends its efforts to the 4IntShip project, the ship with Intranet, Internet, Integration and Intelligent. CSBC no longer just focus on ship itself, also try to find out the green energy solution and for public service. Attentions will be paid on power generated by ocean current, tidal waves and offshore wind, for boarder marine designs and services to the world.
IWSH 2017-308

Uncertainty Analysis in CFD for Resistance of Japan Bulk Carrier with and without an Energy Saving Device
Li Xia1*, Zao-jian Zou1, Jin Chen1, Lu Zou1 and Lei-lei Li2
1Shanghai Jiao Tong University, School of Naval Architecture, Ocean and Civil Engineering, 800 Dongchuan Road, Shanghai 200240, China
2ChangJiang Ship Design Institute, Department of R&D 387 Linjiang Road, Wuhan 430062, China
*Corresponding author, sherlock@sjtu.edu.cn

Resistance prediction for a ship is one of the most important tasks at the design stage. It is closely related to efficiency of navigation and environmental protection. Along with the rapid development of computer science and technology, Computational Fluid Dynamics (CFD) technology has been widely and successfully applied in ship hydrodynamics problems. However, accuracy of CFD is still limited and the accuracy obtained from computations is always a concern. For this reason, the uncertainty analysis in CFD is arousing great concern.

In this paper, resistance is predicted for the test case Japan Bulk Carrier (JBC) with and without Energy Saving Device (ESD). The analyses are carried out by utilizing a commercial CFD software package. Furthermore, a Verification and Validation (V&V) procedure is applied to the resistance result in order to assess the uncertainties and numerical errors.

IWSH 2017-309

Wave Amplitudes in an Open-type Circular Caisson due to Resonance Effects
Yan-Xiang Lin, Da-Wei Chen, Jiahn-Hong Chen*, Kuan-Juan Pan
Department of Systems Engineering and Naval Architecture, National Taiwan Ocean University, 2 Pei-Ning Road, Keelung 20224, Taiwan
*Corresponding author, b0105@mail.ntou.edu.tw

An open-type circular caisson with 60 degree of opening inlet for a model scale of 1:30 has been conducted a serial of simulation testing for evaluating feasibility in coastal waters nearby north-eastern Taiwan. This type of caisson has a property in concentrating the wave energy from incident waves to increase the energy capture for wave energy converters. Moreover, by analysing the cumulative annual wave power from field-measured data in north-eastern Taiwan show that the 1-4 m of wave heights with 6.5-10.5 s of wave periods are the typical conditions in this region. the wave conditions in north-eastern Taiwan could be induced about 0.8 to 1.45 for incident waves of 3.0 m and with about 0.70 to 1.20 for incident waves of 2.5 cm. Comparisons of the incident waves with different incoming angles show that the incident waves with 0 degrees have slightly larger Amplification Factors than that with 15 degrees but the similar standard deviations of wave heights and wave periods.

IWSH 2017-312

Numerical Investigation on Global Wave Loads of a High Speed Monohull in Irregular Waves
Chung-Ju Tsai, Ming-Jyun Dai, Chih-Chung Fang*
National Taiwan Ocean University, Department of Systems Engineering and Naval Architecture 2 Pei-Ning Road, Keelung 20224, Taiwan
*Corresponding author, ccfang@mail.ntou.edu.tw

The 3D translating pulsating source distribution technique (3DT) is used to predict the wave
loads of a high-speed monohull (RD-200) in irregular waves. The numerical results are compared with experimental values obtained from tests carried out at the SSPA with a two-segmented model. These test results cover different wave directions at two different forward speeds. A short-term statistical analysis of wave load response for the RD-200 in random seas has been carried out by using the 3DT techniques. In general, the comparison of the wave load responses between numerical predictions obtained from 3DT and experimental data carried out in SSPA generally shows good agreement for the RD-200 at V=15 and 20 knots in the irregular waves (H_{1/3} = 3.05, T_0 = 9sec ). Furthermore, the wave loads of RD-200 including the shear forces, bending moments and torsional moments at a transverse section of the RD-200 are predicted at various wave heading and ship speeds in oblique waves. The Polar Plot has been developed to display the boundaries between ship speeds and ship-wave heading angles associated with wave load responses in a particular sea state for discussions. The characteristics of transverse sectional wave load responses of the monohull have been discussed.

**IWSH 2017-313**

**Stochastic Parametric Roll Motion of a Vessel under Random Wave**

Liyuan Wang\(^1,2\), Yougang Tang\(^1,2\)*, Xiaorui Zhang\(^1,2\)

\(^1\)Tianjin University, School of Civil Engineering, 92 Weijin Road, Tianjin, 300072, China

\(^2\)Tianjin University, State Key Laboratory of Hydraulic Engineering Simulation and Safety, 92 Weijin Road, Tianjin, 300072, China

*Corresponding author, tangyougang_td@163.com

In this paper, considering the ship under random wave, the nonlinear differential equation of parametric roll motion is established, and the probability density function of random roll amplitude under stochastic wave is obtained by stochastic averaging method. This method is very effective in calculating the stability of the ship in random waves.

**IWSH 2017-314**

**Numerical Simulation of Cavitating Flow over a Projectile with Adaptive Shape**

Tezhuan Du\(^1,2\)*, Yiwei Wang\(^1,2\), Chenguang Huang\(^1,2\)

\(^1\)Institute of Mechanics, Chinese Academy of Sciences, Beijing, 100190, China

\(^2\)School of Engineering Science, University of Chinese Academy of Sciences 100049, China

*Corresponding author, dutezhuan@imech.ac.cn

A fluid-structure coupling model is established to study the performance of adaptive shape on control of cavitation. The proposed model is then applied to simulate cavitating flows over a hollow projectile with adaptive shape. The adaptive shape is achieved by deformation of elastic membrane under internal-external pressure difference. The membrane bulges and recovers while cavitation grows and collapses respectively. Conversely, deformation of membrane affects the evolution of cavitation. Different Young modulus are adopted to analysis the interaction between cavitating flow and elastic membrane. The results show that adaptive shape can reduce the total volume of cavitation and collapse pressure under specific parameters.

**IWSH 2017-316**

**An Empirical Formula for the Fundamental Resonance Frequency in a gap between two 2D Barges**

B. Teng\(*\), L.F. Cong, Y. Gou

State Key Laboratory of Coastal and Offshore Engineering, Dalian University of Technology, Dalian 116024, China
For wave interaction with two 2D floating barges, an empirical equation is proposed for predicting the resonance frequency in the gap between them. The equation is set up on the oscillation theory of the trapped water with the influence of added mass and radiation damping. Through numerical examination, it found that the added mass plays a key role in the resonance frequency, but the radiation damping has less effect on it. Numerical examinations are carried to study the relation of added mass with gap width, barge width and barge draft, then an empirical formula for added mass is set up with those factors. Furthermore, an empirical formula for the resonance frequency inside the gap is derived. It is shown that the new empirical formula is simple and has good accuracy for wide ranges of gap width, barge width and barge draft.

**IWSH 2017-317**

**Numerical Study on the Wind Field Development for CSBC 14,000 TEU Container Vessel**

Min-Mei Shih, Chen-Yue Lin, Yung-Ta Huang, Sheng-An Hsieh
CSBC Corporation, Department of Design 3 Jhonggang Road, Siaogang District, Kaohsiung 81234, Taiwan

*Corresponding author, 108054@csbcnet.com.tw*

For the merchant ship of displacement type, the air drag above waterline accounting for total resistance is about 3-5% for tankers and bulkers, and 5-7% for container carriers. This portion normally won’t affect the guarantee ship speed for relatively slow speed ships during sea trial validation, but might be an issue for medium high speed ships like container carrier. To obey the ISO15016 sea trial rule, the ship speeds should be corrected by the true wind speed. Therefore, how to measure the true wind speed on board of ship becomes an important issue and need room for investigation. In this paper, CFD techniques are introduced to simulate the wind field for a 14,000 TEU class container vessel under the head wind condition to compare the variation of wind field between different anemometer positions. By CFD analysis result and actual measured wind speed data during sea trial, the ISO15016 correction for future sea trial speed can be strengthened and persuasive.

**IWSH 2017-318**

**A CFD-based Comparative Study for Open and Ducted Propeller**

Jie Gong¹, Chunyu Guo*¹, Kewei Song¹, Tiecheng Wu¹, Zhe Guo²

¹College of Shipbuilding Engineering, Harbin Engineering University, 145# Nantong street, Harbin 150001, China
²School of Economics and Management, Harbin Engineering University, 145# Nantong street, Harbin 150001, China

*Corresponding author, guochunyu_heu@outlook.com*

In this study, we numerically investigate the different wake characteristics of the open and ducted propellers set in open-water test conditions. Detached Eddy Simulation (DES) approach is employed to track the main features of the dynamic characteristics in the propeller wake, under different loading conditions. The eddy viscosity model of Spalart & Allmaras and an unsteady sliding mesh technique have been used in the simulations, which are performed on topologically similar hybrid-grids to ensure a consistent comparison. Compared with the experimental results, CFD calculations can give a very good prediction of the hydrodynamic performance of all the cases, which indicate certain credibility for the wake analysis. It can be observed that consecutive tip vortices induced by different blades are convected along the longitudinal direction for the open propeller, resulting in a regular spiral trajectory. Meanwhile, the merging and grouping of tip vortices is delayed downstream as the advance coefficient increasing. However, due to the strong interaction between
the duct and the propeller, a massive secondary vortices are wrapping around the tip vortices, which have interacted and merged with wake vortices for the ducted propeller. The vortex morphology is discontinuous and an irregular spiral trajectory can be observed. In particular, the shed vortex induced by the flow separation of the duct will occupy the main vortex system at the highest advance coefficient.

IWSH 2017-320

A Computational Study on System Dynamics of an Ocean Current Turbine
Jo-Ti Wu\textsuperscript{1}, Jiahn-Horng Chen\textsuperscript{1,}\textsuperscript{*}, Ching-Yeh Hsin\textsuperscript{1}, Forng-Chen Chiu\textsuperscript{2}
\textsuperscript{1}National Taiwan Ocean University, Department of Systems Engineering and Naval Architecture 2 Pei-Ning Road, Keelung 20224, Taiwan
\textsuperscript{2}National Taiwan University, 1 Roosevelt Road, Sec. 4, Taipei 104, Taiwan
\textsuperscript{*}Corresponding author, b0105@mail.ntou.edu.tw

The development of marine energy has become a new frontier of technology research all over the world. Many technological challenges have been addressed. Coping with these challenges, various converters were devised for harnessing different marine energies in the past decade. Among them are the ocean current turbine systems which, in fact, represent another level of deep-water technology. In Taiwan, the floating Kuroshio turbine (FKT) system is under development. In the present study, the system dynamics subject to various scenarios were investigated. We integrated several commercial and in-house packages. The system buoyancy and weight and their centers were estimated using the Rhino software. The system hydrodynamic coefficients were obtained through WAMIT, system drag coefficient through FLUENT, turbine propulsive force through lifting surface code, and system dynamics through OrcaFlex. Detailed study will be presented in the paper. The results show that the system is well designed with excellent stability.

IWSH 2017-321

Sloshing in a Partially Filled Membrane Tank with a Tank Length and Tank Breadth that are almost Equal
Makoto Arai\textsuperscript{*}, Gustavo Massaki Karuka, Tomomi Yoshida, Yuka Akiyama
Yokohama National University, Department of Systems Design for Ocean-Space 79-5 Tokiwa-dai, Hodogaya-ku, Yokohama 240-8501, Japan
\textsuperscript{*}Corresponding author, m-arai@ynu.ac.jp

It is known that there is a possibility that severe sloshing can occur in partially filled cargo tanks of membrane LNG carriers and that it may cause damage to the tank structures. Recently, new uses of LNG cargo transportation have arisen, such as the use of LNG carriers in the shuttle transportation of natural gas from floating LNG, the adoption of LNG as a fuel for ships, etc. In these new applications, partial filling of the cargo or fuel tank is inevitable. In this paper, experimental and numerical studies of sloshing in a prismatic tank with a tank length and tank breadth that are almost equal were carried out. A series of model experiments was carried out with partially filled model tanks using the motion bed facility. Pressures, longitudinal and transverse forces, and liquid motion in the tanks were measured under regular and irregular excitation. A 3D finite-difference-based solver was used for the numerical simulation. In prismatic-shaped membrane LNG tanks, it is well known that an almost two-dimensional violent free surface motion in the longitudinal or transverse direction of the tanks can easily occur when the tanks are excited at near resonance frequencies. However, we observed that, if the tank length and tank breadth are almost equal, a strong rotational motion of the free surface in the tank, i.e., swirling, occurred. The conditions that induce the sloshing and swirling in the prismatic tanks were studied, and the tank length-to-breadth ratio that causes the swirling motion was identified.
**Flow Mechanism of Impact Wave Forces on a Comb-type Breakwater under Nonbreaking Waves**

Zhuo Fang 1, Zhipeng Zang 2, Ningchuan Zhang 3

1Transport planning and Research Institute, Ministry of Transport 1 Shugungxili, Chaoyang District, Beijing 100028, China
2State Key Laboratory of Hydraulic Engineering Simulation and Safety, Tianjin University 92 Weijin Road, Nankai District, Tianjin 300072, China
3State Key Laboratory of Coastal and Offshore Engineering, Dalian University of Technology 2 Linggong Road, Dalian, Liaoning, 160023, China

*Corresponding author, zhuofang@tpri.org.cn

The interactions between waves and a comb-type breakwater were numerically simulated in a 3D numerical wave flume to investigate the impact wave forces on the comb-type breakwater due to the chamber composed of the side plate and the superstructure. The synchronous analyses on the wave profile, the velocity vectors, the vorticity contours and the wave pressure distribution were conducted to reveal the flow mechanism of the impact wave force. The previous studies observed the impact wave force on a vertical wall with an overhanging cantilever slab occurring under breaking or broken waves. However, in the present study, the impact wave force was found even under non-breaking waves on the similar structure, which is an extension of conclusions of the previous studies. The critical conditions for the impact wave force were further investigated. A composite governing parameter, which included the effects of the water depth, the bottom of the superstructure and the incident wave height, was proposed. It is found that the impact wave force occurs when the incident wave height is comparable to the clearance between the still water level and the bottom of the superstructure for non-breaking waves.

**Resistance and Hull Form Optimization for Vietnamese Fishing Vessels**

Tran Dinh Tu 1, Jiahn-Horng Chen 1, Tran Gia Thai 2

1National Taiwan Ocean University, Department of Systems Engineering and Naval Architecture 2 Pei-Ning Road, Keelung 20224, Taiwan
2Nha Trang University, Department of Naval Architecture 2 Nguyen Dinh Chieu Street, Nha Trang City, Vietnam

*Corresponding author, b0105@mail.ntou.edu.tw

In the present study, the resistance of a traditional Vietnamese fishing vessel was studied numerically. A finite volume method was used to discretize the governing equations for incompressible viscous flow around the ship. The Reynolds-Averaged Navier-Stokes equations were solved with the Realizable k-ω model and SST k-ω model. Moreover, the volume of fluid method was employed to capture the free surface. The SIMPLE algorithm was chosen to iterate the velocity and pressure fields. The results showed that wave-making resistance was about twice as much as the frictional resistance. And we found that Realizable k-ω model is more realizable and accurate.
Based on the Maneuvering Mathematical Modeling Group (MMG) and the Runge-Kutta Method, a mathematical model for simulation of ship maneuverability is established. On this basis, a prediction program is compiled on platform Visual Basic 6.0. The turning motion, Zig-zag and crash stopping ability of a container ship are simulated by this program. Compared with the model test results, the error is less than 10%. In view of the admissible accuracy, the program is capable of predicting ship maneuverability.

IWSH 2017-327
Simulation of Violent Free Surface Flow by AMR-IBM Method
Changhong Hu*, Cheng Liu
RIAM, Kyushu University, Fukuoka, Japan
*Corresponding author, hu@riam.kyushu-u.ac.jp

A novel CFD approach based on adaptive mesh refinement (AMR) technique is being developed for numerical simulation of strongly nonlinear wave-body interaction problems. CIP method is applied to the flow solver and THINC/SW is implemented as the free surface capturing scheme. The PETSc library is adopted to solve the linear systems. The linear solver is redesigned and modified to satisfy the requirement of the AMR mesh topology. In this paper our CFD method is outlined and some newly obtained results are presented to demonstrate its performance.

IWSH 2017-328
Horseshoe Vortex Suppression Due to a Strake
Jun Pei Lee, Jiahn-Horng Chen*, Ching-Yeh Hsin
National Taiwan Ocean University, Department of Systems Engineering and Naval Architecture 2 Pei-Ning Road, Keelung 20224, Taiwan
*Corresponding author, b0105@mail.ntou.edu.tw

In the present study, we conduct a parametric study by computations to investigate horseshoe vortex suppression due to a boundary layer flow past a wing of finite span by a leading-edge strake. Both boundary layer flows over flat and curved walls are employed for the study. The cross section of the wing is a combination of an ellipse at the nose and a NACA0020 airfoil at the tail which join each other at the location of maximum thickness. The Reynolds number is $5 \times 10^5$, based on the chord length of the wing, $C$. The maximum thickness is 0.235$C$ and the span of the wing is 0.705$C$. Two different strake forms are studied. The cross-sectional strake shape on plane parallel to the flat plate is an ellipse; however, its shape on symmetric plane can be linear or elliptical. The ratio of the strake height to strake length in front of the leading edge varies from 0.25 to 1.0. And the absolute height also varies from 0.25$T$ to 1.0$T$, where $T$ is the maximum thickness of the wing. In total, there are 48 cases in this study. The Spalart-Allmaras model (1-equation model) is employed for turbulence effect. The computational results show that different ratio leads to different flow development. Some of them can effectively suppress the horseshoe vortex. The detailed flow field near the leading edge of the wing and the wake development are also studied for different cases.

IWSH 2017-330
Resistance Prediction of Viscous Flow by OpenFOAM Solver
Tran Gia Thai1*, Le Van Toan2
1Nha Trang University, Faculty of Transportation Engineering No 2 Nguyen Dinh Chieu Street, Nha Trang, Vietnam
2Ho Chi Minh City University of Transport, Faculty of Naval Architecture No 2, D3 Street, Ward 25, Binh Thanh District, HCM City, Vietnam
The resistance prediction of the ship is one of the most crucial stages in ship design. This paper presents the numerical prediction of resistance for two typical prototypes of the fishing boat at the coastal area of Vietnam. The finite volume method was utilized to discretize the governing equations for incompressible viscous flow around the ship. The Reynolds-Averaged Navier-Stokes equations were solved by SST k-ω model. Furthermore, we employ the volume-of-fluid method and PIMPLE algorithm which were implemented in OpenFOAM Solver to simulate the viscous flow with free surface around the ship. The main contribution of this study is to suggest the k and ω value for the turbulent condition at the inlet boundary domain for small and low-speed fishing boat. As a result, the total resistance for the model and full-scale ship, which includes viscous and pressure resistance were obtained for the range of speed. The essential features of the flow field around the hull were presented. The computational results also were validated by the experimental data and the maximum deviation around 3% in total resistance.

IWSH 2017-332
Manoeuvring Simulations of a VLCC in Adverse Weather Conditions
Mochammad Zaky*, Hironori Yasukawa
Hiroshima University, Kagamiyama 1-4-1, Higashi-Hiroshima, 739-8527, Hiroshima, Japan
*Corresponding author, d156358@hiroshima-u.ac.jp

In this paper, the effect of load conditions of a VLCC on manoeuvring in stormy conditions is investigated by an MMG-based simulation method [8]. In calm water, turning performance in ballast load condition (NB) becomes worse than that in full load condition (DF) since the course stability in NB is better than that in DF. The steady-state sailing condition in adverse weather conditions is quite different between DF and NB: absolute value of the check helm becomes small in NB but the hull drift angle becomes large. The relative drift direction in turning to the wave directions is 20°-30° in NB and DF with rudder angle 35° and almost the constant for any wave and wind directions. Drifting displacement in turning of NB becomes larger than that of DF at the same environmental condition, although the drifting displacement is almost the constant for any wind and wave directions.

IWSH 2017-334
Analysis of the Contribution of Rim-Driven Thruster System to Energy Efficiency Design Index
Lan Jiafen1,2,3, Yan Xinping1,2,3, Ouyang Wu1,2,3*
1School of Energy and Power Engineering, Wuhan University of Technology, 1178 He-Ping Road, Wuhan 430063, China
2National Engineering Research Center for Water Transport Safety, 1178 He-Ping Road, Wuhan 430063, China
3Key Laboratory of Marine Power Engineering &Technology (Ministry of Communications), 1178 He-Ping Road, Wuhan 430063, China
*Corresponding author, ouyangw@whut.edu.cn

Rim-Driven Thruster (RDT) is a new-concept electric thruster, which integrates the motor and propeller directly without intermediate transmission mechanism, has attracted much attention recently. The RDT has the advantages of compact structure, high system power density, and so on. The purpose of this paper is to study the energy-saving characteristic of RDT compared with the conventional propulsion system based on Energy Efficiency Design Index (EEDI). In this paper, taking the 76000 DWT grade bulk carrier as an example, the differences of structure and energy transfer mode between the traditional propulsion system and RDT are compared. The key param e-
ters of propulsion efficiency and transfer efficiency of these two propulsion systems are analyzed. The EEDI of the target ship equipped with two propulsion systems is calculated respectively. The uncertainty factors and errors of EEDI are discussed. The results show that compared with efficiency, EEDI is more systemic and accurate to describe the energy-saving characteristic of RDT. The EEDI value of the ship is significantly reduced by using RDT, and the target ship can meet the requirements of the baseline for a longer time.

IWSH 2017-335

Application of the Pendulum Theory to Overturning Model of a Rectangular Hull
Yuji Doya*, Tatsuo Sawada
Keio University, Department of Mechanical Engineering 3-14-1 Hiyoshi, Kohoku-ku, Yokohama 223-8522, Japan
*Corresponding author, y-doya@keio.jp

Knockdown pontoons with rectangular hulls are often used in work barges in the construction business field. In this paper, we present a novel method of construction that uses knockdown pontoons. The presented method is a self-dumping pontoon system in which the hull is overturned to unload the earth and sand. We model a series of hull movements as a rigid pendulum, thus developing new model equations and solving them by the Runge-Kutta method. Model experiments were also performed, and the calculation results are in agreement with the experiment results. Lastly, we discuss the criteria for hull overturn in detail.

IWSH 2017-336

Simulation Based Study of Supercavitation Turbulence
Han Liu1, 2, Zuoli Xiao2, Lian Shen1*
1Department of Mechanical Engineering and St. Anthony Falls Laboratory, University of Minnesota
2State Key Laboratory of Turbulence and Complex System, Peking University
*Corresponding author, shen@umn.edu

Supercavitation uses a bubble of gas inside a liquid large enough to encompass an object travelling through the liquid so that the skin friction on the object can be greatly reduced and high speed can be obtained. In this study, direct numerical simulation is used to investigate the turbulence happens in supercavitation. The study builds on an in-house simulation code that uses the coupled level set and volume of fluid method to accurately capture the interface between the water and gas phases. A ventilated disk cavitator is used for the bubble generation, and it is modelled by a sharp interface immersed boundary method. Based on the simulation data, we define two characteristics structures near the closure area of cavity, the shear layer (SL) and the jet layer (JL), which have different flow patterns. By analysing turbulent kinetic energy (TKE) and its budget, the different roles played by SL and JL in TKE transportation and generation process are discussed.

IWSH 2017-337

Numerical Simulation of Interactions among Air, Water, and Rigid/Flexible Solid Bodies
Sida He1, 2, Zixuan Yang1, 2, Lian Shen1, 2*
1Department of Mechanical Engineering, University of Minnesota, Minneapolis, 55455, USA
2St. Anthony Falls Laboratory, University of Minnesota, Minneapolis, 55414, USA
*Corresponding author, shen@umn.edu

We develop a novel numerical method for simulating the fluid-structure interactions (FSI) among air, water, and rigid/flexible solid bodies. The simulation code is developed in house. The fluid flow is
solved using a finite difference method. The air and water are simulated as a coherent system. A coupled level-set (LS) and volume-of-fluid (VOF) method is implemented to capture the interface between air and water. Six degrees of freedom are considered for the motion of rigid bodies. A finite element method is utilized to solve the arbitrarily large deformation of flexible plates. To account for the interaction between fluids and solid bodies, the influences of the presence of the solid bodies on fluid flows are captured by the immersed boundary (IB) method, while the fluid force exerted on solid bodies is determined by the dynamics-based extrapolation of pressure and viscous stress onto the body surfaces. We validate our code with canonical test cases of FSI problems with single-phase fluid, including the vortex-induced vibration of a rigid circular cylinder, rigid cylinder falling in water, and the vibration of a cantilever mounted behind a square cylinder caused by vortex shedding. The accuracy of the code for simulating FSI problems with two-phase fluids is tested in the context of violent breaking waves impinging onto rigid/flexible bodies, and the deformation of flexible plate below waves induced by dam break. The present simulation results are in good agreement with those obtained from laboratory experiments and previous numerical simulations.

**IWSH 2017-338**

*Analysis of Ocean Waves in 3 Sites Potential Areas for Renewable Energy Development in Indonesia*

Inovasita Alifdini*, Nabila Alia Pangestu Iskandar¹, Adhitya Wisnu Nugraha¹, Denny Nugroho Sugianto², Anindya Wirasatriya², Adrian Bela Widodo³

¹Diponegoro University, Oceanography Department
²Diponegoro University, Oceanography Department, Center for Coastal Disaster Mitigation and Rehabilitation Studies
³Diponegoro University, Electrical Engineering Department Jl. Prof. Soedarto SH Tembalang, Semarang 50275 Indonesia

*Corresponding author, inovasita@student.undip.ac.id or inovasita@gmail.com*

Indonesia is a maritime country that has many potential areas for the development of wave energy. Three of them are western coast of Sumatera Island, southern coast of Bali Island and northern coast of Papua Island. In the present study, we examined the potential wave energy in these areas by choosing 1 site in each area. They were Meulaboh (Western part of Sumatra Island), South Kuta (Southern part of Bali Island), and Manokwari (Northern part of Papua Island). We analyzed wave condition for each monsoon from 2012 to 2017 based on forecasting result using Sverdrup, Munk, and Bretschneider (SMB) method. The electrical power calculation was done by using Floating Oscillating Water Column (OWC) formulas. Based our analysis, the significant wave average in each sites was Meulaboh (i.e. 1.41 m, 5.73 s), South Kuta (i.e. 2 m, 6.86 s) and Manokwari (i.e. 1.7 m, 6.52 s). In addition, the highest significant wave average in 3 sites was in the West Monsoon and the lowest was in the East Monsoon. The availability of wave which could be generated electricity from each sites (during 6 years) was Meulaboh 25.65 %, South Kuta 50.87 % and Manokwari 36.5 %. The average of electrical power that could be generated per OWC per year from 3 sites were Meulaboh 636.93 kW, South Kuta 761.79 kW and Manokwari 721.54 kW. Thus, among the 3 sites that we examined, South Kuta Bali was the best site for developing wave energy.

**IWSH 2017-343**

*Experimental Investigation of Motion Responses of Moored Twin Barge Model in Regular Wave in Square Tank*

Van Minh Nguyen, Myung-Jun Jeon, Hyeon Kyu Yoon*

Changwon National University, 20 Changwondaehak-ro Uichang-gu, Changwon-si, Gyeongsangnam-do 51140, Korea
Motion response of floating structures is significant concern in marine engineering. These floating structures can be disturbed by wave, wind and current. This can result in undesirable motions of the vessel which may challenge the operability. For a floating structure, the mooring lines are provided in order to remain in its position and it should be able to induce a restoring force when the vessel displaced. Therefore, it is important to investigate the tension of mooring lines and motion responses of twin barge in moored conditions to guarantee the safety of the twin barge during operation. It is necessary to make clear the characteristics of motion responses of a moored barge under different loading conditions. In this study, measurements of motion responses of the moored twin barge were carried out in regular waves in seven wave directions. The twin barge motions and tension of mooring lines in various wave frequencies are investigated. In this experiment, square tank allows the twin barge model carried out in various wave directions. In addition, the motion components roll, pitch, and heave are completely free. In contrast, the surge, sway, and yaw components are fixed. In the following step, a time domain analysis is carried out to obtain the responses of the structured in the moored condition. The Response Amplitude Operator (RAO) for different wave heading are analyzed. The results of the experiment show that motion components of twin barge have a clear effect on the tension of mooring lines.

IWSH 2017-345

Hydrodynamics of a Two-ship Encounter Problem
Touvia Miloh1*, Ioannis K. Chatjigeorgiou2
1Faculty of Engineering, Tel Aviv University Ramat Aviv, 69978, Israel
2School of Naval Architecture and Marine Engineering, National Technical University of Athens 9 Heroon Polytechniou Ave, Zografos Campus, 15773, Athens, Greece
*Corresponding author, miloh@eng.tau.ac.il

In this study, we tackle with one single effort, three fundamental problems in ship hydrodynamics, namely the attraction force exerted on a single ship moving rectilinearly close and parallel to a rigid wall (berthing), the determination of the associated added-mass coefficients (manoeuvring) and finally evaluating the forces exerted on a ship in a two-ship crossing scenarios. Since the forward ship speed in this case is relatively low, we ignore free-surface effects (double-ship model) with respect to inertia. The novelty of the present approach relies on using Weinblum’s proposition to assimilate the ship hull by an ‘equivalent’ tri-axial ellipsoid, as well as to the employment of the Lagally theorem and ellipsoidal harmonics expansion techniques.

IWSH 2017-347

A Numerical Study on Wake Structure and Hydrodynamic Characteristics of Fishlike Undulation
Zeyu GUO1, Zuogang CHEN1,2*
1State Key Laboratory of Ocean Engineering, Shanghai Jiao Tong University, Shanghai 200240, China
2Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration (CISSE), Shanghai 200240
*Corresponding author,zgchen@sjtu.edu.cn

To investigate the hydrodynamics of the undulatory swimming fish, a key issue of the numerical analysis is to figure out the correlation between the undulatory locomotion and flow characteristics. In the paper, a novel dynamic-grids generation method is implemented to deal with the moving and deforming boundaries in unsteady flow field. This method is based on the structured grids in order to
simulate the wake and the boundary layer precisely. The analysis of hydrodynamic coefficients reveals that the undulation results in a significantly increase of frictional force in laminar flow ($Re\leq10^4$). However, the undulation has an adverse effect on friction when fish is swimming in turbulence flow ($Re\geq10^6$). The vorticity distributions and $Q$-criterion are both used to accurately capture the shedding vortexes in the wake. Furthermore, these vortex pairs have a substantial impact on the turbulence state and the wake, in which the turbulent kinetic energy and turbulent viscosity ratio both decrease at $Re\geq10^6$. The wake of an undulatory swimmer presents different vortex patterns with various kinematic parameters. A wide range of marine engineering application, such as autonomous underwater vehicles (AUVs), could benefit from these combined findings of undulatory locomotion.

IWSH 2017-348

Analysis of Dissipation on Gap Resonance between Side-by-side Barges
X.Y. Feng1*, X.B. Chen1,2, H. Liang1, F. Dias3,4
1Deepwater Technology Research Centre, Bureau Veritas Singapore, 117674, Singapore
2Professorship at Harbin Engineering University, China
3School of Mathematics and Statistics, University College Dublin, Belfield, Dublin 4, Ireland
4Center of Mathematics and Their Applications (CMLA), ENS Cachan, CNRS, Universite Paris-Saclay, 94235 Cachan, France
*Corresponding author, xingya.feng@sg.bureauveritas.com

Dissipation effect on two side-by-side barges at gap resonance is firstly explored by a two-phase CFD model. Flow characteristics, including flow separation and vortex shedding, in the vicinity of the gap inlet is demonstrated for cases of both sharp and rounded bilges. It is shown that large amount of energy is dissipated through the flow separation and vortex shedding at the bilges. In order to consider the dissipation in the framework of potential flow model, a dissipative surface covering the inlet of the gap at the bilges is defined, across which there exists a pressure drop representing the dissipative effect. We obtain new boundary integral equations by applying Green’s second identity over the boundaries and the dissipative surface in the fluid. The associate dissipation coefficient is then calibrated by the CFD model and measurements. It is shown that the dissipative surface is effective to account for the viscous effect on gap resonance. A simple relationship can be constructed between the dissipation coefficient and incident wave steepness through CFD computation using varying wave steepness. The results show that it is accurate to reproduce RAOs by the potential flow model using the dissipation coefficient predicted from the constructed relationship.

IWSH 2017-349

Ventilated Cavitating Turbulent Flows in a Francis Turbine Simulated by Two Numerical Methods
Xianwu LUO1*, An YU1, Liang ZHANG2, Hong WANG3, Lin WANG4
1State Key Laboratory of Hydroscience & Engineering, Tsinghua University, Beijing, 100084, China
2Yalong River Hydropower Development Company, LTD., 288 Shuanglin Road, Chengdu, China
3Institute of Nuclear and New Energy, Tsinghua University, Beijing, 100084, China
4State Key Laboratory of Hydroscience & Engineering, Tsinghua University, Beijing, 100084, China
*Corresponding author, luoxw@tsinghua.edu.cn

Francis turbines operating at partial flow conditions usually have vortex ropes in the draft tube, which is responsible for large pressure fluctuations, runner blades breakdowns and power swing phenomena. This unsteady phenomenon is harmful to the safe operation of hydropower stations. For actual operation, air admission is used to alleviate the effect of strong vortex rope. This paper presents the ventilated cavitating turbulent flows in a model hydro turbine simulated by two numerical
methods, i.e. the conventional model and a newly proposed method for ventilated cavitating flows. Both methods are based on a homogeneous assumption, where the mixture includes the vapor and the water, and its physical property is determined by the local vapor volume fraction. Compared to the conventional method, the newly proposed model applies a surface tension source term to reflect the effect of interfaces between different phases in the momentum equation, and the FBDCM turbulence model and a modified Zwart cavitation model. The comparison among the experimental data and the simulation results indicate that the proposed method has great advantage in the simulation of multiphase turbulence flow, and can accurately reflect the unsteadiness of the cavitating turbulent flow and capture the clearer phase interfaces between the water and vapor or air. Further investigation also depicts that the proposed model is suitable for the vortex breakdown in the draft tube of a Francis turbine with particular emphasis on vortex control by using air injection from the main shaft center. The results show that aeration with suitable air flow rate can alleviate the pressure fluctuations in the draft tube. The preferable vortex rope distribution can suppress the swirl at the smaller flow rates, and is helpful to alleviate the pressure fluctuation in the draft tube.

**IWSH 2017-350**

**CFD Predictions of Resistance and Viscous Flow Field around Ships for the JAPAN Bulk Carrier (JBC) without an Energy Saving Device**

Tiecheng Wu, Chunyu Guo*, Wanzhen Luo, Jie Gong, wenxuan She

Harbin Engineering University, College of Shipbuilding Engineering, Nantong street 145#, Harbin, China

*Corresponding author, guochunyu_heu@outlook.com or guochunyu@hrbeu.edu.cn

In this study, the resistance and viscous flow field around ships are predicted for the new test case JAPAN Bulk Carrier (JBC) for the Japan 2015 Workshop on CFD in Ship Hydrodynamics using CFD method. A numerical method based on the solution of the unsteady Reynolds-averaged Navier–Stokes equation is used to solve the fluid motion and viscous flow around ships. The Volume of Fluid (VOF) method with a HRIC (High Resolution Interface Capturing) scheme is used to capture the air/water interface of the free surface. The ship motion with sinkage and trim is evaluated using dynamic overset grid technique. The results obtained under the w/oESD conditions are validated against model test data, and good agreement is observed between the computational and experimental fluid dynamics results. The hydrodynamic performance (resistance, wave pattern, nominal wake, and axial velocity contours,) of JAPAN Bulk Carrier (JBC) is been study and analysis.

**IWSH 2017-352**

**The Numerical Simulation Based Fluid-structure Interaction Analysis of a Deep Sea Mining Vehicle**

Changsheng Fang, Hao Sun, Ligang Yao*, Zhiwei Liao

School of Mechanical Engineering and Automation, Fuzhou University, No. 2, Xueyuan Road, Fuzhou, Fujian, 350116, China

*Corresponding author, ylgao@fzu.edu.cn

As a crucial component in marine mineral resources exploitation, the cobalt crust-mining vehicle is capable of exploiting resources in the scenarios of high pressure, oxygen deficit, temperature variation and complicated ocean current. Particularly, the vehicle has presented great commercial and industrial potential in deep-sea exploration. To quantify the effects caused by different working situations, this work modeled and simulated a mining vehicle in deep sea. In the software COMSOL Multiphysics 5.0, the performance of the vehicle was evaluated based on the Finite Element Method (FEM). The complicated 3D geometrical model of the framed-shaped vehicle was set up followed by transient fluid-structure interaction (FSI) simulation. The arbitrary Euler-Lagrange technique was
leveraged to describe the 3-D fluid domain. Furthermore, since the dipping conditions and the static pressure were fundamental cases during the work of a deep-sea cobalt crust mining vehicle, the parameters of the fluid domain and structure were investigated. Also, the influence rule of different dipping speed, main frame of the elastic modulus, current velocity and the diameter of the pipe of these two conditions were studied. Then, based on the fluid-solid coupling law of the deep-sea cobalt crust-mining vehicle, three mining vehicle owner frame outlines were compared and optimized. In parallel, the contour of different mining vehicles, and the hydrodynamic performance of the mining vehicle was profiled and analyzed according to the frictional resistance and viscous pressure resistance. The presented model and simulation can be employed in many deep-sea oriented engineering applications and hold a potential for optimal underwater vehicle design.

**IWSH 2017-353**

**Numerical Study on the Bifurcation Mechanism of Air Layer on a Flat Plate**

Hyun-Jun Kim, Dong-Young Kim, Hyun-Woo Shin, Sang-Hyun Kim, Kwang-Jun Paik*

Inha University, 100 Inha-ro, Nam-gu, Incheon 22212, South Korea

*Corresponding author, kwangjun.paik@inha.ac.kr

There is no need to emphasize the importance of the frictional resistance of ships to both naval architecture and ship owners because it is known that frictional resistance takes about 70 percent of the total resistance of most merchant ships. Therefore, the development of air lubrication system for ships has been pursued to save fuel costs by reducing the frictional resistance. The air layer is diffused and bifurcated due to the interaction between water and air layer. In this study, the diffusion angle and pattern of air layer generated on the surface of a flat plate were investigated numerically using CFD. Before the simulations, grid resolution test was performed to evaluate the sensitivity of the grid. Air diffusion angle and pattern according to the variation of the water inflow velocity and air injection speed were compared with experimental data and the diffusion patterns of air layer were investigated through the velocity contour and vector as well as vorticity contour to explain the bifurcation mechanism of air layer.

**IWSH 2017-357**

**Numerical Simulation of Flow and Hydrodynamic Noise of NACA0012 Foil Using Large Eddy Simulation**

Chengyu Sun¹, Dan Yang¹*, Yongliang Xiong²

¹School of Naval Architecture and Ocean Engineering, Huazhong University of Science and Technology, Wuhan, China

²School of Civil Engineering and Mechanics, Huazhong University of Science and Technology, Wuhan, China

*Corresponding author, dan_yang@hust.edu.cn

With the mechanical noise and propeller noise of the ship has been effectively controlled, the study of hydrodynamic noise generation and control has attracted attention. The purpose of this paper is to numerically study the characteristic of hydrodynamic noise from NACA0012 foil in the uniform flow. The numerical simulation of flow field and hydrodynamic noise is based on the Large Eddy Simulation (LES) methods and Ffowcs Williams-Hawkins (FW-H) equations respectively. The influence of different inflow velocity (1 m/s to 5 m/s) and attack angles (α=0° to 10°) are analysed systematically. The simulation results show that the peak frequency of the noise spectrum is agreed with that of the vortex shedding from the trailing edge. With the increase of the inflow velocity, the frequency of the trailing edge vortex shedding gradually increases, and the Strouhal number also increases. Under the same inflow velocity, with the angle of attack increases, the frequency of the vortex shedding decreases.
A Fully Coupled Simulation Method for Floating Offshore Wind Turbine Dynamics Using a Boundary Element Method in Time Domain

Stefan Netzband*, Christian W. Schulz, Moustafa Abdel-Maksoud
Hamburg University of Technology, Am Schwarzenberg-Campus 4, 21073 Hamburg, Germany
*Corresponding author, stefan.netzband@tu-harburg.de

Predicting the dynamic behaviour of a floating offshore wind turbine (FOWT) is extremely demanding for common simulation methods because the floating structure motion is strongly influenced by complex aeroand hydrodynamics, inertia, and mooring loads. In order to provide a framework that can capture these effects in time domain, the in-house first-order panel method panMARE is further developed. Both the aerodynamics of the rotor and the hydrodynamics of the platform are simulated with panMARE and an integrated sixdegrees-of-freedom solver is applied to predict the motions over time.

The hydrodynamic loads on the platform are simulated using the current floating condition including the instantaneous wetted surface while considering added mass effects and Froude-Krylov forces. Drag effects are considered using body-dependent coefficients.

Regarding the aerodynamics, the potential theory based method is capable of modelling the threedimensional flow field of a wind turbine and therefore can consider the dynamics of rotor motions and the blade-wake interaction. Especially in the case of a pitching platform the effect of blade-wake interaction can have a significant impact on the aerodynamic loads. This effect can be captured by panMARE. Aerodynamic drag forces due to friction are incorporated using a Reynolds-number-dependent friction model.

In this paper, the motion of the DeepCWind semisubmersible platform is compared with the results of the published code-to-code comparison OC4 Phase II and therefore evidence is provided for the validity of our method. The comparison is carried out for several load cases. The results show that panMARE is capable of providing accurate simulations for decay test cases as well as capturing the influence of wind-wave and windcurrent misalignments in time domain while modelling the wind turbine wake in a realistic manner. In addition, an analysis of the FOWT in yawed conditions is included in order to investigate the self-aligning capabilities and effects due to yawing in detail.

Experimental and Theoretical Investigation of Liquid Sloshing in a Rotating, Laterally Oscillating Cylindrical Container

Y. Saito, T. Sawada*
Department of Mechanical Engineering, Keio University 3-14-1 Hiyoshi, Kohoku-ku, Yokohama 223-8522, Japan
*Corresponding author: sawada@mech.keio.ac.jp

We examined wave phenomena pertinent to water in a rotating, laterally oscillating cylindrical container. In particular, we measured the time-dependent dynamic water pressure and pressure change by fast Fourier transform analysis. The swirling of water in the container had three frequency components; the frequency responses of each frequency component are reported herein. When swirling occurs in a rotating cylindrical container, it was found that the wave rotating in the same direction as the rotation of the cylindrical container and the wave rotating in the opposite direction to the cylindrical container exist at the same time. The swirling direction was determined by the relationship of these magnitude.
Experimental Study on the Performance of an Overtopping Wave Energy Converter
Jing-Fa Tsai*, Chi-Pen Lu
National Taiwan University, Department of Engineering Science and Ocean Engineering No.1, Sec.4, Roosevelt Road, Taipei 10617, Taiwan
*Corresponding author, jftsai@ntu.edu.tw

A 1/25 model of overtopping wave energy converter, which was designed by the National Chung-Shan Institute of Science & Technology, was made to conduct the performance test in the towing tank of National Taiwan University. The test result show that the non-dimensional overtopping flow rate is an exponential function of the non-dimensional parameter freeboard over significant wave height $Rc/Hs$. However, the motion of the overtopping wave energy converter has significant effect on the overtopping flow rate. The non-dimensional mooring force of the overtopping wave energy converter is also an exponential function of the non-dimensional parameter freeboard over significant wave height $Rc/Hs$.

Design and Experiment of the Improved Parallel Correction Guidance Autopilot System
Sin-Der Lee*
National Taiwan Ocean University, Department of Transportation Science 2 Pei-Ning Road, Keelung 20224, Taiwan
*Corresponding author, leesd@mail.ntou.edu.tw

This research work is on the development of an improved parallel correction (IPC) guidance method for the marine vessel path following maneuver. A serial of numerical simulations are used to perform a systematic investigation into the effectiveness of the proposed improved parallel correction guidance autopilot design concept. In my previously research proposed the parallel correction (PC) guidance methods. During experimental testing that we found some interesting results, which encourage me to make some modification from original parallel correction (PC) version into improved parallel correction (IPC) guidance method. This study evaluated the performance between parallel correction (PC) guidance performance and improved parallel correction guidance methods. The proposed new guidance law not only uses the crosstrack errors as feedback information but also adopted temporality guiding waypoint to correct the vessel heading angle and is used to steer the vessel back to the desired path. This will improve the ability of the vessel to quickly maneuver back to its desire trajectories and it also can enhance the energy saving efficiency. The feasibility of the proposed guidance autopilot system was verified by performing a series of path following maneuvers under strong environmental disturbance condition. Specifically, the performance of different guidance methods will be compared in terms of cross track error, rudder activity, heading deviation and the rate of convergence of the vessel toward the desired path during the path following maneuver and the results might provide a realistic guide to the characteristics of autopilot system associated with different guidance methods.

Numerical Study of Cavitating Flow around a Hydrofoil NACA0015 with Super Hydrophobic Surfaces
Xianquan Lv1, Dan Yang1*, Yongliang Xiong2
1Huazhong University of Science and Technology, School of Naval Architecture and Ocean Engi-
neering 1037 Luoyu Road, Wuhan 430074, China
2Huazhong University of Science and Technology, School of Civil Engineering and Mechanics 1037 Luoyu Road, Wuhan 430074, China
*Corresponding author, dan_yang@hust.edu.cn

The present study focuses on the effect of hydrophobic surfaces on the cavitating flow. A hydrofoil NACA0015 with attack angle 8 degree was chosen to carry out the numerical simulation here. The main influence of hydrophobic surfaces is the boundary slip, which was implemented with Navier linear slip BC in the study. Considering the flow of cavitation has multi phases, a boundary condition with shear stress was adopted here. By using the effective multiphase flow model and the powerful user defined functions (UDF), the commercial CFD software ANSYS-FLUENT was adopted to do the simulation. The multiphase model used is the mixture method and the mass transfer model used is the Schnerr-Sauer model. The Reynolds number here is about $5 \times 10^4$, so the turbulence model RNG $k-\varepsilon$ and the enhanced wall functions was implemented. Flow with cavitation number 0.5, 0.8, 1.2 and boundary slip length ranges from 0 to infinity were calculated respectively. The force coefficient, Strouhal number, slip velocity and the cavity structure were investigated. The results show that super-hydrophobic surfaces have stronger influence on the cavitating flow with higher cavitation numbers.

IWSH 2017-374

Neumann-Kelvin Theory versus Neumann-Michell Theory for the Ship Wave Problem by a Vertical Surface-piecing Cylinder

Hui Liang¹, Xiaobo Chen¹,²*
¹Deepwater Technology Research Centre (DTRC), Bureau Veritas Singapore, 117674, Singapore
²College of Shipbuilding Engineering, Harbin Engineering University, Harbin, China
*Corresponding author, xiao-bo.chen@bureauveritas.com

The present paper is devoted to study the wave-resistance problem of a vertical surface-piecing cylinder. The free-surface Green function is used to establish the boundary integral equation. The cylinder surface is meshfree over which the physical quantities are analytically expanded into the Fourier-Laguerre series. The Green function is thus integrated over the entire cylindrical surface and a closed waterline instead of discretized panels and waterline segments so that the singular and highly oscillatory features of the Green function are analytically integrated. To construct the boundary integral equation, both Neumann–Kelvin (NK) theory and Neumann–Michell (NM) theory are used. By calculating the condition numbers of coefficient matrices, it is demonstrated that the NK theory produces singular and ill-conditioned coefficient matrices while NM theory is better-behaved.

IWSH 2017-375

Vertical Bending Moments of a Container Ship in Modulational Wave Trains

H. Houtani¹,²*, T. Waseda², K. Tanizawa³
¹National Maritime Research Institute, Fluids Engineering and Ship Performance Evaluation Department 6-38-1, Shinkawa, Mitaka, Tokyo, 181-0004, Japan
²The University of Tokyo, Graduate School of Frontier Sciences 5-1-5, Kashiwanoha, Kashiwa, Chiba, 277-8563, Japan
³National Maritime Research Institute, Fluids Engineering and Ship Performance Evaluation Department 6-38-1, Shinkawa, Mitaka, Tokyo, 181-0004, Japan
*Corresponding author, houtani@nmri.go.jp

To clarify the structural responses of a container ship to freak waves, we have conducted a towing tank experiment using an elastic model ship (Houtani et al. at Violent Flows 2016) and mod-
ulational wave trains as a model of freak waves. The length between perpendiculars of the model ship and the carrier wavelength of the modulational wave trains were 4.0 m. Our experimental results revealed that the maximum sagging moments of a ship largely vary as a function of the timing of the encounter with a freak wave because the wave geometry largely varies within a few wave periods. Accordingly, a positive correlation between the rear wave height (defined with the crest and the trough behind the crest) and the maximum sagging moments were observed. However, a different response of the maximum sagging moments to the rear wave height was also observed in some cases.

To clarify the cause of this different response, additional analysis was conducted by decomposing the vertical bending moments into wave-frequency components and whipping components. Our analysis revealed that the wave-frequency components of the vertical bending moments depend not only on the rear wave height but also on the phase of the ship's motions relative to the wave. The phase of the ship's motions was evaluated by applying the Hilbert transform to the measured time series of the ship's motions. Moreover, the relative phase was strongly correlated with the front wave height. We also showed that the whipping components of the vertical bending moments depended on both the rear wave height and also the dynamics of the wave at the forward perpendicular (FP) of the ship.

**IWSH 2017-376**

**CFD Application for Sloshing Flow in a Rectangular Tank under Forced Excitation**

Jeoungkyu Lee, Jieung Kim, Yonghwan Kim*

Seoul National University, Department of Naval Architecture and Ocean Engineering, 1 Gwanak-ro, Gwanak-gu, Seoul, Korea

*Corresponding author, yhwankim@snu.ac.kr

In this paper, two-phase sloshing flows inside two-dimensional rectangular tank are simulated for predicting the slosh-induced impact phenomena. A 2-D rectangular tank filled with water and air is modelled, which are under motion excitation which generates single impact on the tank top corner with and without air pocket. The computations are carried out using a commercial software, STAR-CCM+. Adaptive mesh model is applied for the simulation model, which is divided into multiple zones with different grid sizes. The volume of fluid (VOF) approach is used for simulating the flows and segregated implicit method is selected to solve the transport equations. The physical test conditions are divided into compressible and incompressible. In case of compressible air condition, ideal gas and segregated fluid isothermal model are used. The value of impact pressure is measured at the top right corner of the tank. In each condition, grid resolution and time segment are divided into seven cases maintaining the CFL condition. The results of CFD calculation are compared with the existing benchmark test.

**IWSH 2017-377**

**Accuracy Improvement of SVR in Identification of Hydrodynamic Coefficients with Denoising**

Yan Jiang¹, Zao-Jian Zou¹,²*, Xue-gang Wang¹,³

¹School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China

²Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai Jiao Tong University, Shanghai 200240, China

³CCCC Fourth Harbor Engineering Institute Co., Ltd, Guangzhou 510230, China

*Corresponding author, zjzou@sjtu.edu.cn

The success in predicting ship manoeuvrability with system based (SB) approaches requires
accurate hydrodynamic coefficients. Support vector regression has been used to identify the hydrodynamic coefficients in both Abkowitz and MMG type mathematical models. However, the performance of SVR suffers from the measurement noise existing in model test data. In this paper, Wavelet based denoising technique and moving average filter are proposed aiming to improve SVR's estimation accuracy in identifying the hydrodynamic coefficients from noise polluted model test data as training sample. Validations are carried out based on the simulated data of a KVLCC2 tanker using MMG model. The performance of wavelet denoising technique with respect to enhancing LS-SVM predicting accuracy is compared with moving average filter.

IWSH 2017-378

Multi-objective Optimization of Principal Ship Dimensions for Improving Operational Efficiency in Waves

Yoo-Won Jung, Yonghwan Kim*

Seoul National University, Department of Naval Architecture and Ocean Engineering 1, Gwanak-ro, Gwanak-gu, Seoul, 151 - 744, Korea

*Corresponding author, yhwankim@snu.ac.kr

In actual seaways, a ship experiences a loss of speed caused by environmental loads due to waves and wind, hence not only calm-water resistance but also speed loss in waves should be considered in hull-form design in order to improve operational efficiency in waves. This study introduces a basic study on hydrodynamic optimization of ship dimensions to take into account such hydrodynamic performance in waves. A multi-objective optimization method was used in the optimization process to minimize ship's total resistance in a seaway and its speed loss by additional resistance. In this process, added resistance is predicted using numerical methods: slender-body theory and Maruo’s far-field formula, and an empirical formula is adopted for short wave range. In addition, the speed loss in waves is estimated by using powerspeed curves in the representative sea condition as well as calm sea condition. The optimization solutions on a Pareto front set are compared to basic ship in terms of hull form, and their performances in waves are evaluated.

IWSH 2017-379

A NURBS-based Numerical Calculation of Radiation Problem with Forward Speed

Tian-long Mei¹, 2, Zao-Jian Zou¹, 2 *, Jing-ping Wu³, 4

1School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China
2Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai Jiao Tong University, Shanghai 200240, China
3Departments of Naval Architecture, Ocean and Structural Engineering, School of Transportation, Wuhan University of Technology, P. R. China, Wuhan 430063, China
4Key Laboratory of High Performance Ship Technology of Ministry of Education, Wuhan University of Technology, P. R. China., Wuhan 430063, China

*Corresponding author, zjzou@sjtu.edu.cn

When potential theory is applied in solving oscillating motion of a floating body, the key is how to obtain add mass and damping coefficient. In this paper, a desingularized Rankine panel method based on NonUniform Rational B-Splines (NURBS) is developed for 3D radiation problem with nonlinear wave-making effect. NURBS is used to precisely describe the body surface; the boundary integral equations are numerically solved by distributing sources a small distance inner body surface and above the free surface. The nonlinear wave-making boundary value problem (BVP) is solved by applying linear-superposition of first-order and second-order wave-making BVPs, then the radiation
BVP with forward speed are calculated based on the basic flow of nonlinear wave-making (NWM). In order to validate the efficiency of the present method, firstly, non-linear wave-making problems of a submerged moving sphere a and a Wigley hull are calculated; secondly, the radiation (with $m$ terms) problems of a submerged moving sphere and another modified Wigley I hull are solved by the non-linear wave-making (NWM) linearization method, the wave patterns of wave-making and radiation are also presented. The numerical results of hydrodynamic coefficients show good agreement with other results in literatures, which verify the reliability of the numerical calculation method in this paper.

**IWSH 2017-380**

**Computational Simulation of a Submarine Propeller Using Overlapping Grid Method**

Peng Wei*

School of Naval Architecture and Ocean Engineering, Huazhong University of Science and Technology, Wuhan, Hubei, P.R. China

*Corresponding author, weipeng2016@hust.edu.cn

Simulations of the submarine propeller E1619 are presented using the overlapping-grid flow solver HUST-ship. A numerical code based on the solution of the Reynolds averaged Navier-Stokes equations for the purpose of simulation on hydrodynamic performance of marine vehicles. Propeller open water curves were obtained for two grids for a wide range of advance coefficients, the viscous flow around the propeller was analyzed and results compared with available experimental data. A study was presented for one advance coefficient ($J=0.85$) on three time-step sizes and the effect of the turbulence model on the wake was analyzed comparing results with RANS, DES and DDES. The results show that refine grid could capture wake flow better than coarse grid and grid refinement shows more obvious effect on torque but not on thrust. Time step also shows weak influence on thrust and torque but time step should not be set too long. RANS, DES and DDES show weak effect in tip vortex but RANS approach shows more intense vorticity near the hub. The wake velocities are compared against experimental data for $J=0.74$, showing good agreement.

**IWSH 2017-381**

**The Research and Verification of Wave-generating and Wave-absorbing in 3-D Numerical Wave Tank**

Jingjing Lu¹, Peng Wei¹*, Zhiguo Zhang²

¹School of Naval Architecture and Ocean Engineering, Huazhong University of Science and Technology, Wuhan, Hubei, P.R. China

²School of Naval Architecture & Ocean Engineering, Huazhong University of Science and Technology, Wuhan, Hubei, 430074, P.R. of China

*Corresponding author, m201671573@hust.edu.cn

In recent years, wave-generating technology has been playing a more and more essential role in hydrodynamic fields for a good variety of advantages. There is no denying that it has made a rapid progress with the development of computer technology. This paper primarily aimed at building a numerical wave tank to simulate the generation and maintenance of waves. Based on level-set method to track free surface, by defining inlet boundary conditions, the three-dimensional numerical wave tank is developed, and the RANS equation is dispersed by finite difference method, in order to generate waves as specified and suppress reflected wave effectively, different methods of wave elimination are adopted near to the exit of numerical tank and the advantages and disadvantages of the two modes are discussed. Numerical tests show that wave-generating and propagating in the present 3-D numerical wave tank are of good stability and reliability, which lays a certain foundation for the study of numerical prediction of ship’s seakeeping.
IWSH 2017-382

Vortex Structure Behind the Cylinder Mounted on Flat Plate
Jiajun Chen*
School of Naval Architecture and Ocean Engineering, Huazhong University of Science and Technology, Hubei, P.R. China
*Corresponding author, chenjiajun@hust.edu.cn

The flow behind the circular cylinder fixed on the plate in a cross flow has been numerically investigated. In this paper, mainly study the vortex structure behind the finite length cylinder mounted on the plate, which the Reynolds number equal to $6 \times 10^4$, and then, for the better research, the effect of Reynolds number on vortex also has been done to find the difference. And the cylinder mounted on the plate with the aspect ratio (Height:Diameter) equal to 7, this geometry has been selected from previous work. 3D Detached Eddy Simulation (DES) has been adopted to investigate the flow field for the different Reynolds number. The mesh and time step independence have been made to guarantee the reliable results. The results of the simulation clearly show the details of the flow structures. Including the wall near horseshoe vortices and the flow behavior of the flow over the top of the cylinder. The features of the vortex structure and the interaction between the vortexes have been investigated in instantaneous flow in this paper.

IWSH 2017-383

Study on Steady Flow Effects in Numerical Computation of Added Resistance of Ship in Waves
Jae-Hoon Lee, Beom-Soo Kim, Yonghwan Kim*
Seoul National University, Department of Naval Architecture and Ocean Engineering 1, Gwanak-ro, Gwanak-gu, Seoul, Republic of Korea
*Corresponding author, yhwankim@snu.ac.kr

In this study, steady-flow effects are investigated in the numerical computation of added resistance on ship in waves. For a ship advancing forward, a time-domain 3D Rankine panel method is applied to solve the ship motion problem, and added resistance due to waves is calculated by a near-field method, the direct integration of the second-order pressure on hull surface. In the linear potential theory, steady flow is approximated by the basis potential of uniform flow or double-body flow in order to linearize the boundary conditions. By applying the two different linearization schemes, coupling effects between steady and unsteady solutions are examined. Also, in order to analyze the steady-flow effects on hull geometry, computation results for two realistic hull forms, KVLCC2 tanker and DTC containership, are compared. Particularly, the $m_i$ term which represents the coupling effects in body boundary condition is evaluated considering the geometry of non-wall-sided ship. Lastly, the characteristics of linearization schemes are discussed for the disturbed waves around ship and the components of added resistance.

IWSH 2017-386

Numerical Analysis the Dynamic Performance of Composite Propeller
Fanchen Zhang¹, Dakui Feng¹*, Peng Wei²
¹School of Naval Architecture and Ocean Engineering Huazhong University of Science and Technology Wuhan, Hubei, 430074, P.R of China
²School of Naval Architecture and Ocean Engineering Huazhong University of Science and Technology Wuhan, Hubei, 430074, P.R of China
*Corresponding author, feng_dk@hust.edu.cn
The marine propeller is regarded as critical component with regard to the performance of the ships and torpedoes. In general screw propellers are used for these purposes. Traditionally marine propellers are made of manganese-nickel-aluminum-bronze (MAB) or nickel-aluminum-bronze (NAB) for superior corrosion resistance, high-yield strength, reliability, and affordability. More over metallic propellers are subjected to corrosion, cavitation damage; fatigue induced cracking and has relatively poor acoustic damping properties that can lead to noise due to structural vibration. Moreover, composites can offer the potential benefits of reduced corrosion and cavitation damage, improved fatigue performance, lower noise, improved material damping properties, and reduced lifetime maintenance cost. In this work, the INSEAN 1619 large screw 7 blade propeller is analyzed using the fluid and structural analysis code. The fluid analysis is carried out using the general purpose CFD software Fluent and structural analysis is carried out using ANSYS Workbench. This paper will describe the composite propeller performances (propeller characteristics) under the elastic deformation. The same model propellers with the different Young’s modulas are studied numerically. In order to confirm the propeller characteristics these propellers, the open water model tests results were used to validate the numerical method used in this paper.

IWSH 2017-387

Transient Diffraction Waves by Fourier-Laguerre Expansions
R.P. Li1,*, H. Liang2, X.B. Chen1,2, W.Y. Duan1
1Harbin Engineering University, Harbin, China
2Deepwater Technology Research Centre, Bureau Veritas, Singapore
*Corresponding author, liruipeng@hrbeu.edu.cn

We consider transient wave diffraction around an infinite vertical cylinder in deepwater. Within the framework of boundary element method (BEM), velocity potential and its normal derivative on the cylinder are expressed by Fourier series in polar direction and Laguerre function in vertical direction. By applying the Green’s theorem in the domain external to the cylinder, we obtain so-called Dirichlet-to-Neumann (DtN) operator representing the relationship between the velocity potential and its radial derivative on the cylinder. There is no need to compute the transient Green function itself but its integration involving the Fourier-Laguerre base function on the cylindrical surface is evaluated. DtN is applied to obtain the transient waves diffracted from the incoming waves. For the selection of the incoming waves, not only the steady-state plane progressive waves without wave front, but also the transient waves with wave front generated by a wave-maker are considered. The latter case is more general but few analysis in previous studies is made due to its complexity in numerical computations. The analytical diffraction potential includes both instantaneous and memorial components and induced loads are evaluated. By comparing with analytical solutions with BEM solutions including diffraction wave loads on the cylinder, the DtN relation is then validated.

IWSH 2017-391

Numerical Investigation on Vortex-induced Motions of a Buoyancy Can
Kangdi Xie, Weiwen Zhao, Decheng Wan*
Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China
*Corresponding author, dcwan@sjtu.edu.cn

This paper presents three-dimensional (3D) Computational Fluid Dynamic (CFD) simulation to analyse the flow induced response (FIR) especially the yaw motion of a Buoyancy Can. The numerical cases are conducted with a Buoyancy Can under different reduced velocities utilizing naoe-FOAM-SJTU, a solver based on the open source toolkit OpenFOAM. SST-DDES model is applied to
handle the flow separation, and oversets grid method is utilized to solve a large amplitude 6 Degrees-Of-Freedom (6-DOF) motions. Free decay test and vortex-induced motion (VIM) test are built numerically. In VIM cases, the responses of trajectory, amplitude, frequency are calculated in a series of reduced velocities. With the increase of reduced velocity, yaw frequency is increased, which is similar with surge and sway frequency. And yaw frequency is equal to the sway frequency, which is consistent with Kang's experimental results \cite{1, 2}. Furthermore, comparing two cases, one fixed in rotation and the other one free in rotation, we can obtain a conclusion that release in the degree of rotation can decrease the sway amplitude but make no difference in the surge amplitude.

**IWSH 2017-392**

**Numerical Validation of Aerodynamics for Two in-line Model Wind Turbines Using Actuator Line Model and CFD Technique**

Yong Ai, Ping Cheng, Decheng Wan*

Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China

*Corresponding author, dcwan@sjtu.edu.cn

In this present study, a numerical validation of aerodynamics for two in-line model wind turbines using actuator line model and CFD technique. The actuator line model is used to simulate the blades of two in-line model wind turbines while the hub, nacelle and tower of both two wind turbines are not included in present study. The SST k-ω turbulence model is applied to solve the RANS equation due to the closure problem. The uniform free-stream flow condition at a speed of 10 m/s at the reference height of hub is applied to the inlet. Upstream and downstream wind turbines are running at tip speed ratio 6 and 4 respectively. The result from the present simulation is compared to the experiment data. From the comparison, the results from the present study show a good agreement with the experimental results especially for the aerodynamic loads prediction taking the aerodynamic power and thrust into account yielding a maximum error of 3% for the upstream wind turbine and maximum error of 10% for the downstream wind turbine. Another conclusion can be easily drawn that although difference in wake prediction exists in the simulation for two in-line wind turbines model comparing to the blind test2, the actuator line model still can yield the distribution characteristics of the mean wake velocity and mean turbulent stress. Such as the number and position of peaks, the wake width is also can be captured with acceptable accuracy.

**IWSH 2017-393**

**CFD-based Hull Form Multi-objective Optimization for Better Resistance and Wake Performances**

Xinwang Liu, Aiqin Miao, Decheng Wan*

State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean & Civil Engineering, Shanghai Jiao Tong University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai 200240, China

*Corresponding author, dcwan@sjtu.edu.cn

In this paper, a ship model is used as the parent ship, and the optimization of the whole ship at a specific speed (Fr=0.26) is carried out aiming at the resistance performance and the wake performance. A RANS-based CFD solver naoe-FOAM-SJTU and a practical hydrodynamic optimization tool OPTShipSJTU are applied for the hull form optimization. Here, the free-form deformation method and shifting method are used as parametric hull surface modification techniques in order to generate a series of hull forms subjected to geometric constraints, and a multi-objective genetic algorithm, which is NSGA-II, is adopted to obtain the Pareto-optimal front. Moreover, to reduce the computa-
tional cost, the Kriging model which is constructed based on several sample designs is introduced. Finally, the optimal hull form can be obtained to be the reference of the hull form design, and the numerical results further prove the availability and reliability of the optimization tool, OPTShip-SJTU.

IWSH 2017-396

Numerical Calculation and Analysis of Hydrodynamic Performance of a Cylindrical Oscillating Float Wave Energy Absorption Device

Di Wang, Ke Xia, Decheng Wan*

Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China

*Corresponding author, dcwan@sjtu.edu.cn

The point-absorber wave energy device has attracted researchers’ attention for its efficiency and stability. In order to study the response of point-absorber wave energy generation device in wave, the motion of cylindrical floating body in wave is simulated in this paper. The CFD solver naoe-FOAM-SJTU based on open source platform OpenFOAM is used to simulate the motion performance of cylindrical power generation device under the regular wave. The wave response of the cylindrical floating body is analyzed and compared with the test. The motion of the single floating body and two floating bodies in the wave is simulated respectively. The interaction of the two floating bodies, the movement characteristics of the body under different conditions are analyzed. The damping coefficient is introduced to reflect the influence of generator on the motion of cylindrical floating body in wave. Through the calculation of theoretical wave energy conversion efficiency, the factors affecting the conversion efficiency of wave energy are discussed. Key words: wave power converter; naoe-FOAM-SJTU solver; wave load; energy conversion efficiency

IWSH 2017-399

Dynamic Pressure Distribution and Wave Forces on Offshore Spar-type Wind Turbines in Diffracted Wave Field

S. Kimura, Y. Nihei, S. Srinivasamurthy

Osaka Prefecture University, Department of Aerospace and Marine System Engineering 1-1 Gakuen-cho, Nakaku, Sakai, Osaka 599-8531, Japan

*Corresponding author, swb03045@edu.osakafu-u.ac.jp

Various floating platform concepts and designs for FOWT have been proposed over the years to utilize the ocean environment. With innovative and complex designs, evaluating hydrodynamic forces becomes challenging and it is inevitable to develop suitable methodology for accurate assessment of loads. It is important to understand the behaviour of such complex floating structures experimentally to comprehensively identify the potential challenges. In this research, a methodology for estimating wave forces on an AdvancedSpar type platform consisting of small and large cylinders is proposed based on experimental findings. A scaled-model (1/100) tank test is performed on an Advanced Spar-type platform with pressure gauges to estimate wave forces and compared with Morison's equation. The effect of small and large cylinders is discussed from the viewpoint of wave force contribution and a hybrid calculation method is found to be effective to accurately assess wave loads. It is found that the total wave force measured from the experiment is less than the Morison’s equation calculations while the maximum error ratio reduces from 46% to 24% (almost by half) when we adopt the proposed hybrid calculation method. Further attempts are made to correlate the observations for D/L>0.2 based on incident waveforms.
Numerical and Experimental Study of Two-Stage Savonius Vertical Axis Marine Current Turbine on the Effect of Deflector

M.A. Ismail¹, O.B. Yaakob¹,²*, Yasser M. Ahmed³
¹Faculty of Mechanical Engineering Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
²Marine Technology Centre, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
³Faculty of Engineering, Alexandria University, Alexandria 21544, Egypt
*Corresponding author, omar@fkm.utm.my

Renewable energy sources including natural ocean and tidal current sources seem promising and have been widely explored. For vertical axis tidal current turbine application, Savonius-type rotor suits Malaysian sea conditions that are normally associated with low current speed and shallow water depth. However, the rotor suffers from a low efficiency. The present study seeks to optimize the rotor through the usage of deflector. Numerical approach was carried out using commercial Computational Fluid Dynamic (CFD) software while model testing was carried out at the Marine Technology Centre (MTC) facility. The results indicate an improvement of typical two-stage Savonius turbine performance by more than 20% increment by using the deflector as an added outer structure. The research work results have been thoroughly validated between experimental and numerical data. Some recommendations on future work have also been made for further knowledge enhancement of two-stage Savonius turbine performance.

Comparison of Different Added Power in Waves Prediction Methods

Chen weimin, Li Jianpeng, Dong Guoxiang, Xing Lei
State Key Laboratory of Navigation and Safety Technology Shanghai Ship and Shipping Research Institute 600 Minsheng Rd, Shanghai, China
Corresponding author, wmchen@sssri.com

In order to predict the speed loss in the actual sea states more precisely, delivered power shall be measured more accurately as an input. Therefore, based on a 50000DWT tanker, various results obtained from different prediction methods were compared by a series of model tests performed in calm water and in waves. It is shown that speed loss deprived from RTIM method in regular waves test could satisfy the engineering requirements most.

Study on Visualization of Ship Collision Risk in Complex Intersection Waters

Liu Yi¹,², Hu Youxi¹,², Zhu Jiao¹,², Liu Jingxian¹,²,³*
¹School of Navigation, Wuhan University of Technology, Wuhan 430063, China
²Hubei Inland Shipping Technology Key Laboratory, Wuhan University of Technology, Wuhan 430063, China
³National Engineering Research Center for Water Transport Safety, Wuhan University of Technology, Wuhan 430063, China
*Corresponding author, 825802999@qq.com or ljxteacher@sohu.com

The complex intersection waters are the waters with complex environment, diverse types of ships, traffic control capacity of a limited number of waterways or custom route intersection waters, the ship between the easy to produce mutual influence, mutual control, resulting in the ship collision accident increased probability. In this paper, the accident risk analysis and the method of system
engineering are used to identify the impact factors of ship collision risk in complex confluence waters, and the risk of ship collision in complex intersection waters is evaluated from the navigation conditions. By using the fuzzy comprehensive evaluation method of qualitative and quantitative combination, the risk of collision of each unit grid is evaluated by the grid of the evaluation target, and the risk distribution of the ship collision risk level is obtained. Combined with the risk characteristics of the intersection of the main channel of Yantai Port and the main channel of Yantai Port, the evaluation model is applied to the waters, the traffic flow data are collected. The ship collision risk assessment is carried out on 150 units of the West Port area, Visualization the risk distribution of ship collision risk in the intersection of West Harbor, and the reliability of the evaluation result through the comparison of accident data.

IWSH 2017-411

Solving Inverse Problems of Wave Equation by a Boundary Functional Method
Chein-Shan Liu1,2*, Yung-Wei Chen3
1Center for Numerical Simulation Software in Engineering and Sciences, Hohai University, College of Mechanics and Materials, Nanjing, Jiangsu 210098, China
2National Taiwan Ocean University, Department of Mechanical and Mechatronic Engineering, 2 Pei-Ning Road, Keelung 20224, Taiwan
3National Taiwan Ocean University, Department of Marine Engineering, 2 Pei-Ning Road, Keelung 20224, Taiwan
*Corresponding author, csliu58@163.com

In the paper, we solve the inverse problems of wave equation to recover unknown space-time dependent functions of wave speed and wave source, without needing of initial conditions and no internal measurements being required. After a homogenization technique, a sequence of spatial boundary functions are derived, which satisfy the homogeneous boundary conditions, and are at least the fourth-order polynomials. The boundary functions and zero element constitute a linear space, and then a new boundary functional is proved in the linear space, of which the energy is preserved for each dynamic energetic boundary function. The linear systems and iterative algorithms used to recover unknown wave speed and wave source functions with dynamic energetic boundary functions as bases are developed, which converge fast at each time step. The input data are parsimonious, merely the measured boundary strains and the boundary values and slopes of unknown functions to be recovered. The accuracy and robustness of present methods are confirmed by comparing the exact solutions with the estimated results under large noises up to 20%.

IWSH 2017-412

Mesh Properties for RANS Simulations of Aerofoil-Shape Rudder Hydrodynamics
Jialun Liu1*, Robert Hekkenberg2, Zhonglian Jiang1, Xiumin Chu1
1National Engineering Research Centre for Water Transport Safety, Wuhan University of Technology, Wuhan, Hubei, 430063, China
2Delft University of Technology, Mekelweg 2, 2628 CD, Delft, The Netherlands
*Corresponding author, jialunliu@whut.edu.cn

A good mesh is a prerequisite to achieving reliable results from Computational Fluid Dynamics (CFD) calculations. Relevant mesh properties include mesh types, computation domain shapes and sizes, element sizing, and cell growth rate. However, in literature, no clear consensus about what these properties should be was found. In this article, we strive to determine what the suitable mesh properties are for the analysis of aerfoil-shaped ship rudders as rudder profiles are commonly adapted from aerfoils, for instance, the NACA series. This paper presents a step-by-step study of
mesh properties and their impacts on the accuracy of solutions of rudder hydrodynamic coefficients. A classic NACA0012 profile is chosen as an example. Commercial packages ANSYS ICEM and Pointwise are applied for meshing. ANSYS Fluent is used as the numerical solver. Suitable mesh properties are summarised.

**IWSH 2017-413**

**Evolution of an ISW with Different Modes Propagating across Vertical Cylinder**
Ming-Hung Cheng¹*, Robert R. Hwang¹, Chih-Min Hsieh²
¹National Taiwan Ocean University, Department of Systems Engineering and Naval Architecture 2 Pei-Ning Road, Keelung 20224, Taiwan
²National Kaohsiung Marine University, Department of Maritime Information and Technology No.482, Jhongjhou 3rd Rd., Kaohsiung 80543, Taiwan

*Corresponding author, chengminghung@gmail.com

An internal solitary wave (ISW) has significant ramification not only in marine ecology but also on engineering works in the ocean. As an ISW propagates around large vertical cylinders, this interaction has been investigated in some previous studies. For small vertical cylinders, such as deep-sea oil drilling rigs or pipelines, the strong velocity difference induced by an ISW propagating causes the threat of the structures. Although the interaction between surface waves and small cylinder has been vigorously studied in several decades, the investigations on the interaction between an ISW and the obstacle is less to be found. Moreover, the waveform types of ISWs are found as mode-1 and mode-2 in stratified water and have different effect on the actsives based on the present field observations. In this paper, a finite volume method solving the Navier-Stokes equations using Improved Delayed Detached Eddy Simulation turbulence model is adopted to discuss the evolution as an ISW with mode-1 or mode-2 type propagates across a vertical cylinder. Herewith, the mode-2 ISW with large amplitude defined by Brandt and Shipley (2014) is employed in this paper. Numerical results reveal that significant differences of density and pseudo net pressure are investigated at both sides of the cylinder in different waveform type. Based on the comparisons between the results of mode-1 and mode-2 ISW cases, the magnitude of the vortices and pseudo net pressure difference in mode-1 case is larger than mode-2 ones. Hence, the mode-1 ISW has a more significant influence on wave-cylinder interaction than mode-2 ISW.

**IWSH 2017-415**

**Internal Solitary Wave Transformation over the Slope: Asymptotic Theory and Numerical Simulation**
Changhong Zhi, Ke Chen*, Yunxiang You
Shanghai Jiao Tong University, School of Systems Naval Architecture, Ocean & Civil Engineering 800 Dongchuan Road, Shanghai, China

*Corresponding author, raulphan@sjtu.edu.cn

The propagation and evolution of long nonlinear internal solitary waves over slope-shelf topography is theoretically and numerically studied in a two-layer fluid system of finite depth. The variable Korteweg–de Vries (vKdV) and variable extended Korteweg–de Vries (veKdV) equations are derived for the weak and moderate nonlinear waves, respectively. The numerical method is developed from finite difference/volume (FD/FV) scheme to solve the nonlinear equations. The transformation of solitary waves is observed when they propagate past the slope. The elevation of rear face of the front wave grows with the increase of the slope inclination. The results also show that the transformed waves can be described by the steady solution of the corresponding theoretical model (vKdV, veKdV) by considering the depth condition beyond the shelf.
The Effect of the Sloshing Tank on the Ship Rolling Motion in Beam Waves
Ming-Chung Fang1*, Jiun-Ting Lin1, Zi-Yi Lee2
1National Cheng Kung University, Department of Systems Engineering and Naval Mechatronic Engineering 1 University Road, Tainan 700, Taiwan
2National Kaohsiung Marine University, Department of Naval Architecture and Ocean Engineering No.142, Haijhu Rd., Nanzih Dist., Kaohsiung City 81157, Taiwan
*Corresponding author, fangmc@mail.ncku.edu.tw

Based on the well-developed ship motion simulation program, the coupled effect of the sloshing tank on board with of the ship motions in waves has been investigated. The strip method and potential theory are adopted to treat the ship motion in waves and the liquid sloshing behaviors are included by using the commercial software Fluent, which is based on the viscous flow theory and solved by finite fluid volume to track the free surface position. Incorporating the liquid sloshing force and moment on the tank wall with the ship motion in waves, we can establish the coupled motion prediction model of the sloshing tank and ship hull in waves.

From the present study, we find the sloshing tank with different liquid level will make different effect on the ship motions with respect to different wave periods, especially for rolling motion. The key factor for increasing or reducing the roll motion depends on the phase difference between the ship motion and liquid sloshing motion, which is very important to be paid attention while any of the ship liquid tanks is fully filled in waves.

Nominal Wake Fluctuations due to Waves with Different Amplitudes
Emel Tokgoz, Ping-Chen Wu*, Yasuyuki Toda
Osaka University, Department of Naval Architecture and Ocean Engineering 2-1 Yamadaoka, Suita, Osaka 565-0871, Japan
*Corresponding author, Ping-Chen_Wu@naoe.eng.osaka-u.ac.jp

This paper reports the wave amplitude dependency on ship advancing in regular head waves. The flow field around the fully-loaded KVLCC2 (KRISO very large crude oil carrier 2) at design Froude number Fr=0.142 and the vertical ship responses are predicted using CFD (Computational Fluid Dynamics). The selected wave lengths are λ/L=1.1, 1.6 and 2.0 and five different wave amplitudes are investigated for each wave length. The influence of the wave amplitude on the velocity distribution on the propeller plane is determined. The axial velocity distribution is Fourier analyzed to obtain the harmonic contents of it. Moreover, the propeller performance in waves is predicted using the body-force distribution model. As future work, the experiments will be carried out for validation by means of new force measurement system.

Dual BEM for Wave Diffraction of a Thin Structure with an Arbitrary cross Section
Ching-Yi Tu1*, Ching-Yun Yueh1, Chih-Ting Chang1, Shih-Hsuan Chuang2
1National Taiwan Ocean University, 2 Pei-Ning Road, Keelung 202, Taiwan
2Department of Hydraulic and Ocean Engineering R&D Foundation, National Cheng Kung University, Tainan 701, Taiwan
*Corresponding author, tje1108@gmail.com

In this paper, under the usual assumptions of linear wave theory and potential flow, a numerical solution has been obtained using the dual boundary element method (DBEM). The method is proposed to investigate wave diffraction of the thin structure with an arbitrary cross section such as
arc-shaped, V-shaped, open circular shell, open square shell, cosine-type, etc. We compared the non-dimensional free surface elevations of the open circular shell with a frontal breakwater obtained by Hariri Nokob and Yeung (2015) \cite{1}, and excellent agreement was observed. The numerical results show that the method adopted here is simple, convenient, flexible, and is a powerful and efficient numerical scheme to address various ocean engineering problems concerning impermeable or porous thin structures. Moreover, it also has been found that the porous structures can reduce the oscillation of free surface elevations. This may provide useful hydrodynamic information for the design of offshore structures and harbor engineering.

**IWSH 2017-420**

Green Water over a fixed Deck by Immersed Boundary Method

Bin Yan\textsuperscript{1} *, Wei Bai\textsuperscript{2}, Ser Tong Quek\textsuperscript{1}
\textsuperscript{1}Department of Civil and Environmental Engineering, National University of Singapore, Kent Ridge, Singapore 117576, Singapore
\textsuperscript{2}School of Computing, Mathematics and Digital Technology, Manchester Metropolitan University, Chester Street, Manchester M1 5GD, UK
*Corresponding author: binyan@u.nus.edu

Interaction between marine structures and free surface flows is a classical hydrodynamic problem and has a wide range of applications in many ocean and coastal engineering problems. Green water is one common occurrence of such problems. When the water wave impacts the structure, and runs up over the deck of the structure, green water occurs. Due to the strong nonlinearity caused by the decreasing water depth, the green water problem is very complicated and difficult. As the wave breaks and overtops on a marine structure, the flow becomes multi-phased and chaotic as a large aerated region is formed in the flow in the vicinity of the structure.

In the present paper, an immersed boundary method is applied to investigate the green water over a fixed deck. One important part of this paper is the use of experiment data of green water over a fixed deck to validate the present immersed boundary method. In addition to the investigation of green water loads, pressure on the deck induced by wave impact, the volume of green water affecting on the deck stability is another important part of the present paper, which it is difficult to estimate especially in experiments and observations. Furthermore, the comparison with experiment indicates immersed boundary method can simulate green water well and effectively.

**IWSH 2017-423**

Dynamic Truncation Analysis for Mooring Lines with Multi-Objective Optimization Method

Gang Ma, Yue Jiang, Hongwei Wang*, Liping Sun
Harbin Engineering University, 145 Nantong Street, Nangang District, Harbin, Heilongjiang Province 150001, China
*Corresponding author, wanghongwei@hrbeu.edu.cn

Truncation is needed for mooring model test in ocean engineering basin due to the limitation of its depth and width. The static truncation is usually used in model test, but it cannot simulate the inertial forces and drag forces. The nonlinear dynamic characteristics should be included within truncation design in time domain. The main problem is that dynamic similarity cannot be easy to be met over long time history, or by equivalent method. So a dynamic truncation method is presented to solve this problem. The multi-objective optimization method and slender rod model are adopted to fulfill the dynamic truncation. The multi-objective optimization is realized with Non-dominated Sorting Genetic Algorithm-II (NSGA-II). Here presents a verification method to verify the effective of the proposed method. The population size of NSGA-II is a main factor. The parameter population size is set to 100, 200 and 300 in order to ensure the stability of the calculation. Sinusoidal motion with dif-
The foundations of offshore wind turbine and met mast appear the scour phenomenon subjected to the interaction between structure, wave-current and sea bottom characteristics. This scour phenomenon could result in damages or dumping of offshore wind turbines and met mast. Firstly, hydraulic model test with the monopile foundation of offshore wind turbines and sea observation towers is carried out in this study to investigate the maximum scour depth and the potential scour area around the monopile foundation exposed to the attack of different ocean currents. Experimental result indicates that the scour depth initially develops in the up-current side of monopile. A semi-rounded scour hole gradually develops in the up-current side of monopile. Meanwhile, a cone-shaped scour hole also develops in the down-current side of monopile, and there have the phenomenon of sediments deposition at the far downstream from scour hole. To sum up all the experimental data and use hyperbolic model analysis, the maximum scour depth estimation cannot only verify the scouring of the experiment has reached equilibrium stage, but also provide the conservative scouring protection design information of monopile foundation. Furtherly through dimensionless analysis, the relationship between Reynolds number \(R_e = UD/\nu\) and relative scour water depth (the maximum scour depth over water depth, \(d_{s,max}/h\)) can be obtained as a regression curve. Then this regression curve can use as maritime engineering application to estimate the maximum scour depth around a monopile foundation under different conditions of current speed (\(U\)), water depth (\(h\)) and pile diameter (\(D\)).
temporal resolutions, there is a practical limit on the refinement due to available computer resources. At least a 2nd-order time-stepping scheme is needed for accurate wave propagation. However, there is a trade-off between the numerical stability and the high-order accuracy of the scheme. To improve the stability and the robustness of the code, we are experimenting with a modified Crank-Nicolson scheme where the scheme parameter is non-uniformly distributed in the domain. The results of this numerical experiment and comparison with standard Euler and three-step backward schemes are reported. The application case consists in KCS test case of Tokyo 2015 CFD Workshop. A proper grid refinement study is performed in order to assess the accuracy of the code. Some general recommendations for time-stepping schemes for seakeeping are provided.

**IWSH 2017-428**

**Analysis Aero Dynamic Performances of an Offshore with Effects of Wind Attacked Angle and Hull Shape by Using CFD**

N.C. Cong¹,², B.D. The¹, L.T. Thai¹, Ph.A. Tuan¹, N.V. He¹*

¹Hanoi University of Science and Technology, 1 Dai Co Viet, 10000, Hanoi, Vietnam
²Vietnam Maritime University, 484 Lach Tray, Bach Dang, Ngo Quyen, Hai Phong, Vietnam

*Corresponding author, he.ngovan@hust.edu.vn

In this paper, we propose a study on effects of hull shape and wind attacked angle on aero dynamic performances of above water surface hull of an offshore by using a commercial code Computation Fluid Dynamic (CFD). From the results of computation the aero dynamic performances of original model, the authors propose several new models with a different hull shape structure and the same main dimensions with those of the original one. All models are computed by the CFD to find the best one with the smallest effects of hull shape and wind attacked angle on aero dynamic performances of the offshore. The target of study is reducing effects of hull shape and wind attacked angle on aero dynamic performances of the offshore. From the results of comparison among models, the effects of hull shape and wind attacked angle on aero dynamics performances are shown. The conclusions of paper are that how effects of hull shape and wind attacked angle on aero dynamic performances of the offshore. From the research, the authors propose some comments on effects of hull shape and wind attacked angle on aero dynamic performances of the offshore. The results of this study may be useful for the research and design offshore engineer as well as oil rigs engineering.

**Keywords:** Offshore; offshore engineering; CFD; aero dynamics performance

**IWSH 2017-430**

**The Numerical Study on the Cavitation-structure Interaction of the Elastic Cylinder Shell Exiting Water**

Hu Shi-liang¹*, Lu Chuan-Jing², Chen Jing-pu³

¹China Ship Scientific Research Center, 185 Gao-Xiong Road, Shanghai, China
²Shanghai Jiao Tong University, 800 Dong-Chuan Road, Shanghai, China
³China Ship Scientific Research Center, 185 Gao-Xiong Road, Shanghai, China

*Corresponding author, hushiliang542@qq.com

The numerical method of fluid-structure interaction (FSI) is applied to simulate the process of the elastic plat-head cylinder shell exiting water vertically with initial attitude angle. The FSI solver is based on the partitioned strategy called closely coupled approach, in which the fluid domain is calculated with the finite volume method, and the structure is simulated by the finite element method. The modelling results of the flexible cylinder shell exiting water show that the effect of fluid-structure interaction could change the collapse process of the shoulder cavity and shorten the cavity collapse time. The collapse pressure of the cavity on the shell would decrease when the structure oscillation...
and the hydrodynamic loads have the same direction, while the collapse pressure would rise when their directions are opposite. Besides, the time history of the lift coefficient on the shell surface indicates that the oscillation characteristic of the hydrodynamics loads would be dominated by the vibration of the structure.

**IWSH 2017-434**

**Numerical Study on the Effects of Bilge Keel, Mooring and Riser Arrangement on FPSO Motion and Green Water Assessment**

Shuo Wang*, Xin Wang, Wai Lok Woo  
Newcastle University, Newcastle upon Tyne, NE1 7RU, United Kingdom  
*Corresponding author: S.Wang38@newcastle.ac.uk

Numerical case studies are carried out on a floating, production, storage and offloading unit (FPSO) model with a few different arrangements of the bilge keel, mooring and risers, using a practical numerical approach proposed and validated in our recent work. The aim of the case studies is to obtain more understanding on how the arrangement of appendages and attachments may affect the motion response and relative wave elevation on the FPSO.

First case study is performed on a spread mooring FPSO to investigate the effect of asymmetric risers at port side. Further study is carried out to investigate the effects of truncated bilge keel design combining with asymmetric riser arrangement. Numerical analysis is also carried out on a turret mooring FPSO in head sea condition to study the heave-pitch coupling effect induced by mooring and risers. Comparisons of numerical results between different models indicate that, the FPSO’s motion is affected by asymmetrically arranged appendages and attachments in a complicated way, which depends on the wave conditions. The relative wave elevation of the FPSO is also affected by appendages and attachments in a similar way, but not synchronous to the effect on motion response.

**IWSH 2017-436**

**Kinematics of Breaking Waves**

De Wang Chia1*, Longbin Tao1, Arun Kr Dev1, Xin Wang1, Yali Zhang2  
1School of Marine Science and Technology Newcastle University, United Kingdom  
2Lloyd’s Register, Global Technology Centre 1 Fusionopolis Place #09-11 Galaxis, Singapore 138522, Singapore  
*Corresponding author, d.w.chia@ncl.ac.uk

Wave breaking induces significant additional loads on offshore structures and increases the operational risk of the platforms. Understanding the kinematics of breaking waves not only helps to predict the inception of this extreme event but may also improve the estimation of the impact loads during such event.

One of the important kinematic properties of breaking waves is the wave celerity. A constant phase speed has been used for the wave breaking criterion by many researchers. However, this approach does not consider the variation of wave celerity at different phases before breaking. Hence, this paper examines the aspects of the wave breaking criterion and dynamics of wave celerity before breaking wave breaking.

Using a JONSWAP focused spectrum, breaking waves with different intensities were created at the Wind Wave Current Tank of Newcastle University. By using a high-speed camera and wave probes located at various positions of the breaking vicinity, the dynamics of the wave celerity during different phases of wave breaking were captured.

Based on the third order Stokes theory, a semi-empirical formula for the wave celerity estimation was also established. The estimated results were compared against the experimental data.
The kinematic wave breaking criterion has been established by analysing the experimental results and the comparison between this criterion and the ones proposed in the literature has also been presented in this paper.

IWSH 2017-437

The Study of Nonlinear Internal Wave in Two-layer Fluids due to a Point Vortex

Zhen Wang*, Changhong Wu
School of Mathematical Science, Dalian University of Technology, Dalian, China
*Corresponding Author, wangzhen@dut.edu.cn

The method of boundary integral method is employed to investigate the interfacial wave of stratified fluid due to a point vortex. When the point vortex is located in the upper layer, an integral-differential equation is formulated for the nonlinear interfacial wave, based on the incompressible potential flow theory, with the nonlinear boundary conditions at the interface. The fully nonlinear numerical results for the interfacial wave are compared when the point vortex is located in the upper layer liquid and in the upper layer liquid, respectively. And the linear analytical solutions for the interfacial wave are also compared about the location of the point vortex. It is demonstrated that when the point vortex strength and the stratified flow condition are kept same, the interfacial wave amplitude for the point vortex is located in the upper layer is far less than that for the point vortex is located in the lower layer. And its amplitude ratio increases with the density of the upper layer and lower layer, but it's always less than 1.

IWSH 2017-439

Bifurcation Analysis of Vortex-Induced Vibration for Top Tension Riser

Yuancen Wang*, Zhiqiang Wu, Xiangyun Zhang, Jiangtao Li
Tianjin University, School of Mechanical Engineering No.135 Yaguan Road, Tianjin 300350, China
*Corresponding Author, wangyuancen@sina.cn

The top-tensioned riser (TTR) is one of the most frequently used equipment in deep-sea petroleum engineering. At present, the research of its vortex-induced vibration (VIV) is mainly focused on numerical calculation and simulation, with few analytic analysis. In this paper, the nonlinear dynamic equation of VIV about TTR is developed by introducing the third-order variable lift coefficient hydrodynamic model. The amplitude-frequency response equation under 1:1 primary resonance excitation is deduced based on the single mode discrete results. The nonlinear dynamic responses characteristics of the TTR vibration under the excitation of vortex-induced resonance are discussed. Through the calculation of engineering examples, the bifurcation theory is used to analyse the singularity of the analytic solution of the system. Obvious hysteresis and solitary solutions from the results of singularity analysis are found. Such results can provide the guidance for the design and optimization about the riser structural parameters.

IWSH 2017-442

Parametric Study on the Flow Passing the Cylinder Mounted on the Edge of Backward-facing Step

Trieu V. Nguyen1*, Jiahn H. Chen2
1The University of Danang, University of Science and Technology, Faculty of Transportation Mechanical Engineering 54 Nguyen Luong Bang St., Danang City, Vietnam
2National Taiwan Ocean University, Department of Systems Engineering and Naval Architecture 2 Pei-Ning Road, Keelung 20224, Taiwan
*Corresponding author, nvtrieu@dut.udn.vn
In the present paper, the flow passing a circular cylinder mounted on the edge of a backward-facing step is numerically investigated by using Computational Fluid Dynamics (CFD). The k-epsilon realizable turbulence model (RKE) is employed to solve the Reynolds Averaged Navier-Stokes Equations (RANS). The numerical results show that the wake and vortex structures behind the cylinder are changed due to the backward-facing step.

IWSH 2017-443

Fully Coupled Effects on Waves and Barge with Single Sloshing Tank by CFD Methods
Yuan Zhuang, Decheng Wan*

Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China
*Corresponding author: dcwan@sjtu.edu.cn

With the demands for natural gas and sources in deep sea, the ship with partially filled tanks becomes more and more popular. The ship motion is an external excitation for sloshing tank, and sloshing tank would influence ship motion in return. In this paper, fully coupled effects on ship motion and a single sloshing tank is discussed using our in-house CFD solver naoe-FOAM-SJTU, which is developed and based on open source toolbox OpenFOAM. The coupling effects are observed between waves and the box with two tanks, and only one tank is partially filled. Only sway motion is released to discover the relationship between external wave and inner tank sloshing. To avoid the box move along with wave, a spring was considered. The results were compared with existing experimental results, and sloshing force and moment were included.

IWSH 2017-449

Investigation of the Seakeeping Performance of Twin Hull Vessels by Different Computational Methods
Chun-Ta Lin1*, Tsung-Yueh Lin1, Cheng-Wen Lin1,2, Yu-Wen Hsieh2, Ling Lu2, Ching-Yeh Hsin2
1CR Classification Society, Department of Research 8th Fl., No.103, Sec. 3, Nanking E. Rd., Taipei, Taiwan
2National Taiwan Ocean University, Department of Systems Engineering and Naval Architecture 2 Pei-Ning Road, Keelung 20224, Taiwan
*Corresponding author, ctlin@crclass.org

The seakeeping performances of wave piercing high speed catamaran (CAT-I) was computed and analyzed by different computational methods in this paper, and the purpose is to investigate the hydrodynamic characteristics of twin hull vessels for installing the motion suppressing devices such as RCS (Riding Control System). The potential flow method and viscous flow RANS (Reynolds Averaged Navier-Stokes) method were used for computations. The comparisons of results from different computational methods serve two objectives. The first one is for the verification purpose, and to ensure the accuracy of computations. Secondly, by comparing results with experimental data, the results not only show the numerical errors, but also reveal the different features of each method. For example, the viscous and nonlinear effects of forces and flow field can thus be identified. From the preliminary results, the comparisons between numerical predictions and experimental data show a good agreement for seakeeping performances, as well as some differences near the resonance frequencies. Overall, the viscous flow RANS method demonstrates better predictions in seakeeping based on the verification and validation analysis. This may due to the limitations of potential flow method, and further analysis is still ongoing.
Investigation of the Pre-Swirl Stator Effect to the Ship Resistance in Calm Water and in Waves
Yao-Tang Mao¹, Yu-Wen Hsieh¹, Ling Lu¹, Ching-Yeh Hsin¹*, Min-Mei Shih², Sheng-Ann Shieh²
¹Department of Systems Engineering and Naval Architecture, National Taiwan Ocean University, Keelung, Taiwan
²CSBC Corporation, Department of Design, 3 Jhonggang Road, Siaogang District, Kaohsiung 81234, Taiwan
*Corresponding author, hsin@mail.ntou.edu.tw

In this paper, the resistance of a container ship with a pre-swirl stator is computed, and the pre-swirl stator effects to the ship resistances in calm water and in waves are then investigated. The fuel efficiency in seaway becomes more important due to environmental regulations, and speed loss in seaway is required to evaluate as weather factor in EEDI. To obtain the weather factor, the simulations of self-propulsion tests both in calm water and in seaway have to be carried out. The speed loss and weather factor thus can be computed from ship speeds obtained at the same power in both calm water and seaway. Therefore, both the ship resistances in calm water and in seaway have to be computed. In this paper, the hull resistance in calm water will be computed by two different approaches. One is that the “double-body model” is used in the viscous flow computations to compute the viscous drag, and the wave-making resistance is computed by the potential flow boundary element method. The other one is that free surface effect is included in the viscous flow computations, and ship hull forces in calm water are only computed by the viscous flow method. The computations of hull resistances in sea way are also carried out by two approaches, which the added resistance is computed by a strip theory and by a viscous flow RANS method respectively. The computational results of added resistance by viscous flow RANS method are first verified by using both KCS and KVLCC2 ship models, and the results from RANS method are very close to those of strip theory and experiments. The hull resistances of a container ship designed by CSBC which has an energy-saving device, pre-swirl stator, are then computed and investigated. The results from strip theory and RANS method are first compared to each other, and the added resistances in waves are then investigated by the comparison between those with and without the pre-swirl stator.

Three-dimensional Numerical Study on Green Island Wake
Tien-Hung Hou¹, Shih-Chun Hsiao¹, Chia-Cheng Tsai², Tai-Wen Hsu³*
¹National Cheng Kung University, 1 University Road, Tainan 701, Taiwan
²National Kaohsiung Marine University, 142 Haihuan Road, Kaohsiung 811, Taiwan
³National Taiwan Ocean University, 2 Pei-Ning Road, Keelung 202, Taiwan
*Corresponding author, twhsu@mail.ntou.edu.tw

Kuroshio-induced island wake downstream of Green Island, Taiwan, is numerically investigated using POM (Princeton Ocean Model). The horizontal space and vertical space of 0.005 degree and 33 levels were specified to achieve a higher resolution of the island wake. Results from the HYCOM (hybrid coordinate ocean model) are used as the initial and boundary conditions in the model simulation. Numerical results are fairly compared with in-situ measurements of ADCP satellite image and simulating results of HYCOM. Characteristics of vortex street such as the aspect ratio, width, Strouhal number and period at different water depth were studied. Numerical results provide useful information for the development of power plant platform in Green Island.
### Added Resistance of Wigley-based Catamaran Advancing in Large Waves

Zi-Hao Zhang, Guang-Hua* He, Li-Min Chen  
Harbin Institute of Technology, Weihai, School of Naval Architecture and Ocean Engineering, 2 Wenhuaxi Road, Weihai 264209, Shandong  
*Corresponding author, ghhe@hitwh.edu.cn

When a ship is advancing in waves, series of strongly nonlinear phenomena make problems much more complex. To predict the seakeeping performance of ship in head waves with forward speed accurately, a numerical model is established by utilizing multiphase flow software, FLNE/Marine. The ship motion and added resistance of the Wigley catamaran in linear and nonlinear waves are simulated. To make a more realistic simulation, the solid-liquid-gas three-phase flow coupling model based on the viscous theory is developed by using the BRICS compressible discrete scheme as the free-surface capturing scheme which reduces numerical diffusion near the free-surface. A comparison of results between the monohull and catamaran is carried out and it is confirmed that the catamaran has a higher seakeeping performance than the monohull. This research can provide the references for the design of ship principal dimensions and structural strength.

### IWSH 2017-458

**Nonlinear Numerical Analysis of the Flow of Fluid Trapped inside a Narrow Gap by a Particle Method**

Cezar A. Bellezi\(^1,2\), Liang-Yee Cheng\(^2,3\) *, Makoto Arai\(^4\)  
\(^1\)Department of Naval and Offshore Engineering, University of Sao Paulo Av. Prof Mello Moraes, 2231, Cidade Universitária, São Paulo, Brasil  
\(^2\)Numerical Offshore Tank Laboratory (TPN-USP), University of Sao Paulo Av. Prof. Mello Moraes, 2231, Cidade Universitária, São Paulo, Brasil  
\(^3\)Department of Construction Engineering, University of Sao Paulo Av. Prof. Almeida Prado, trav. 2, n. 83, Cidade Universitaria, São Paulo, Brasil  
\(^4\)Faculty of Engineering, Division of Systems Research, Yokohama National University 79-5 Tokiwadai, Hodogaya-ku, Yokohama, Japan  
*Corresponding author, cheng.lee@usp.br

The flow inside a narrow gap is a phenomenon observed in several applications of the naval and offshore industry. The resonance of such flow could amplify the motion of the vessels and impair its operation or, in extreme conditions, cause collisions. The present work aims to investigate the dangerous transversal resonance modes of the flow in a narrow gap. Numerical simulations using a particle-based meshless method are adopted in order to capture nonlinear aspects of the flow. The investigation is carried out by using a two-dimensional numerical towing tank with a vertical wall in the left side and a slope beach in the right side. A moving square shaped barge is positioned close to the left wall separated by a narrow gap. The flow in the gap is investigated by a parametric analysis considering the length of the gap, the sway motion period and amplitude. The obtained results show that the piston-type resonance is dominant for short gap lengths and motion periods around 15 seconds. Significant nonlinearities are observed for the large amplitude motions, such as two resonance peaks and the formation of vortex in the lower edges of the barge.

### IWSH 2017-459

**Evaluation of Acceleration in Retreated Flow during Run-down Phase of Solitary Wave over Steep Slope**

Wei-Ying Wong, Chang Lin*, Ming-Jer Kao  
National Chung Hsing University, Department of Civil Engineering, 145, Xingda Road, South District,
Calculation of acceleration from the data of velocity profiles and/or fields measured is relatively difficult. The use of distinct time/spatial intervals can result in very different values of acceleration due to drastically temporal and spatial change of flow velocity in a highly unsteady flow, like the one in the run-down phase of a solitary wave traveling over a sloping bottom.

The experiments of velocity field measurements were carried out in a wave flume. The free surface elevations of the solitary waves propagating over a beach having a slope of 1:3 were measured by using two capacitance-type wave gauges. A flow visualization technique using particle trajectory method and a high-speed particle image velocimetry (HSPIV) system were employed to observe qualitatively the flow fields and to measure quantitatively the 2-D velocity fields. Then the local acceleration and two convective accelerations are calculated accordingly based on several trials of different spatial and time intervals used in the velocity fields. The method and procedure for computing the reasonable values of these accelerations is outlined. The calculated accelerations at different times and positions (heights) are then dealt with a partially depth-averaged operation to obtain the depth-averaged local and convective accelerations, together with the counterpart of pressure gradient.

**IWSH 2017-460**

**Nonlinear Effects on Ship Hydroelastic Responses with Forward Speed in Large-amplitude Waves**

Ryuta Tanaka, Takuya Taniguchi, Masashi Kashiwagi*
Osaka University, Department of Naval Architecture and Ocean Engineering, 2-1 Yamada-oka, Suita, Osaka 565-0871, Japan

*Corresponding author, kashi@naoe.eng.osaka-u.ac.jp

Estimation of wave-induced ship responses like springing in large-amplitude waves becomes more important for reliable fatigue assessment. In the present study, so-called weakly nonlinear Rankine panel method in the time domain is applied to calculate ship hydroelastic responses, taking account of nonlinear effects in the restoring and Froude-Krylov forces with forward speed. In this method, we consider memory effects in hydrodynamic forces through the impulse-response function to be computed with the Fourier transform of damping force coefficients in the frequency domain. The mode superposition method is used with Euler-Bernoulli’s beam theory for representing the structural deformation. Some calculation results based on the method are presented and the nonlinear effects on wave-exciting force and elastic responses with forward speed are discussed.

**IWSH 2017-461**

**Numerical Analysis of Ship Generated Unsteady Waves based on a Cartesian-Grid Method**

Kyung-Kyu Yang, Masashi Kashiwagi*
Osaka University, Department of Naval Architecture and Ocean Engineering 2-1 Yamada-oka, Suita, Osaka 565-0871, Japan

*Corresponding author, kashi@naoe.eng.osaka-u.ac.jp

The wave-making resistance of a ship in waves can be related to ship-generated unsteady wave pattern. In the present study, the unsteady wave-pattern analysis is applied to calculate the added resistance in waves for the modified Wigley model. In the present numerical method, a first order fractional-step method is applied to the velocity-pressure coupling in the fluid domain, and the volume-of-fluid (VOF) method is adopted to capture the fluid interface. The ship is embedded in a Cartesian grid, and the volume fraction of the ship inside the grid is calculated in order to identify the
different phases in each grid. The sensitivity of the location of measuring position as well as the number of solution grids is examined. The computed added resistance by direct pressure integration and wave pattern analysis is compared with experimental data. In addition, nonlinear characteristics of the added resistance in waves along with the unsteady wave-pattern are investigated.

IWSH 2017-462
Case Study on the Wave-current Interaction and its Effects on Ship Navigation
Chen Chen
JSPP Researcher, Department of Naval Architecture and Ocean Engineering, Osaka University
Corresponding author, cc198895@hotmail.com
The East China Sea, where both the strong Kuroshio Current and powerful low pressures exist, is an inevitable ocean area for various ships sailing between Japan and other Asian and European countries. The safety and economics of such shipping behaviors are often affected by the strong dynamics of the environmental matrix. The wave conditions are usually significant under high ocean winds and strong currents induced by the earth rotation as well as certain topography, leading to interactions between waves and currents. In this study, the third generation wave model SWAN are used to study about the wave propagation and wave-current interactions, following by their effects on the ship navigation discussed.

IWSH 2017-464
Experimental Research on the Resistance and Motion Attitude Variation of Ship-wave-ice Interaction in Marginal Ice Zones
Wan-zhen Luo, Chun-yu Guo*, Tie-cheng Wu, Yu-min Su
College of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China
*Corresponding author, guochunyu_heu@outlook.com
The interaction of waves and ice floes with the expansion of the marginal ice zones increasingly threatens the safety of ice-going ships. Therefore, research on the resistance of ice-going ships in marginal ice zones is necessary. Model testing is a feasible and practical method for conducting such research. In this study, the Harbin Engineering University towing tank equipped with a wave maker is used to simulate the special environment of the marginal ice zone using paraffin material as the model ice. A ship model resistance test is conducted under the combined effects of waves and ice floes. The study results show that the motion of the ship model is more unstable in the marginal ice zone than that in ice floes. Furthermore, the degree of the ship–ice interaction increases under the wave–ice interaction. The quantity of the ice floes submerging into the water from the bow and sliding along the bottom to the stern significantly increases, which may have negative effects on the propulsive efficiency of the ship propeller. The test result shows that the total ship resistance is not equal to the sum of the open-water resistance, wave-added resistance, and ice floe resistance because of the combined action of the waves, the ice floes, and the ship. The increase in the resistance resulting from the combination of the three components must be considered when designing an ice-going ship. Consequently, the parameters associated with the wave length, wave height, and ice concentration are the major factors influencing the added coupling resistance.

IWSH 2017-465
Review of VIV of Cylindrical Structure in Current and Suppression Methods
Liping Sun¹, Daming Wang¹*, Shuhong Chai²
¹Harbin Engineering University · Chuanhai Buiding, No. 145 Nantong Ave., Harbin, China
²Australian Maritime College, Univ. of Tasmania · Launceston Tasmania 7250, Australia
Steel risers are usually arranged in deep water with a certain configuration (catenary, lazy wave, etc) and are in responsible for carrying the oil or gas from the wells to the platforms or seabed which play a key role in the exploration process. These structures may be subjected to VIV (Vortex Induced Vibrations) due to the interaction between structures and water. Vortex Induced Vibrations (VIV) are motions induced on bodies interacting with an external fluid flow, produced by periodical vortex shedding on the structures. Due to the changing of the pressure distribution along the surface of the structure, there are different forces developed on each side of the body, thus leading to motion transverse to the flow. In specific range of the reduced velocity, “Lock-in” phenomenon may happen, which can cause serious vibrations. These undesired vibrations can reduce their lifetime due to fatigue. And as maintenance, repair and installation in oil exploration industry are quite demanding concerning costs and time, any increase in structural reliability has large economic benefits. In the past decades, both experimental and numerical methods are widely used to analyze VIV on risers. Various types of fairings and shrouds are invented to suppress VIV and have achieved satisfying results. This paper focuses on reviewing the work relevant with both VIV phenomenon and VIV analysis methods and suppression devices.

**IWSH 2017-466**

**Study on the Flow Field between the Main- and Demi-Hull of a Trimaran**

Lianzhou Wang, Chunyu Guo*, Zuotian Zhang, Yumin Su
Harbin Engineering University, College of Shipbuilding Engineering 145 Nan-Tong Street, Harbin 150001, China
*Corresponding author, guochunyu_heu@outlook.com

The interactions between the main- and demi-hull of trimarans have long been a major topic of interest, and obtaining detailed circumferential flow fields is very important to trimaran research. Based on the principles of viscous flow, in this study, the unsteady wake flow field of a trimaran was calculated using Reynolds-Averaged Navier-Stokes (RANS) equations, and the heaving and trimming of the trimaran were considered. Then, a particle-image velocimetry (PIV)-based experimental method was designed to obtain the detail flow field between the main- and demi-hull of the trimaran. As the test object, a trimaran model was adopted with one demi-hull that was manufactured using polycarbonate material. This material has excellent light transmission characteristics (90%) and refractive index close to that of water (polycarbonate: 1.58, water: 1.33), which helped to rectify problems affecting laser-sheet light-source transmission and high-speed camera recording. A nonstandard calibration was implemented for the PIV flow-field measurement system, an inverse three-dimensional (3D) distortion coordinate system was established, and the corresponding coordinates were obtained using optics calculations. Further, the spatial mapping for the PIV system was corrected and consequently the real flow field was obtained. The numerical results are in good agreement with the experimental data, which demonstrates that the numerical and experimental methods employed in this study can provide an important reference for obtaining the detailed flow-field information between the main- and demi-hull of trimarans.

**IWSH 2017-467**

**The Analysis of Nominal Wake Flow Characteristics in Short Wave**

Tian Liu¹, Chun-Yu Guo*, Hao-Hao Hao²

¹Harbin Engineering University, College of Shipbuilding Engineering, Nan-tong street, Harbin 150001, China
²Harbin Institute of Technology, School of Energy Science and Engineering, West straight street, Harbin 150001, China
KCS standard ship model is studied without considering its motions in this paper. The variation of the constraint model’s nominal wake field in a short regular wave encounter period is investigated by using Reynolds Averaged Navier-Stokes method combined with Shear Stress Transport turbulence model and its features are analysed. The results show that: the time history of axial nominal wake fraction is the same as the periodic variation of the incident wave. The variation of the wave wake fraction is relatively small in a single wave period. Wave wake fraction in negative value results from the wave drift in the direction of the wave direction, which is approximately equal to the total average flow. When the wave crest and wave trough locate in the propeller disk, the axial nominal wake field have obvious variability and are significantly affected by the wave steepness. Furthermore, the wave length has a certain influence on the axial nominal wake field for area below the propeller disk. Nominal flow field is discussed under short heading wave, providing a valuable reference for the problem about difference between the actual sea condition and the model test for the wake field and the propeller exciting force.