籌備委員會工作組員

主任委員	陳建宏
會 務 組	張建仁 總幹事 李耀輝
論 文 組	辛敬業 總幹事 周一志
秘 書 組	陳乃綺、廖嘉慧
會 計 組	余麗秀

Scientific Committee

- 方銘川（成功大學）
- 朱德祥（中國船舶科學研究中心）
- 何友聲（上海交通大學）
- 李家春（中國科學院力學研究所）
- 呂學信（高雄海洋科技大學）
- 吳有生（中國船舶科學研究中心）
- 邱逢琛（台灣海洋科技研究中心）
- 周連第（《水動力學研究與進展》編輯部）
- 柯永澤（台灣海洋大學）
- 黃正弘（成功大學）
- 黃正利（聯合船舶設計發展中心）
- 黃煌煇（成功大學）
- 黃榮鑑（永達技術學院）
- 郭真祥（台灣大學）
- 曾國正（台灣國際造船公司）
- 楊德良（台灣大學）
- 廖朝軒（台灣海洋大學）
- 劉格非（台灣大學）
- 戴世強（上海大學）
- 蔡進發（台灣大學）
- 繆國平（上海交通大學）
- 蕭葆羲（中央研究院）

注 意 事 項

- 敬請注意**大會會場**與**分場報告**地點不同。
- 由於海大校本部多處施工，停車請至工學院臨海停車場。
- 論文發表者請於各場次開始前十分鐘先請服務同學將簡報檔案複製至電腦內。
- 每一篇論文報告十五分鐘，第十三分鐘按鈴一聲提醒，第十五分鐘按鈴三聲提醒，五分鐘發問討論。
- 場次主持人請於場次開始前先確認論文發表者是否到場並說明論文發表及時間規定。
- 場次主持人請負責各會場秩序與論文發表時間之控制。
- 十月十三日(星期二)晚宴於下午六點三十分於基隆港海產樓舉行，附免費停車場，地址：基隆市信二路181號；電話：(02)24278026。
- 敬請各位與會貴賓於會議全程及晚宴務必佩帶名牌。

議 程 表

10 月 13 日

08:30~17:00	報到	早上：延平技術大樓 B1 下午：工學院一樓大廳
09:00~09:20	開幕	延平技術大樓 B1 演講廳
09:20~11:00	大會演講	延平技術大樓 B1 演講廳
11:00	團體照	延平技術大樓大門
11:10~12:40	休息與午餐	學生活動中心 3F 多功能展演廳
13:00~14:20	分場報告：A 場次	工學院 101 室、系工系 103、205 室
14:20~14:40	咖啡茶點時間	工學院一樓大廳
14:40~16:20	分場報告：B 場次	工學院 101 室、系工系 103、205 室
16:20~16:40	咖啡茶點時間	工學院一樓大廳
16:40~18:00	分場報告：C 場次	工學院 101 室、系工系 103、205 室
18:30~	晚宴	基隆港海產樓

10 月 14 日

08:30~12:00	報到	延平技術大樓 B1 & 工學院一樓大廳
09:00~10:40	大會演講	延平技術大樓 B1 演講廳
10:40~11:10	咖啡茶點時間	工學院一樓大廳
11:10~12:10	分場報告：D 場次	工學院 101 室、系工系 103、205 室
12:10~13:10	休息與午餐	工學院
13:20~14:40	分場報告：E 場次	工學院 101 室、系工系 103、205 室
14:40~15:00	咖啡茶點時間	工學院一樓大廳
15:00~16:20	分場報告：F 場次	工學院 101 室、系工系 103、205 室
16:25~17:00	研討會閉幕	工學院 101 室

附註：場次 1(工學院 101 室)、場次 2(系工系 103 室)、場次 3(系工系 205 室)

研討會議程

2009 年 10 月 13 日

08:30~17:00	**報到**
09:00~09:20	**開幕**
09:20~11:00	**大會演講　主持人：邱逢琛**
09:20~10:10	**波浪之調變與碎波** 成功大學副校長 黃煌輝
10:10~11:00	**基於 CFD 的船舶水動力學性能的計算與預報發展趨勢** 上海交通大學教授 繆國平
11:00~13:00	**休息、拍照與午餐**

13:00~14:20 Session A

Session A1 主持人：蕭松山

13:00~13:20 **隨機波作用下圓弧面防波堤波浪反射試驗**
吳米玲，蔣學煉，李炎保

13:20~13:40 **Trapping and near-trapping by arrays of cylinders in water waves using the addition theorem and superposition technique**
Yi-Jhou Lin, Ying-Te Lee, Jeng-Tzong Chen

13:40~14:00 **斜坡底床地形上大型圓柱波力之雙互換邊界元素法解析**
蕭松山，溫志中，張君名，謝宜辰

14:00~14:20 **Numerical modeling of a dual pontoon floating structure with a liquid container**
Chai-Cheng Huang, Chih-Ting Feng, Hung-Jie Tang

Session A2 主持人：方銘川

13:00~13:20 **A pressure correction method for solving NS equation and the application for sloshing**
Xiaoyu Guo, Benlong Wang, Hua Liu

13:20~13:40 **台船公司創新的貨櫃輪節能 10%技術**
陳柏汎，李志義，陳良駿

13:40~14:00 **貨櫃船三維興波流場之實驗研究**
鄭再峯，蔡進發

14:00~14:20 **微泡減阻方法應用於貨櫃船之數值模擬計算**
郭真祥，蔡進發，黃少廷，陳彥均，陳柏汎，陳良駿

Session A3 主持人：王道增

13:00~13:20 **The trajectories of particles in nonlinear interfacial waves**
Hung-Chu Hsu, Hwung-Hweng Hwung, Ray-Yeng Yang

13:20~13:40 **Influences of discharge reductions on salt water intrusion and residual circulation in Danshuei river estuary**
Wen-Cheng Liu, Wei-Bo Chen, Ming-Hsi Hsu

13:40~14:00 **Numerical simulation of the seabed oil spill under a variety of sea conditions**
Qing-jun Gao, Ying-jie Zhao, Tian-yu, Mao

14:00~14:20 **Harmonic decomposition by HHT on regular waves propagating over a rectangular submerged obstacle upon a fluidized bed**
Yung-Lung Chen, Shiaw-Yih Tzang, Shan-Hwei Ou

14:20~14:40 咖啡茶點時間

14:40~16:20 Session B

Session B1 主持人：楊瑞源

14:40~15:00 **台灣近年海嘯之數值模擬分析**
張國棟，謝佳紘

15:00~15:20 **Development of early warning modeling system for sudden chemical discharge incidents in plain river network**
Zuxin Xu, Chen Wang, Hailong Yin

15:20~15:40 **Experimental study on the sedimentation of cohesive sediment**
Wen-Yang Hsu, Yu-Shiuan Huang, Hwung-Hweng Hwung, Ray-Yeng Yang, Hung-Chu Hsu

15:40~16:00 **河口地區河流泥沙的環境效應**
王道增，樊靖郁，鐘寶昌

16:00~16:20 **Improved technique for controlling oscillation of coastal morphological modeling system**
Ming-Chung Lin, Yun-Chih Chiang , Sung-Shan Hsiao

Session B2 主持人：鄒早建

14:40~15:00 **應用多目標最佳化方法設計可適應多速度域之翼型**
黃正利，辛敬業，程俞樺，金尚聖

15:00~15:20 **非平面螺槳應用於貨櫃船之效率提升與激振力控制**
柯永澤，辛敬業，莊靖秋，陳柏汎

15:20~15:40 **Efficiency and weight: considerations for improved propeller design**
Jyh-Yih Li, Chi-Shin Chang, I-Chang Huang, Sing-Kwan Lee

15:40~16:00 **根據參數化環流分佈之螺槳設計方法**
錡瑞晴，辛敬業，李志義，陳柏汎

16:00~16:20 **空化數值分析應用於快艇螺槳**
涂景欽，張方南，金尚聖，張冠凱

Session B3 主持人： 陳柏汎

14:40~15:00 **NARMAX 模型在深水模型試驗中的應用**
范菊，袁夢，朱仁傳，繆國平

15:00~15:20 **Applications of the modified Trefftz method to the simulation of sloshing behaviours**
Yung-Wei Chen, Chein-Shan Liu, Chun-Ming Chang, Jiang-Ren Chang

15:20~15:40 **超大型貨櫃輪周圍風場與斜航性能之數值模擬**
林辰岳，陳柏汎

15:40~16:00 **水下拖纜系統之數值模擬**
呂學信，張博超

16:00~16:20 **The modified Trefftz method with a new concept for solving large amplitude sloshing problems**
Yung-Wei Chen, Chein-Shan Liu, Wei-Chung Yeih, Jiang-Ren Chang

16:20~16:40 咖啡茶點時間

16:40~18:00 Session C

Session C1 主持人：呂學信

16:40~17:00 **建橋對橋區河段水流及通航的影響分析**
范平易，鄒早建，汪淳，童朝鋒

17:00~17:20 **潮汐對台中港港池水質影響之研究**
溫志中，劉勁成，蔡立宏

17:20~17:40 **應用資料驅動模型反演海域潮流場數值模型開邊界條件方法初探**
李明昌，張光玉，周斌

Session C2 主持人：郭真祥

16:40~17:00 **鯨豚式推進器之研發歷程**
吳俊概

17:00~17:20 **渦流場中魚遊推進的水動力學研究**
潘定一，余剑聖，鄧見，邵雪明

17:20~17:40 **三維柔性尾鰭弦向展向變形對推進性能的影響研究**
王志東，叢文超，陳裴，老軼佳

17:40~18:00 **Numerical simulation of burst-and-coast swimming fish**
Meng-Hsuan Chung

Session C3 主持人：杜廣生

16:40~17:00 **相位轉移法應用於小水面雙體船之耐海性分析**
李子宜，方銘川，林岱陵

17:00~17:20 **應用船舶迴旋特性尋找最適化操舵角度之研究**
方志中，游坤達

17:20~17:40 **微氣泡在剪切流層中運動之趨勢**
張育齊，李耀輝，周一志，陳柏汎

17:40~18:00 **Experimental on the Drag Reduction Technique in Towing Tank**
Jing-Fa Tsai, Chi-Tran Chen, Yu-Ming Wang, Po-Fan Chen

18:30 晚宴

2009 年 10 月 14 日

08:30~12:00 **報到**

09:00~10:40 **大會演講　主持人：王偉輝**

09:00~09:50 **大會演講：船舶水動力學研究的若干進展**
中國船舶科學研究中心副所長 顏開

09:50~10:40 **大會演講：從創新與整合談螺槳設計**
財團法人聯合船舶設計發展中心董事長 黃正利

11:10~12:10 Session D

Session D1 **主持人：吳祚任**

11:10~11:30 **新型 TLP 平臺水動力性能預報**
匡曉峰，繆泉明，楊爍，何再明

11:30~11:50 **湖泊水動力與底泥污染物釋放的關係研究**
王超，王沛芳，牛淮金，歐陽萍，李金

11:50~12:10 **臺灣四周海域之潮殘餘流探討**
莊文傑，廖建明

Session D2 **主持人：陳明志**

11:10~11:30 **高速公路中央分隔帶對廂式貨車行車氣動特性影響的研究**
杜廣生，李莉，李永衛，岳建雄

11:30~11:50 **計算流體力學於風力發電機葉片氣動力性能計算之研究**
郭真祥，楊淳宇

11:50~12:10 **The method of fundamental solutions for solving the streamfunction-vorticity formulation of the Navier-Stokes equations**
Chia-Ming Fan

Session D3 **主持人：柯永澤**

11:10~11:30 **An envisaged inception process for the new type of cavitation identified from the three gorges turbines**
Shengcai Li

11:30~11:50 **非定常自然空泡流動現象研究**
陳瑛，魯傳敬，郭建紅，潘展程

11:50~12:10 **水中單一空蝕氣泡之產生與受壓破裂流場的量測研究**
趙勝裕，楊昇學，葉克家

12:10~13:20 休息與午餐

13:20~14:40 Session E

Session E1 主持人：萬德成

13:20~13:40 **附加分離盤對雷諾數 1000 和 10000 情況下隔水管流動控制的數值模擬研究**
王嘉松，蔣世全，譚波，谷斐

13:40~14:00 **Numerical study of wave-bathymetry interactions using shallow-water equations**
Shin-Jye Liang, Wei-Chun Wu

14:00~14:20 **Dynamic coupling of multi-phase fluids with a moving obstacle**
Mei-Hui Chuang, Tso-Ren Wu, Chih-Jung Huang, Chung-Yue Wang, Chia-Ren Chu

14:20~14:40 **Development of a level set method with better volume preservation to predict interface in two-phase flows**
C. H. Yu, P. H. Chiu, Tony W. H. Sheu

Session E2 主持人：吳寶山

13:20~13:40 **螺旋槳葉片環量分佈數值分析**
洪方文，董世湯

13:40~14:00 **螺旋槳尾流場 PIV 測量與分析研究**
張軍，洪方文，李廣年，張國平，陸林章

14:00~14:20 **Propulsion calculation of container ship equipped with vortex fin near stern**
Shiu-Wu Chau, Jhih-Hong Fan, Hao-Hsiang Hsu, Po-Fan Chen, Jyh-Yih Li

Session E3 主持人：繆泉明

13:20~13:40 **船舶在尾隨浪拍擊下的模型試驗與回應預報**
邱　強，楊大明，駱寒冰，萬正權

13:40~14:00 **球形液滴衝擊自由液面之數值模擬**
廖清標，洪梓銘，葉忠訓

14:00~14:20 **Flow induced by relatively moving plates**
Chi-Min Liu

14:20~14:40 **Initial stage about three-dimensional waves generated by a submerged body moving in water**
Chih-Hua Chang

14:40~15:00 咖啡茶點時間

15:00~16:20 Session F

Session F1 主持人：莊文傑

15:00~15:20 **錢塘江出海碼頭水域水動力條件數值模擬**
魯海燕，潘存鴻，韓海騫，吳輝

15:20~15:40 **Tsunami dispersion effect in propagation from Manila Trench to Taiwan**
Dong-Jheng He, Tso-Ren Wu

15:40~16:00 **Experimental investigations of tide effect on coastal groundwater table**
Longhua Wu, Shui-ying Zhuang

16:00~16:20 **海嘯湧潮對近岸結構物之影響**
魏妙珊，吳祚任

Session F2 主持人：蔡進發

15:00~15:20 **Prediction of fully nonlinear wave loads on ships by CIP based Cartesian grid method**
Changhong Hu

15:20~15:40 **串列式雙振動翼推進性能探討**
邱逢琛，張政傑

15:40~16:00 **耦合液艙晃蕩的船舶運動性能研究**
鄒康，繆泉明

16:00~16:20 **船舶作橫蕩和首搖運動數值模擬研究**
韓陽，吳寶山，潘子英

Session F3 主持人：趙勝裕

15:00~15:20 **Density/viscosity blockage method for the viscous flows with complex immersed interfaces**
萬德成

15:20~15:40 **Flow characteristics of a pair of rotating side-by-side circular cylinders**
Farida R Purnadiana, Dedy Z Noor, Ming-Jyh Chern

15:40~16:00 **The method of fundamental solutions for the multi-dimensional wave equations**
D. L. Young, M. H. Gu, C. M. Fan

16:00~16:20 **Streamwise dynamics controlled jet spreading, mixing and physical source of the vortices**
Amalendu Sau, Robert R. Hwang

16:25~17:00 閉幕典禮

論文摘要

大會演講（2009/10/13　09:20~11:00）

Observations on the Evolution of Wave Modulation

Hwung-Hweng Hwung[1]
[1]Department of Hydraulic and Ocean Engineering, National Cheng Kung University
[*]**E-mail:** hhhwung@mail.ncku.edu.tw

Wave modulation and breaking are an interesting and complicated nonlinear wave motion. In order to understand the characteristics of wave modulation and breaking, a series of laboratory experiments on the long-time evolution of nonlinear wave trains in deep water was carried out in a super wave flume (300x5.0x5.2 m) at Tainan Hydraulics Laboratory of National Cheng Kung University. Two typical wave trains, namely uniform wave and imposed sideband wave, were generated by a piston-type wavemaker. Detailed discussions on the evolution of modulated wave trains, such as transient wavefront, fastest growth mode and initial wave steepness effect, are given and the results are compared with existing experimental data and theoretical predictions.

Present results on the evolution of initial uniform wave trains cover a wide range of initial wave steepness $(k_c a_c = 0.12 - 0.30)$ and thus, greatly extend earlier studies that are confined only to the larger initial wave steepness region $(k_c a_c > 0.2)$. The amplitudes of the fastest growth sidebands exhibit a symmetric exponential growth until the onset of wave breaking. At a further stage, the amplitude of lower sideband becomes larger than the carrier wave and upper sideband after wave breaking, which is known as the frequency downshift.

Experiments on initial imposed sideband wave trains with varied initial wave steepness illustrate that the evolution of the wave train is a periodic modulation and demodulation at post-breaking stages, in which most of the energy of the wave train is transferred cyclically between the carrier wave and two imposed sidebands. Meanwhile, the wave spectra show both temporal and permanent frequency downshift for different initial wave steepness, suggesting that the permanent frequency downshift induced by wave breaking observed by earlier researchers is not permanent. Additionally, the local wave steepness and the ratio of horizontal particle velocity to linear phase velocity at wave breaking in modulated wave group are very different from those of Stokes theory.

Keywords: modulation; recurrence; sideband instability; frequency downshift

基于 CFD 的船舶水动力学性能的计算与预报发展趋势

缪国平　朱仁传　范菊
上海交通大学船舶海洋与建筑工程学院
Email: renchuan@sjtu.edu.cn

船舶水动力性能的研究是船舶综合航行性能预报技术的基础，也是新型船舶与海洋工程结构物研究开发首先需要解决的关键问题之一。

船舶水动力学在发展过程中形成了四大传统的研究领域，分别是船舶阻力、船舶推进、船舶操纵和船舶耐波性。由于历史的原因，上述四个领域是相对独立地发展起来的。事实上，船舶是一个矛盾的综合体，一种性能的提高有时会不可避免地影响另一种性能，它们既相对独立又互相耦合。例如，快速性的提高往往会影响耐波性。航向稳定性的提高反过来使回转性能恶化。这中间当然有个设计的权衡问题。但从研究角度来看，从孤立研究某一性能到综合研究是目前的发展趋势之一。以往在船舶阻力、推进和操纵的研究中一个基本的前提是在静水状态中进行的，在许多场合，由于波浪的影响，上述静水状态的研究成果显得不够充分或者说不能很好地反映实际现象和解决实践中出现的问题。我们必须考虑波浪与船体边界层的干扰、波浪中的失速、波浪对推进器效率的影响、波浪中的操纵性能、波浪中的稳性等等一些非线性耦合问题。许多本来可以以定常问题来描述的数学问题变成了非定常问题。

近年来，随着计算机科学技术飞速发展，计算速度和容量大幅度提高，计算流体力学（CFD）在自由面追踪方面出现了很多技术，如 VOF 法、Level Set 方法、SPH 方法等等，为带自由面的流动研究注入了新的活力，应用 CFD 技术进行船舶与海洋工程结构物流动数值模拟和水动力性能预报的优越性和应用前景日益显现。从研究角度而言，原先难以求解的复杂物体粘性流动问题、强非线性问题、瞬态响应问题在现阶段都可以用 CFD 技术一定程度上直接求解，例如船舶阻力与带自由液面高雷诺数三维流场的精细结构、船-桨-舵干扰与螺旋桨性能数值模拟、甲板上浪与砰击、液舱晃荡、舰首飞溅、极端海况下船舶运动响应预报等等。一些传统的基于势流或理想流体理论处理的问题，如船舶耐波性预报与研究，

也开始向在粘性流场中直接求解的方向发展；尽管目前离工程要求的实用化和反应的快速化上仍然有很大的距离，但 CFD 在水波与结构物的相互作用方面巨大的潜力已经得到广泛的认同。

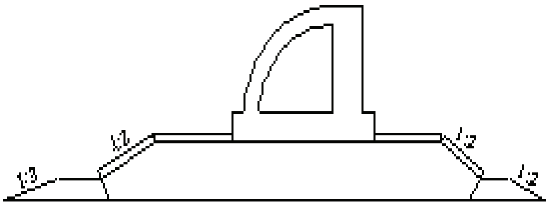
从研究线性或（和）定常问题向研究非线性或（和）非定常问题发展、从理想流体研究向粘性流体研究发展形成了目前另两大发展趋势。

Session A1 (2009/10/13 13:00-14:20)

隨機波作用下圓弧面防波堤波浪反射試驗

吳米玲[1] 蔣學煉[2] 李炎保[3*]
[1] 武漢港灣工程設計研究院
[2] 天津城市建設學院
[3] 天津大學 港口工程系
[*]**E-mail:** yanbao.li@263.net

圓弧面防波堤是在半圓型防波堤基礎上發展的新型防波堤。其斷面由 1/4 圓圓弧體和基床組成（如圖示）。圓弧面防波堤具有半圓型堤所特有的波浪作用力小，穩定性好；結構簡單，圓弧拱圈受力性能好，造價低；施工安放後即可抵禦大浪襲擊以及視覺美觀等特點，但較之半圓堤可減少將近一半的拋石基床體積。筆者們曾對圓弧面防波堤的應用可行性和規則波作用下水力特性進行過物理模型試驗和數值類比。本文介紹為這一新型防波堤工程應用所進行隨機波試驗成果。試驗對比了圓弧面防波堤與半圓型堤在 JONSWAP 譜隨機波作用下的反射與波浪力特性，應用概率分佈法、譜法分析試驗成果，討論了波浪力概率分佈特徵、波力譜、綜合反射係數、反射係數譜特徵與影響因素。



圓弧面防波堤斷面示意圖

關鍵詞：圓弧面防波堤，隨機海浪，波浪反射，波浪力，譜分析法

Trapping and near-trapping by arrays of cylinders in water waves using the addition theorem and superposition technique

Yi-Jhou Lin[1], Ying-Te Lee[1], and Jeng-Tzong Chen[1,2*]
[1]Department of Harbor and River Engineering, National Taiwan Ocean University
[2]Department of Mechanical and Mechatronics Engineering, National Taiwan Ocean University
[*]**E-mail:** jtchen@mail.ntou.edu.tw

In this paper, we employ the addition theorem and superposition technique to examine the near trapping mode of water wave problems. The scattering of water waves by arrays of vertical circular cylinders is solved by using the null-field integral equations in conjunction with degenerate kernels and Fourier series to avoid calculating the Cauchy and Hadamard principal values. In the implementation, the null-field point can be exactly located on the real boundary without singularity owing to the introduction of degenerate kernels for fundamental solutions. An adaptive observer system in the polar coordinates is considered to fully employ the property of degenerate kernels. This method can be seen as a semi-analytical approach since errors attribute from the truncation of Fourier series. The physical-resonance phenomena of near-trapped modes are our concern. The nonuniqueness solution of fictitious frequency arising in the boundary integral equation (BIE) is also addressed. Several examples are given to demonstrate the validity of the proposed approach.

Keywords: Addition theorem, Superposition technique, Null-field integral equations, Fourier series, Trapped modes

斜坡底床地形上大型圓柱波力之雙互換邊界元素法解析

蕭松山 [1*] **溫志中** [2] **張君名** [1] **謝宜辰** [1]
[1] 台灣海洋大學　河海工程學系
[2] 弘光科技大學 環境與安全衛生工程系
[*]**E-mail:** sshsiao@mail.ntou.edu.tw

關於海洋波浪作用於大型結構物之波力計算，以往的邊界元素法波浪模式一般以 Berkhoff (1972) 推導出的緩坡方程式(Mild-slope equation, MSE)作為主要控制方程式。但此控制方程式(MSE)在推導過程中忽略底床高階項，無法反應擾動地形處對波浪之影響。緣此，本文將以 Chamberlain and Porter (1995) 提出之修正型緩坡方程式(Modified mild-slope equation, MMSE)保存了底床高階項，更能準確計算出波場中之波浪變化，應用雙互換邊界元素法(Dual Reciprocity Boundary Element Method, DRBEM)建立更有效之數值模式，以解析斜坡底床上之波浪折、繞射問題。經由本研究計算結果與前人研究之驗證，顯示本模式之適用性相當良好。
關鍵詞：波力，大型結構物，緩坡地型，修正型緩坡方程式

Numerical modeling of a Dual Pontoon Floating Structure with a Liquid Container

Chai-Cheng Huang[1*]**, Chih-Ting Feng**[1]**, and Hung-Jie Tang**[1]
[1]Department of Marine Environment and Engineering, National Sun Yat-Sen University
[*]**E-mail:** cchuang@mail.nsysu.edu.tw

This paper is to establish a numerical model to study the wave-induced dynamic properties of a dual pontoon floating structure with a liquid container on the top. This model has to deal with two kinds of fluid domain: one is the incident wave domain, while the other is the excited water wave domain inside a sloshing container. The boundary integral equation method (BIEM) with linear element scheme is applied to establish a 2D fully nonlinear numerical wave tank (NWT). The nonlinear free surface condition is treated by combining three schemes such as the Mixed Eulerian and Lagrangian method (MEL), the fourth-order Runge-Kutta method (RK4) and the cubic spline scheme. The second-order Stokes wave theory is adopted to give the velocity on the input boundary. Numerical damping zones are deployed at both ends of the NWT to dissipate or absorb the transmitted and reflected wave energy. The velocity potential and acceleration potential are solved simultaneously to obtain the instantaneous forces of waves interacting with the floating structure. In this study, the acceleration potential method and the modal decomposition method are used to solve the external forces on the floating body as well as the sloshing impact induced by the water inside the container. Summing the total external forces on the floating platform, we may form a system of motion equations. To establish an accurate and yet efficient numerical model is the goal of this paper, and it may lay a good foundation for the future study on the renewable wave energy and structural motion damping system.
Keywords: BIEM, NWT, sloshing, acceleration potential method, modal decomposition method

Session A2 (2009/10/13　13:00-14:20)

A pressure correction method for solving NS equation and the application for sloshing

Xiaoyu Guo, Benlong Wang, Hua Liu[*]
Department of Engineering Mechanics, Shanghai Jiao Tong University
[*] **E-mail:** hliu@sjtu.edu.cn

This paper presents a family of algorithms for solving time-dependent Navier-Stokes equations. The first order scheme, the improved first order scheme and the second order scheme are presented. To evaluate the efficiency and capability of these methods, two different cases are simulated and the results are compared with those of other methods. It is demonstrated that, these methods are stable and efficient. The second order method is highly accurate and has the full second order convergence rate for both velocity and pressure. Based on the new method, a numerical investigation of flip through in depressurized conditions has been dedicated to highlight the role of the air-entrapment during the impact of a breaking wave against a vertical wall. Furthermore, a comparison between the numerical results and experimental results by Miozzi (2007) is done.
Keywords: Navier-Stokes equations; Stability; Accuracy; Efficiency; Convergence rate.

台船公司創新的貨櫃輪節能 10%技術

陳柏汎 [1*]，李志義 [1]，陳良駿 [1]
[1] 台灣國際造船公司　設計處基本設計課
[*]**E-mail: 096946@csbcnet.com.tw**

為降低燃油消耗、減少船舶運輸成本及對環境的汙染，台船公司積極整合台灣及國外造船產、官、學界資源，共同致力於船舶節能技術的開發，成立相關研發案，以現有設計優良之 1,700TEU 新型貨櫃輪減少 10%馬力為目標，簡稱 ES-10 計畫。以阻力節能 5%、推進裝置節能 5%，即節能 10%為目標，開發降低阻力及提高推進效率的相關技術，包括：船舶線形改善、高效率新型螺槳、反動舵、舵翼及預旋流產生裝置，希望經由理論分析、計算流力應用及大規模水槽試驗等方式，進行相關技術的研發工作。

先以分案方式與國內大學造船相關系所進行個別研發，輔以於國內水槽執行之模型試驗驗證，逐步找出單項最佳化設計，再到德國漢堡水槽(HSVA)進行最後之試驗驗證，確認單項性能之後，再組合各有效的省能源裝置進行節能效果最大化的試驗。最後根據 1,700TEU 貨櫃輪原始船艉的節能試驗結果，選出 3 種最佳的省能源裝置組合與構形，應用在新設計的節能船艉上，加上最佳航行俯仰節能，總計節省 10.3%馬力，順利的達成 ES-10 減少 10%馬力的目標。

關鍵詞：節能，預旋流產生裝置，反動舵，Energy Saving

貨櫃船三維興波流場之實驗研究

鄭再峯　蔡進發
台灣大學　工程科學及海洋工程學系
E-mail: d96525011@ntu.edu.tw

本研究利用六隻波高計建立一三維波形量測系統用來量測拖曳水槽中船模的三維興波波形，並使用此系統量測兩艘貨櫃船 RD542_0 與 RD542_1 的三維興波波形，最後也利用所量測的波高資料計算此兩艘貨櫃船的興波阻力係數。

本研究的三維波形量測結果和計算結果比較發現兩者的資料相當吻合，只有在波峰及波谷處量測值稍低於計算值，兩艘貨櫃船的興波阻力係數雖然幾乎相同，但從不同橫向位置的波高比較圖可以發現 RD542_1 的艉浪低於 RD542_0 的艉浪，顯示消浪艉的觀念是可以降低艉浪。

本研究由量測的波高值計算興波阻力係數，發現其計算值和由阻力實驗由形狀因子所求的的興波阻力計算值的誤差約在 7%以內。

關鍵詞：貨櫃船，興波阻力係數，消浪艉

微泡減阻方法應用於貨櫃船之數值模擬計算

郭真祥 [1] ，蔡進發 [1]，黃少廷 [1]，陳彥均 [2]，陳柏汎 [3]，陳良駿 [3]
[1] 台灣大學　台灣大學工程科學與海洋工程學系
[2] 龍華科技大學　多媒體與遊戲發展科學系
[3] 台灣國際造船公司　設計處基本設計課
[*]**E-mail: r96525043@ntu.edu.tw**

本研究主要之目的為利用計算流體力學中的二相流方法模擬微氣泡流體噴入貨櫃船船體周圍流場產生減阻之效果，其中將氣泡噴口位置設計五個不同位置，再針對各種不同設計方案的計算結果加以比較分析，做為微泡噴口位置選用之依據。貨櫃船船型採用台船公司所設計的 RD542_2 船型，藉助於 CFD 計算軟體 COMET 的應用，探討於不同含氣率之設定下，微泡對減阻效果的影響，計算條件採用 Menter’s Baseline k-ω 紊流模型，含氣率設定為 10%到 90%之間，其中發現在航速為 20 節時含氣率為 10%時，有較佳的減阻效果。另外為了瞭解微泡在實船應用上的表現，將五號開口(約位於由 FP 往後算起 1/6 船長之位置)應用到實船尺寸上做計算，並比較實船尺寸與船模尺寸減阻上的差異，由計算的結果發現：微氣泡在大尺寸的船殼上較無法持續在船殼上產生作用，日本學者 Watanabe 教授及 Kodama 教授所作的實驗也顯示出這樣的結果；接著分析在 10 節、20 節、30 節三種不同航速下微泡產生的減阻效果，結果顯示在較高速(30 節)情況下使用高含氣率的微泡產生的減阻效果較佳(約降低 10%摩擦力)，而在低速(10 節)時則沒有明顯減阻效果。

關鍵詞：微泡減阻、計算流體力學、CFD、二相流

Session A3 (2009/10/13 13:00-14:20)

The Trajectories of Particles in Nonlinear Interfacial Waves

Hung- Chu Hsu [1*], Hwung-Hweng Hwung[2], Ray-Yeng Yang[1]
[1] Tainan Hydraulics Laboratory
[2] Department of Hydraulic and Ocean Engineering, National Cheng Kung University
[*]**E-mail:** hchsu@thl.ncku.edu.tw

This paper uses a modified Euler-Lagrange transformation method presented by Chen and Hsu (2007) to obtain the second-order trajectory solution in a Lagrangian form for the water particles in nonlinear interfacial waves. We impose the assumption that the Lagrangian wave frequency is a function of wave steepness and an arbitrary vertical position for each water particle. Expanding the unknown function in a small perturbation parameter and using a successive expansion in a Taylor series for the water particle path and the period of a particle motion, the second order asymptotic expressions for the Lagrangian particle trajectories, the mass transport velocity and the period of particle motion can be derived directly in Lagrangian form. The mean level of the particle motion in Lagrangian form differ from that of the Eulerian. Finally, the second-order asymptotic solution obtained is uniformly valid in contrast with early works containing resonant terms presented by Wiegel (1964) (Eqs. (B.1) and (B.2) in Appendix B.) based on a straightforward expansion for two-dimensional progressive waves.
Key words: Lagrangian approach, Eulerian approach, Lagrangian wave frequency, Lagrangian mean level, particle trajectory, mass transport

Influences of Discharge Reductions on Salt Water Intrusion and Residual Circulation in Danshuei River Estuary

Wen-Cheng Liu[1,*], Wei-Bo Chen[2], and Ming-Hsi Hsu[2]
[1]Department of Civil and Disaster Prevention Engineering, National United University
[2]Department of Bioenvironmental Systems Engineering, National Taiwan University
[*]**E-mail:** wcliu@nuu.edu.tw

Reservoirs are the most important and effective water storage facilities in modifying uneven distribution of water both in space and time. They not only provide water, hydroelectric energy and irrigation, but also smooth out extreme inflows to mitigate floods or droughts. However, they naturally led to the imposition of man-made changes on rivers, such as reservoir regulation, bathymetric change, and freshwater withdrawals. In these cases, upstream changes of the river runoff will inevitably have consequences on the estuarine portion of the river.

A three-dimensional hydrodynamic model was established and applied to study the salt water intrusion in the Danshuei River estuarine system of northern Taiwan. The river system has experienced dramatic changes in the past half century because of human intervention. The construction of two reservoirs (i.e. Shihmen Reservoir and Feitsui Reservoir) and water diversion in the upper reaches of the river system significantly reduces the freshwater inflow. The changes had contributed farther to the intrusion of tidal flow and salt water in the upstream direction. The model was calibrated and verified with available hydrographic data measured in 2001, 2002 and 2008. The model was then used to probe the change in salt water intrusion as a result of reservoir construction. The model simulations indicate that more tidal energy propagates into the estuarine system after reservoir construction because of the substantial increase in river cross-sections. The residual circulation before reservoir construction was weaker than that after reservoir construction. The limits of salt water intrusion after reservoir construction extended farther inland 2~3 km than those after reservoir construction. This case study offers the quantitative estimate of the salinity and residual circulation changes due to human interface in this nature system.
Keywords: Three-dimensional hydrodynamic model, freshwater reduction, reservoir, salt water intrusion, residual circulation.

Numerical Simulation of the seabed oil spill under a variety of sea conditions

GAO Qing-jun, ZHAO Ying-jie, MAO Tian-yu
Laboratory of Environmental Protection in Water Transport Engineering,
Tianjin Research Institute of Water Transport Engineering,
E-mail: zhonghedaibiao@yahoo.com.cn

Most of the oil spill models to predict the trajectory and fate of the spilled oil were concerned at the surface or near surface spills, only a few were about submarine spills. However, these underwater oil spill models were mainly considered in calm water. So a model was developed to predict the underwater oil spill due to the barge oil from wreck and the leakage of the seabed pipeline under kinds of sea conditions. Calculate the trajectory of the underwater spill oil under current, wave, wind and joint action of them, and analogy analysis the related pollution range. The computed result can be scientific basis of oil spill response decision making.
Keywords：Underwater oil spill; FLUENT; Tidal current; Wave; Wind

Harmonic Decomposition by HHT on Regular Waves Propagating over a Rectangular Submerged Obstacle upon a Fluidized Bed

Yung-Lung Chen [1*], Shiaw-Yih Tzang [1] and Shan-Hwei Ou [2]
[1]Department of Harbor and River Engineering , National Taiwan Ocean University
[2]Department of Environmental Resources Management, Tajen University
[*]**E-mail: d92520007@mail.ntou.edu.tw**

Wave flume experiments have been previously conducted to investigate wave reflection and decay of non-breaking regular waves propagating over a rectangular submerged obstacle (S.O.) upon a fluidized bed (Tzang et al., 2008). They have found that waves could be reduced by as maximum as 50% near the obstacle due to the occurrence of soil fluidization. To further detail the transformations of the wave harmonics induced by interactions among waves, the obstacle and the fluidized bed, the Hilbert-Huang Transform (HHT) method is applied to analyze their wave data for both unfluidized (UF) and resonantly fluidized (RF) responses. The Ensemble EMD (EEMD) purposed by Wu and Huang (2009) has been validated in analyzing the non-stationary pore pressure data in a RF response (Chen et al., 2008). Thus, the established scheme is adopted in this study to first decompose the frequency shifted waves into intrinsic mode functions (IMF) in the vicinity of the obstacle. Then, the Hilbert spectra are calculated to illustrate the temporal variations of each harmonic.
Keywords: HHT, Submerged obstacle, Fluidized bed

Session B1 (2009/10/13　14:40-16:20)

台灣近年海嘯之數值模擬分析

張國棟 [1*] ，謝佳紘 [1]
[1] 國立高雄海洋科技大學　海洋環境工程系
[*]**E-mail: ktchang@mail.nkmu.com.tw**

本研究模擬近年台灣地區海底斷層引發之海嘯，使用美國康乃爾大學之 COMCOT 模式(Cornell Multi-grid Coupled Tsunami Model, 1997)，輸入 Harvard CMT(Centroid Moment Tensor) 之斷層活動參數，模擬海嘯於台灣沿岸傳遞的變化情況。模式利用多重網格藕合有限差分處理 (multi-grid coupled finite difference scheme)模擬海嘯由外海傳播到近岸的溯上過程，分別考慮線性淺水波 (linear shallow water wave) 方程及非線性淺水波 (non-linear shallow water wave) 方程，並採用蛙躍式 (leap-frog scheme)的有限差分格點系統來執行，海岸線附近則使用移動邊界(moving boundary) 處理，可使海嘯波於近岸地區的模擬狀況較為符合實際情形。

模擬結果與 1986/11/15 花蓮海嘯實測水位紀錄相驗證，結果顯示數值模擬海嘯水位變化情況和實際觀測數據相當一致，証實了模式的可靠性。並模擬 1978/12/23 台東成功、2002/03/31 花蓮秀林及 2006/12/26 屏東恆春外海海嘯之傳播情形，模擬結果展釋出各海嘯沿台灣四周岸際傳遞的變化情形，以及海嘯波於近海受到水深地形變化影響所產生的波形變動。本研究成果可提供未來台灣地區海嘯預警之參考。
關鍵詞：海嘯數值模擬，海嘯溯上，移動邊界

Development of Early Warning Modeling for Sudden Chemical Discharge Incidents in Plain River Network

Xu Zuxin, Wang Chen, Yin Hailong*
College of Environmental Science and Engineering, Tongji University
E-Mail:yinhailong@tongji.edu.cn

Plain river network is usually characterized with developed industries leading to large amounts of potential environment risks. The dense population in the region intensifies the potential risks. Previous modeling is usually suitable for the scenario discharging in the single river or large open waters such as costal waters or sea one river, but it is incapable of simulating the sudden chemical discharging in the cross-linked river network. At the same time the various types of risks including floating, solute and settling chemicals make the conventional spilling oil or solute advection-diffusion model insufficient. So the comprehensive early warning modeling technique for floating, solute and settling chemicals' sudden discharge in the plain river network is presented. Firstly, to overcome the difficulty of chemical discharging model in simulating the sluice gates controlling flowing field in the crossed rivers, a water flowing model based on SMS for the plain river network equipped with sluice gates was established, and the simulated hydrologic boundary conditions near the sluice gates was further input into DELFT3D for simulating chemical discharging in waters to achieve the integration of complex flowing field with the chemical discharging concentration field in the plain river network. Secondly, taking Minhang District in Shanghai for study area, the odium cyanide, toluene and paint which represent floating, solute and settling chemicals respectively were screened from the industrial risks; correspondingly the early warning model for sudden odium cyanide, toluene and paint discharging in Shanghai plain river network were established, which proved the reliability of developed model for a variety of sudden chemicals discharging in the plain river network.
Keywords: water environment early warning model; sudden chemical discharge; plain river network

Experimental study on the sedimentation of cohesive sediment

Wen-Yang Hsu[1*], Yu-Shiuan Huang[1], Hwung-Hweng Hwung[1], Ray-Yeng Yang[2] , Hung- Chu Hsu[2]
[1]Department of Hydraulic and Ocean Engineering, National Cheng Kung University
[2] Tainan Hydraulics Laboratory
[*]**E-mail: n8895110@mail.ncku.edu.tw**

An experiment study was carried out to investigate the settling behavior of cohesive sediments in static water. Six optical backscatter sensors (OBS) were used to monitor changes of suspended sediment concentration (SSC) at different levels. Settling velocity of each level in vertical direction was determined by depth-integrated mass balance equation. Additionally, acoustic backscatter system was used to provide clear insight of shock wave. The fall speed of interface was calculated by tracing peak response of backscatter and which is significantly different with settling velocity derived from mass balance equation. Because of the complexity involved in the settling velocity measurements, a simultaneous application of different types of instruments seems to be the more appropriate approach to understand the aggregate dynamics and settling behavior. Compared with velocity of shock wave, OBS approach has ability to display variations of settling velocity due to floc dynamics in vertical column. Not all cases had a good linear relationship between settling velocity and velocity of shock wave. Within a limited range of SSC, however, these two velocities can be well correlated. Moreover, the criteria for occurrence of shock wave and translation of hindered settling were obtained. An empirical formula is also proposed to calculate the settling velocities at different SSC.
Keywords: cohesive sediments, settling velocity, optical backscatter sensors

河口地区河流泥沙的环境效应

王道增，樊靖郁，鐘寶昌
上海大学　上海市应用数学和力学研究所
Email: dawang@staff.shu.edu.cn

河口地区河流泥沙的环境效应正愈来愈为人们所重视，通常河口地区人口稠密，经济发达，水环境污染严重；同时河流水体中含有丰富的细颗粒泥沙。泥沙和水体共同成为污染物的载体，泥沙在水体中

的悬浮、沉降、输移以及它们对污染物质的吸附及解吸，共同影响着污染物在河流水体中的输移和转化过程，最终影响水体的生态系统，这就是泥沙的环境效应。

本文针对河口地区河流受到潮汐水流的影响以及含有丰富的细颗粒泥沙特征，通过实验研究和数值模拟对河口地区河流水体中的泥沙环境效应进行了研究，得到了以下主要结果:

1) 河口地区河流受到潮汐水流影响，复杂的水动力条件，使得泥沙频繁处于悬浮、沉降、输移的动态过程中。泥沙的运动特征或存在状态对污染物的吸附和释放有直接的影响。
2) 河口地区河流中悬移质泥沙浓度对水体中各种污染物含量有直接关系。悬移质浓增加时，水流中的 COD、BOD 以及氨氮值增加，反之减少。
3) 河口地区河流一般都有一层受污染的底泥，在潮汐水流作用下会频繁起动、悬浮、沉降、再悬浮。如此，受污染底泥对上覆水体的二次污染就有两种形式: 一是静态释放，底泥未受到冲刷起动，污染物从底泥孔隙水中直接向上覆水体扩散释放; 二是动态释放，底泥颗粒起动再悬浮后，在水体中通过泥沙颗粒表面污染物大量释放，造成水环境迅速恶化。

最后，本文结合上海市苏州河水环境综合整治，通过水槽试验研究了苏州河泥沙在潮汐水流作用下的悬浮、沉降、输移特性和得到泥沙颗粒的沉降速度、起动流速、水流离散系数等参数; 并在此基础上建立数学模型进行数值模拟。实验研究和数值模拟主要结果得到苏州河现场实测资料验证。

關鍵詞：河口，泥沙，潮流，水環境

Improved Technique for Controlling Oscillation of Coastal Morphological Modeling System

Ming-Chung Lin[1], Yun-Chih Chiang[2] , Sung-Shan Hsiao[3*]
[1]Department of Engineering Science and Ocean Engineering, National Taiwan University
[2]Center for General Education, Tzu Chi University
[3]Department of Harbor and River Engineering, National Taiwan Ocean University
[*]**E-mail: sshsiao@mail.ntou.edu.tw**

Numerical modeling of coastal morphodynamic evolution is a powerful tool for planning and design of coastal engineering. Coastal morphological modeling system is based on sediment conservation law, which couples modules for waves, waves driven currents, and sediment transport rates. The numerical scheme for sediment conservation law and nonlinear coupling between these modules can lead to dispersions, diffusions, spurious oscillations and stability problems which are still not well developed.

In this paper, we analyze the diffusions of the morphological modeling and develop the technique of controlling oscillation in spatial and temporal discretization. The bed-slope updated techniques, WENO schemes, and 2-steps 3-time-levels temporal discretization are employed to avoid the oscillations and improve the stability of the coastal morphological model with accuracy up to $O\left(\Delta t^2, \Delta x^5, \Delta y^5\right)$.

With these methods, the model shows good performances for coastal areas with complex topography (Fig. 1), as shown in Fig. 2. The numerical stability for the long-term morphological simulations are examined, the modeling system can be used with large time grid intervals well.

Keywords: Morphological modeling system, Controlling oscillation, Complex topography.

Session B2 (2009/10/13 14:40-16:20)

應用多目標最佳化方法設計可適應多速度域之翼型

黃正利[1]，辛敬業[2]，程俞樺[2]，金尚聖[1*]
[1]財團法人聯合船舶設計發展中心
[2]台灣海洋大學　系統工程暨造船學系
[*]**E-mail: samking@mail.usddc.org.tw**

本文研討一套可設計出適應多速度域翼型之流程，產出翼型能擁有良好的性能，在遭遇不同的入流狀況時能維持性能的穩定性。本文探討三種設計邏輯，分別是給定壓力分布、應用拉格朗日法於單一目標最佳化、應用拉格朗日法於多目標最佳化。而相同的最佳化設計需求是在滿足升力的條件下使阻力降到最小，且均採用黏性流方法來計算其性能與空化狀況，本文中所假設的三個設計例在計算中均發現有效。此設計流程未來可擴展至螺槳整體設計中，解決螺槳在斜軸操作或多域操作速度時的缺陷。

關鍵詞：翼型設計、多目標最佳化、拉格朗日法、黏性流

非平面螺槳應用於貨櫃船之效率提升與激振力控制

柯永澤 [1*]，辛敬業 [1]，莊靖秋 [1]，陳柏汎 [2]
[1] 台灣海洋大學　系統工程暨造船學系
[2] 台灣國際造船股份有限公司
[*]**E-mail: kehr@mail.ntou.edu.tw**

本文主要目的是建立一完整泛用之非平面幾何螺槳之設計理論，此理論於升力線設計階段，容許在徑向上有大幅度變化的歪斜（skew）與傾斜（rake）存在，並且考慮螺槳三個方向誘導速度之影響，而能對非平面螺槳進行設計。關於螺槳徑向負載之最佳化，將採用變分法（Calculus of Variations）進行求解，而最後的螺槳幾何設計，則需借助 BEM 的計算作決定。

由理論推導發現，非平面螺槳之速力圖以及螺距之計算與傳統螺槳大不相同，對於非平面螺槳葉尖向低壓面傾斜將使螺槳其葉尖設計螺距與拱高大幅下降。由實驗結果發現非平面螺槳之葉尖渦空泡確實不容易發生，證實其有很強之葉尖端板效應，此對螺槳誘發振動與噪音之控制非常有利，而效率亦比傳統螺槳明顯增加，對於 1700TEU 貨櫃船螺槳的設計於模型尺寸可增加效率約 2.28%，推至實船尺度可增加效率約 3.59%，並證實本文所發展的理論可以相當準確的預測該等螺槳的受力與效率。
關鍵詞：非平面螺槳、效率、空化、激振力控制

Efficiency and Weight: Considerations for Improved Propeller Design

Jyh-Yih Li[1*], Chi-Shin Chang[1], I-Chang Huang[1] Sing-Kwan Lee[2]
[1] CSBC Corporation, Taiwan
[2] ABS
[*]**Email: 091930@csbcnet.com.tw**

The large fluctuations of fuel prices in conjunction with strong trends towards using less energy while also curbing emissions has brought to the forefront the need to increase energy efficiency in the shipping industry. During each phase of a vessel's lifecycle— namely design, operation on sea, and maintenance—there are many ways of reducing fuel consumption and cutting emissions. However, the most cost-effective method providing the greatest energy savings can be achieved at the basic design stage. Fundamentally, low resistance and high propulsion efficiency are the two main aspects in energy-saving ship design. The focus in this paper will be the latter, high propulsion efficiency.

In this study, an integrated propeller design for an 8200 TEU containership will be implemented to maximize its energy-saving potential for propulsion. Through the comprehensive investigation of the possibility of enlarging the propeller diameter and reducing the blade area ratio as well as reducing skewness, an optimal design for the propeller efficiency will be obtained. However, this is not the end of the study. To minimize the propeller fabrication cost, the propeller blade will be thinned as much as possible to reduce the amount used of the expensive propeller alloy. In order to achieve this design intention, a series of propeller stress analyses needs to be performed to determine the minimum allowable thickness distribution on blade. The resulting data will provide the necessary facts towards ensuring that the propeller is fatigue failure safe for normal ahead operation and yielding failure safe under emergency stop operation.

In an integrated design, propeller induced vibration should also be considered. As it is well-known, a large containership is high powered with a relatively fast service speed. Propeller cavitation cannot be avoided in a practical design. Due to the large ship size, the ship structure becomes more flexible and more prone to the vibration problem from the cavitating propeller excitation. Thus, to control the vibration so that it will be within an acceptable level during operations, propeller induced hull vibration analysis will also be performed providing feedback to the necessary modifications of the propeller design.

A Propeller Design Procedure Based on the Parametric Circulation Distributions

Juei-Ching Chi[1], Ching-Yeh Hsin[1], Jyh-Yih Li[2] and Po-Fan Chen[2]
[1]Department of Systems Engineering and Naval Architecture, National Taiwan Ocean University
[2]CSBC Corporation, Kaohsiung, Taiwan
***E-mail: hsin@mail.ntou.edu.tw**

In this paper, a graphical user interface (GUI) program is developed, and a parametric design of the

circulation distribution is established under this platform. For propeller designs, the high efficiency, high speed and low vibration and noise are always important; however, different design concepts such as tip unloading to reduce the tip cavitation, root unloading to reduce root erosion are more popular as the increase of ship speed and energy-saving requirements. The presented paper has developed a propeller design program to integrate different design methods, and to present the design results visually through the graphical user interface. The optimum circulation distribution can be obtained through this design program, and can then be adjusted by methods of splines, B-splines and Chebyshev polynomials. The lifting surface and boundary element methods are then used to design the camber and pitch distributions. Adjustment of the circulation distribution can result in different design results, and the presented method provides a tool for designers to immediately visualize the effects. The parametric studies of the circulation distributions have been carried out in the paper, and performances corresponding to different circulation distributions have been demonstrated. This design method is proved to be effective, and it is found that through the parametric design process, a more efficient propeller other than that designed by the optimum circulation distribution can be obtained.
Keywords: Propeller Design , Circulation Distribution, Lifting line method, BEM, GUI

空化數值分析應用於快艇螺槳

涂景欽[1*]，張方南[1]，金尚聖[1]，張冠凱[1]
[1]財團法人聯合船舶設計發展中心
[*]**E-mail: jctu@mail.usddc.org.tw**

快艇螺槳由空化所造成之空蝕現象一直是亟待解決之問題，對於螺槳根部空化現象之控制，業界過去主要依靠各廠家之經驗並輔以實船測試驗證。如能建立控制螺槳空化的能力，則可避免螺槳產生空化所造成效率下降及與空蝕所帶來的結構破壞。由於空化乃至空蝕皆為流體中極為複雜之現象，藉由實驗觀測與數值計算分析兩者並行，可以獲得更多相關資訊。首先由數值計算預估螺槳可能之空化現象，再經由螺槳實驗所取得之高速攝影照片或影像比對計算結果，比較數值分析所得到壓力與速度場資訊，配合實驗觀測的空泡型態與發生位置，可取得細部流場觀察空化之成因並推論空蝕發生之可能性，最後改良設計使其更能控制空化而減少空蝕對螺槳的影響。
關鍵詞：空化、螺槳、計算流體力學

Session B3 (2009/10/13 14:40-16:20)

NARMAX 模型在深水模型试验中的应用

范菊，袁梦，朱仁传，缪国平
船舶海洋与建筑工程学院 上海交通大学
Email: fanju@sjtu.edu.cn

深水试验中由于模型试验池的深度限制，试验需要采用混合模型，使用截断锚系。混合模型试验主要由被动式和主动式两种。本文采用主动式。通过 NARMAX 模型理论，对单根张紧锚泊线进行研究，锚泊线在模型池底部被截断。将锚泊线上端点的运动时历作为系统的输入，截断点的运动和张力时历等作为系统的输出。根据 NARMAX 模型理论，相应的的系统模型可被识别。得到的系统模型表达式，可用来预报锚泊线截断点的运动和张力的时历。将模型预报结果与原时历进行了比较。从结果可见，模型预报结果与原时历相当吻合，NARMAX 模型的预报效果良好。通过模型识别方法，根据锚泊线上端点的运动，可控制在模型池底部截断点的运动和张力等，这对深水系泊模型试验的实施提供有效的手段。

Applications of the Modified Trefftz Method to the Simulation of Sloshing Behaviours

Yung-Wei Chen[1], Chein-Shan Liu[2], Chun-Ming Chang[3], and Jiang-Ren Chang[1*]
[1]Department of Systems Engineering and Naval Architecture, National Taiwan Ocean University
[2]Department of Mechanical and Mechatronic Engineering, National Taiwan Ocean University
[3]Department of Harbor and River Engineering, National Taiwan Ocean University
[*]**E-mail: cjr@mail.ntou.edu.tw**

This paper develops a modified Trefftz method (MTM) associated with the group-preserving scheme (GPS) to deal with the two-dimensional non-linear sloshing problem. For non-linear sloshing phenomena, the conventional numerical method may encounter numerical instabilities and numerical dissipation because the coefficient matrix obtained from the conventional numerical method is ill-posed, and the free surface exhibits nonlinear kinematics. To overcome these problems, we introduce the concept of controlled volume into the numerical procedure to prevent the elevation from vanishing, and we take the characteristic length into account to maintain the stability of the new numerical method. Moreover, the use of the GPS eliminates the need to use second-order derivatives, thus increasing numerical efficiency, and weighting factors can be used to observe the vanishing of the velocity potential. Comparisons of results of the present study with those in the literature show that the numerical results are better than those obtained by using previous methods. The method developed here is a simple and stable way to cope with the non-linear sloshing problems.
Keywords: Laplace equation, Modified Trefftz Method, Mixed-boundary value problem, Sloshing

超大型貨櫃輪周圍風場之數值模擬

林辰岳[1*]，陳柏汎[1]
[1]台灣國際造船公司設計處 基本設計課
[*]**E-mail:** Phobia91@hotmail.com

隨著全球貿易量的增加與新巴拿馬運河的拓寬，新一代巴拿馬極限型貨櫃輪船深已將近 30 公尺，加上甲板上裝載 8~9 層貨櫃，有必要研究這種超大型貨櫃輪航行時受強風作用的影響，因此本研究選定台船公司最新開發的 12,600 TEU 新巴拿馬極限型超大型貨櫃輪作為船舶周圍風場的研究目標，建構船舶滿載水線以上三維模型以計算流力(CFD)程式分析在不同貨櫃裝載量下各風向角的風場分佈，計算出船舶風場中關鍵的四種分量：縱向受力、橫向受力、平擺力矩、橫傾力矩，做成無因次化圖表，可與現有設計資料作比較，並可作為日後設計超大型貨櫃輪的參考。

本研究採用計算流體力學軟體 STAR CCM+作為計算工具，其理論與方法為有限體積法與以壓力為基礎分離求解法來求解三維 Navier-Stokes 方程式，紊流模式採 k-ω SST 模型來模擬，採用 GRIDGEN 軟體產生數值計算的網格。超大型貨櫃輪周圍風場之數值模擬結果如進一步結合水線以下船舶斜航的流場分析結果，將可作為設計者評估船舶受風時之航向穩定性及設計超大型貨櫃輪舵面積的參考。
關鍵詞：新巴拿馬極限型、STAR CCM+、船舶風場

水下拖纜系統之數值模擬

呂學信，張博超
國立高雄海洋科技大學造船工程系
[*]**E-mail:** **ssleu@mail.nkmu.edu.tw**

本研究的目的在探討船舶轉彎圓形路徑拖纜之平衡運動。研究中假設纜索本身為一纜索物體系統，視為一完全柔性且不可拉伸之細長結構體，兩端包括拖曳端之「拖曳點」及尾掛端之「掛物點」。當拖曳點以一定速度作適當半徑之圓周運動，運動中此纜索的諸外力包含：張力、重力、慣性力及水下環境外力等達成平衡時，重要影響參數間之關係所展現的螺旋運動圖騰是本研究的重點。本研究相關的八個重要參數包括：纜索總長、纜索端點之垂向深度(Depth)、拖曳點圓周運動速度、纜索張力、末端掛物點之掛重、掛重距轉動軸的距離、拖曳轉動半徑及掛物點和拖曳點之相位角等。本研究為一初始邊界值問題的探討，研究過程以朗其古塔(Runge-Kutta)數值方法，模擬水下圓形拖曳纜索的動力行為。所得之數值結果，將顯示纜索端點之垂向深度、掛重距轉動軸的距離、拖曳點纜索張力、掛物點之相位角、拖曳點圓周運動速度、拖曳轉動半徑等之相對關係，其結果有利於水下纜索物體系統定位問題的探討。

本研究獲致以下諸結論：(一) 拖曳半徑愈大者其對時間模擬的穩定性愈佳，數值收斂迅速。(二) 纜索端點之垂向深度 Vert. depth[d/L]和拖曳速度成反向關係。(三) 索端點之垂向深度 Vert. depth[d/L]和掛重距轉動軸的距離 Dtox[r/R]對拖纜半徑呈相反趨勢。另外，本研究利用數值模擬計算，求得纜索端點之垂向深度、掛重距轉動軸的距離、及掛物點和拖曳點之相位角三數值，可得知水面下之末端點和水面上拖曳基礎點之相對位置，因此有利於水下纜索物體系統定位問題的探討。

本研究目前完成三個測試速度有和四個迴轉半徑的模擬計算分析，今後速率若增為五個，迴轉半徑提昇至纜索長甚至超過，且以單一迴轉後直航進一步和半單一迴轉後逆向直航比較，其結果將和 WHOI

Grosenbaugh (2007)之研究一致，有利後續之比較研究之進行。
關鍵字：纜索物體系統、垂向深度、水下定位

The Modified Trefftz Method with a New Concept for Solving large amplitude sloshing problems

Yung-Wei Chen[1], Chein-Shan Liu[2], Wei-Chung Yeih[3], and Jiang-Ren Chang[1*]
[1]Department of Systems Engineering and Naval Architecture, National Taiwan Ocean University
[2]Department of Mechanical and Mechatronic Engineering, National Taiwan Ocean University
[3]Department of Harbor and River Engineering, National Taiwan Ocean University
[*]**E-mail:** cjr@mail.ntou.edu.tw

This paper is concerned with the accurate and stable analysis of large amplitude liquid sloshing in two-dimensional tank under the forced excitation. For large amplitude sloshing behaviours, numerical instabilities and numerical dissipation usually arise seriously due to ill-posedness of the coefficient matrix and non-conservation of mass. To overcome these problems, we introduce the concept of controlled volume into the numerical procedure to prevent the elevation from vanishing, and we take the automatic characteristic length into account to maintain the stability of the new numerical method. Besides, the use of the GPS eliminates the need to use second-order derivatives and thus, increasing numerical efficiency, and weighting factors can be used to observe the vanishing of the velocity potential. Comparisons of results of the present study with those in the literature show that the numerical results are better than those obtained using previous methods. The method developed here is a simple and stable way to cope with the non-linear sloshing problems.
Keywords: Laplace equation, Modified Trefftz Method, Mixed-boundary value problem, Sloshing

Session C1 (2009/10/13　16:40-18:00)

建桥对桥区河段水流及通航的影响分析

范平易[1]，邹早建[1]，汪淳[1]，童朝鋒[1]
[1]上海交通大学 船舶海洋与建筑工程学院
[*]**E-mail:** fpylove@sjtu.edu.cn

桥梁的建设，会对水流的流态产生一定的影响，由此可能会带来一些负面效应。建立数学模型对桥区河段水域进行数值模拟，是研究这些影响的一种最常用的方法。本文利用 Telemac-2D 软件建立了江阴至青龙港（北支）、杨林（南支）的长江口河段的二维有限元水动力数学模型，并用实测的潮位和流速资料对模型进行了验证。验证结果表明，建立的模型较好地反应了该河段的水动力特性。在此基础上，以苏通大桥为例，建立桥墩模型并进行数值模拟，利用建桥前后的计算结果，进行水动力分析，总结出建桥前后水动力变化的一般规律；并结合相关规范，就建桥对通航的影响进行了研究，得出了一些结论。本软件在国内使用甚少，本文对相关参数包括边界条件的选择和处理具有一定的指导意义，其计算结果可以为后续的研究提供参考。
关键词：Telemac-2D，水流，数值模拟，桥墩，通航

潮汐對台中港港池水質影響之研究

溫志中[1*]，劉勁成[2]，蔡立宏[3]
[1]弘光科技大學　環境及安全衛生工程學系
[2]交通大學　土木工程學程學系
[3]交通部運輸研究所　港灣技術中心
[*]**E-mail:** wen1558@sunrise.hk.edu.tw

台中港位於台灣中西部海域，現有港口朝西北西方向，寬度三百五十公尺。港域面積現有水域 640 公頃和陸域 2500 公頃。由於台中港區受潮汐變化影響甚劇，潮波運動特性複雜，本研究擬以此主題，探討在潮波運動特性下港持區域內水質之變化趨勢。本研究擬以水動力模擬，以數值模擬計算方式，探討

流體之運動與動力特性。在研究中考量時間、地形、水深、底床摩擦與渦度效應等之影響，用丹麥水力研究所 (DHI: Danish Hydraulic Institute) 研發完成之 MIKE21_HD 水動力計算模式(DHI, 2002)，配合 Matsumoto 等人(2000) 針對 NAO Tide 提出的 NAO.99b 模式的輸出結果作為驅動 HD 模式進行天文潮推算所需的動力邊界條件，進而針對臺灣環島海域進行潮汐與潮流數值模擬計算。利用水動力計算所得之結果，探討在不同潮汐水位變化下，台中港港池內之水質變化。

關鍵詞：水動力模式，潮位，港灣，水質

应用数据驱动模型反演海域潮流场数值模型开边界条件方法初探

李明昌　張光玉　周斌
交通部天津水运工程科学研究院
*E-mail: lmcsq1997@163.com

利用POM（Princeton Ocean Model）等数值模型计算海域潮流场需要开边界条件来驱动模型，开边界条件的确定是主要难点之一。本文建立基于人工神经网络的数据驱动模型为海域潮流场数值模型验证提供开边界条件。通过实际海域计算验证模型，取得了良好的效果。

关键词：数据驱动模型，人工神经网络，POM 模型，反演，开边界

Session C2 (2009/10/13　16:40-18:00)

鯨豚式推進器之研發歷程

吳俊概
長榮大學 科技工程管理學系
E-mail: wowakai@mail.cjcu.edu.tw

本文介紹一種可改善船舶推進效率與操縱性能之新式推進器—鯨豚式推進器之發展歷程。該推進器為一擺動翼片式仿生機構，其組成包含一主支架、一旋轉臂、一推拉桿、一翼片、一驅動單元及一垂直轉向軸系，其動作原理為模仿高泳速狀況下鯨豚類之尾部擺動動作。該推進器推進效能之證實，已歷經數艘模型船之實作與測試。測試結果顯示，該推進器設計兼具推進之有效性及結構之強健性。再者，該推進器所具備之 360 度可轉向功能，已使其在某些應用上，可望優於傳統的螺旋槳推進器。

關鍵詞： 仿生、擺動翼片、推進器、鯨豚式、高效率

涡流场中鱼游推进的水动力学研究

潘定一，余釗聖，邓见，邵雪明*
浙江大学力学系，流体传动及控制国家重点实验室
***Email:** mecsxm@zju.edu.cn

鱼群在游动过程中可以利用身体的侧线来感知和利用流场中的涡漩，从而提高推进效率。实验研究中发现，当将鱼体置于 D 型半圆柱产生的尾流场中时，鱼体可以通过弯曲身体，而获得逆流向前的推进力。本文中，首先对 D 型半圆柱产生尾流场中的鱼型体波动进行了数值模拟，模拟中，取雷诺数 1500，选取 NACA0012 翼型作为鱼体横截面。结果显示，当鱼体以一定的频率和相位进行波动时，鱼体推进力的获得与鱼体与 D 型半圆柱之间的距离有关。根据鱼体所获得推进力的变化，以及流场中涡结构的变化，可将 D 型半圆柱的尾流场区域依次定义为：吸附区、推进力获得区和涡流场弱影响区。在吸附区，由于鱼体距离 D 型半圆柱较近，对于 D 型半圆柱的涡脱落过程产生较大影响，一般当鱼体位于此区域时，会直接被吸附至半圆柱附近。在推进力获得区，由半圆柱脱落的涡，依次附于鱼体的上下表面，鱼体在此区域可获得较大的推进力，此时由于在鱼体表面的涡作用，造成鱼体表面的流体流动方向与来流方向相反，从而使得鱼体所受粘性摩擦力也表现为推进力。在涡流场弱影响区，鱼体所获推进力较推进力获得区有所降低，半圆柱脱落涡不在附于鱼体表面。文中给出了相应的涡结构示意图和鱼体所受推进力随与半圆柱距离的变化曲线。其次，文中同时给出了 D 型半圆柱尾流场中，仿生拍动翼的运动的数值模拟。

仿生拍动翼是俯仰和升沉的组合运动，是鲸鱼等大型海洋鱼类的月牙尾的简化模型，文中，主要给出了不同俯仰角以及升沉振幅时，仿生拍动翼在 D 型半圆柱尾流场中所受推进力的变化情况，同时给出了相应了涡结构示意图和推进力变化曲线。本文采用沉浸边界法进行数值模拟，与传统的贴体网格法或者非结构网格法相比，沉浸边界法应用笛卡儿网格，大大缩短了前处理所耗费的时间，并且消除了由于网格质量而带来的计算误差。沉浸边界方法尤其适合于模拟运动边界问题，与动网格法相比，沉浸边界法具有非常高的计算效率，无须在每个时间步重新生成网格。

三维柔性尾鳍弦向展向变形对推进性能的影响研究

王志东，丛文超，陈裴，老轶佳

江苏科技大学 船舶与海洋工程学院

*E-mail: cywzd@sina.com

根据鱼类尾鳍摆动式推进模式，建立了三维柔性尾鳍数值计算模型，利用 FLUENT 软件对柔性摆动尾鳍的推进性能进行了数值计算。其中，湍流模拟选用 $k-\omega$SST 模型，弦向变形采用非均匀载荷悬臂梁方程，展向变形采用 BOSE 模式，探讨了变形系数、变形长度对三维柔性尾鳍水动力性能的影响。结果表明，当弦向变形系数为 0.0025，弦向变形长度为整个弦长时，推进效率可达 66%，平均推力系数值为 0.5047; 在相同计算工况下，弦向展向同时变形能够获得更高的推进效率。

关键词： 柔性尾鳍；弦向变形；展向变形；推进特性

Numerical Simulation of Burst-and-Coast Swimming Fish

Meng-Hsuan Chung

Institute of Ocean Engineering and Technology, National Kaohsiung Marine University

E-mail: mhsuan@mail.nkmu.edu.tw

Burst-and-coast swimming performance in fish-like locomotion is studied via two-dimensional numerical simulation. The numerical method is the collocated finite-volume adaptive Cartesian cut-cell method developed previously. The NACA00×× airfoil shape is used as the equilibrium fish-body form. Swimming in burst-and-coast style is computed assuming the burst phase is composed of a single tail beat. Swimming efficiency is evaluated in terms of mass-specific cost of transport instead of Froude efficiency. Effects of Reynolds number (based on body length and burst time), duty cycle, and fineness ratio (body length over largest thickness) on swimming performance (momentum-capacity and mass-specific cost of transport) are studied quantitatively. The results lead to a conclusion consistent with the previous findings that larval fish seldom swims in burst-and-coast style. Given mass and swimming speed, the lowest cost can be reached if fish swim in burst-and-coast style with fineness ratio of 8.33, a value larger than that derived from the simple hydromechanical model proposed in literature. The calculated amount of energy saving in burst-and-coast swimming is comparable with the real-fish estimation in literature. Finally, wake-vortex structures of both continuous and burst-and-coast swimming have been presented and studied.

Keywords: Burst-and-Coast, Efficiency, Mass-specific cost of transport, Fineness ratio, Wake vortex

Session C3 (2009/10/13　16:40-18:00)

相位轉移法應用於小水面雙體船之耐海性分析

李子宜 [1*] ，方銘川 [1] ，林岱陵 [1]

[1] 國立成功大學　系統及船舶機電工程系

*E-mail: p1896101@mail.ncku.edu.tw

本文採用三維脈動型源點配合小板法作為基礎，解析單邊船 體 的入射、繞射及輻射問題，以求解在流場中所產生的壓力，並將得到的壓力利用座標轉換的方式，加入相位角的觀念，推導出雙體船 的各項流體動力係數，求解船 舶 之六度運動量以及非線性力 。由於本方法僅需計算單一船 體再加以轉換，即

可得到雙體船 間無交互作用情況下之流體動力係數，因此對於有考慮兩船 體間交互作用影響的計算方式而言，本文所推導出的方法可有效地節省計算時間及計算機資源。

本文以小水面雙體船(SSP)船型進行理論分析。另外，亦加入了黏性阻尼效應及穩定翼效應來修正得到的運動量。最後，再探討不同的入射波角、船 體間距與速度，對雙體間交互作用的影響，以達到相位轉移法之最佳實用性。

關鍵詞：相位轉移法、非線性力、雙體船、交互作用

應用船舶迴旋特性尋找最適化操舵角度之研究

方志中[*]，游坤達
國立臺灣海洋大學運輸與航海科學系
[*]**Email: ccfang@mail.ntou.edu.tw**

本研究應用野本謙作(NOMOTO, 1964)所提出的船舶操縱一階線性近似式，探討線性模式中之迴旋性指數(K 值)、延遲性指數(T 值)與船舶迴旋角速度間之關係，再應用航海技術中新航向距離之規劃，探討船舶在不同船速避碰時所需之舵角。船舶操縱數值模擬採用日本 MMG 模式，對一艘聯合船舶設計發展中心所設計之 5550TEU 的大型貨櫃船進行定迴旋運動，進而模擬出不同船速、舵角下之迴旋運動反應，並藉由分析各時間點之迴旋角速度，計算出不同舵角下之 K 值與 T 值，並建立大型貨櫃船在各種船速、舵角之迴旋係數迴歸模式，並對大型貨櫃船之迴旋特性進行討論。最後將以一簡化之例子，探討船舶在已知改向角度與所需要之新航向距離，在不同船速時需要使用多少舵角才能達到避碰之功效。

關鍵詞：一階線性近似式、船舶操縱、新航向距離、迴歸模式。

微氣泡在剪切流層中運動之趨勢

張育齊 [1]，李耀輝 [1]，周一志 [1*]，陳柏汎 [2]
[1] 國立台灣海洋大學　系統工程暨造船學系
[2] 台灣國際造船股份有限公司
[*]**E-mail: ycchow@ntou.edu.tw**

微氣泡在流場中運動的趨勢是同時具有學術上及工業上的重要性。因此，我們建構了一座具有上下兩層流道的剪切流水槽(shear-layer water tunnel)，並以調整上下層流道的流速來分別模擬具有正值或負值的速度梯度之剪切流流況，再利用電解(electrolysis)的方式產生微氣泡於流場中並擷取其影像來分析微氣泡的速度、直徑等參數。我們並運用質點影像測速法(Particle　Image Velocimetry, PIV)來量測剪切流的背景流場分佈。實驗顯示氣泡在具有正值的速度梯度之剪切流的上升趨勢小於在具有負值的速度梯度之剪切流，我們並建立一簡化的氣泡動力學理論來釐清造成此趨勢之關鍵物理機制。

關鍵詞：微氣泡，剪切流，速度梯度，氣泡動力學，質點影像測速法

Experimental on the Drag Reduction Technique in Towing Tank

Tsai Jing-Fa[1*], Chen Chi-Tran[1], Wang Yu-Ming[1], Chen Po-Fan[2]
[1]Department of Engineering Science and Ocean Engineering, National Taiwan University
[2]CSBC Taiwan, Kaohsiung, Taiwan.
[*]**E-mail: jftsai@ntu.edu.tw**

A porous plate micro-bubble injecting system and air/water mixture injecting system were designed with a flat plate drag measuring system to study the frictional drag reduction effect in towing tank. The diameter of the bubble generated by the air/water mixture system is about 0.08mm. And the diameter generated by the 10 micron porous plate micro-bubble injecting system is about 1.2mm. The test results show that the maximum drag reduction effect of the air/water mixture injecting system is only 7% for the limited airflow rate 2.5 liter per minute. The maximum drag reduction by the porous plate micro-bubble injecting system is about 31%. And the effect of the drag reduction increases with increasing air content and velocity.

Keywords: Micro-bubble, Drag Reduction

大會演講（2009/10/14　09:00~10:40）

船舶水动力学的若干研究进展

颜开
中国船舶科学研究中心
E-mail: yank@cssrc.com.cn

本文综合性地叙述了中国船舶科学研究中心近几年来在船舶水动力学领域在以下几方面取得的主要研究进展情况：（1）数值船舶水动力学；（2）新型推进器水动力学；（3）高速船水动力学。
关键词：计算流体力学; 高性能船; 船舶推进器。

從創新與整合談螺槳設計

黃正利 [1*]
[1] 財團法人聯合船舶設計發展中心
[*]**E-mail:** usddc@mail.usddc.org.tw

良好的螺槳設計為提升推進效率最重要的核心能力，而船舶中心發展許多螺槳設計工具，諸如圖表設計、理論設計，可以廣泛的應用於船用螺槳的開發。隨著對螺槳性能之需求日益增加，螺槳所遭遇之運作環境也日益嚴苛，因此近年來船舶中心致力於加強船舶核心技術研發及設計整合能力提昇，積極發展數值計算方法，搭配中心發展之計算工具與商用軟體解析螺槳流場以預測性能並協助改善設計缺失。在集合業者需求與意見後，著手發展適合應用於螺槳由設計至製造的開發平台，期望藉由整合各設計與計算軟體，以及中心近年來之 RANS 計算經驗，配合 3D 建模技術與製造端考量，使設計更能符合需求，可針對未來設計方案做初步預估以減少開發時程中各方面之消耗，避免可能發生的不良現象，更可以以此為基礎發展更新一代的船用推進螺槳。
關鍵詞：船用螺槳設計、製造整合、計算流體力學

Session D1 (2009/10/14　11:10-12:10)

新型 TLP 平台水动力性能预报

匡晓峰 [1*]**，缪泉明** [1]**，杨烁** [1]**，何再明** [1]
[1]中国船舶科学研究中心
[*]**E-mail:** Frank_kuang@yahoo.com.cn

张力腿平台（以下简称“TLP”）作为深水浮式平台的一种典型形式，与半潜式平台（SEMI）和深吃水浮筒式平台（SPAR）相比，能提供更好的平台运动性能，更稳定的生产操作条件, 特别是在水深 1000m～2000m 范围内，其经济优势更为明显，是优先选用的深水浮式平台型式。结合国内外新一代 TLP 平台形式的特点，提出了一种新型的 TLP 平台概念，希望通过圆盘压载，在保持 TLP 平台良好的运动和受力总体性能的基础上，有所改善，同时采用规则的外形构造，便于建造和海上拖运。通过三维势流理论，对平台的水动力特性，载荷以及运动响应进行了数值建模计算，分析结果表明，其运动响应较小，水动力性能良好。

湖泊水动力与底泥污染物释放的关系研究

王超，王沛芳[*]**，牛淮金，欧阳萍，李金**
河海大学浅水湖泊综合治理与资源开发教育部重点实验室，河海大学环境科学与工程学院
Email: pfwang2005@hhu.edu.cn

水动力对湖泊水体的作用，通过风生流、波浪等对水体特别是底泥产生作用，致使湖泊水环境中的

污染物质发生掺混、再悬浮等，强烈地影响着湖泊的水环境质量。湖泊底泥释放的磷为春季藻类爆发提供了重要的营养物质。当前部分学者开始重视水动力条件对的湖泊水环境中，已有的研究表明，太湖流域盛行的风能促使湖泊底泥中磷向水体中悬浮释放，而且不同季节和风速条件下，悬移和释放量各异。实际上，风的作用是通过引起湖泊水流流态和流速改变，从而致使湖泊表层底泥上浮。但是，风速是如何影响水流条件的，不同风速下浅水湖泊水流流速垂向分布规律如何，与表层底泥中污染物释放的相关性如何，还鲜有报道。

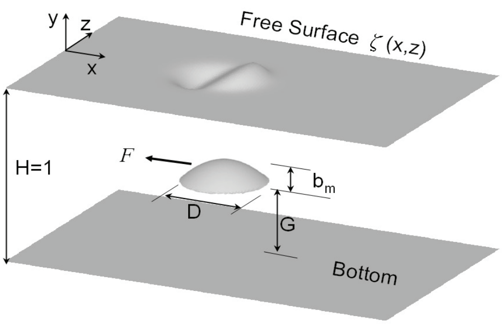
本文采用 10cm 厚的湖泊原状底泥，通过环形风动水槽的试验，研究了 3.38m/s、5.69m/s、7.56m/s 3 种风速条件下，水槽中水体流速的垂向分布规律及底泥中颗粒物含量和磷的悬浮过程，分析了不同深度水体中污染物质浓度的变化规律，并计算了由于风速的影响使得底泥 SS 和 P 向上覆水中释放的总量。上覆水采用自来水，水样观测点分别位于底泥以上 4cm、水表面以下 4cm、中间点 3 个位置。ADV 流速测点为底部以上和水面以下各 5cm 布设、中间每隔 2cm 设置一个测点。结果表明：(1) 测流的结果分析说明，不同强度的风速对水体产生了不同程度的流速效应。但是从总体规律来看，流速值在底部最小，之后逐渐增大，在靠近水面 10cm 左右处达到最大值，近水面处流速有减小趋势。并且，风速在 3.38m/s 时深度方向上各层流速值分布较散，表层流速值较小；而风速为 7.56m/s 时，底部和上部流速值点分布相对集中，与最大流速差异较小。(2) 分别在 3 个风速条件下，观测水体中不同深度水层水样中悬浮物浓度和磷的浓度值。取样时间为风机开始运行后的第 0h、0.5h、1h、1.5h、2.5h、4.5h、7.5h。结果发现：随着风动时间的增加，水体中各层 SS 含量均在增加，说明风浪对底泥上浮具有直接的作用；同时，由于流速对中上部水层影响最大，故中层水样中 SS 含量也较大，上层水体中 SS 含量最低；然而随着时间增加，风速扰动影响累积效应，使得中层和上层水体中 SS 含量增加显著，与底层含量差别减小。(3) 总磷的变化规律基本与 SS 含量的变化规律一致，上层水体中 TP 浓度最高、底层 TP 浓度最低。但是，由于最大风速试验中，水体中藻类的干扰，使得 TP 浓度出现随时间延长降低的异常现象。这也为今后更深入的研究提供了契机。(4)根据以上资料，计算各种风速强度下，水槽底泥中 SS 和 TP 释放总量，可知：3.38m/s、5.69m/s、7.56m/s 风速影响下，实验条件下 SS 的释放总量分别达到 98.1g、115.9g、138.8g，TP 释放量为 5.6mg、28.6mg、65.4mg。掌握湖泊水动力对水体 SS 含量和磷浓度的影响，有助于对太湖水体富营养化的认识及水质改善方法的研究。

臺灣四周海域之潮殘餘流探討

莊文傑[1*]，廖建明[2]
[1*]港灣技術研究中心，交通部運輸研究所
[2]臺灣颱風洪水研究中心籌備處，財團法人國家實驗研究院
[*]**E-mail: jye@mail.ihmt.gov.tw**

針對臺灣鄰近海域，本研究應用POM (Princeton Ocean Model)三維海洋環流數值模式，首先進行年期間的逐時潮流模擬計算，接著，配合月期間大、小潮的潮汐變化特性，分別計算其1 日、7日、15 日之Euler 及Lagrange 潮殘餘流(tidal residual currents)特性與型態，最後，評估其對海岸長期沖淤趨勢的影響。相關研究結果顯示：Euler 及Lagrange 潮殘餘流，型態與特性甚相近似；在臺灣西海岸，潮殘餘流主要係以和緩的流速，順沿海岸向北流動；在臺灣中西部外海，潮殘餘流明顯會以近於10 cm/s 的較大流速，順沿雲彰隆起水下大型沉積沙體地形，蜿蜒向北流動；在澎湖群島及臺灣灘等淺水海域，潮殘餘流流速普遍最強勁，同時具有順時針方向旋轉的大型環流型態。此外，與臺灣鄰近海域在水深50 米內所自然陳現的水下大型沉積沙體相對照驗證，可概略說明，潮殘餘流對漂沙物質的長期輸運作用，不僅與水下大型沙體的沉積形成有關，更與近岸淺水海域的長期輸沙優勢方向與海灘的自然沖淤趨勢相吻合。

關鍵詞： 潮殘餘流；潮流；海岸沖淤、水下沉積、臺灣海域



Session D2 (2009/10/14　11:10-12:10)

高速公路中央分隔带对厢式货车行车气动特性影响的研究

杜广生，李莉，李永卫，岳建雄
山东大学，能源与动力工程学院
Email: du@sdu.edu.cn

行驶汽车与中央分隔带之间存在相对运动，在以往对汽车高速行车气动特性的研究中，均未考虑中央分隔带对于汽车外流场的干扰作用，使模拟结果与真实情况存在偏差。本文把一种利用动网格技术进行非稳态计算的方法应用到行驶车辆非稳态气动特性的模拟计算中，研究了植株型中央分隔带对厢式货车单车行驶及尾随行驶气动特性的影响，通过与无中央分隔带时相同工况汽车气动特性的比较，得出结论：厢式货车阻力系数在有中央分隔带时较无分隔带时偏大一些；厢式货车周围流场在靠近分隔带一侧呈现较大负压，货车有向分隔带靠近趋势；中央分隔带的存在使尾随行驶两车的侧向力系数和升力系数都有所增加，影响了车辆的行驶稳定性，且后车所受影响较大。本文为汽车外流场数值模拟计算中是否要考虑中央分隔带等道路设施因素提供理论依据，对汽车的安全行驶有重要指导意义。
关键词：中央分隔带，非稳态气动特性，数值模拟，尾随行车

計算流體力學於風力發電機葉片氣動力性能計算之研究

郭真祥 [1] ，楊淳宇 [1*]
[1] 台灣大學　工程科學及海洋工程學系
[*]**E-mail:** newabner@yahoo.com.tw

風力發電機葉片的氣動力效率為影響風力發電機發電效率的主因，因此在設計時如何正確的評估葉片的氣動力性能便相當重要。葉片動量元素法[1]常被應用在評估葉片的氣動力性能，但該方法忽略流體沿徑向的運動，無法直接計算因為葉片尖端渦漩所造成之影響，會有高估之情況產生。因此本文以計算流體力學(CFD)方法對風力發電機進行模擬計算，以得到較好的評估結果。

首先對有限長度的翼形進行模擬計算，並與實驗[2]比對，結果顯示翼端的渦心軌跡與實驗相近。接著以美國國家再生能源實驗室(NREL)的風力發電機 PHASEII 為對象，利用模擬有限長度翼形之經驗進行模擬計算。

模擬計算所得之扭力落在實驗[3]量測之範圍內，且最大功率系數的發生位置都在風速為 12m/s 之下，與實驗值相吻合。最後利用模擬所得到在葉片表面的壓力分佈，計算此一負荷對葉片所造成的變形，並對變形後的葉片再進行一次模擬，結果顯示在扭力輸出和沿軸向的作用力皆會有所下降，但幅度不大。

本文利用 CFD 方法對有限長度翼形與風力發電機進行模擬計算，所得的結果能夠描述葉片尖端的渦漩現象，而風力發電機葉片扭力輸出也在實驗的量測範圍內，顯示 CFD 方法能夠有效評的估風力發電機的氣動力性能。
關鍵詞：風力發電機，翼形，計算流體力學(CFD)

The Method of Fundamental Solutions for Solving the Streamfunction-Vorticity Formulation of the Navier-Stokes Equations

Chia-Ming Fan
Department of Harbor and River Engineering, National Taiwan Ocean University
E-mail: cmfan@ntou.edu.tw

In this paper, the method of fundamental solutions is proposed for solving the streamfunction-vorticity formulation of the Navier-Stokes equations. For two-dimensional viscous incompressible fluid flow, the governing equations based on the primitive variables can be transformed to the streamfunction-vorticity formulation, which only includes two dependent variables. Therefore, by adopting this formulation, the simulation will become more efficient and cost less computer resources. The method of fundamental solutions is one of the promising meshless methods and expresses the numerical solutions by linear combination of the fundamental solutions and the radial basis functions. In comparing with the mesh-dependent methods, the time-consuming mesh generation and numerical integrations can be avoided by using the method of

fundamental solutions. Several numerical examples are adopted to demonstrate the efficacy and stability of the proposed meshless method for analyzing both two-dimensional steady and transient flow fields. In addition, several factors that might influence the numerical performance are examined in the numerical tests. From the numerical results, it's believed that the method of fundamental solutions can successfully solve the viscous incompressible fluid flow without mesh and numerical quadrature.
Keywords: method of fundamental solutions, streamfunction-vorticity formulation, Navier-Stokes equations, meshless method, radial basis functions

Session D3 (2009/10/14 11:10-12:10)

An Envisaged Inception Process for the New Type of Cavitation Identified from the Three Gorges Turbines

Shengcai Li
School of Engineering University of Warwick
E-mail: S.Li@warwick.ac.uk

An new type of cavitation has been identified from the Three Gorges turbines. These Francis turbines (710 MW) developed by world leading manufacturers are the world largest, reflecting cutting edge technology, but all developed a strange pattern of damage. This unknown phenomenon has puzzled professionals owing to its highly multi-disciplinary attributions. The worrying fact is that the manufacturers and turbine laboratories involved have all failed to predict this phenomenon either in their model tests or CFD simulations. Therefore, it is not an isolated technical issue but a universal and fundamental phenomenon, presenting a challenge to the community of hydrodynamic studies.

Multi-disciplinary analysis indicates that it is a new type of cavitation induced by boundary-layer turbulence production. A thorough study is being performed to verify its triggering mechanism. This research has now become the headline, see http://www.waterpowermagazine.com/home.asp and (*Innovation & Research Focus*, 75 NOV 2008) http://www.innovationandresearchfocus.org.uk. The outcome of this study is likely to make a great impact on the theoretical advance of fluid science as well as on many important applications. This paper is a brief update of the on-going studies performed at the Cavitaion Laboratory of Warwick University, focusing on possible inception process.

Firstly, the history of discovery of this cavitation damage will be briefly introduced, showing the complexity of this phenomenon. This damage occurs only on the foil's lower surface*** in the form of horizontal strips, starting from the favorable pressure gradient (FPG) zone extending into adverse pressure gradient (APG) zone, bearing heated signs. These damaged strips are approximately in the flow direction with span-wise regularities.

Secondary, based on the knowledge of both the statistic characteristics of nucleation in boundary layer and the streak growth, breakdown and turbulence production in boundary-layer, the envisaged inception process will be described.

Finally the author will explain why this process will produce such strange pattern of cavitation cavitation (damage).

***Here, the lower surface refers to the pressure side of a foil if the angle of attack is positive as conventionally defined in aerofoil aerodynamics.

非定常自然空泡流动现象研究

陈瑛，鲁传敬，郭建红，潘展程
上海交通大学 船舶海洋与建筑工程学院
Email: cyofjs@sjtu.edu.cn

本文基于所开发的空泡流动数值模拟方法和计算软件，研究了文丘里管内和绕水翼的非定常空泡流动现象及其机理。

对文丘里管内空泡流动现象的研究表明，不同的收缩－扩张角下分别产生准定常空泡形态和非定常空泡脱落。定常空泡大小对空化数敏感，大空化数下的流场压力脉动与准定常空泡振荡之间存在对应关系，且与非定常空泡的压力周期性振荡存在本质的不同；非定常空泡的周期性演化过程与实验现象相符，空泡长度、泡内速度和相分数分布、空泡脱落频率等与试验结果接近，压力振荡频率与空泡脱落频率一

致，表明了空泡非定常运动对流场结构产生的规律性影响。
对绕 NACA 翼型空泡流动现象的研究表明，湍流条件与层流条件下的空泡流动特性有较明显的差异：湍流空泡主体长度比层流的大很多，空泡脱落点明显后移，认为这是由于湍流边界层分离点比层流靠后的原因；湍流空泡结构更为复杂。对于空泡周期性断裂、脱落、下泄、溃灭过程中流场特性的分析表明，流场涡结构的变化与空泡脱落之间存在紧密的联系，云雾空化是具有大尺度涡结构的多相湍流流动，揭示了空泡非定常运动的一些机理问题。非定常空泡的时均升阻系数比定常空泡的略大，且振荡频率与空泡脱落频率一致，非定常过程的 Strouhal 数与试验结果相当接近。

水中單一空蝕氣泡之產生與受壓破裂流場的量測研究

趙勝裕[1*]**，楊昇學**[2]**，葉克家**[2]
[1] 國立台灣海洋大學　系統工程暨造船學系
[2] 國立交通大學　土木工程學系
[*]**E-mail:** syjaw@ntou.edu.tw

為了能仔細探討空蝕氣泡破裂流場的特性與氣泡破裂過程中、形成逆向噴流的原因，本研究以旋轉U型管的設備，利用離心力的作用，於轉軸中心位置產生單一空蝕氣泡，並以不同強度壓力波擊破位於固體邊界不同距離處之空蝕氣泡，藉由高速攝影機拍攝得的影像以分析氣泡破裂流場特性。實驗中發現，空蝕氣泡受壓、變形後，會於中心軸處產生噴流，此噴流突破氣泡的表面張力界面後，會產生Kelvin-Helmholtz vortex。當氣泡中心距離固體邊界的位置介於一倍至三倍氣泡半徑之間時，則於中心軸噴流突破氣泡表面張力界面後、會撞擊固體界面，產生滯流環。滯流環內部流體被往內推擠、於滯流環中心處形成逆向噴流。在氣泡中心距離固體邊界約為三倍氣泡半徑的臨界值位置，則可發現，當壓力波強度由小逐漸增大、氣泡破裂流場的特性會由只產生Kelvin-Helmholtz vortex、繼而出現Richtmyer-Meshkov instability 現象、與產生逆向噴流等三種流況。若氣泡中心距離固體邊界的位置小於一倍氣泡半徑，氣泡表面張力界面直接與固體邊界接觸，則氣泡中心軸噴流於撞擊固體界面後，無法形成滯流環，只能由內向外往輻射方向反彈、造成空蝕氣泡往輻射方向碎裂。各種複雜的空蝕氣泡破裂流場特性，都清楚的呈現於本研究中。
關鍵詞：空蝕氣泡、噴流、逆向噴流、Kelvin-Helmholtz 渦流

Session E1 (2009/10/14　13:20-14:40)

附加分离盘对雷诺数 1000 和 10000 情况下隔水管流动控制的数值模拟研究

王嘉松[1]**，蒋世全**[2]**，谭波**[3]**，谷斐**[3]
[1] 上海交通大学船舶海洋与建筑工程学院工程力学系
[2] 中国海洋石油研究中心
[3] 上海交通大学机械与动力工程学院动力机械及工程教育部重点实验室
[*] **E-mail:** jswang@sjtu.edu.cn

涡激振动可能导致海洋深水隔水管疲劳破坏，如何有效抑制隔水管涡激振动现象，正引起国际上相关科学与工程界的高度关注。本文利用计算流体力学研究在雷诺数 1000 和 10000 情况下附加分离盘对深水隔水管流动控制的效果。采用新建立的不可压缩湍流有限体积 TVD 算法，计算分离盘长与隔水管直径之比，L/D 在几种典型情况下的隔水管系统关键水动力学参数。与经典圆柱绕流的实验及数值模拟结果对比表明该算法具有较高的精度；分离盘对隔水管具有明显的升阻力降低及频率控制效果，在所给定的雷诺数和几何结构下，最大减阻效果可达 26%，升力在一定时间后几乎完全消失。
关键词： 隔水管；涡激振动；分离盘；TVD；流动控制

Comparison of Shallow-Water Equations and Boussinesq Equations for Wave Propagation over a Submerged Bar

Shin-Jye Liang*, **and Yuting Chen**
Department of Marine Environmental Informatics, National Taiwan Ocean University
***E-mail:** sjliang@ntou.edu.tw

Both numerical models based on non-dispersive nonlinear shallow-water equations (SWE) and dispersive Boussinesq equations (BE) are used to simulate a relatively long, unidirectional, non-breaking waves propagating over a submerged trapezoidal bar (Beji and Battjes, 1993). The SWE model is based on the least-squares finite-element formulation of the non-conserved shallow-water equations (Liang and Tsai, 2009); While BE model is based on the finite-difference approximations of Boussinesq equations with improved dispersive characteristics (Beji and Battjes, 1994). The Computed results are compared with experiments. Essential features of wave evolution are well reproduced in both models. Spectrum analysis shows that bound harmonics generation in the shoaling region (upslope) and their release (wave decomposition) in the deepening (downslope) region are clearly demonstrated in both models. However, due to the neglecting of effects of dispersion and vertical accelerations, predictions of SWE model are not as accurate as those of BE model.

Keywords: Shallow-Water Equations, Boussinesq Equations, dispersion, shoaling, de-shoaling, wave decomposition

Dynamic Coupling of Multi-phase Fluids with a Moving Obstacle

Mei-Hui Chuang [1*], Tso-Ren Wu [1], Chih-Jung Huang [2], Chung-Yue Wang [2], Chia-Ren Chu [2]
[1] Graduate Institute of Hydrological and Oceanic Sciences
[2] Department of Civil Engineering, National Central University
***E-mail:** 966205004@cc.ncu.edu.tw

The numerical simulation of water waves interacting with a moving solid body is important for many science and engineering applications, such as landslide generated tsunamis, caisson work, ship maneuvering, and wave energy. However, to analyze unsteady flows with free surfaces poses a great challenge to numerical simulations, because both of the free-surface boundary and the moving solid boundary are presented in the fluid system, and both elements are parts of the solutions.

In order to solve this problem, we coupled moving-solid algorithm and discrete element method (DEM) to simulate the dynamics of fluid-solid interaction. This two-way coupled model allowed solid bodies moving in the multi-phase fluid to generate realistic motion for both the fluid and solid objects. We discretised the Navier-Stokes equation using finite volume method (FVM) for the fluid part. The Volume-of-Fluid (VoF) method was used to track the fluid-fluid and fluid-solid interfaces. The moving-solid algorithm which conserved mass and momentum was adopted to transfer the force from solid motion to the multi-phase fluid. To predict the movement of the solid body, discrete element method model was adopted. The multi-phase fluid force acting on the solid bodies and solid surfaces were integrated to calculate the movement of the solid objects. By balancing forces, discrete element method projected the position of solid bodies. The moving-solid algorithm was implemented straightforward the solid objects could be rigid or deformable. The contact analyses among discrete bodies were also included in the simulation algorithm. The coupled model was then used to study the landslide generated waves and a floating cube. Detailed results will be presented in the full paper.

Keywords: Moving solid algorithm, Discrete Element Model (DEM), Volume of Fluid (VoF), Landslide Tsunami, Floating cube

Development of a level set method with better volume preservation to predict interface in three-dimensional two-phase flows

C. H. Yu[1], P. H. Chiu[1], Tony W. H. Sheu [1,2,3*]
[1] Department of Engineering Science and Ocean Engineering, National Taiwan University
[2] Taidai Institute of Mathematical Sciences (TIMS), National Taiwan University
[3] Center for Quantum Science and Engineering (CQSE), National Taiwan University
***Email :** twhsheu@ntu.edu.tw

A two-step conservative level set method is proposed in this study to simulate the gas/water two-phase fluid flow. For the sake of accuracy, the spatial derivative terms in the equations of motion for fluid flows are approximated by the coupled compact scheme. For accurately predicting the modified level set function, the dispersion-relation-preserving advection scheme is developed to preserve the true dispersion relation of the first-order derivative terms shown in the pure advection equation cast in conservative form. For the purpose of preserving its long-time accurate Casimir functionals and Hamiltonian in the transport equation for the level set function, the time derivative term is discretized by the sixth-order accurate symplectic Runge-Kutta scheme. To avoid contact discontinuity oscillations, a proper nonlinear compression flux term and an artificial damping term are added to the second-step equation of the modified level set method for sharply resolving the interface.

For the verification of the proposed dispersion-relation-preserving scheme applied in non-staggered girds for solving the incompressible flow equations, three benchmark problems have been chosen in this study. The conservative level set method with good area preservation proposed for capturing the interface in incompressible fluid flows is also verified by solving bubble rising in water problems. Good agreements with the referenced solutions are demonstrated for the investigated problems.
Keywords: conservative level set method; coupled compact scheme; dispersion-relationpreserving; Casimir; Hamiltonian; sixth-order accurate; symplectic Runge-Kutta; area preservation

Session E2 (2009/10/14 13:20-14:40)

螺旋桨叶片环量分布数值分析

洪方文　董世汤
中国船舶科学研究中心
Email: hongfangwen@sina.com

利用环量分布来控制桨叶的径向负荷是螺旋桨设计中一个重要的环节，然而很少通过模型试验来验证设计结果的环量分布。在试验方法中一般是测量尾流中的切向速度分布计算环量及应用 Stokes 定理来分析叶片上的环量分布，作者应用此方法对 CSSRC 用 LDV 测量过尾流场的一个螺旋桨进行了环量分布分析，同时也利用 Stuart Jessup 的测量结果对 DTMB4119 桨的环量分布进行了分析，均发现在外半径区环量分布有一突起的小峰(a hump)，因此对此产生疑虑。本文对上述二个推进器用 CFD 粘流数值手段做了流场分析计算，在此基础上采用二种方法进行环量分布分析。第一种方法即用上述传统方法分析，其结果也在外半径区环量分布有一突起的小峰，这说明小峰的出现并非流场试验测量误差所致，是分析方法所导致的。另一种方法，直接围绕桨叶各半径上叶剖面进行环量计算，其环量分布结果曲线光顺，上述小峰消失，说明小峰并非叶片环量分布实际如此。本文发表了上述分析结果，并对原因作出分析，分析了应用 Stokes 定理的传统分析方法导致误差的原因，提出应该直接环绕叶剖面计算分析环量。

螺旋桨尾流场 PIV 测量与分析研究

张军 [1]，洪方文 [1]，李广年 [2]，张国平 [1]，陆林章 [1]
中国船舶科学研究中心
浙江海洋学院船舶与建筑工程学院
*Email : zhangjuncssrc@sina.com

螺旋桨尾流场测量对于其水动力性能设计、空泡与噪声性能评估都具有重要意义。由于其自身的复杂性，长期以来一直是螺旋桨试验研究的难点。本文以 DTMB P4119 桨为研究对象，运用 2D-PIV 技术在大型空泡水筒中开展了均匀来流下螺旋桨尾流场测试研究，测量了螺旋桨不同相位下水平轴平面速度场，获得了螺旋桨尾流场轴向和径向速度分布。同时，针对同一对象在同一工况下开展了 LDV 与 PIV 的比对试验研究。比较结果表明，两者在流场宏观量和微观结构上都获得了很好的一致。本文还分析梢涡、尾涡的结构及其在尾流中的生成、演化情况，以及螺旋桨尾流场随进速系数的变化情况。试验研究表明，螺旋桨尾流场 PIV 测试技术具备定量获取螺旋桨尾流场复杂流动细观结构的能力，为螺旋桨设计研究和 CFD 计算验证提供了有效的试验技术手段。
关键词：PIV，螺旋桨，尾流场，梢涡，尾涡

Propulsion Calculation of Container Ship Equipped with Vortex Fin near Stern

Shiu-Wu Chau[1*], Jhih-Hong Fan[2], Hao-Hsiang Hsu[2], Po-Fan Chen[3] and Jyh-Yih Li[3]
[1] Department of Mechanical Engineering, National Taiwan University of Science and Technology
[2] Department of Mechanical Engineering, Chung Yuan Christian University
[3] Department of Design, CSBC Corporation
[*]**E-mail:** chausw@mail.ntust.edu.tw

The numerical study on propulsion prediction of a modern container vessel equipped with vortex fins installed near stern at model scale is discussed in this paper. The turbulent flow around ship is calculated by solving the Reynolds-averaged continuity and Navier-Stokes equations incorporated with an adequate turbulence model, where the double-model assumption is employed and the rudder effect is neglected. An effective body force model is employed to describe the propeller effect, where the propeller performance is calculated by a lifting-surface model. The correct rotational speed of propeller at ship's design speed is determined by the force balance condition, which gives a propeller thrust equal to the ship resistance. The propulsion calculation is then conducted to analyze the propulsion performance of the same hull form but with different vortex fin arrangements. The prediction obtained from the proposed method shows the required power is a combined result of the variation of resistance and propulsion efficiency due to the induced vortex created by vortex fins, which is validated by the corresponding experimental measurements. Revealed by the numerical calculations, the size, geometry, location and orientation of vortex fin could have substantial influences on the propulsion performance of studied container ship.
Keywords: Propulsion Calculation, Vortex Fin, Container, RANSE Computation, Body Force Approach.

Session E3 (2009/10/14　13:20-14:40)

船舶在尾随浪拍击下的模型试验与响应预报

邱强，杨大明，骆寒冰，万正权
中国船舶科学研究中心　江苏科技大学
E-mail: qiuqiangone@163.com

某些船舶有时需要在特定的海况中尾随浪航行，可能发生尾随浪拍击，引起船体抖动，造成部分设备不能正常工作，甚至个别设备损坏，工作人员的能力不能正常发挥。

第 23 届 ITTC（2002 年）的载荷和响应委员会，在对将来研究的建议和要求中指出，现代客轮和 Ro-Pax 船舶，经常在其肥大型艉部受到砰击载荷作用，这是一个令人头痛的问题，建议开展艉部砰击的研究。

Kapsenberg (2003)对一艘豪华游轮进行了艉部砰击载荷和振动响应的模型试验研究。在试验中，发现该游轮在尾随浪零航速时会出现严重的艉砰击现象。提出了一种用试验测量艉砰击压力场的方法，在**刚性船模**的艉底部布置了大量压力传感器，用来测量砰击压力场的时间和空间分布，再通过积分方法得到了砰击力。本文介绍**分段弹性船模**在尾随浪航行下的砰击载荷测量概况，主要介绍 1、在水池条件下再现船舶模型尾随浪砰击的抖动现象及船艉压力，船舯弯矩及运动测量结果。2、探讨船舶在尾随浪航行时的船体艉砰击合成弯矩幅值的概率预报方法.

球形液滴衝擊自由液面之數值模擬

廖清標[1]，洪梓銘[2*]，葉忠訓[2]
[1]逢甲大學　水利工程與資源保育學系教授
[2]逢甲大學　水利工程與資源保育學系碩士生
[*]**E-mail:** m9607775@fcu.edu.tw

液滴撞擊液面後觀察液面的流場情形，在以往大都利用攝影圖像來量化並且分析其流況。本文利用數值計算方法來模擬球形液滴以自由落體高速衝擊自由液面後之流場運動情形。此流場屬於自由液面流問題，也因為牽涉到兩流體間之自由液面，如水與空氣，就會使得在移動液面的處理上具有挑戰性。本研究中以不可壓縮 Navier-Stokes 方程式和等位函數法，在簡單均勻卡式網格下利用有限差分來求解，其

在空間有二階精確度，在時間項離散則採用二階 Adams-Bashforth 法來做。由於追蹤包含表面張力之兩流體界面需要求高精確，故利用五階 WENO 法對 Level Set 方程式求解，時間項則採用三階 Runge-Kutta 法。數值模式的驗證，已經由陳(2004)驗證二維和三維 Zalesak 問題，模式的運用效果不錯。等位函數法在模擬兩相流問題已經相當成熟，利用來計算液滴以自由落體高速衝擊自由液面時，自由液面所產生的合併及射流等現象。模式採以 Liow(2001)的實驗設置與數據作為參考，將整個流場進行無因次化即可計算並模擬其流況。在低衝擊速度下時可以觀察到受到撞擊之液面形成孔穴及合併的過程；逐漸提高衝擊速度可以發現液面孔穴形成時間拉長及產生射流與皇冠狀液面的現象。觀察模擬的流場運動情形，並與實驗結果相比較，其現象大致上是符合。
關鍵詞：球形液滴，自由液面，等位函數法，孔穴，皇冠狀液面

Flow Induced by Relatively Moving Plates

Chi-Min Liu
General Education Center, Chienkuo Technology University
E-mail: cmliu@ctu.edu.tw

A theoretical study on flow induced by relatively moving plates is presented. The viscous fluid over a horizontal plate is initially at rest, and is driven by a suddenly moving half-plate with another plate being still at all times. Several mathematical techniques and integral transforms are used to solve the exact solution. The so-called exact solution is capable of capturing not onlythe steady-state phenomenon, but also the transient solution. Applications of present solutions are rather wide. For example, the earthquake-induced flow by two relatively moving plates can be well understood by using present solutions.
Keywords: Relatively moving plates, viscous flow

Initial Stage about Three-DimensionalWaves Generated by a Submerged body moving in water

Chih-Hua Chang
Department of Information Management, Ling-Tung University
***E-mail:** changbox@mail.ltu.edu.tw

The theme of this article is to develop a three-dimensional fully-nonlinear water-wave model for an object moving in water with various speeds (such as subcritical flow, critical flow and supercritical flow). The transient boundary-fitted grid is built to fit the moving boundaries. The Laplace equation for potential-function and complete nonlinearity in boundary conditions are solved through a finite-difference algorithm. The whole computational domain is to observe a chopping spheroid with height $m\ b$ =0.15 (in dimensionless form based on the reference length of still water depth) as well as a bottom-base radius of D=10 moving with a distance G off bottom in the water with respect to a fixed frame (see Figure 1). The three-dimensional wave generation from a submerged moving body can be simulated successfully. To compare it with a two dimensional case, it can be obtained that good agreement results are presented.When G=0, a few various Froude numbers (F) are computed to distinct the free-surface evolving patterns in three dimensional situation (see Figure 2). Especially, the three cases of F (F=0.5, 1.0, 1.5) are emphasized. Furthermore, the F (F=0.5, 1.0, 1.5) in G=0, 0.2, 0.3, 0.4 are also studied to take count of the G effect. The results show that the G effect is weak for supercritical flow, whereas, it appears strong influence on critical flow and subcritical flow.
Keywords: 3D nonlinear wave, moving body, wave generation

Session F1 (2009/10/14 15:00-16:20)

钱塘江出海码头水域水动力条件数值模拟

鲁海燕，潘存鸿，韩海骞，吴辉
浙江省水利河口研究院
Emil: Luhy@zjwater.gov.cn

钱塘江河口潮强流急、涌潮汹涌、主槽摆动频繁，且纵剖面上存在一个长达130km的沙坎，这三个因素制约了港口、码头的发展。钱塘江河口经过近50年的综合治理，目前尖山河段旧仓至曹娥江口河段南汊主流一直存在，且萧山廿二工段前沿南岸水深比断面平均水深大一倍左右，为这一带开发港口资源带来了契机。本文建立了平面二维涌潮数值模型，并经过多次水文实测资料验证，计算分析了拟建港区的水动力情况。为码头工程的选址和可行性研究提供了科学依据。
关键词： 钱塘江；码头；水动力条件；数值模拟

Tsunami Dispersion Effect from Manila Trench to Taiwan

Dong-Jheng He [1*]，Tso-Ren Wu [1]
[1] Graduate Institute of Hydrological and Oceanic Sciences, National Central University
[*]**E-mail:** 966205006@cc.ncu.edu.tw

Manila Trench had been identified by USGS as a high-risk earthquake zone. An earthquake of magnitude 9 or higher could be generated (Huang et al., 2008) and followed by gigantic tsunamis. The tsunami arrival time from the Luzon source region to Taiwan ranges from 20 to 40 minutes only. With such a short warning time, a fast and accurate numerical prediction is the key for hazard mitigation. Tsunami simulation is a common method in tsunami researches. Models based on shallow water equation (SWE) are even popular and useful in tsunami hazard prediction. Ignoring dispersion effect, without high-order partial derivative terms, SWE can be solved explicitly and requires no iterations. However, the effect from dispersion, ignored in SWE, might plays a role when tsunamis enter the complex shallow water area. Taiwan, connecting to Eurasian Plate and Philippine Plate, is surrounding by the complex bathymetry. Boussinesq model, on the other hand, is able to calculate the dispersion effect, but requires interaction,

In this discussion, both SWE and Boussinesq based numerical models, COMCOT and COULWAVE, are performed in the same earthquake-tsunami scenario. Manila Trench is divided into 33 sub-faults (Megawati et al., 2008) for the worst case scenario, a magnitude 9.0 earthquake. The comparisons focus on the wave height, arrival time, and wave period, analyzing with Fast Fourier transform. The results show that further the tsunami travels, stronger the dispersion effect becomes. The predicted arrival time is about 2 minutes earlier without the dispersion terms. Then dispersion effect becomes strong in terms of wave period and wave height after the leading tsunami waves. The detailed analysis is presented in this paper. It is concluded that Non-dispersive SWE model is a suitable choice for tsunami warning, however from the research point of view, an accurate simulation which includes most of the physical phenomena is suggested.
Keywords: Taiwan tsunami, dispersion, COMCOT, COULWAVE, Manila

Experimental Investigations of Tide Effect on Coastal Groundwater Table

Wu Longhua[1*], Zhuang Shui-ying [2]
[1] Center for Eco - Environmental Modeling, Hohai University
[2] Pearl River Water Resources Scientific Research Institute
[*]**E-mail:** jxbywlh2000@yahoo.com.cn

Coastal groundwater table fluctuates with the sea tide. The groundwater table fluctuation will directly affect sediment transport, seawater intrusion, and substance movement. During flood tides, seawater will intrude into the unconfined aquifer and lead to raise of local the groundwater table. During the ebb tide, the groundwater will expel from coastal unconfined aquifer, and the seepage face is engendered to influences the sediment transport in the beach. Based on the research of Grant (1948), when the groundwater table is higher than the average sea water level, the beach is easier eroded. Contrarily, if the groundwater table is lower than the average sea level, the sediment easy silting. Otherwise, in being mathematical model of groundwater, the sea level

fluctuates of tide is usually ignored. The average sea level is used to the boundary condition of mathematical model merely. Because of the periodically fluctuates of sea level, the mean period's groundwater table in unconfined aquifers is higher than the still groundwater table in seacoast (abbreviated as over height) (Philip J R. 1973, Nielsen P1990, Li L , Barry D A, et al 2000, Ataie-Ashtiani B , et al 2001). When the tide's swing is 4~5m., the over height can be attained 2~3m.If the over height is ignored, forecast the groundwater resource gross will be brought error. In this article, based on the experimental observation, study on effect of tide for coastal groundwater table fluctuations.

To simulate the natural tidal, a tide simulation system based on two-way water pump technique was designed and development. It is consisted of mechanical tide generating system and automatic control system. Adopted serial communication between industrial control computer and controller to remote control the test process real time. All the experimental apparatus are under the control of the control center with telecommunications. The sand flume's length is 30m, its width is 1.2m, and height is 1.5m. The system maximal amplitude of tide is 0.25m. The experimental results indicated that, Everywhere in the seacoast, groundwater table fluctuations are asymmetric, and they spread much farther than the falling phase.

Amplitude attenuation of groundwater table fluctuations is quickening as the onshore distance increases. The mean period's groundwater table in unconfined aquifers is higher than the still groundwater table in seacoast (i.e. over height). The largest over height has reached about 10% for the aquifers thickness.

The swing and frequency of tide are very important factors for the over height of coastal groundwater table. The frequency of tide does more impact to over height than the swing of tide under similar conditions. And the same time, the over height relates to the aquifers thickness too.

Keywords: Tide simulation system, Coastal, Groundwater table, Over height

海嘯湧潮對近岸結構物之影響

魏妙珊 [1*]，吳祚任 [1]
1 國立中央大學　水文與海洋科學研究所
***E-mail:** 966205001@cc.ncu.edu.tw

本文以數值模擬研究潰壩湧潮，並分析其與位於結構物之交互作用。分析得知，湧潮撞擊結構物時，將於結構物前產生兩個方向相反且尺度與湧潮臨前深度相仿之漩渦。結構物初始受力最大處位於臨前湧潮高度，證實主破壞力為動壓力。

關鍵詞：湧潮，浪與結構物之交互作用，海嘯，潰壩

Session F2 (2009/10/14　15:00-16:20)

Prediction of Fully Nonlinear Wave Loads on Ships by CIP based Cartesian Grid Method

Changhong Hu
RIAM, Kyushu University, Fukuoka
E-mail: hu@riam.kyushu-u.ac.jp

Strongly nonlinear free surface problems in ship and ocean engineering include violent sloshing of a liquid tank, slamming, water on deck, wave impact by green water and capsizing of ships or ocean structures in rough seas. Numerical simulation of such strongly nonlinear problems is still a challenging subject. The major difficulty is that the topology of free surface may be largely distorted or broken up, which makes it impossible to apply the conventional numerical method such as potential flow solver by BEM. In this paper a new CFD method for the strongly nonlinear seakeeping problems, the CIP based Cartesian grid method, will be introduced. The CIP (Constrained Interpolation Profile) algorithm is adopted as the base scheme. The CFD method has been developed for years in RIAM, Kyushu University, to make it not only be able to handle complicated flow phenomena but also be relatively simple in scheme which can perform three-dimensional simulations in an acceptable spatial and temporal resolution at reasonable cost. Recent obtained CFD results on a modern container ship model which are running at constant forward speed in regular waves will be presented. The frequency response characteristics of the wave-induced motions and various wave loads, such as the vertical bending moment and the shear force on three cross-sections of the ship, are compared to the experiments and the results obtained by linear-theory based numerical methods.

Keywords: CIP based Cartesian grid method, Strongly wave-body interaction, Nonlinear Wave Loads

串列式雙振動翼推進性能探討

邱逢琛 [1,2]　**張政傑** [3]
[1] 國立台灣大學工程科學及海洋工程學系 教授
[2] 國家實驗研究院台灣海洋科技研究中心 主任
[3] 國立台灣大學工程科學及海洋工程學系 研究生
E-mail: g41010943@gmail.com

本研究為了開發不干擾水下滑翔機滑翔，且能於需要時協助水平推進之助推器，乃模仿魚類推進機制，提出串列式雙振動翼助推器之設計，並應用計算流體力學套裝軟體 Fluent，以動態網格進行非穩態流場模擬，進行二維串列式雙振動翼的流體動力分析，探討振動翼之振幅、相位、頻率、速度及翼板間距等參數對於推進性能之效應，及其產生推力之力學機制。串列式雙振動翼助推器包含做為前緣渦產生器(Leading Edge Vortex Generator)功能的前振動翼、渦操控器(Vortex Manipulater)功能的後振動翼，以及介於二者間的固定翼。本研究藉由數值計算分析探討的結果將作為後續水槽試驗規劃之依據，最終將能建立可作為開發仿生助推器之分析工具，並促成具水平移動能力的水下滑翔機的實現。
關鍵詞：水下滑翔機、振動翼、助推器

耦合液舱晃荡的船舶运动预报

邹　康 [1]**，缪泉明** [1*]
[1] 中国船舶科学研究中心
[*]**Email:** qmmiao@yahoo.com

带液舱船舶（如 LNG 船）的运动会激起液舱的晃荡，反过来液舱晃荡又影响着船舶的运动。因此，要准确预报带液舱船舶的运动响应，以及液舱自由面运动对舱壁的冲击载荷，就必须要考虑液舱晃荡和船舶运动的耦合效应。本文研究耦合液舱晃荡的船舶运动响应问题。采用时域内的三维面源法来求解船舶运动问题，液舱晃荡采用商业 CFD 软件同步进行求解。液舱晃荡诱导的力和力矩作为外力加入运动方程中求解，同时把求解得到的运动作为晃荡的外部激励输入进行液舱晃荡的模拟。计算结果和一艘搭载一个矩形液舱的 S175 船水池试验数据进行了比较，吻合较好。
关键词： 船舶运动；晃荡；耦合效应

船舶作横荡和首摇运动数值模拟研究

韩　阳，吴宝山，潘子英
中國船舶科學研究中心
Email: hanyang702@yahoo.com

CFD是预报操纵性水动力系数的最为广泛的手段之一；对纯横荡和纯首摇运动的水动力预报而言，PMM试验可能是目前最可靠的方法。随着计算流体力学技术的不断发展，基于RANS的CFD已经可以有效地应用于模拟这些复杂的操纵运动。本文以ITTC推荐标模KVLCC2为研究对象，采用非定常RANS方程求解船舶的操纵运动，研究船舶作纯横荡和纯首摇运动时的水动力及水动力系数。

Session F3 (2009/10/14　15:00-16:20)

Density/Viscosity Blockage Method for the Viscous Flowswith Complex Immersed Interfaces

Decheng Wan
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University

Immersed boundary problems have important applications in a variety of physical and engineering areas such as flows around high-rise buildings, fluid-structure interaction, multiphase flows, stratified flows, bubble dynamics, melting and solidification, crystal growth, etc. The numerical investigation of these physical problems has to take into account the effect of the immersed boundaries, especially may require extremely to

regeneration or deformation of the grid when the immersed interfaces or boundaries are moving. Generally speaking, there are two main methods to simulate fluid flows with complex immersed boundaries. One is body-conformal approach which always keeps mesh line fitting with the immersed body boundaries. Another one is fixed grid approach in which the mesh is fixed and the immersed solid boundaries are allowed to move freely through the mesh.

In this paper, we present a density/viscosity blockage method based on the fixed grid approach. In the density/viscosity blockage method, we use infinity values of density or viscosity to replace a solid body which originally stands in the flow field. The discontinuities of density/viscosity across the immersed interface are smoothed and kept at every grid level for multigrid procedure. The influence of the boundary is transmitted to the fluid through source terms in the transport equations. The incompressible Navier-Stokes equations are discretized using nonconforming finite element technique, stabilization techniques for convective terms via streamline diffusion is adopted, the fractional-step-θ -scheme and adaptive time step control are employed for time advancement. The moving blockage area or immersed boundary can be easily reformed by using an Eulerian-Lagrangian approach. For simplification, in this paper we do not take interface tracking procedure, instead the velocity values of evolution of the interface is specified in advance.

Four numerical examples by using the proposed density/viscosity blockage method are presented: 1) Flow around single or multiple circular cylinders in channel; 2) Flow around an oscillating cylinder in channel; 3) Flow generated by a heating plate in an enclosure box; 4) Oscillating rotation of a heating plate in an enclosure box. The numerical examples illustrate the presented density/viscosity blockage provides a robust and efficient approach to simulate the flows with complex immersed interfaces.

Flow characteristics of a pair of rotating side-by-side circular cylinders

Farida R Purnadiana, Dedy Z Noor*, Ming-Jyh Chern
Department of Mechanical Engineering, National Taiwan University of Science and Technology
* **E-mail: D9503802@mail.ntust.edu.tw**

Numerical investigation of the characteristics of two-dimensional flow around two rotating circular cylinders in a side-by-side arrangement in which the upper cylinder rotates counterclockwise and the lower cylinder rotates clockwise is presented in this paper. In order to investigate the effects of the rotation and the gap between two cylinders, numerical simulations are performed using the immersed boundary method at various rotational speeds, , for three different dimensionless gaps (*g**) of 3, 1.5 and 0.7 at Reynolds number 100. For the case with *g** = 0.7 drag coefficient increases as increases. The wake behind those cylinders changes from flip-flopping pattern to in-phase synchronized pattern at = 2.0. For the case with *g** = 1.5 the wake changes from the in-phase synchronized pattern to anti-phase synchronized pattern at = 1.5. For the case with *g** = 3.0 drag coefficient is inversely proportional to and the wake is still in the anti-phase pattern. For all the range of gaps lift coefficient is proportional to .

Keywords: rotating cylinders, side-by-side, immersed boundary method, vortex shedding.

The method of fundamental solutions for the Multi-dimensional wave equations

D. L. Young[1*], M. H. Gu[1] and C. M. Fan[2, 3]
[1] Department of Civil Engineering and Hydrotech Research Institute, National Taiwan University
[2] Department of Harbor and River engineering, National Taiwan Ocean University
[3] Computation and Simulation Center, National Taiwan Ocean University
***E-mail: dlyoung@ntu.edu.tw**

In this paper, a meshless method is developed to solve the multi-dimensional wave equations. The proposed method is based on the method of the particular solution (MPS), the method of fundamental solutions (MFS) and the Houbolt method. The wave equation is considered as the Poisson-type equation with the time-dependent loading. The Houbolt method is used to avoid the difficult problems of dealing with the initial conditions for forming the linear system. This paper considers three numerical examples, such as the wave vibration and wave propagation problems. Numerical validations have proven that the proposed method is a highly efficient numerical tool for solving wave equations in engineering and sciences.

Streamwise dynamics controlled jet spreading, mixingand physical source of the vortices

Amalendu Sau and Robert R. Hwang*
Institute of Physics, Academia Sinica
***Email:** phhwang@gate.sinica.edu.tw

Within the framework of the present investigation we perform Direct Numerical Simulations to study the role of streamwise dynamics in the jet spreading and mixing, and provide a clear understanding about the source of these important vortices in two-step rectangular sudden expansion flows. The present setup is observed to generate higher mass entrainment, and the system generated passive forcing provided the necessary impetus for the sustained growth/evolution of the streamwise vertices, which played the key role in the mixing enhancement.

On the other hand, by suitably placing two tiny rectangular `tabs' on the inlet channel walls, the nature of growth and the dynamics of the streamwise extended three-dimensional vortical rollers could be effectively controlled. These vortices through their inflow/outflow type dynamics are found to effectively control the downstream jet spreading; and the associated physical processes either lead to quick axis switching of the jet section, or stop axis switching altogether. In addition, an extensive pressure analysis, as presented here, suggests that the transverse pressure gradient skewing is probably the major source of streamwise vorticity generation for the flow. With the help of simulated transverse pressure distribution over the channel, we establish here a physical mechanism that efficiently predicts inception and dynamics of all the streamwise vortices. Our pressure analysis even successfully identifies every local change in streamwise dynamics during downstream evolution.