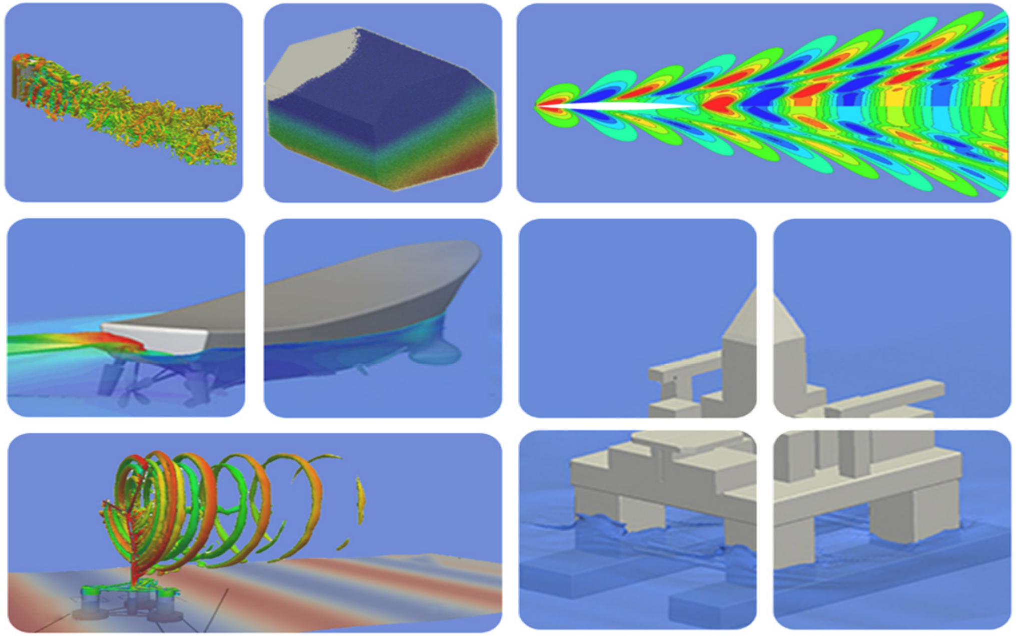
上海交通大学
Shanghai Jiao Tong University

船舶海洋与建筑工程学院
School of Naval Architecture, Ocean & Civil Engineering

海洋工程国家重点实验室
State Key Laboratory of Ocean Engineering

高新船舶与深海开发装备协同创新中心
Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration



联系方式 / CONTACT US

上海市闵行区东川路800号木兰船建大楼
Dongchuan Road 800, Minhang District, Shanghai

邮编/Postcode : 200240
电话/Tel : (+86) 21-34207971
传真/Tax : (+86) 21-34207991
电邮/Email : dcwan@sjtu.edu.cn
网址/URL : http://dcwan.sjtu.edu.cn

上海交通大學
SHANGHAI JIAO TONG UNIVERSITY

船海计算水动力学研究中心

CMHL COMPUTATIONAL MARINE HYDRODYNAMICS LAB
SHANGHAI JIAO TONG UNIVERSITY

CMHL COMPUTATIONAL MARINE HYDRODYNAMICS LAB
SHANGHAI JIAO TONG UNIVERSITY

目录
Contents

船海计算水动力学研究中心简介
Introduction to CMHL

上海交通大学船海计算水动力学研究中心（CMHL）成立于2006年。CMHL研究中心针对高端海洋装备设计数值化、精细化、智能化和系统综合的需求，瞄准学术前沿，始终致力于船舶与海洋工程计算水动力学的先进数值方法研究，高性能计算软件的自主研发，以及高新船舶、海洋装备、深海潜器、海洋能源和空间开发利用中高效CFD数值模拟的应用研究。CMHL研究中心近五年发表国内外学术论文500余篇，承担十多项国家和省部级的科研基金项目，拥有二十多项专利和软件著作权，多人次荣获国内外重要奖项。针对船舶与海洋工程结构物智能优化设计需求，CMHL研究中心自主研发了船舶与海洋工程水动力学综合计算软件仿真系统，包括：船舶与海洋工程水动力学求解器naoe-FOAM-SJTU，深海立管涡激振动求解器viv-FOAM-SJTU，深海浮式平台涡激运动求解器vim-FOAM-SJTU，海上浮式风机求解器FOWT-UALM-SJTU，船型优化求解器OPTShip-SJTU，水动力学无网格粒子法求解器MLParticl-SJTU，流固耦合求解器MPSFEM-SJTU等。十余年来，CMHL研究中心培养了一大批优秀毕业生，目前在读博士研究生17人，硕士研究生24人。

The Computational Marine Hydrodynamics Laboratory (CMHL) of Shanghai Jiao Tong University (SJTU) was established in 2006. CMHL is devoted to do the research of state-of-the-art, high fidelity and next generation CFD methods, to do the development of integrated CFD solvers for marine structures design under the framework of digitization, refinement, intelligence and system synthesis, as well as to do the applications of efficient and reliable CFD investigations on advanced and innovated ship, marine structures, ROV, offshore renewable energy, etc. CMHL has published more than 500 academic research papers, and has more than 20 patents and software registered certificates in recent five years, many members have won important awards at home and abroad. The research achievements have been closely followed and highly recognized by domestic and foreign counterparts. CMHL owns high performance computing (HPC) clusters and independent server room, and has developed the integrated CFD solver system, aiming at improving the virtual design for new concept of marine structures. The solver system includes CFD solver for marine hydrodynamics, naoe-FOAM-SJTU, fluid-structure interaction solver for vortex-induced vibration of deep-sea risers, viv-FOAM-SJTU, CFD solver for vortex-induced motions of floating platforms, vim-FOAM-SJTU, aero-hydrodynamic solver for floating offshore wind turbines, FOWT-UALM-SJTU, solver for ship hull optimization, OPTShip-SJTU, meshless particle CFD solver for marine hydrodynamics, MLParticle-SJTU, FSI solver based on meshless particle and finite element method, MPSFEM-SJTU, etc. In recent years, a lot of talents were trained and graduated from CMHL. Currently there are 17 doctoral students and 24 master students at CMHL.

负责人：万德成教授
Leader of CMHL: Professor Decheng WAN

万德成教授，上海交通大学船舶海洋与建筑工程学院教授，博士生导师，教育部长江学者奖励计划特聘教授，上海东方学者(跟踪计划)特聘教授，上海交通大学特聘教授，上海市优秀学术带头人，上海浦江人才基金获得者，教育部新世纪优秀人才计划入选者，英国Strathclyde大学兼职教授，大连理工大学兼职教授。上海交通大学科学技术发展研究院副院长，先进技术与装备研究院常务副院长，船海计算水动力学研究中心(CMHL)主任。

万德成教授主要研究方向为船舶与海洋工程水动力学，计算流体力学的基础理论及其应用，船舶与海洋工程数值水池研究与开发，基于CFD船型与海洋结构物优化设计，新概念与绿色海洋结构物水动力性能预报，海洋浮式风机平台耦合动力分析及海上新能源开发装置水动力性能分析，水下潜水器与仿生流体力学分析，立管涡激振动和平台涡激运动分析，非线性波浪载荷、波浪爬高、上浪拍击，内外流耦合与液舱晃动，平台运动响应、气隙，以及重叠网格overset grid技术，无网格SPH和MPS方法，高性能并行计算与GPU技术等。已发表论文500余篇，其中SCI和EI收录150余篇，自主研制开发了船舶与海洋工程非定常粘性流动求解软件系统naoe-FOAM-SJTU和MLParticle-SJTU，获国家软件著作权16项。主持负责过国家自然科学基金课题、国家863课题、工信部高技术船舶创新专项课题、教育部博士点基金等多项国家科研课题。 2017年和2014年两次获上海交通大学TOP1%优异学士论文指导老师奖；2016年获上海市高校精品课程奖和上海交通大学教学成果一等奖；2016年被评聘为上海交通大学二级教授；2016年、2013年、2010年三次被评为上海交通大学优秀教师。

目前担任国际离岸与极地工程学会(ISOPE)执委会委员、ISOPE水动力学委员会主席、技术委员会委员(TPC)，国际船模拖曳水池会议(ITTC)顾问委员会委员，国际船舶CFD会议指导委员会委员，全球华人计算力学学会常务理事，韩国KAIST大学海洋系统工程系国际咨询委员会委员，军委科技委GF科技海军水面平台技术专业组专家，中国造船工程学会船舶力学学术委员会CFD学组副组长，中国海洋学会军事海洋学专业委员会委员，中国力学学会计算力学专业委员会新型数值方法及应用专业组委员，上海市船舶与海洋工程学会设计专业学术委员会副主任，上海力学学会理事，上海市欧美同学会留英分会副会长。现担任6个国际杂志《Ocean Engineering》,《Applied Ocean Research》,《Journal of Ocean and Wind Energy》,《Journal of Shipping and Ocean Engineering》,《International Journal of Naval Architecture and Ocean Engineering》,《Journal of Marine Science Research and Oceanography》,《Current Trends in Oceanography and Marine Sciences》编委，以及8个国内杂志《Journal of Hydrodynamics》(副主编)，《Journal of Ocean Engineering and Science》(副主编),《水动力学研究与进展》(执行编委),《海洋工程》,《船舶力学》,《中国舰船研究》,《力学季刊》,《Journal of Marine Science and Applications》编委。

Prof. Decheng Wan received his Ph.D from Shanghai Jiao Tong University (SJTU), China in 1994. He became a lecturer of Shanghai University in 1994, and was promoted to be an associate professor of Shanghai University in 1996. After successively worked as a research fellow of the Royal Society at University College London, UK, a senior research fellow at National University of Singapore, and a Wissenschaftliche Angestellter at Dortmund University, Germany, he returned to Shanghai and was appointed as a full professor of Shanghai Jiao Tong University in 2006. He was selected as a distinguished professor of Shanghai Eastern Scholar in 2008, and promoted as a chair professor of Chang Jiang Scholar of China in 2014, and distinguished professor of Shanghai Jiao Tong University in 2015. Currently, Prof. Wan is Vice Director of Office of Research Management, Acting Director of Office of Advanced Technology Research, Head of Computational Marine Hydrodynamics Laboratory at SJTU.

Prof. Wan’s research interest is mainly on Computational Marine Hydrodynamics, Simulation Based Design for Offshore and Polar Structures, Renewable Energy in Deep Sea, numerical marine basin, nonlinear wave theory, wave loads on structures, numerical analysis of riser vortex-induced vibration (VIV) and platform vortex-induced motion (VIM), fluid-structure interaction, offshore wind turbine and other offshore renewable resources, meshless method , as well as high performance computation on complex ship and ocean engineering flows, etc. In these areas, he has published over 500 papers and carried out more than 30 projects on marine hydrodynamics and computational hydrodynamics.

Prof. Wan is Board of Directors and Chair of International Hydrodynamic Committee (IHC) of International Society of Offshore and Polar Engineering (ISOPE), Member of Advisor Committee of International Towing Tank Conference (ITTC), Member of Energy Saving Method Specialist Committee and CFD Specialist Committee of International Towing Tank Conference (ITTC), Member of Steering Committee of CFD Workshops in Ship Hydrodynamics, Standing Council Member of Association of Global Chinese Computational Mechanics, Member of External Advisory Committee (EAC) of the Department of Ocean Systems Engineering (OSE) of Korea Advanced Institute of Science and Technology (KAIST).

Prof. Wan is associate editor of Journal of Hydrodynamics, Journal of Ocean Science and Engineering, as well as member of editorial board of Ocean Engineering, Applied Ocean Research, Journal of Ocean and Wind Energy, Journal of Shipping and Ocean Engineering, International Journal of Naval Architecture and Ocean Engineering, Journal of Ship Mechanics, Journal of Marine Science and Applications, Journal of China Ship Research as well as Journal of Chinese Quarterly of Mechanics.

团队成员
Team Members

王建华
博士，助理研究员

Jianhua WANG
Ph.D.
Research Associate

研究方向
- 船舶计算水动力学
- 船桨舵相互作用
- 波浪中船舶操纵

Research Fields
- Computational fluid dynamics
- Ship hull-propeller-rudder interaction
- Ship maneuvering in waves

科研情况
王建华博士主要从事船舶计算水动力学、基于重叠网格技术的船桨舵相互作用分析、波浪中船舶操纵运动数值模拟等方面的研究，并参与完成了上海交通大学CMHL研究中心多个船舶与海洋工程计算水动力学软件的开发工作。负责和参与了多项船舶水动力性能预报的基金项目。王建华博士曾多次参加船舶与海洋工程水动力学领域顶级的国际会议，并作交流报告。共发表学术论文30余篇。

Research achievements
Dr. Jianhua Wang is mainly engaged in the research of CFD study of ship hull-propeller-rudder interaction and ship maneuvering in waves based on overset grid method. Dr. Wang was one of the main developers of the in-house CFD solvers in CMHL of SJTU. In addition, he has participated in several projects related to ship hydrodynamics. He has attended and given many presentations at international conferences, such as SNH, IWWWFB, ISOPE, etc. He has published more than 30 journal and conference papers.

邹璐
博士，讲师

Lu ZOU
Ph.D.
Lecturer

研究方向
- 计算流体动力学方法在船舶与海洋结构物水动力问题中的应用
- 限制水域船舶操纵性
- 数值计算不确定度量化分析

Research Fields
- Application of CFD methods in ship and ocean engineering problems
- Ship maneuvering in restricted waters
- Uncertainty quantification in CFD simulations

科研情况
邹璐博士现为上海力学学会青年工作委员会副主任、中国造船工程学会船舶力学学术委员会操纵性学组秘书。主要研究方向为计算流体动力学方法在船舶与海洋结构物水动力问题中的应用，特别在限制水域船舶操纵性预报、数值计算不确定度量化分析等方面取得了一系列研究成果。曾主持了国家自然科学基金青年科学基金等项目，并作为核心成员参与了工信部数值水池创新专项等课题；曾获上海市力学学会优秀青年学者奖。

Research achievements
Dr. ZOU is currently the deputy director of Youth Affairs Committee in Shanghai Society of Theoretical and Applied Mechanics, the secretary of Ship Maneuverability Group of Academic Committee on Ship Mechanics, the Chinese Society of Naval Architects and Marine Engineers. Her main research interests include ship maneuvering and navigation in restricted waters, uncertainty quantification in numerical simulations, etc. She has presided over projects supported by the National Natural Science Foundation of China, and participated as a core member in the innovation specific project of numerical tank funded by the Ministry of Industry and Information Technology.

团队成员
Team Members

CMHL中心拥有硕士和博士研究生40多名，其中博士研究生约20名。他们来自上海交通大学、大连理工大学等国内船海专业著名高校，有着踏实全面的科研能力和杰出的创新能力。CMHL中心为研究团队提供了丰富的高水平交流机会，包括前往知名企业和院所考察，参加国内外高水平学术会议，邀请国内外知名专家学者来中心交流。每年，从CMHL中心毕业的研究生达数十名，他们不仅去往船海知名研究所，还广泛就业于风电、IT、金融等行业。

CMHL center has more than 50 master students and doctoral studentss, among which about 20 doctoral candidates. They come from Shanghai Jiao Tong University, Dalian University of Technology and other famous universities, and have practical and comprehensive scientific research ability and outstanding innovation ability. CMHL center also provides the research team with abundant high-level communication opportunity, including visit to famous enterprises and research institutions, high-level academic conference, and invitation of well-known experts and scholars around the world to CMHL center. Each year, more than ten students graduate from CMHL center. They not only go to famous research institutes in ship and ocean engineering, but also work in wind power, IT, finance and other industries.

CMHL主要成员
Members of CMHL

CMHL中心的交流活动
Communication activities of CMHL

研究方向
Research Directions of CMHL

上海交通大学船海计算水动力学研究中心（CMHL）针对高端海洋装备设计数值化、精细化、智能化和系统综合的需求，瞄准学术前沿，致力于船舶与海洋工程计算水动力学的先进数值方法研究，高性能计算软件的自主研发，以及高新船舶、海洋装备、深海潜器、海洋能源和空间开发利用中高效CFD数值模拟的应用研究。

Computational Marine Hydrodynamics Lab (CMHL) is devoted to do the research of state-of-the-art, high fidelity and next generation CFD methods, to do the development of integrated CFD solvers for marine structures design under the framework of digitization, refinement, intelligence and system synthesis, as well as to do the applications of efficient and reliable CFD investigations on advanced and innovated ship, marine structures, ROV, offshore renewable energy, etc.

上海交大CMHL团队研究方向
Research Directions of CMHL at SJTU

高性能数值计算方法研究
Advanced and High Performance CFD Methods for Marine Hydrodynamics

先进CFD方法研究 State-of-the-art CFD Methods	动态重叠网格方法 Dynamic Overset Grid Methods	多级物体运动求解 Complex 6DoF Motion System with Hierarchy of Bodies	高雷诺数分离涡方法 Detached Eddy Simulations	湍流转捩模型 Turbulence Transition Models
高效CFD方法研究 High Fidelity CFD Methods	多分辨率粒子方法 Multi-resolution Particle Methods	动态负载平衡技术 Dynamic Load Balancing	GPU并行加速技术 GPU Computing	异构众核并行技术 Many-core Computing
下一代CFD方法研究 Next-generation CFD Methods	自适应直角网格方法 Adaptive Cariestian Methods	无网格粒子方法 Meshless Particle Methods	多种数值方法耦合 Multi Numerical Methods Coupling	多相系统耦合求解 Multi-System Interactions Computing

船海计算水动力学计算软件自主研发
Development of Integrated In-House CFD Solvers for Marine Hydrodynamics

基于OpenFOAM平台的软件开发 Software Development based on OpenFOAM	无网格粒子法求解器开发 Development of Meshless Particle Method Solvers	船型综合优化软件开发 Development of Hull Form Optimization Softwares
用户界面开发技术 Development of Graphic User Interfaces	跨平台软件集成技术 Cross Platform Software Integration Techniques	基于人工智能的CFD软件开发 CFD Development based on Artificial Intelligence

船舶与海洋工程高性能CFD方法和软件应用研究
Applications of Efficient and Reliable CFD Analysis Tools on Marine Hydrodynamics

船舶工程领域 / Naval Architecture

高精度阻力与兴波预报 Resistance and Wave-making Prediction	高速船舶首破波研究 Breaking Bow Waves of High-speed Ships	复杂海况下船舶运动和波浪增阻预报 Sea-keeping and Wave Added Resistance in Rough Sea	甲板上浪和砰击预报 Green Water and Slamming
船舶节能装置精细化预报 Prediction of Energy Saving Devices	基于NM理论线型高效优化 Hull Form Optimization based on Potential Theory	基于CFD船舶线型综合性能优化 Multi-objective Optimization for Ship Hull by CFD	船桨舵相互作用 Ship Hull-Propeller-Rudder Interaction
约束船模操纵运动预报 Simulations of Captive Ship Model Tests	静水和波浪中的船舶自航和操纵 Self-propulsion and Maneuvering in Calm Water and Waves of Free Running Ship	水下潜器水动力性能预报 Prediction of Underwater Vehicle Hydrodynamic Performances	多体船性能预报 Prediction of Multi-hulls Hydrodynamics Performances

海洋工程领域 / Ocean Engineering

数值造波与数值消波 Wave Generation and Absorption	极端波浪与非线性波浪载荷 Extreme Waves and Nonlinear Wave Loads	系泊系统与浮式结构物耦合分析 Coupled Numerical Analysis of Mooring System and Floating Structures	液舱晃荡研究 Liquid Tank Sloshing
粘流与势流耦合研究 Coupling of Viscous and Potential Flow	深海立管涡激振动分析 Vortex-Induced Vibrations of Risers	深海浮式平台涡激运动分析 Vortex-Induced Motions of Floating Platforms	多浮体间相互作用 Interactions between multiple floating structures
带液舱的LNG船运动响应 Motion Response of LNG with Liquid Tanks	MPS-FEM流固耦合研究 Fluid-Structure Interaction based on MPS-FEM	单点系泊FPSO模拟分析 Simulation of Single-point Mooring FPSO	物体出入水问题研究 Water-entry-exit Problems

海洋新能源领域 / Offshore Renewable Energy

风机及风场数值模拟 Numerical Simulations of Wakes flows of Wind Turbines and Wind Farms	浮式风机耦合模拟 Numerical Modeling of Wind-Wave-Current Interactions for Floating Offshore Wind Turbines	气动-水动-系泊系统全耦合求解方法 Coupled Aero-Hydro-Mooring Analysis Method	波浪能装置模拟 Simulations of Wave Energy Converter

计算资源

Computational Resources of CMHL

上海交通大学船海计算水动力学研究中心拥有由CPU+GPU组成的异构高性能计算集群和独立机房。集群目前拥有118台节点，其中CPU节点108台，GPU节点2台，IO节点4台，管理节点2台，高性能工作站2台，CPU数量达到2432核，存储能力为630TB。2块NVIDIA K40 GPU，4块NVIDIA P100 GPU，节点之间使用千兆以太网及FDR 56Gbps InfiniBand网络高速互联。机房配备有动态环境监测、气体消防、精密行间空调以及大功率UPS。

The high performance computing (HPC) cluster of computational marine hydrodynamics laboratory (CMHL) at Shanghai Jiao Tong University is heterogeneous high performance computing system composed of CPU and GPU. The HPC cluster now has 118 nodes, including 108 CPU nodes, 2 GPU nodes, 4 IO nodes, 2 management nodes and 2 high performance workstations. The total number of CPU cores is 2432, and the storage size is 630TB. Two NVIDIA K40 GPU and four NVIDIA P100 GPU are used. The nodes are inter-connected by Gigabit Ethernet and FDR 56Gbps InfiniBand high-speed network. The HPC server room is equipped with dynamic environmental monitoring system, gas fire control system, precision air conditioner and high power uninterrupted power supply (UPS).


高性能计算集群
HPC cluster


计算节点
Computing nodes


行间精密空调
Precision air conditioner


不间断电源
UPS


气体消防柜
Gas fire control cabinet

设备名称 Device Name	详细配置 Specifications	数量 Number
CPU Node	CPU: 2×Intel Xeon E5520 (4 Cores, 2.26GHz) Mem: 24GB (2GB×12) DDR3 1066MHz	16
	CPU: 2×Intel Xeon E5-2680 v2 (10 Cores, 2.80GHz) Mem: 64GB (8GB×8) DDR3 1866MHz	34
	CPU: 2×Intel Xeon Gold 5120 (14 Cores, 2.20GHz) Mem: 128GB (16GB×8) DDR4 2666MHz	58
GPU Node	CPU: 2×Intel Xeon E5-2680 v2 (10 Cores, 2.80GHz) Mem: 64GB (8GB×8) DDR3 1866MHz GPU: 2×NVIDIA Tesla K40	1
	CPU: 2×Intel Xeon E5-2650 v4 (14 Cores, 2.20GHz) Mem: 128GB (16GB×8) DDR4 2400MHz GPU: 4×NVIDIA Tesla P100	1
Management Node	CPU: 2×Intel Xeon E5520 (4 Cores, 2.26GHz) Mem: 24GB (2GB×12) DDR3 1066MHz	1
	CPU: 1×Intel Xeon E5-2640 v4 (10 Cores, 2.40GHz) Mem: 64GB (16GB×4) DDR4 2400MHz	1
IO Node	CPU: 2×Intel Xeon E5520 (4 Cores, 2.26GHz) Mem: 24GB (2GB×12) DDR3 1066MHz	1
	CPU: 2×Intel Xeon E5-2609 v2 (4 Cores, 2.50GHz) Mem: 48GB (8GB×4 + 4GB×4) DDR3 1600MHz	1
	CPU: 2×Intel Xeon E5-2690 v4 (14 Cores, 2.60GHz) Mem: 128G (16GB×8) DDR4 2666MHz	2
High Performance Workstation	CPU: 2×Intel Xeon E5-2677 (14 Cores, 3.2GHz) Mem: 256GB (32GB×8) DDR4 2400MHz Video Card: NVIDIA Quadro M6000 24GB	2
Parallel Storage	IBM Storwize V3700 150TB	1
	Dell PowerVault MD3460 480TB	1
Precision Air Conditioner	Envicool XR040 40KW 600mm(W)*1000mm(D)*2000mm(H) Inverter compressor EC fans	4
UPS	Schneider G3HT40K 40KVA 500mm(W)*860mm(D)*1300(H)	1

自主研发软件
In-house Softwares

上海交通大学船海计算水动力学研究中心（CMHL）针对船舶与海洋工程结构物智能优化设计需求，自主研发了船舶与海洋工程水动力学综合计算软件仿真系统，包括：船舶与海洋工程水动力学求解器naoe-FOAM-SJTU，深海立管涡激振动求解器viv-FOAM-SJTU，深海浮式平台涡激运动求解器vim-FOAM-SJTU，海上浮式风机求解器FOWT-UALM-SJTU，船型优化求解器OPTShip-SJTU，水动力学无网格粒子法求解器MLParticl-SJTU，流固耦合求解器MPSFEM-SJTU等。

The Computational Marine Hydrodynamics Lab (CMHL) has developed the integrated CFD solver system, aiming at improving the virtual design for new concept of marine structures. The solver system includes CFD solver for marine hydrodynamics, naoe-FOAM-SJTU, fluid-structure interaction solver for vortex-induced vibration of deep-sea risers, viv-FOAM-SJTU, CFD solver for vortex-induced motions of floating platforms, vim-FOAM-SJTU, aero-hydrodynamic solver for floating offshore wind turbines, FOWT-UALM-SJTU, solver for ship hull optimization, OPTShip-SJTU, meshless particle CFD solver for marine hydrodynamics, MLParticle-SJTU, FSI solver based on meshless particle and finite element method, MPSFEM-SJTU, etc.

船舶与海洋工程水动力学求解器
CFD Solver for Marine Hydrodynamics
naoe-FOAM-SJTU

海上浮式风机求解器
Aero-hydrodynamic Solver for Floating
Offshore Wind Turbines FOWT-UALM-SJTU

船型优化求解器
Ship Hull Optimization
Solver OPTShip-SJTU

船舶与海洋工程
水动力学综合软件仿真系统
Integrated CFD Solver System

流固耦合求解器
FSI Solver Based on Meshless
Particle and Finite Element
Method MPSFEM-SJTU

深海浮式平台涡激运动求解器
CFD Solver for Vortex-Induced Motions of
Floating Platforms vim-FOAM-SJTU

深海立管涡激振动求解器
Fluid-Structure Interaction Solver for Vortex-Induced
Vibrations of Deep-sea Risers viv-FOAM-SJTU

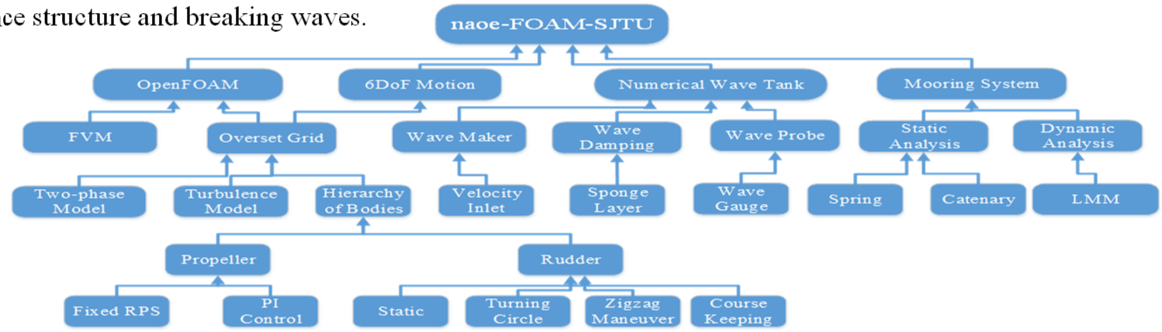
无网格粒子法求解器
Meshless Particle CFD Solver for Marine
Hydrodynamics MLParticle-SJTU

研发团队
Research team

船舶与海洋工程水动力学求解器 naoe-FOAM-SJTU
CFD Solver for Marine Hydrodynamics naoe-FOAM-SJTU

针对船舶及海洋工程的典型应用问题，CMHL研究中心开发了基于OpenFOAM的求解器naoe-FOAM-SJTU。求解器包含六自由度运动模块，系泊系统、重叠网格模块及数值造波模块，可用于模拟在多种海洋环境下船舶及海洋结构物在海洋环境中的大幅度复杂运动。求解器成功的应用于预报船舶静水阻力，耐波性，船舶操作性能的预报。求解器可用于大规模并行计算，在CMHL超算中心的支持下，实现了对湍流结构，波浪破碎等精细物理结构的模拟仿真。

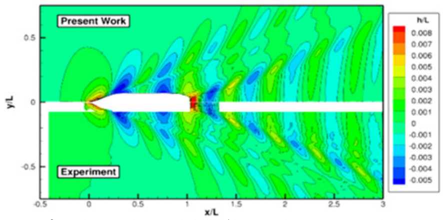
CMHL developed naoe-FOAM-SJTU solver based on the open source platform OpenFOAM to solve typical hydrodynamic problems of ship and marine engineering. The solver is mainly consist of a 6DOF motion module, a mooring system module, a 3D numerical wave tank module and a dynamic overset grid module. It can be used to simulate the large and complex motion of ships and marine structures in various sea environment. In addition, the developed numerical wave tank module can be used to generate typical regular and irregular waves. The solver has been successfully applied to the prediction of ship resistance, seakeeping, propulsion and maneuverability. With the support of CMHL Supercomputing Center, the solver can carry out large-scale parallel computations to simulate the detailed flows such as turbulence structure and breaking waves.



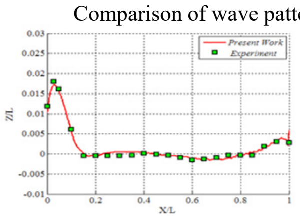
船舶静水阻力预报
Prediction of Ship Resistance

船舶静水阻力计算涉及到对船体兴波特性、船体边界层及尾流场的预报，阻力的计算精度达到5%以内。

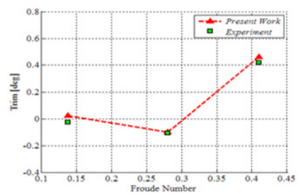
The prediction of Ship resistance includes the simulation of wave pattern, boundary layer distribution and wake field. The prediction error is less than 5%.



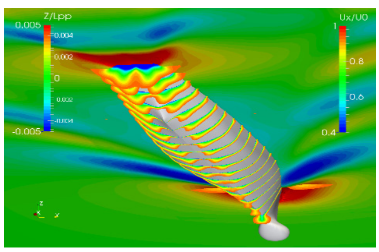
船体兴波数值模拟 实验 VS. 数值
Comparison of wave pattern measurement VS. calculation



船体湿表面计算与实验结果对比图
Comparison of wet surface position of the hull. Measurement VS. Calculation



船体姿态计算与实验结果对比图
Comparison ship's trim Measurement VS. Calculation

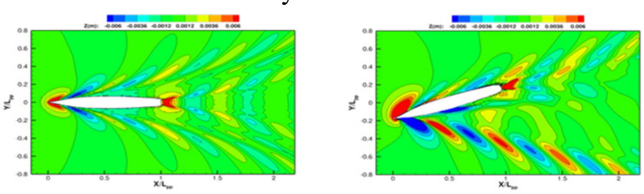


船体表面边界层数值模拟
Numerical simulation of boundary layer distribution

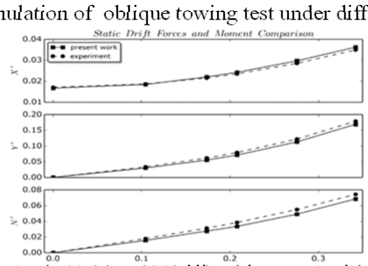
平面运动机构试验数值计算
Simulation of Planar Motion Mechanism

数值斜拖试验可以测定出船体在不同漂角下的侧向力和转首力矩，从而根据操纵性数学模型给出对应的水动力导数值。

The lateral force and moment of the ship under different drift angles can be determined by the numerical oblique towing test, then corresponding hydrodynamic derivatives are obtained according to the mathematical model of maneuverability.



不同漂角的斜拖数值模拟
Numerical simulation of oblique towing test under different drift angles

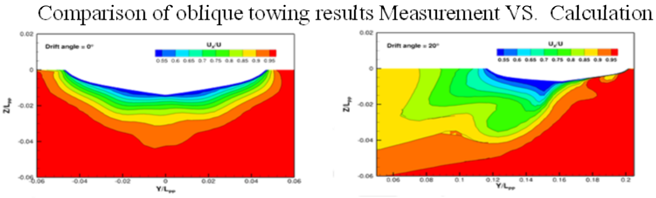


阻力
Resistance

侧向力
Lateral force

转首力矩
Yaw moment

不同漂角的斜拖数值模拟结果与实验结果对比
Comparison of oblique towing results Measurement VS. Calculation



0°漂角工况的尾部伴流
The wake under zero drift angle

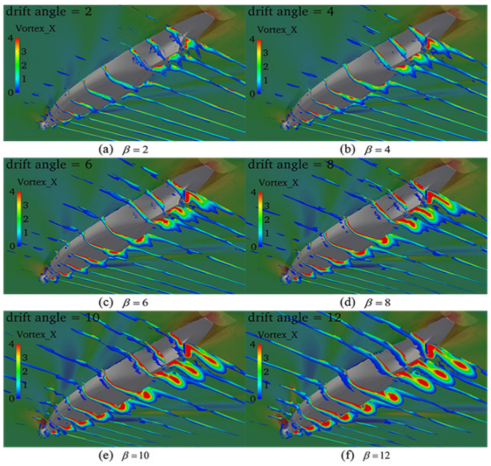
20°漂角工况的尾部伴流
The wake under 20 degree drift angle

naoe-FOAM-SJTU 在预报船舶性能中的应用

Applications of naoe-FOAM-SJTU in Prediction of Ship Performance

斜拖实验中，随着漂角的增加，伴流的分布呈现出非常明显的不对称性，同时伴随着强烈的泻涡现象。

With the increase of the drift angle, the distribution of wake shows obvious asymmetry, accompanied by a strong vortex shedding.

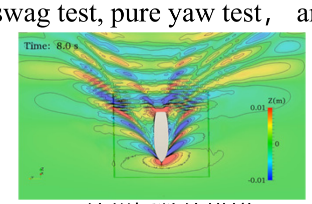


不同漂角下的斜拖实验中船体边界层的结构

Structure of ship boundary layer in oblique towing test under different drift angles

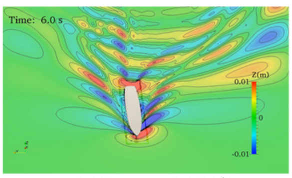
动态约束船模试验数值模拟是给定不同自由度方向上的振荡运动，研究运动船体下的水动力特性。根据不同的运动形式，动态约束船模试验可以分为纯横荡试验、纯摇首试验等。

The numerical simulation of dynamic captive ship model test is used to study the hydrodynamic characteristics of moving ship under the oscillating motion of different freedom. According to the freedoms of motion, dynamic captive ship model test can be divided into pure swag test, pure yaw test， and so on.



纯艏摇数值模拟

Numerical simulation of pure yaw



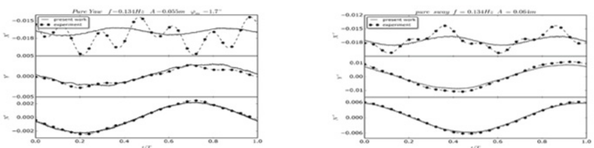
纯横荡数值模拟

Numerical simulation of pure sway

Hydrodynamic derivative	CFD	EFD	Error
$Y_{\dot{v}}'$	3.39e-2	3.75e-2	-9.82%
$N_{\dot{v}}'$	4.11e-3	4.71e-3	-12.6%
Y_v'	4.81e-2	5.85e-2	-17.8%
$Y_{v\|v\|}'$	0.271	0.304	-10.6%
N_v'	5.07e-2	5.35e-2	-5.27%
$N_{v\|v\|}'$	2.16e-2	2.69e-2	-19.6%

水动力导数 数值 VS. 实验

Comparison of hydrodynamic derivatives Calculation VS. Measurement



纵向、横向水动力及转艏力矩计算 VS. 实验（左：纯艏摇 右：纯横荡）

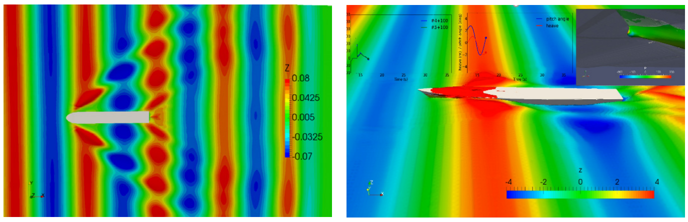
Comparison of the lateral and vertical hydrodynamic force and yaw moment (left: pure yaw, right: pure sway) Calculation VS. Measurement

船舶耐波性预报

Prediction of Seakeeping performance

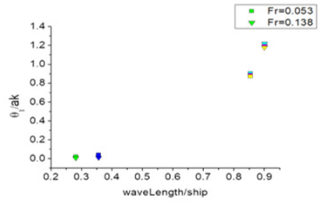
船舶耐波问题的核心是造波和船体运动。求解器的造波模块，可模拟高阶规则波和多种不规则波。

The key point of seakeeping problem is wave generation and ship motions. The wave generation module in the solver can generate high order regular waves and irregular waves.



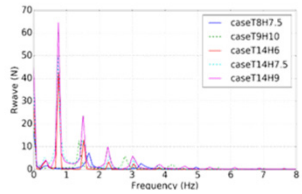
船体在波浪中的兴波

Wave-making of ship in the wave



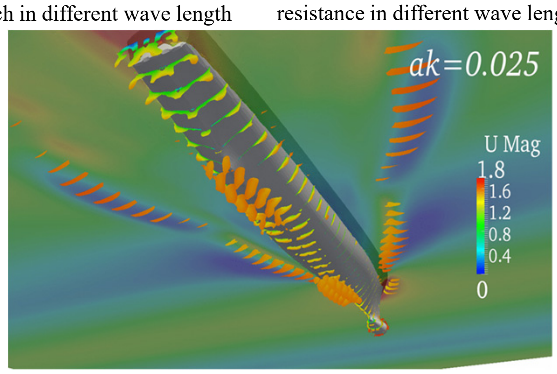
不同波长下纵倾一阶响应幅值

The first harmonic amplitude of pitch in different wave length



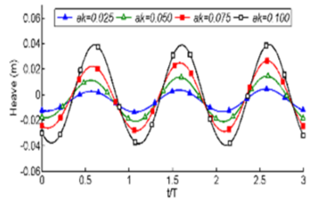
不同波长下阻力的高阶幅值

High order harmonic amplitude of added resistance in different wave length



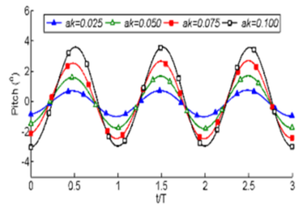
船舶在波浪中运动而产生的泄涡

Vorticity distribution around the hull in waves



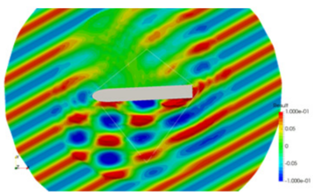
垂荡幅度时历曲线

Time history of heave amplitude



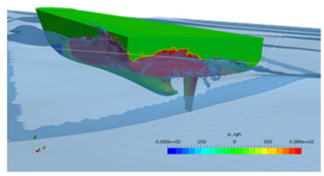
纵摇角度时历曲线

Time history of pitch angle



斜浪中波浪的辐射和衍射

The radiation and diffraction of waves in a oblique wave



斜浪中的尾部砰击现象

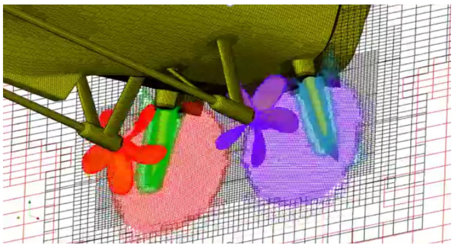
The tail slamming in a oblique wave

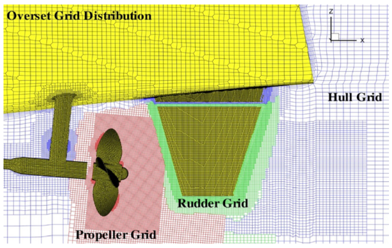
naoe-FOAM-SJTU 在自航船舶中的应用

Application of naoe-FOAM-SJTU for Free Running Ship

通过引入重叠网格模块，naoe-FOAM-SJTU求解器可应用于复杂的多级物体运动计算，并首次在OpenFOAM平台上实现了船桨舵复杂运动的直接数值模拟。求解器结合PI控制器，可以实现螺旋桨和舵不同形式的控制。求解器目前已经成功的应用于静水和波浪中的船舶自航、航向保持、Z形操纵以及自由回转等典型问题。

naoe-FOAM-SJTU can be used to simulate complex motion of ship hull-propeller-rudder system through implementing overset grid. The solver is for the first time ever to accomplish the simulation of free running ship in OpenFOAM. Combined with overset grids, the PI controller is applied to realize the self-propulsion control of propeller and rudder. The solver has been successfully applied to self-propulsion, course-keeping, zigzag maneuver and turning circle maneuver in both calm water and regular waves.



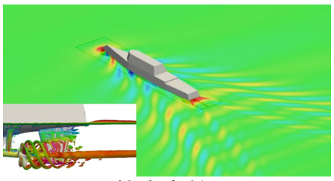


船桨舵周围重叠网格布置
Overset grid distribution around ship hull, propellers and rudders

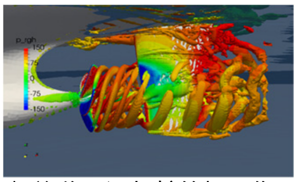
船舶在静水中的自航
Self-propulsion in calm water

对于自航，求解器可以精确预报自航点，精确捕捉尾部船-桨-舵相互作用。

The self-propulsion point can be accurately predicted and the interaction of hull-propeller-rudder can be well resolved.



静水自航
Self-propulsion in calm water

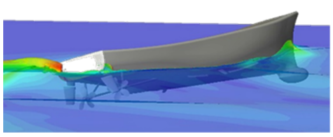


螺旋桨尾涡与舵的相互作用
Interaction between tail vortex and rudder

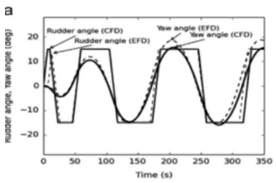
船舶在静水中的操纵
Ship maneuvering in calm water

求解器可进行典型的Z形操纵和自由回转操纵运动的直接数值模拟，并准确预报操纵运动特征参数、船舶六自由度运动以及水动力变化。

The present solver can be used to directly simulate standard zigzag and turning circle maneuvers. The main parameters of maneuverability, 6DoF motions and hydrodynamic forces can be well predicted.

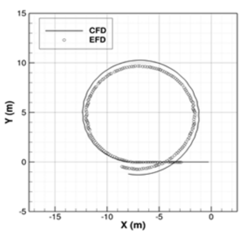


Z形操纵运动
Zigzag maneuver in calm water



舵角以及航向角的变化
Variation of the yaw and the rudder angle

自航回转
Turning circle maneuver in calm water

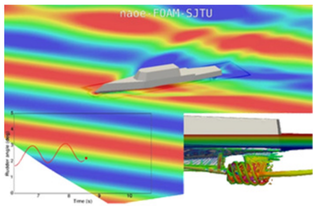


回转轨迹对比
Trajectory comparison of turning circle

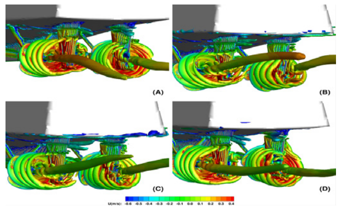
船舶在波浪中的自航
Self-propulsion in waves

结合求解器中的造波模块，可以完成波浪工况中的船舶自航数值模拟，结合航向控制器，同时可以完成航向保持的数值模拟。

Self-propulsion in waves can be directly simulated combining with wave tank module. Meanwhile, free running ship in waves under course keeping control can also be simulated through a heading control module.



首斜浪中的船舶航向保持
Course-keeping in waves

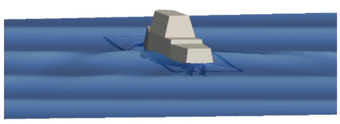


一个波浪周期尾涡结构变化
Vortical structures in a wave period

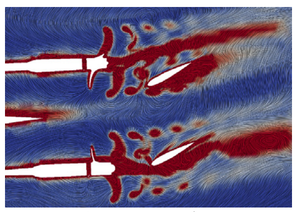
船舶在波浪中的操纵
Ship maneuvering in waves

对于波浪中的Z形操纵问题，可以准确预报波浪对操纵运动特征参数、螺旋桨推力及舵力的影响。而对波浪中的船舶回转模拟，则可以预报出回转降速、轨迹漂移等复杂现象。

The wave effects on main parameters of zigzag maneuver as well as hydrodynamic forces acting on propeller and rudder can be accurately predicted. The speed loss and drift of turning trajectory can be observed through the present simulation of turning circle maneuver in waves.



波浪中的Z形操纵运动
Zigzag maneuver in waves



螺旋桨尾流
Wakes behind the ship propeller

naoe-FOAM-SJTU 在精细流场模拟中的应用

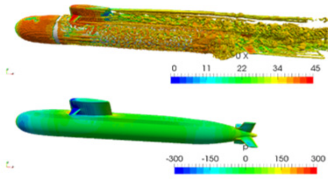
Applications of naoe-FOAM-SJTU in High Resolution Simulations

潜艇围壳湍流结构

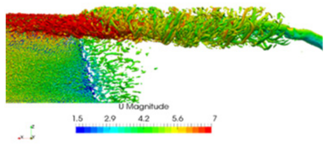
Turbulence structure of submarine sail

利用LES等高精度计算方法，精确捕捉潜艇围壳周围涡结构和表面压力分布，预报了噪声的强度。

LES and other high-precision calculation methods are used to accurately capture the vortex structure and surface pressure distribution around the submarine and predict the noise intensity.



潜艇周围涡结构及表面压力分布
Vortex structure and surface pressure distribution of submarine



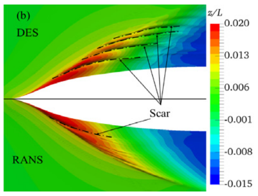
潜艇围壳涡结构
Vortex structure of submarine sail

KCS艏波破碎模拟

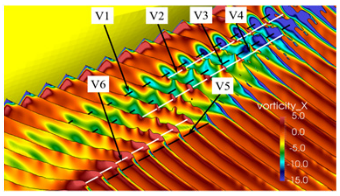
Simulation of bow wave breaking of KCS

波浪破碎机制复杂，对船体阻力性能影响显著。课题组利用OpenFOAM中的高精度VOF模型，预报了高速KCS船的艏波破碎。

The mechanism of wave breaking is complex, which greatly influences the ship resistance. High resolution VOF scheme in OpenFOAM is used to predict the bow wave breaking of high-speed KCS.



KCS船艏破波波模拟
Bow wave breaking of KCS



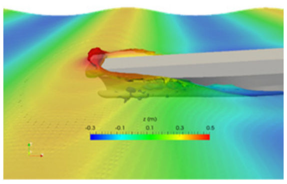
KCS船艏涡量场
Bow vorticity of KCS

波浪砰击

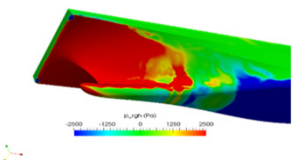
Wave slamming

波浪砰击具有明显的非线性特征。求解器准确的预报了在极限海况下的船艏上浪和瞬时抨击力。

Wave slamming has nonlinear characteristic. The solver accurately predicts the shipping green water and slamming forces in extreme conditions.



甲板上浪
Shipping green water



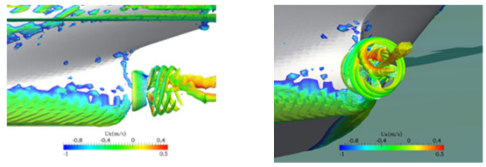
船艏瞬时波浪砰击压力
Instantaneous pressure of bow wave slamming

伴流补偿导管

Wake equalizing duct(WED)

伴流补偿导管有效改善螺旋桨入流均匀性，求解器可以精细捕捉船尾尾涡结构，对导管水动力性能做出准确的数值分析。

Wake equalizing duct(WED) effectively improve the uniformity the propeller inflow. The solver can finely capture the stern vortex structure and apply accurate numerical analysis on the hydrodynamic performance of the duct.



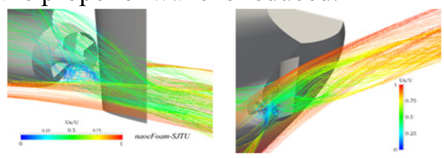
带有伴流补偿导管船尾涡量场
Vortex field in the stern with WED

前置预旋导轮

Pre-Shrouded Vanes(PSV)

前置预旋导轮改善螺旋桨入流均匀性，数值模拟的船尾流线场直观表现其对桨尾流旋转能量损失的降低作用。

Pre-Shrouded Vanes(PSV) can improve uniformity of the propeller inflow, directly shown by the streamline field of numerical simulation, where the energy loss of the propeller wake is reduced.



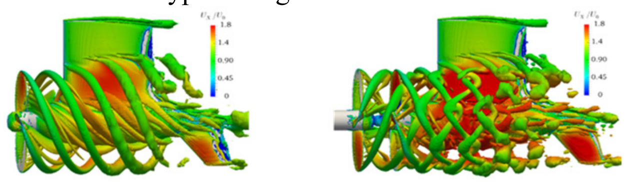
船尾流线场
Streamlines at stern

混合对转桨吊舱推进器

Hybrid CRP podded propulsor

吊舱推进器是具有较好的机动性和较低的振动噪声。通过naoe-FOAM-SJTU计算，可以在普通吊舱推进器的基础上，对其结构进行进一步优化。

Podded propulsor features good mobility and low vibration noise. Further optimization can be done based on the common type through naoe-FOAM-SJTU solver.



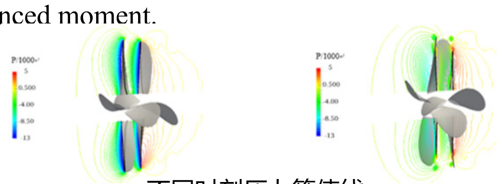
吊舱推进器和混合对转桨吊舱推进器涡量场
Q iso-surfaces of normal hybrid CRP podded propulsor

对转桨

Contra-rotating propellers(CRPs)

对转桨由两个旋向相反的同轴传统螺旋桨组成，求解器可以有效的预报推进系统不平衡扭矩的变化。

Contra-rotating propellers(CRPs) consist of two conventional propellers which rotate coaxially in reverse direction and the present solver can predict variation of unbalanced moment.



不同时刻压力等值线
Pressure contours at different moments

naoe-FOAM-SJTU 在海洋工程中的应用
The Application of naoe-FOAM-SJTU in Ocean Engineering

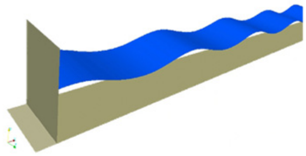
上海交通大学CMHL研究中心基于开源代码OpenFOAM平台上自主开发了船舶与海洋工程数值波浪水池模块、结构物六自由度运动模块以及浮式结构物系泊系统模块。形成了面向船舶与海洋工程水动力学问题的CFD求解器naoe-FOAM-SJTU，可以对海洋工程中的热点问题进行分析。

CMHL Research Institute of Shanghai Jiao Tong University develops the CFD solver for marine hydrodynamics, naoe-FOAM-SJTU, based on the open source platform OpenFOAM, which includes numerical wave tank module, 6DOF motion module for naval structures, and mooring system module. The solver can analyze focused problems in ocean engineering.

数值波浪水池造波模拟
Wave-generation simulation in numerical wave tank

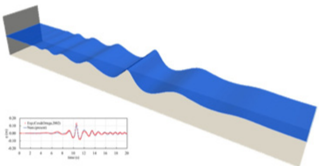
naoe-FOAM-SJTU求解器的数值造波模块，可建立三维数值水池。造波模拟方法包括数值推板造波和输入式数值造波。生成的波形与物理实验吻合较好，可以为数值模拟提供良好的波浪环境。

naoe-FOAM-SJTU solver can establish three dimensional numerical wave tanks using numerical wave tank module. Numerical wave generation methods include piston wave generation method and boundary condition input method. The generated wave pattern is in good agreement with experiment results, which provides reliable wave condition for numerical research.



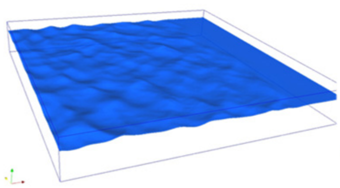
推板式数值造波：周期波

Piston wave generation: regular waves



推板式数值造波：畸形波

Piston wave generation : freak waves



输入式数值造波：多向不规则波

Boundary condition input method: irregular waves

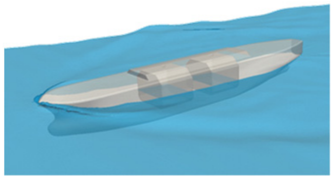
输入式数值造波：三维斜向规则波

Boundary condition input method: three dimensional oblique regular wave

船舶在波浪中的运动模拟
Simulation of ship motion in waves

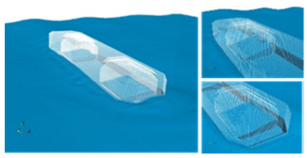
结合求解器中的六自由度模块与造波模块，求解器可以对FPSO、带液舱LNG船在波浪中的运动情况进行模拟。可以模拟甲板上浪等强非线性现象，清晰展示瞬时流场信息。

Combined with 6DOF module and numerical wave tank module, the solver can simulate the motion of FPSO and LNG carriers in waves. Strongly nonlinear phenomenon such as green-water effect can be simulated well, and the instantaneous flow field can be described clearly.



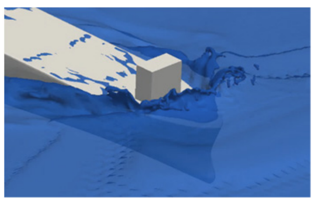
带液舱LNG船在迎浪上运动模拟

Motion of LNG carrier with liquid tank in the head waves



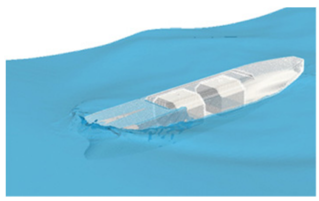
带液舱LNG船在横浪情况下的数值模拟

Numerical simulation of LNG with sloshing tank



波浪中的S175船舶甲板上浪模拟

Green-water effect simulation for S175 ship



波浪中的LNG甲板上浪模拟

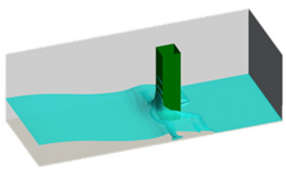
Green-water effect simulation for LNG

naoe-FOAM-SJTU 在海洋工程中的应用
The Application of naoe-FOAM-SJTU in Ocean Engineering

固定式结构物的波浪载荷预报
Wave load prediction for fixed structures

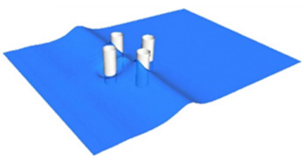
naoe-FOAM-SJTU求解器利用其三维数值水池模块，可以有效模拟溃坝波、孤立波、规则波、不规则波以及聚焦波等波浪工况对结构物的作用情况，以及波浪爬高和砰击问题。并对波浪载荷进行准确预报。

naoe-FOAM-SJTU solver is capable to simulate the interaction between structures and waves, such as dam break wave, isolated wave, regular wave, irregular wave and focusing wave based on the three dimensional numerical wave tank module. The wave run-up and slimming problems can also be solved. Wave loads can be predicted accurately during the simulation.



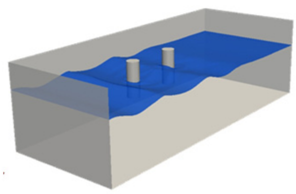
溃坝波对单桩柱的砰击载荷预报

Wave load prediction of dam break wave slamming on a Square column



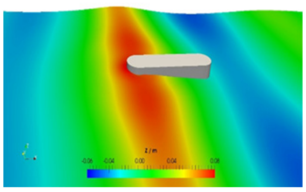
孤立波下多桩柱波浪爬高与波浪载荷预报

Wave run-up and wave loads prediction of multi-column in isolated wave



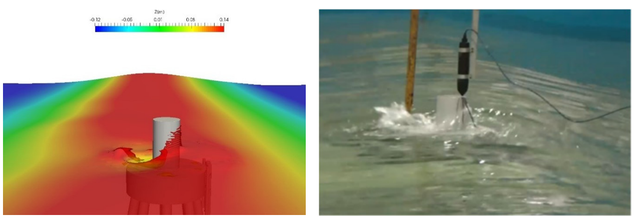
规则波下双桩柱波浪爬高与波浪载荷预报

Wave run-up and wave loads prediction of double columns in regular waves



聚焦波中FPSO标准模型波浪载荷计算

Wave loads calculation on FPSO benchmark model in focusing wave



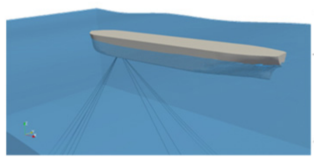
波流联合作用下固定式平台载荷预报

Wave load prediction of fixed platform under wave-current combinations

锚泊下浮式结构物在波浪中的运动响应预报
Motion response prediction for floating structures with mooring system in waves

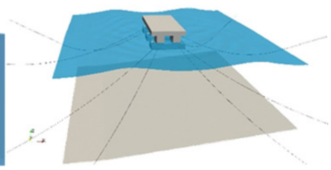
求解器结合造波模块、六自由度运动模块和系泊系统模块，可预报浮式结构物在复杂波浪环境下的运动响应，并对作用在结构物上的波浪力以及锚链力进行精确预报。

The solver takes the advantage of 6DOF module, numerical wave tank module and mooring system module to predict the motion of floating structures under complex wave conditions. The wave loads and mooring forces of structures can be predicted accurately.



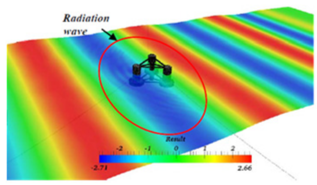
单点系泊FPSO在波浪中的运动模拟

Motion simulation of Single point mooring FPSO in waves



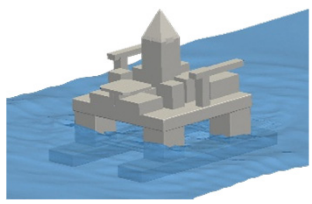
浮式码头-悬链线系泊耦合系统

Floating Wharf - Catenary mooring coupling system



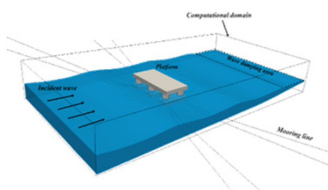
深漂式风力机半潜式平台在波浪中的运动

Motion of Semi-submersible Platform of the DeepCwind Floating Wind Turbine in waves



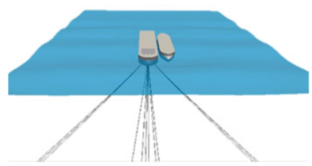
981半潜式平台在不规则波浪上的运动响应预报

Motion prediction for 981 semi-submersible platform in irregular waves



近海淹没地形浅水半潜式平台

Semi-submersible platform in shallow water with submerged terrain near island



单点系泊FLNG旁靠情况的运动响应模拟

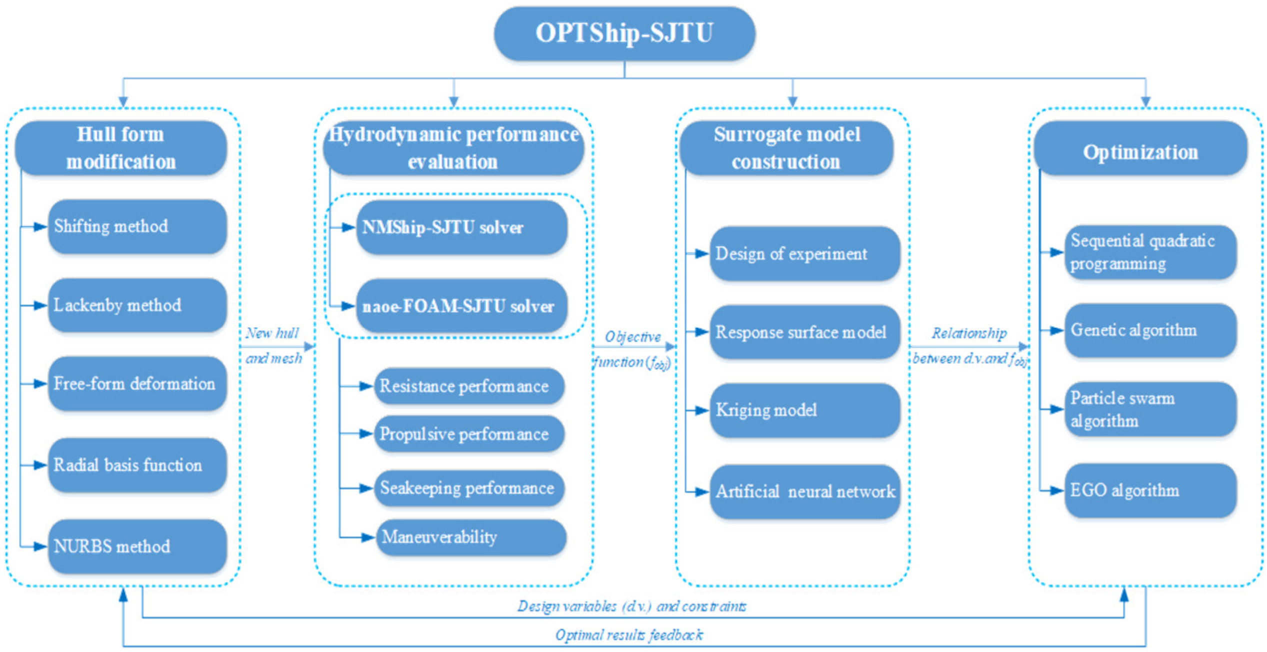
Motion prediction of single point mooring FLNG in side-by-side condition

船型优化求解器 OPTShip-SJTU

Ship Hull Optimization Solver OPTShip-SJTU

上海交通大学CMHL研究中心自主开发了船型优化设计工具OPTShip-SJTU求解器，并已获得国家软件著作权。求解器集成了船型变换模块、水动力性能评估模块、近似模型和优化模块，实现了船型的自动优化设计。求解器已成功应用于单目标（单航速下的阻力性能）船型优化设计以及多目标（多航速下的阻力性能、阻力与伴流综合性能等）船型优化设计。

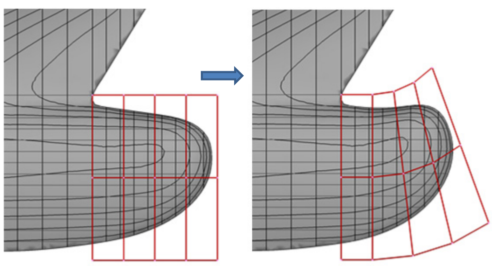
The OPTShip-SJTU solver, developed by CMHL of SJTU, is a self-developed tool for the ship hull form optimization design, which has obtained national software copyright. It integrates a hull form modification module, a hydrodynamic performance evaluation module, a surrogate module and an optimization module, which can achieve the ship hull form optimization design automatically. It has been successfully applied to the single-objective optimization of ship hull form for reducing resistance at a given speed and the multi-objective optimization of ship hull form such as reducing drag at a range of speeds, a comprehensive optimization of ship drag and wake performance at the disk.



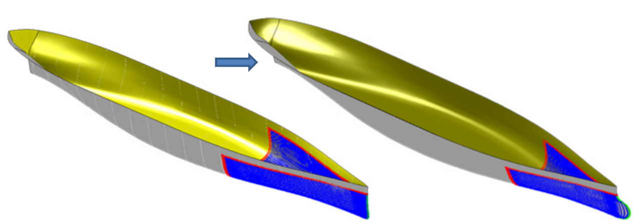
OPTShip-SJTU求解器基本框架
Framework diagram of OPTShip-SJTU

船型变换模块是船型优化设计的基础及关键技术之一。通过尽可能少的控制参数获取尽可能大的变换空间，以保证能够得到更优秀的船型。目前该模块已有的方法有：平移法，径向基函数方法 (RBF)，自由变形方法 (FFD)，NURBS方法等。

Ship hull modification module is the basis and key technology of ship hull optimization design. It uses a few transformation parameters to get the design space as large as possible, which ensures to obtain better ship hull. At present, there are several methods available, such as shifting method, radial basis Function (RBF) method, free-form deformation (FFD) method, NURBS method and so on.



自由变形法船型变换前后
Deformation by FFD method

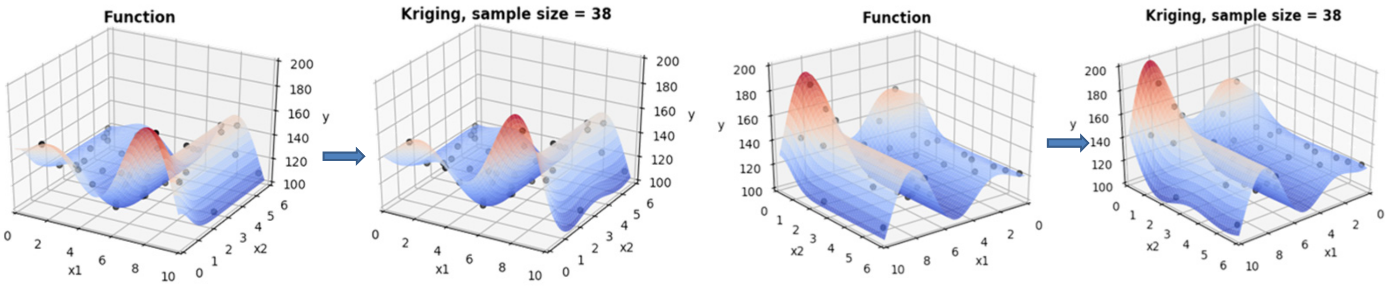


NURBS与RBF方法组合变形法前后
Deformation by NURBS+RBF method

船型优化求解器 OPTShip-SJTU的应用
Applications of Ship Hull Optimization Solver OPTShip-SJTU

优化过程中需要评估大量新船的水动力性能，而数值计算的时间长，费用高。考虑到实际工程周期要求，首先通过试验设计在设计空间中选择足够数量的新船型，由数值方法评估出水动力性能，再采用近似技术构建船型变换参数与水动力性能的近似关系从而生成近似模型，在优化进程中可直接调用近似模型评估任一新船型的水动力性能。

During the optimization process, the hydrodynamic performance of numerous new ship hulls should be evaluated, while numerical evaluation costs long time and high expense. In consideration of practical project period requirement, enough new ship hulls are selected in design space by design of experiments, then numerical method is applied to calculate hydrodynamic performance, and approximate technique is employed to build the approximate relationship between ship transformation parameters and hydrodynamic performance, which is called approximate model. During the optimization process, approximate model can be directly employed to evaluate the hydrodynamic performance of any new ship hull.

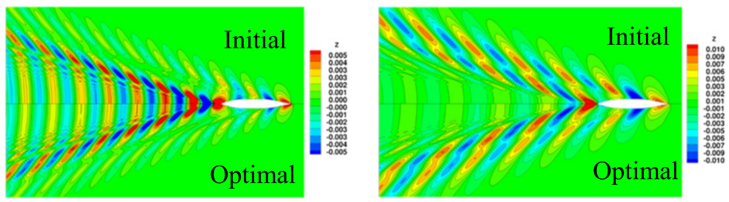


实际函数与近似模型对比
Comparison of function and approximate model

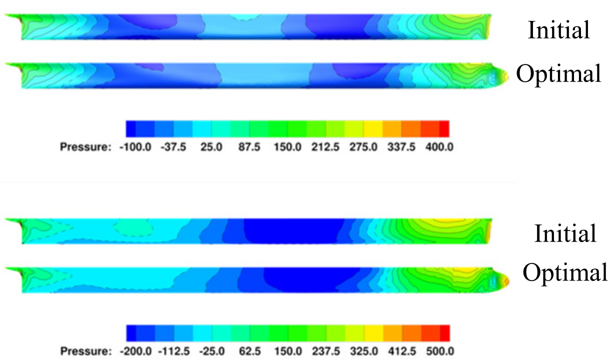
S60船大变形优化
Optimization design with large deformation of S60 hull

求解器可对S60船在两个航速下的兴波阻力(Fr=0.25和Fr=0.316)进行多目标优化设计。船型变换采用NURBS结合FFD的方法对全船大变形，兴波阻力采用势流理论计算，优化求解采用多目标遗传算法。最终得到优化船型兴波阻力系数分别降低25.83%，14.06%。通过母型船和优化船型的自由面兴波对比和压力分布对比可以看出优化后的船自由面兴波高度显著减小。优化船体压力分布更均匀。

The solver can achieve the multi-objective optimization of S60 hull for minimum wave-making resistance at the speeds of Fr=0.25 and Fr=0.316. In ship hull modification module, we combine NURBS and FFD to transform the hull. Potential flow theory is applied to calculate wave-making resistance and multi-objective genetic algorithm is employed to do the optimization. The wave-making resistance coefficients of optimal hulls has reduced by 25.83% and 14.06% at two speeds. Through the comparison of wave pattern and pressure distribution, the wave height has obviously reduced, and the pressure distribution is more uniform.

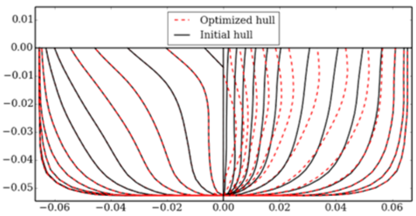


优化前后自由面兴波对比图（左：Fr=0.25；右：Fr=0.316）
Comparison of free surface wave patterns
(Left: Fr=0.25 ; Right: Fr=0.316)

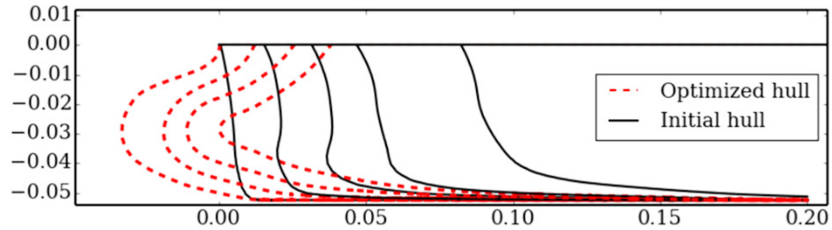


优化前后船体压力分布对比图（上：Fr=0.25；下：Fr=0.316）
Comparison of ship's pressure distribution
(Up: Fr=0.25 ; Down: Fr=0.316)

船型优化求解器 OPTShip-SJTU的应用
Applications of Ship Hull Optimization Solver OPTShip-SJTU



S60船型优化前后船体横剖线对比图
Comparison of body lines

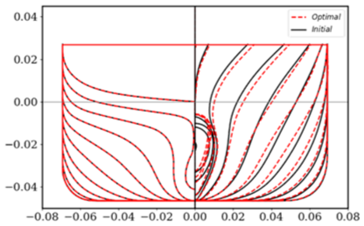


S60船型优化前后船体纵剖线对比图
Comparison of buttocks

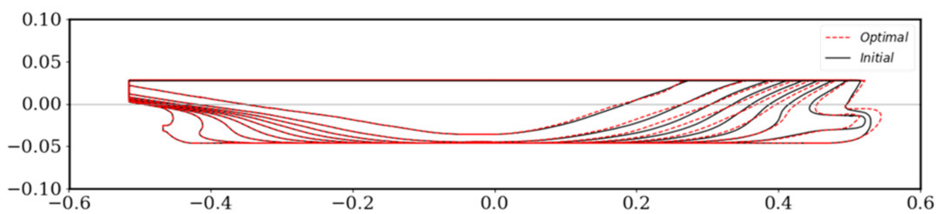
KCS船兴波阻力单目标优化
Single-objective optimization design of wave drag for KCS hull

求解器可对KCS船的兴波阻力 (Fr=0.26) 进行单目标优化设计。船型变换采用平移法和自由变形法对船前半体和球鼻艏进行全局和局部变形，兴波阻力采用势流理论计算，优化求解采用单目标遗传算法。最终得到优化船型兴波阻力系数降低40.09%，经验证最终总阻力值相比降低8.64%。通过母型船和优化船型的船侧波高对比可以看出优化船前肩位置波高降低，船中位置波高也降低。而从压力图也可以看出高压区和低压区压力值明显降低，整个船身压力梯度减小，压力分布更为均匀。

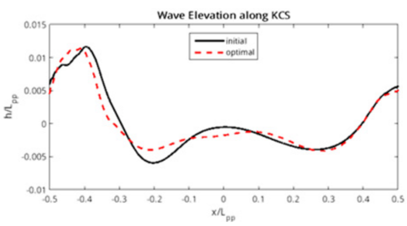
The solver achieves the single-objective optimization of KCS hull for the minimum wave-making resistance at the speed of Fr=0.26. In ship hull modification module, shifting method and FFD method are applied to modify the fore part of the hull and bulbous bow globally and locally. Potential flow theory is utilized to calculate wave-making resistance and single-objective genetic algorithm to solve this optimization problem. The wave-making resistance coefficient of optimal hull reduces by 40.09%. Through verification by CFD, the total drag decreases by 8.64%. Through the comparison of wave elevation along the hull, the wave height at front shoulder and middle of the optimal hull decreases obviously. As shown in the pressure distribution figure, in both high and low pressure regions, the pressure values have obviously decreased, and the pressure gradient along the whole hull becomes smaller, thus the pressure distribution is more uniform.



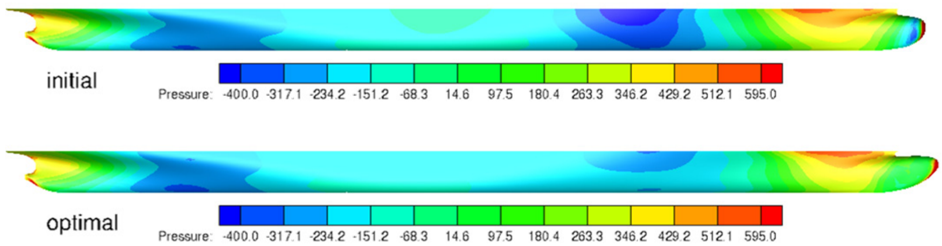
KCS船型优化前后船体横剖线对比图
Comparison of body lines



KCS船型优化前后船体纵剖线对比图
Comparison of buttocks



KCS船型优化前后船侧波高对比图
Comparison of wave height around hull



KCS船型优化前后船体压力分布对比图
Comparison of ship's pressure distribution

船型优化求解器 OPTShip-SJTU的应用

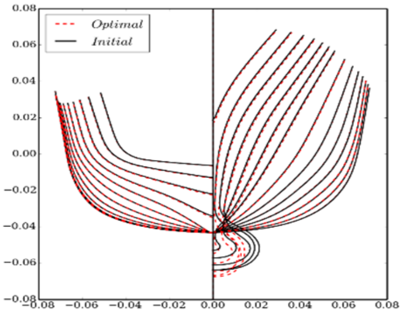
Applications of Ship Hull Optimization Solver OPTShip-SJTU

DTMB5415总阻力多目标优化

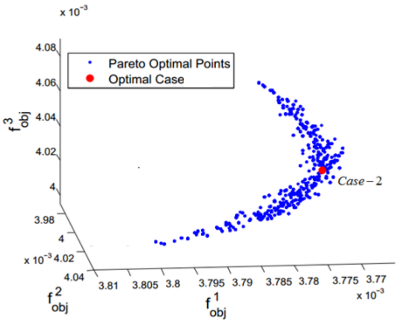
Multi-objective optimization design of total drag for DTMB 5415 hull

求解器可对DTMB5415船在三个航速下的总阻力进行多目标优化设计。采用自由变形方法对船首局部变换，基于RANS方法求解总阻力，优化求解采用多目标遗传算法。得到系列优化船型，总阻力均降低。从船首压力对比图可以看出优化船的高低压区变小，从波形对比图可以看出优化后的船首尾处兴波减小。

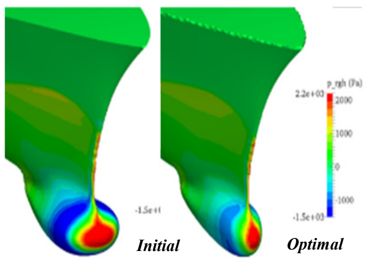
The solver can achieve the multi-objective optimization of DTMB 5415 hull for the minimum total drag at three speeds. FFD method is utilized to modify the bulbous bow locally. Total drag is evaluated by RANS method, and multi-objective genetic algorithm is employed to solve optimization. Series of optimal hulls are obtained with reduced total drags at three speeds. Through the comparison of pressure on bulbous bow, we can see that both high and low pressure region are smaller. Through the comparison of free surface wave pattern , we know the waves in bow and stern decrease.



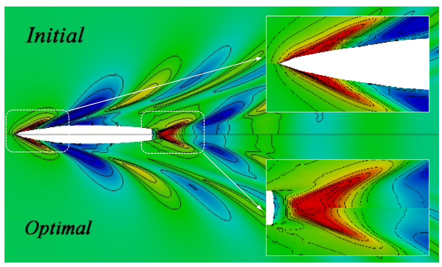
优化前后船体横剖线对比图
Comparison of body Lines



优化收敛得到的Pareto解集
Optimal pareto set



优化前后船体球鼻艏对比图
Comparisons of pressure on bulbous bow



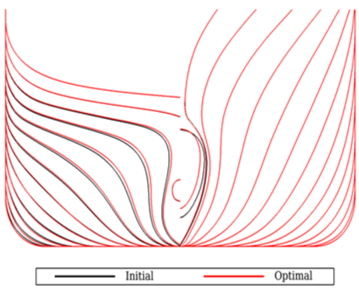
优化前后自由面兴波对比图
Comparison of free surface wave pattern

KCS船总阻力和尾部伴流综合优化

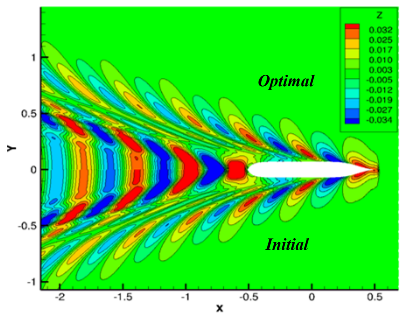
Comprehensive optimization design of drag and wake performance for KCS hull

求解器可对KCS船在航速Fr=0.26下的总阻力和尾部伴流进行多目标综合优化设计。优化船型总阻力降低3.62%，伴流性能改善2.8%。从母型船和优化船型的自由面兴波对比可以看出优化船兴波减小，从船首压力对比可以看出优化船的压力变小，从伴流对比可以看出船尾桨盘面处速度场更均匀，伴流性能更好。

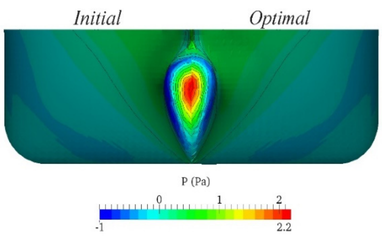
The solver can also achieve the comprehensive optimization of KCS hull for minimum total resistance and best wake performance at the speed of Fr=0.26. The optimal hull has a drag reduction of 3.62% and wake performance improvement of 2.8%. Through the comparison of wave patterns, the wave height of the optimal hull is reduced. Both high and low pressure region are smaller. The velocity field at the propeller disk becomes more uniform, so the wake performance is improved.



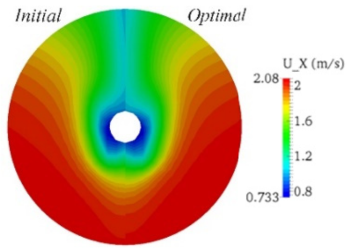
优化前后船体横剖线对比图
Comparison of body lines



优化前后自由面兴波对比图
Comparison of free surface wave patterns



优化前后船艏压力分布对比图
Comparison of pressure distributions on bulbous bow



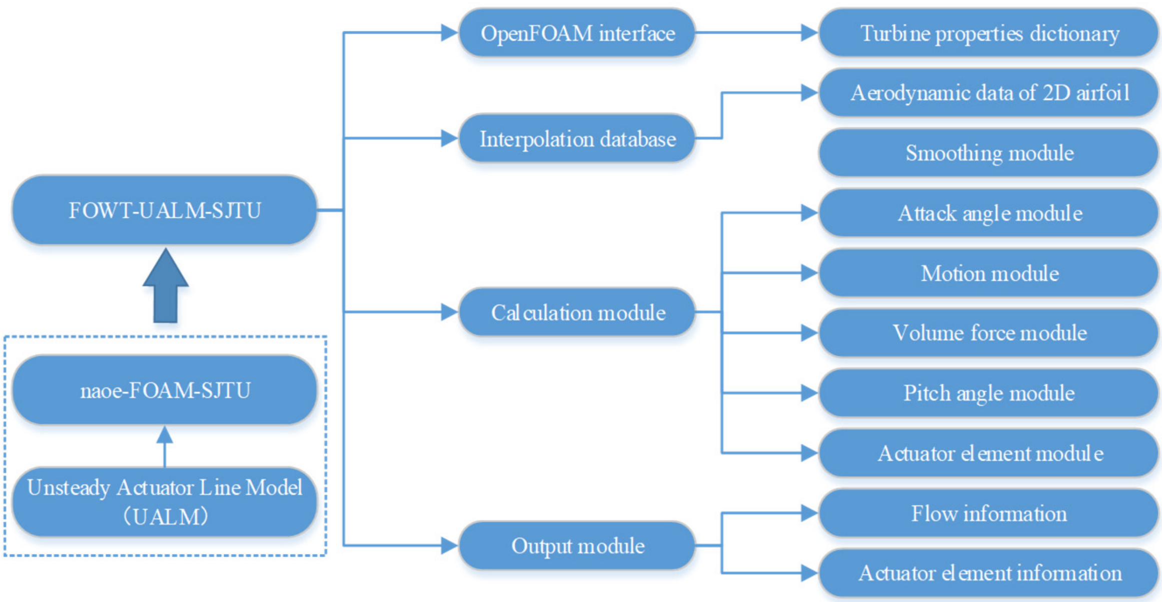
优化前后船艉桨盘面处伴流对比图
Comparison of the wake at the disk

浮式风机气动与水动耦合性能求解器 FOWT-UALM-SJTU

Aero-hydrodynamic Solver for Floating Offshore Wind Turbines FOWT-UALM-SJTU

上海交通大学CMHL研究中心基于非稳态致动线模型，开发了浮式风机气动与水动性能耦合求解器FOWT-UALM-SJTU，并获得了国家软件著作权。该求解器包含OpenFOAM接口、插值数据库、计算模块和输出模块。可实现对风机气动力性能和平台水动力性能的数值模拟，以及海上风机气动与水动耦合动力性能的预报。同时能够实现阵列分布多风机风场复杂流动的数值仿真。

CMHL of SJTU has developed the FOWT-UALM-SJTU solver to achieve fully coupled aero-hydrodynamic simulations of floating offshore wind turbines (FOWTs) based on the unsteady actuator line model (UALM). FOWT-UALM-SJTU solver consists of four main modules: OpenFOAM interface, interpolation database, calculation module and output module. The numerical simulations of aerodynamics of wind turbines and the hydrodynamics of platforms can be achieved, and the coupled aero-hydrodynamics of offshore wind turbines can also be predicted. In addition, this solver is also capable of aerodynamic simulations for large wind farms with multiple wind turbines.



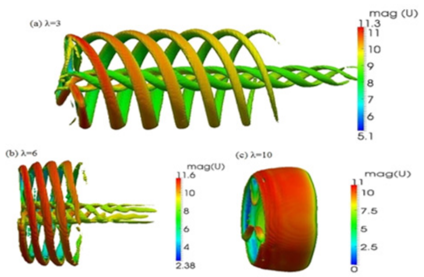
FOWT-UALM-SJTU求解器基本框架
Basic Frame of the FOWT-UALM-SJTU Solver

单风机气动性能数值模拟

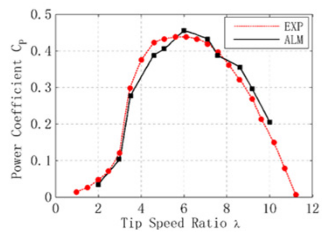
Numerical simulations of a single wind turbine

非稳态致动线模型利用带体积力的致动线模拟真实的风机叶片，可有效提高计算效率，并得到比较准确的气动载荷数值模拟结果，对均匀流下固定单风机气动载荷的预报误差在5%以内。

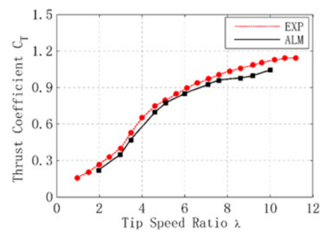
The basic idea of the unsteady actuator line model is to replace the blades of the wind turbine with a series of actuator points withstanding body forces. So this method is well suited for aerodynamics and wake studies while keeping the computing costs at a reasonable level. And the error of aerodynamic loads prediction is within 5%.



不同叶尖速比下单风机尾涡结构
Vortex structure of the single wind turbine under different tip speed ratios



功率系数
Power coefficient



推力系数
Thrust coefficient

基于标准实验“Blind Test 1”单风机数值模拟
Numerical simulation of the single wind turbine based on “Blind Test 1”

浮式风机气动、水动及耦合性能模拟

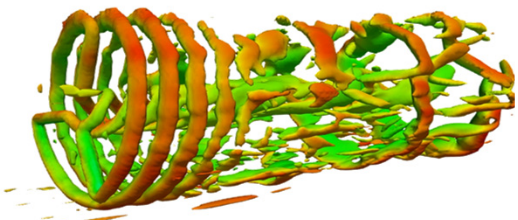
Numerical Simulations for Coupled Aero-hydrodynamics of FOWTs

单风机气动性能数值模拟

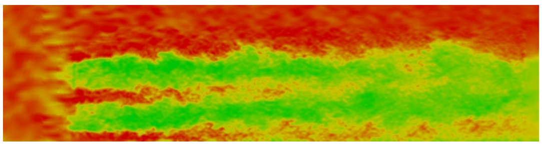
Numerical simulations of a single wind turbine

通过在计算域入口生成满足连续性方程的随机脉动速度模拟湍流入流条件，可以模拟更加真实的大气湍流环境，进而对湍流入流条件下风机的气动特性及复杂尾流特性进行模拟。

By generating random velocity series that satisfies the continuity equation at inlet boundary, the turbulent inflow can be simulated. Then simulations for complicated aerodynamics and wake characteristics of the wind turbine under turbulent inflow condition can be achieved.



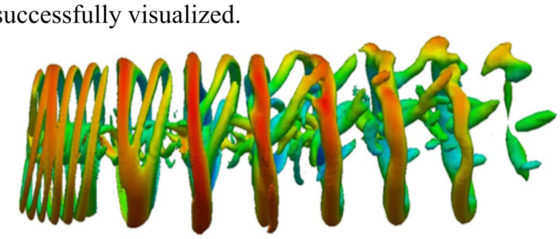
湍流入流下风机尾流涡结构
Vortex structure of wind turbine under turbulent inflow



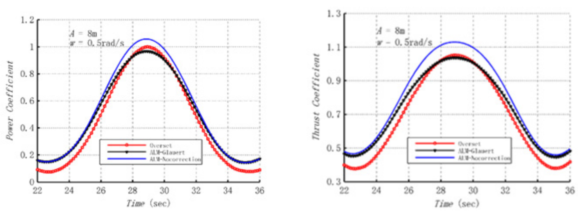
湍流入流下风机尾流速度云图
Wind velocity counters in vertical plane under turbulent inflow

对给定运动下风机的非稳态气动性能进行数值仿真，成功模拟出带纵摇运动风机的复杂气动特性，得到了风机的非稳态气动推力与气动功率，以及复杂的尾流结构。

Furthermore, this method can also simulate the unsteady aerodynamic characteristics of wind turbine under the periodic motions. Unsteady aerodynamic loads and complex vortex structure due to the motion of wind turbine are successfully visualized.



纵摇运动下风机尾涡结构
Vortex structure of wind turbine with pitch motion



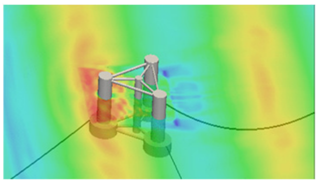
纵摇运动下功率与推力输出时历曲线
Time history curves of aerodynamic loads with pitch motion

平台水动力性能预报

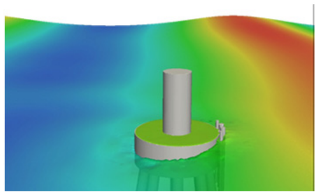
Hydrodynamic simulations of platforms

基于naoe-FOAM-SJTU，求解器可对波浪与固定式、浮式风机支撑平台的相互作用过程进行模拟，预报支撑平台所受的水动力载荷。

naoe-FOAM-SJTU solver can simulate the interaction between the waves and fixed/floating support platform, and the hydrodynamic loads acting on the support platforms are also available.



浮式平台水动力性能预报
Hydrodynamic prediction of floating platform



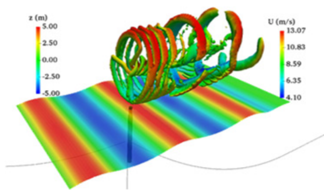
固定式平台水动力性能预报
Hydrodynamic prediction of fixed platform

浮式风机气动-水动耦合数值模拟

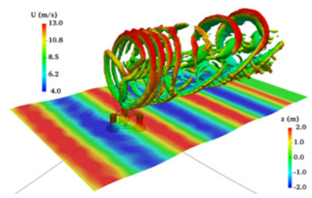
Coupled aero-hydrodynamic simulations of FOWTs

FOWT-UALM-SJTU求解器实现了浮式风机系统的气动-水动全耦合数值模拟，可对不同形式浮式风机的耦合动力响应进行数值预报，并对串列式、错列式两浮式风机的复杂尾流相互作用进行了数值模拟。

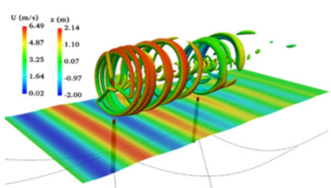
FOWT-UALM-SJTU solver can achieve coupled aero-hydrodynamic simulations of different FOWTs, and the performance of wind turbine-floating platform-mooring system can be predicted. The complicated wake interactions between two FOWTs in both tandem and offset configurations are also studied.



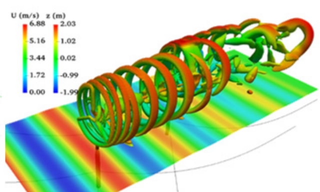
Spar型浮式风机系统气动-水动耦合数值模拟
Fully coupled simulations for spar type FOWTs



半潜式浮式风机系统气动-水动耦合数值模拟
Fully coupled simulations for semi-submersible FOWTs



串列式两浮式风机气动-水动耦合数值模拟
Fully coupled simulations for two FOWTs in tandem configuration



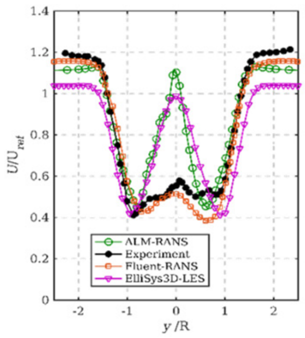
错列式两浮式风机气动-水动耦合数值模拟
Fully coupled simulations for two FOWTs in offset configuration

多风机风电场气动性能模拟

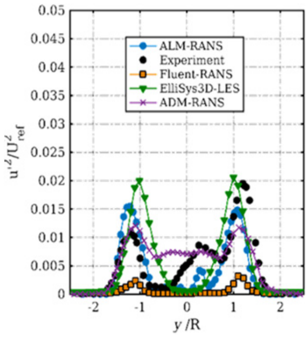
Numerical Simulations for Aerodynamics of Wind Farms

FOWT-UALM-SJTU求解器基于致动线模型，实现两风机在不同布置形式下的数值模拟，得到两风机风电场的气动载荷特性，流场特性，尾涡结构以及复杂尾流相互干扰特性。

FOWT-UALM-SJTU solver can simulate two wind turbines with in-line and offset layouts, and predict the aerodynamic loads, wake characteristics, vortex structure and the complex wake interaction phenomenon.

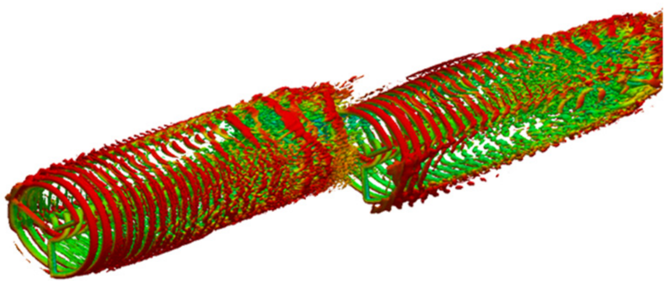


尾流场速度分布
Mean wake velocity profile



尾流场湍流应力分布
Mean turbulent stress profile

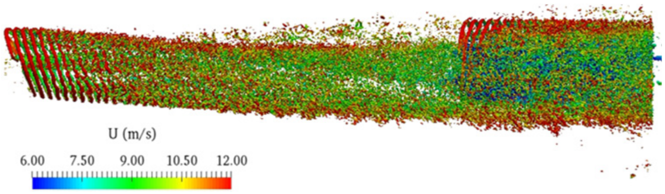
基于标准试验 “Blind Test2” 串列式两风机数值模拟
Numerical simulation of the two in-line turbines based on “Blind Test 2”



错列式两风机尾涡结构
Vortex structure of two offset wind turbines

FOWT-UALM-SJTU求解器可对处于偏航状态的多风机风场进行数值仿真，成功模拟出在风机偏航时尾流偏移、卷曲等流场特性，为通过偏航抑制多风机间尾流干扰提供参考。

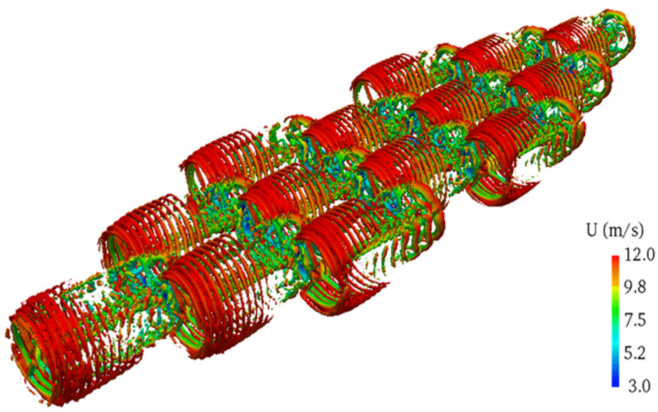
FOWT-UALM-SJTU solver can analyze the wake development and wake interaction among wind turbines in yaw condition. The wake deflection and wake curling can be observed in the numerical simulation.



偏航两风机尾涡结构
Vortex structure of two turbines in yaw

FOWT-UALM-SJTU求解器可对阵列式多风机大型风场进行数值仿真，成功模拟出风场中复杂的尾流相互干扰现象，为风场的优化布置研究提供了合理的参考建议。

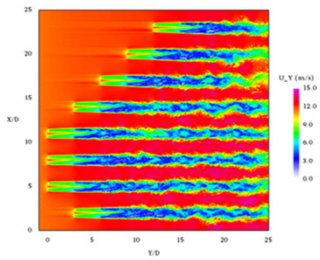
This solver can also create a sound methodology for performing the simulation of large wind farms and improve understanding of the wake interaction, providing the reasonable reference for wind farm layout optimization.



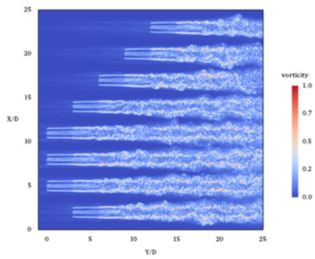
阵列式多风机风场数值模拟
The instantaneous vortex structure of wind farms

基于FOWT-UALM-SJTU求解器对丹麦Lillgrund 风电场进行数值模拟，分析大型风电场中复杂尾流干扰下风机的气动特性。

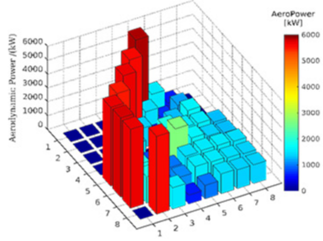
With FOWT-UALM-SJTU solver, a numerical investigation has been executed based on the Lillgrund wind farm layout to discuss the aerodynamic loads, complex wake effects and significant wake interactions.



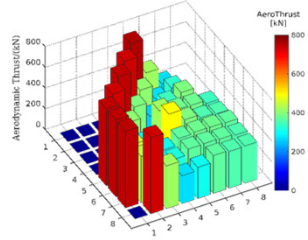
轴向速度云图
Transient velocity contour



流场涡量云图
Transient vorticity contour



平均气动功率输出统计
Time average aerodynamic power



平均气动推力输出统计
Time average aerodynamic thrust

基于重叠网格技术对风力机的数值模拟

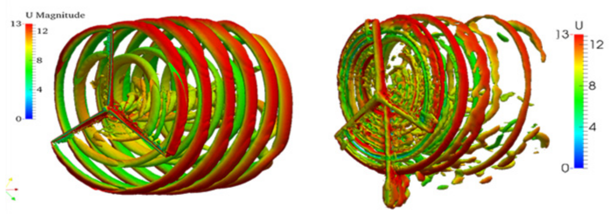
Numerical Simulations of FOWTs Based on the Overset Grid Technique

基于重叠网格技术，对全尺度风机的气动特性进行数值模拟，能够得到更加精确的流场信息，更加真实地模拟了风机的作业状态。可实现风机在考虑塔影效应时的气动载荷预报、考虑剪切风作用的风机非稳态气动性能数值仿真。同时，基于求解器naoe-FOAM-SJTU，结合重叠网格技术实现了浮式风机系统的气动-水动全耦合数值模拟。

The aerodynamics of full-scale wind turbines have been simulated based on the overset grid technique, which can provide more specific flow information in more realistic working conditions. Aerodynamic performance of wind turbine with tower shadow effects or shear wind effect are predicted. By combining overset grid technique and the in-house CFD hydrodynamic solver, naoe-FOAM-SJTU, the fully coupled aero-hydrodynamic simulations of floating offshore wind turbines are achieved.

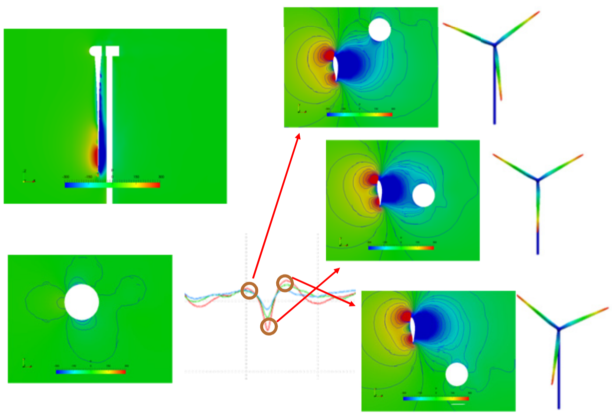
塔影效应影响下的风机气动性能预报

Aerodynamics of wind tirbine with Tower Shadow Effect



有塔架和无塔架的尾流涡结构

Vortex structure of wind turbine with/without tower structure

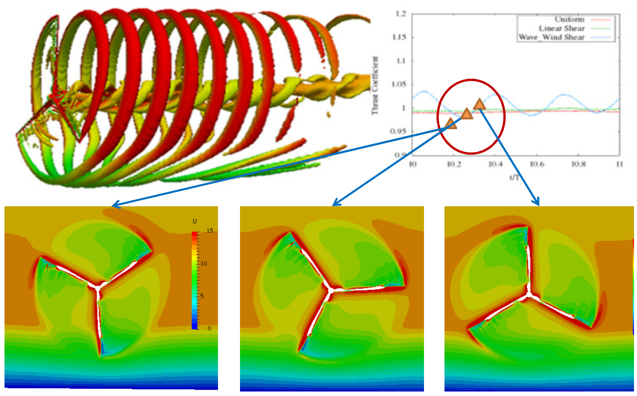


压力分布云图

Pressure distribution

风剪切效应影响下的风机气动性能预报

Aerodynamics of wind turbine with Shear Wind Effect

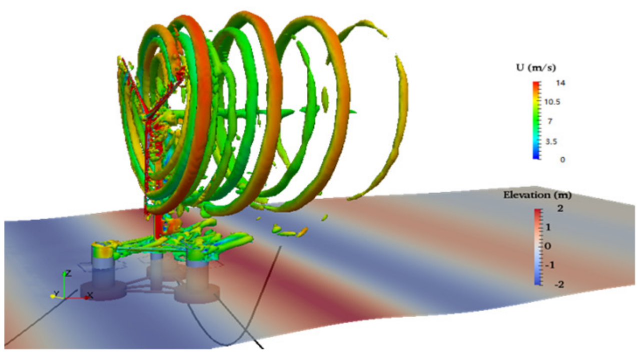


叶片附近流场速度分布

Distribution of flow velocity near turbine blades

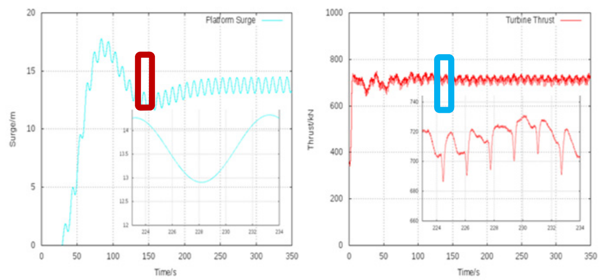
浮式风机系统气动-水动耦合数值模拟

Coupled aero-hydrodynamic simulation of FOWTs



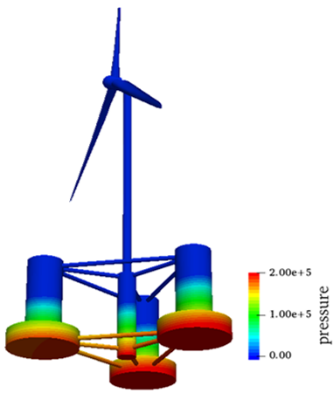
尾流涡结构与自由面可视化

Visualization of wake vortex structure and the free surface



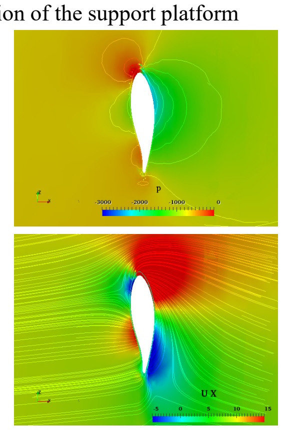
风机载荷与支撑平台运动曲线

Thrust on wind turbine and motion of the support platform



支撑平台表面压力分布

Pressure distribution on platform surface



流场压力和速度分布

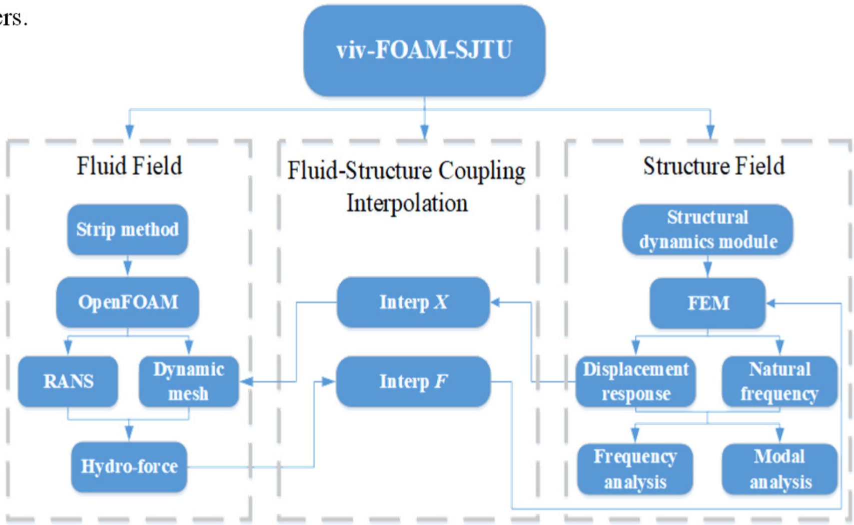
Pressure and velocity distribution near blade

深海立管涡激振动流固耦合求解器 viv-FOAM-SJTU

Fluid-Structure Coupling Solver for Vortex-induced Vibration of Deep-sea Risers viv-FOAM-SJTU

上海交通大学CMHL研究中心基于OpenFOAM，结合流固耦合切片法，自主开发了用于预报深海柔性立管涡激振动的CFD求解器viv-FOAM-SJTU。该求解器由流场模块、流固耦合插值模块、结构场模块三大模块构成，可对不同结构参数、不同流场剖面、平台运动影响及双立管耦合等多种复杂工况下柔性立管涡激振动进行数值模拟。

Based on OpenFOAM and combined with strip method, the CMHL of SJTU have developed the CFD solver viv-FOAM-SJTU for predicting the vortex-induced vibration (VIV) of deep-sea flexible risers. The solver is composed of three modules, including flow field module, fluid-structure coupling interpolation module and structure field module. The solver can simulate VIV of flexible risers in various conditions, such as different structural parameters, different flow field profiles, platform motion effects and coupling of two risers.



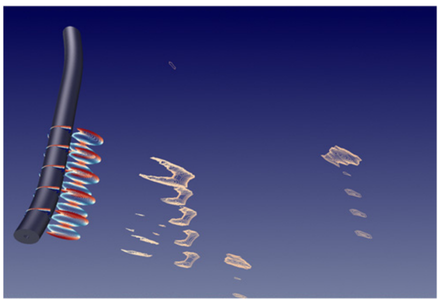
viv-FOAM-SJTU求解器基本框架
Basic framework of viv-FOAM-SJTU solver

阶梯流中柔性立管的涡激振动数值模拟

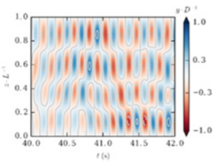
VIV of a flexible riser in stepped flow

viv-FOAM-SJTU求解器对阶梯流中柔性单立管涡激振动标准试验进行了数值模拟，准确预报了柔性立管的主振模态和位移响应。

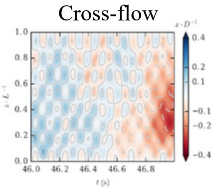
Numerical simulations on the standard experiments of VIV of a flexible riser in stepped flow have been conducted, which accurately predict the dominant vibration mode and displacement response of the flexible riser.



阶梯流中柔性立管的涡激振动数值模拟
VIV of a flexible riser in stepped flow



横流向
Cross-flow



顺流向
In-line

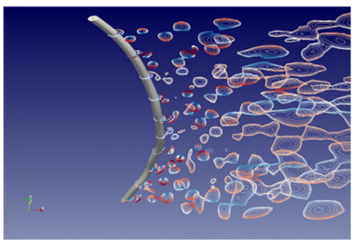
位移时空云图
Spatio-temporal displacement

均匀流中柔性立管的涡激振动数值模拟

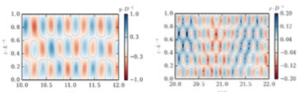
VIV of a flexible riser in uniform flow

viv-FOAM-SJTU求解器可模拟均匀流中不同长细比柔性立管的涡激振动。随长细比增加，立管的主振模态增加，且会产生多模态振动现象。

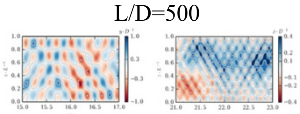
The viv-FOAM-SJTU solver can simulate the vortex-induced vibration of a flexible riser with different aspect ratios in uniform flow. The dominant vibration mode of the riser increases, with the increase of aspect ratio. And the multi-mode vibration phenomenon occurs.



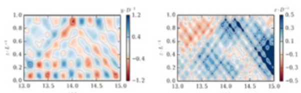
均匀流中柔性立管的涡激振动数值模拟
VIV of a flexible riser in uniform flow



L/D=500



L/D=750



L/D=1000

横流向 Cross-flow　　顺流向 In-line

viv-FOAM-SJTU求解器在平台运动和双立管耦合中的应用

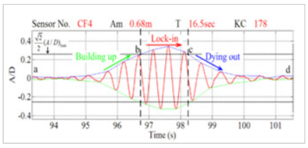
Applications of viv-FOAM-SJTU Solver to Platform Motions Effects and Coupling of Two Risers

振荡流中柔性立管的涡激振动数值模拟

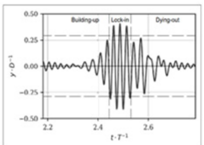
VIV of a flexible riser in oscillatory flow

viv-FOAM-SJTU求解器可模拟振荡流中柔性立管涡激振动的泻涡和振动特性，观察到涡激振动“产生-锁定-消亡”过程。

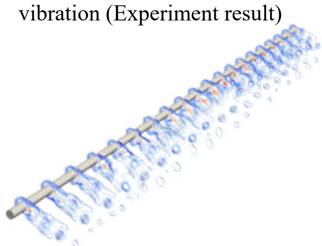
The viv-FOAM-SJTU solver can simulate the vortex shedding and vibration characteristics of a flexible riser in oscillatory flow, the obvious “building up - locking in - dying out” process of VIV is observed.



横流向振动时历曲线（试验结果）
Time history of cross-flow vibration (Experiment result)



横流向振动时历曲线（计算结果）
Time history of cross-flow vibration (Computational result)



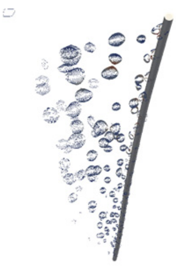
振荡流中柔性立管的涡激振动数值模拟
VIV of a flexible riser in oscillatory flow

平台水平运动下柔性立管的涡激振动

VIV of a flexible riser under platform horizontal motions

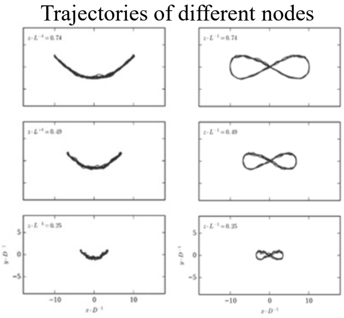
viv-FOAM-SJTU求解器可模拟平台纵荡与横荡耦合抛物线型运动和“8”字型运动时，柔性立管的涡激振动。

The viv-FOAM-SJTU solver can simulate VIV of a flexible riser under the combined surge and sway motions of the platform, including the parabolic type and the “8” type trajectories.



平台水平运动下柔性立管的涡激振动
VIV of a flexible riser under platform horizontal motions

不同节点的轨迹线
Trajectories of different nodes



抛物线型
The parabolic type

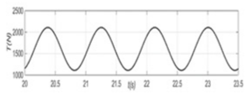
“8”字型
The “8” type

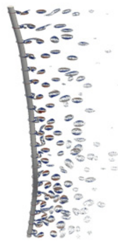
平台垂荡运动下柔性立管的涡激振动

VIV of a flexible riser under platform heave motions

viv-FOAM-SJTU求解器可模拟平台垂荡运动影响下细长柔性立管的涡激振动响应。

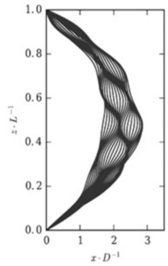
The viv-FOAM-SJTU solver can simulate VIV responses of a flexible riser under platform heave motions.



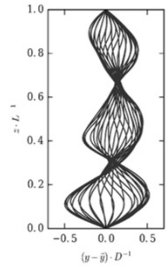


泄涡过程
Vortex shedding process

振动包络线
Vibration envelopes



顺流向
In-line



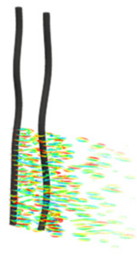
横流向
Cross-flow

阶梯流中柔性双立管的涡激振动数值模拟

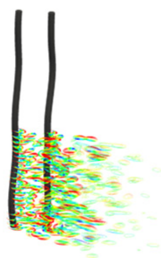
VIV of two flexible risers in stepped flow

viv-FOAM-SJTU求解器可模拟阶梯流中不同布置形式的柔性双立管的涡激振动。

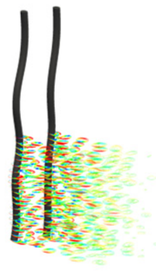
The viv-FOAM-SJTU solver can simulate VIV of two flexible risers in stepped flow with different arrangements.



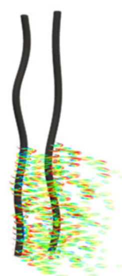
串列布置
Tandem arrangement



错列布置(30º)
Staggered arrangement(30º)



错列布置(60º)
Staggered arrangement(60º)



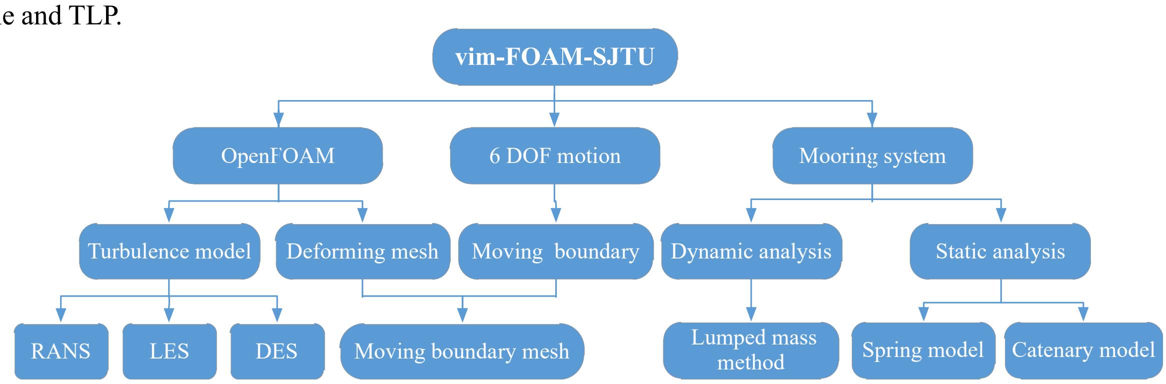
并列布置
Side-by-side arrangement

深海浮式平台涡激运动求解器 vim-FOAM-SJTU

CFD Solver for Vortex-Induced Motions of Floating Offshore Platform vim-FOAM-SJTU

上海交通大学CMHL研究中心在开源代码OpenFOAM平台上自主开发了深海浮式平台涡激运动求解器vim-FOAM-SJTU。该求解器采用分离涡模拟（DES）方法处理高雷诺数下三维流动分离问题，结合物体六自由度运动理论和动边界弹簧网格技术求解海洋平台涡激运动问题，适用于浮筒、Spar、半潜式和张力腿等形式的浮式海洋结构物的静止绕流研究和涡激运动预报。

CMHL of SJTU develops the CFD solver vim-FOAM-SJTU based on open source platform OpenFOAM. The solver utilizes detached-eddy simulation (DES) for modeling massively separated flows. By combining 6DOF module and dynamic deforming mesh technique, it can simulate VIM of floating offshore platforms, including buoy, Spar, semi-submersible and TLP.



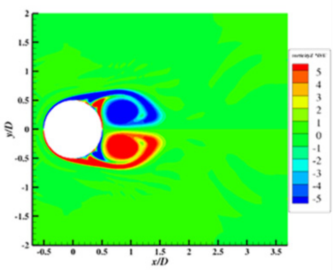
vim-FOAM-SJTU求解器基本框架
Framework for the CFD solver vim-FOAM-SJTU

柱体绕流问题模拟

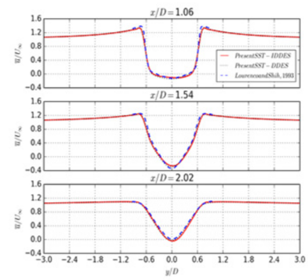
Flow over a bluff body

vim-FOAM-SJTU求解器在圆柱绕流标准试验数值模拟中可捕捉到比时均方法更加精细的涡结构及明显的三维旋涡脱落效应，表面压力系数预报误差5%以内。

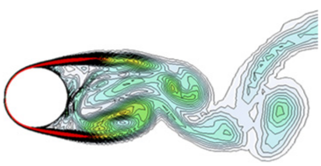
The solver can capture finer turbulence structures and detached eddies in comparison with Reynolds-Averaged methods. The error of predicted surface pressure coefficients is below 5%.



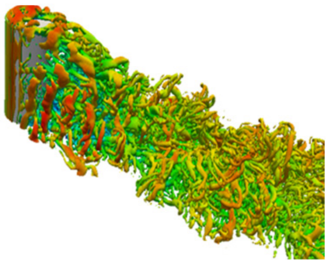
圆柱绕流在z=0水平截面涡涡情况
Instantaneous vorticity Z contour on a horizontal plane (z=0)



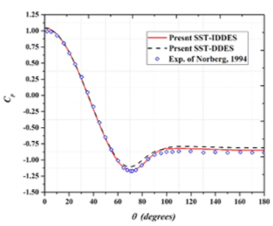
近尾流场三个剖面处的流向速度分布
Mean stream-wise velocity at three locations in the near wake



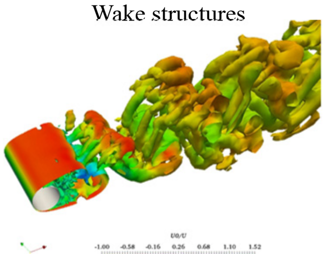
圆柱绕流水平截面涡量场
Flow over a cylinder – Vorticity contour on a slice



有限长度三维圆柱绕流尾涡
Flow over a truncated cylinder – Wake structures



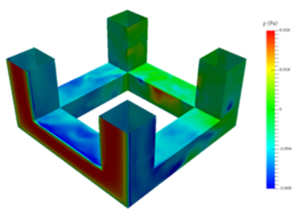
圆柱表面压力系数C_p
Pressure coefficient on the surface of the cylinder



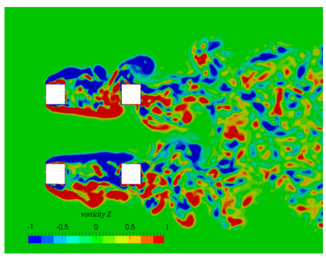
无限长度三维圆柱绕流尾涡
Flow over a infinite cylinder – Wake structures

vim-FOAM-SJTU求解器还可以对多立柱浮体例如半潜式平台进行静止绕流数值模拟。

The solver can also predict the flow over a fixed multi-column structures, such as semi-submersible platform.



半潜式平台静止绕流表面压力分布
Surface pressure of flow over a fixed semi-submersible



半潜式平台静止绕流水平截面涡量场
Vorticity contour of flow over a fixed semi-submersible

vim-FOAM-SJTU应用于浮筒和Spar平台涡激运动问题

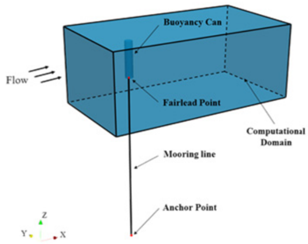
Applications of vim-FOAM-SJTU on VIM of buoy and Spar platform

浮筒涡激运动模拟

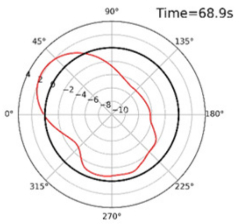
VIM of buoys

vim-FOAM-SJTU求解器可对圆柱浮筒进行涡激运动数值模拟。分析不同工况下浮筒的涡激运动响应特征。可对考虑旋转运动的浮筒进行涡激运动数值模拟。

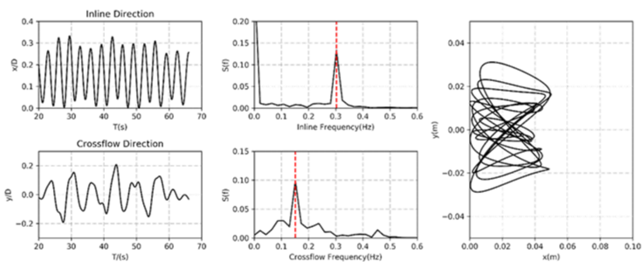
The solver can predict VIM of tethered buoys and analyze the motion characteristics induced by vortices. It can release the rotational degree-of-freedom when performing a simulation to the VIM of a tethered buoy..



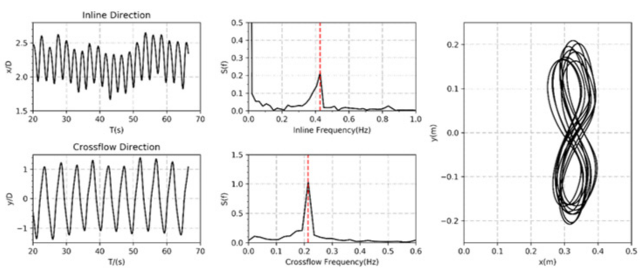
浮筒涡激运动计算模型
Computational model of VIM of a tethered buoy



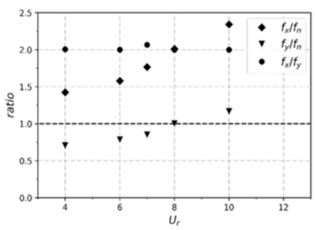
压力系数周向分布图
Surface pressure distribution of VIM of a tethered buoy



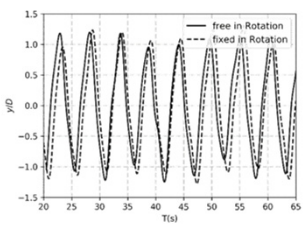
折合速度为4时的运动响应
Motion response at reduced velocity of 4



折合速度为8时的运动响应
Motion response at reduced velocity of 8



不同工况下浮筒涡激运动频率
VIM frequency of the buoy at several reduced velocities



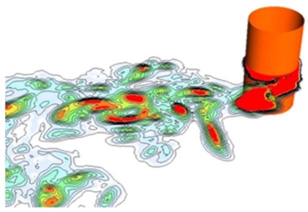
旋转运动对浮筒运动的影响
Effect of rotational degree-of-freedom on the sway of buoy

Spar平台涡激运动模拟

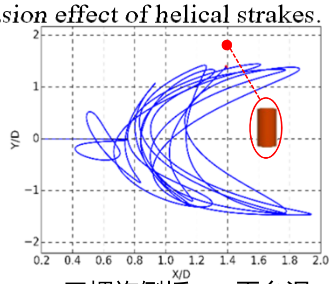
VIM of Spar platform

vim-FOAM-SJTU求解器可用于Spar平台涡激运动预报，分析平台在不同情况下的涡激运动响应，及柱体表面螺旋侧板的减涡效果。螺旋侧板可以显著降低流向和横向运动响应。

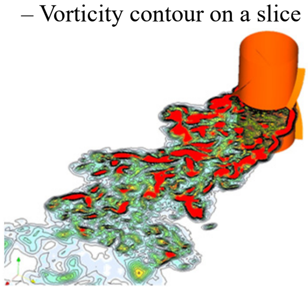
The solver can predict VIM of Spar platform and analyze motion response for various cases, such as evaluating the VIM suppression effect of helical strakes.



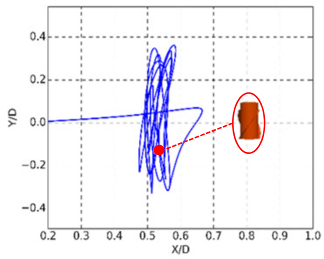
无螺旋侧板Spar平台涡激运动截面涡量图
VIM of Spar without helical strakes – Vorticity contour on a slice



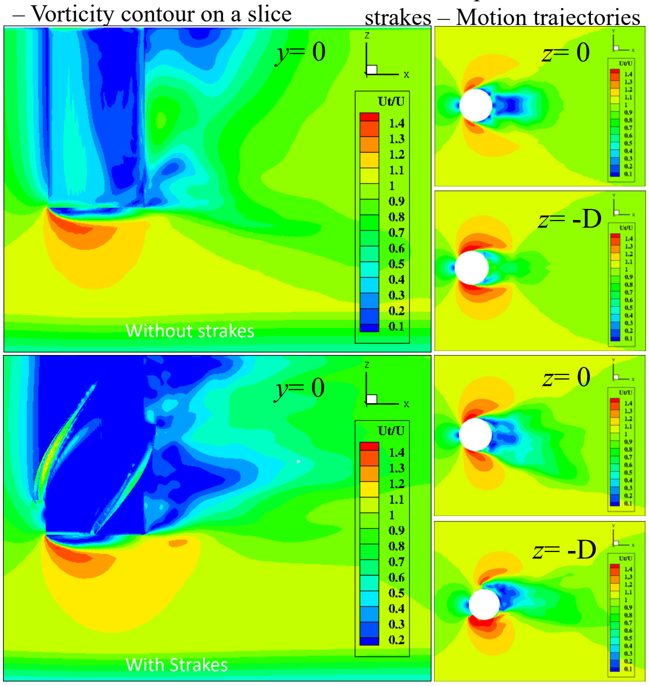
无螺旋侧板Spar平台涡激运动轨迹线
VIM of Spar without helical strakes – Motion trajectories



带螺旋侧板Spar平台涡激运动截面涡量图
VIM of Spar with helical strakes – Vorticity contour on a slice



带螺旋侧板Spar平台涡激运动轨迹线
VIM of Spar with helical strakes – Motion trajectories



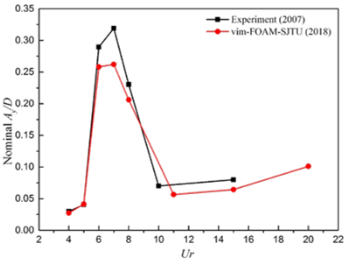
不带螺旋侧板和带螺旋侧板Spar平台的速度云图
Velocity contour on a slice for Spar with/without helical strakes

vim-FOAM-SJTU应用于半潜式平台涡激运动问题
Applications of vim-FOAM-SJTU on VIM of semi-submersible platform

半潜式平台涡激运动模拟
VIM of semi-submersible

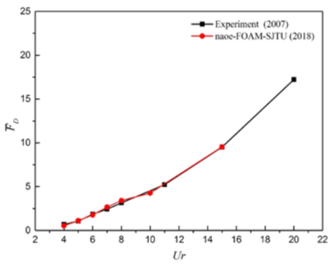
vim-FOAM-SJTU求解器可以模拟多柱式半潜平台涡激运动。拖曳力预报误差在6%左右，涡激运动响应预报误差在10%以内。

The solver can predict VIM of multi-column semi-submersibles. The prediction error of drag forces is below 5%. And the error of VIM response is within 10%.



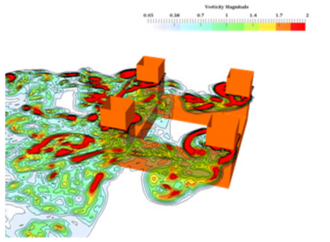
准确预报“锁定”现象

Captured the “lock-in” phenomenon



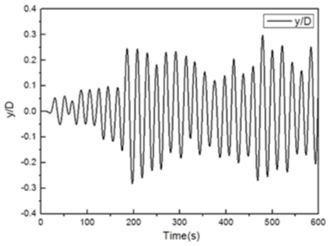
拖曳力随折合速度变化曲线

Drag force at different reduced velocities



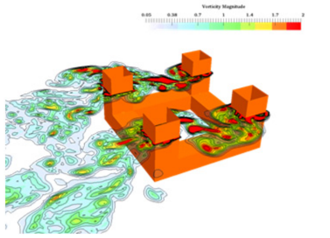
45度来流角工况下半潜平台涡量场

VIM of semi-submersible at 45 degree current heading – Vorticity contour



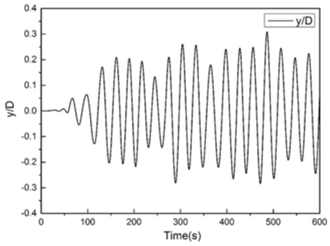
45度来流角工况下半潜平台涡激运动横向运动时历曲线

VIM of semi-submersible at 45 degree current heading – Time histories of transverse motion



0度来流角工况下半潜平台尾涡结构

VIM of semi-submersible at 0 degree current heading – Vorticity contour

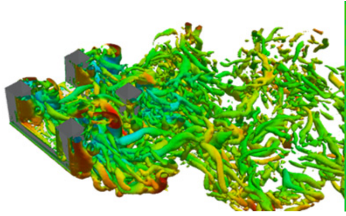


0度来流角工况下半潜平台涡激运动横向运动时历曲线

VIM of semi-submersible at 0 degree current heading – Time histories of sway motion

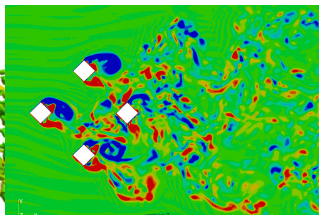
可进行不同来流角度工况下半潜式平台涡激运动的数值模拟，分析平台的运动以及泻涡特征。

The solver can perform VIM simulations under different incident current angles and compare the motion of platform and vortex shedding patterns.



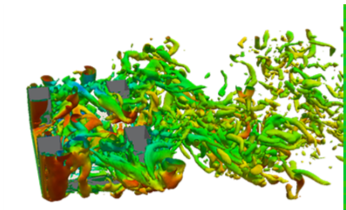
45度来流角工况下半潜平台尾涡结构

VIM of semi-submersible at 45 degree current heading – Wake structures



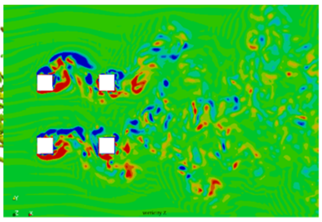
45度来流角工况下半潜平台水平截面涡量场

VIM of semi-submersible at 45 degree current heading – Vorticity contour on half-draft plane



0度来流角工况下半潜平台尾涡结构

VIM of semi-submersible at 0 degree current heading – Wake structures

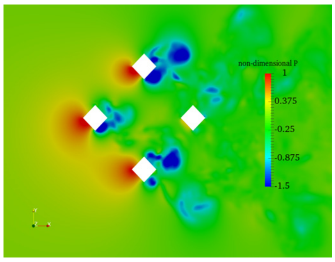


0度来流角工况下半潜平台水平截面涡量场

VIM of semi-submersible at 0 degree current heading – Vorticity contour on half-draft plane

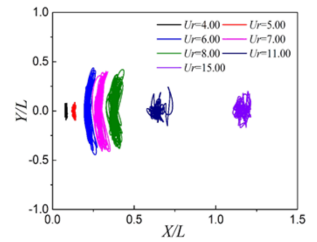
可对不同立柱形状的半潜式平台进行涡激运动数值模拟，从流场的角度分析立柱形状对半潜式平台涡激运动响应的影响。

The solver can analyze the geometric effect of column shape on VIM response of semi-submersibles and explain the difference from the perspective of flow field.



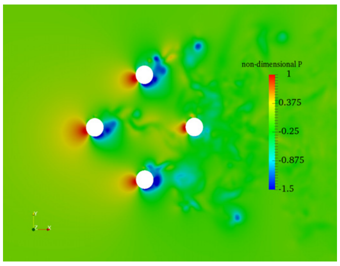
方柱式半潜平台水平截面压力场

VIM of square-column semi-submersible – Pressure contour



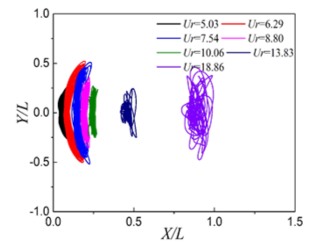
方柱式半潜平台涡激运动轨迹

VIM of square-column semi-submersible – Motion trajectories



圆柱式半潜平台水平截面压力场

VIM of cylindrical column semi-submersible – Pressure contour



圆柱式半潜平台涡激运动轨迹

VIM of cylindrical column semi-submersible – Motion trajectories

vim-FOAM-SJTU应用于对柱式半潜平台涡激运动问题

Applications of vim-FOAM-SJTU on VIM of paired-column semi-submersible platform

对柱式半潜平台涡激运动模拟

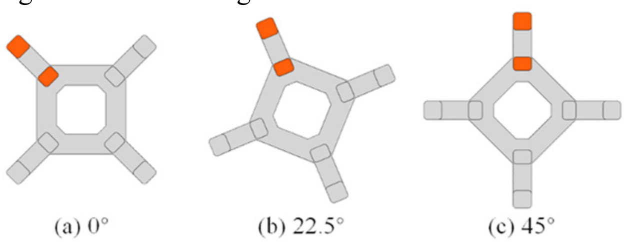
VIM of Paired-Column Semi-Submersible

vim-FOAM-SJTU求解器还可对新型的对柱式半潜平台涡激运动进行模拟，分析其涡激运动响应特性。

The solver can predict the VIM response of a concept design Paired-Column Semi-Submersible (PC Semi) platform.

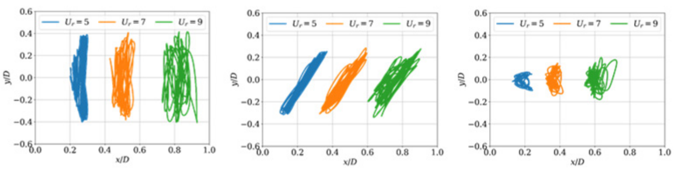
分析对柱式半潜平台在不同来流角度情况下的涡激运动响应特性，得知其在0度和22.5度来流角下的运动轨迹较为规则，而在45度来流角下的运动轨迹程不规则形状。

Comparing the VIM response at different current headings, it shows the VIM trajectories at 0 and 22.5 degree current headings are more regular than that at 45 degree current heading.



对柱式半潜平台来流角度定义

Definition of current heading for Paired-Column Semi-Submersible

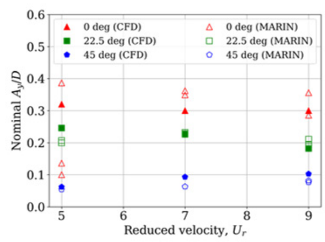


不同来流角度下的涡激运动轨迹

Motion trajectories at different current headings

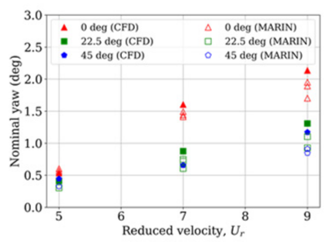
计算得到的标称横向运动和首摇运动响应与模型试验对比吻合良好。

The nominal transverse and yaw responses obtained by the solver are in good agreement with model test.



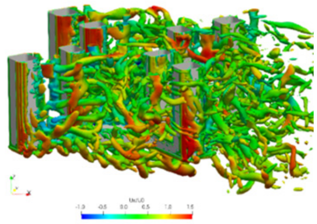
标称横向运动响应随折合速度变化

Nominal transverse response at different reduced velocities



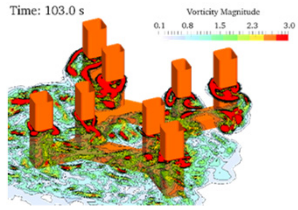
标称首摇运动响应随折合速度变化

Nominal yaw response at different reduced velocities



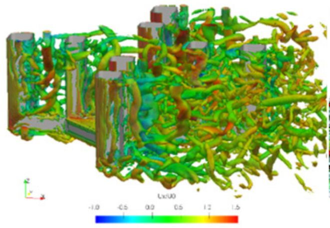
来流角0度对柱式半潜平台尾涡结构

VIM of PC Semi at 0 degree current heading – Wake structures



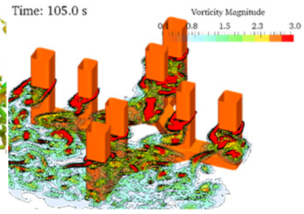
来流角0度z/L=0.5平面内涡结构

VIM of PC Semi at 0 degree current heading – Vorticity contour at half-draft plane



来流角45度对柱式半潜平台尾涡结构

VIM of PC Semi at 45 degree current heading – Wake structures

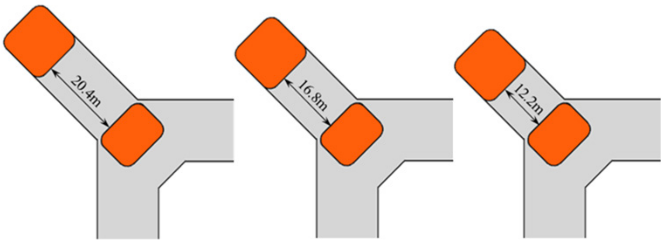


来流角45度z/L=0.5平面内涡结构

VIM of PC Semi at 45 degree current heading – Vorticity contour at half-draft plane

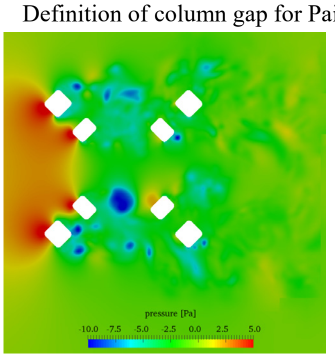
可分析几何外形细微变化对半潜平台涡激运动的影响，如外柱和内柱间距对涡激运动的影响。

The solver can analyze the effects of geometrical subtle variation (such as gap between outer and inner columns) on VIM response of a Paired-Column Semi-Submersible.



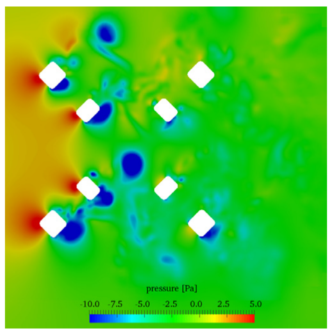
对柱式半潜平台内外柱间距定义

Definition of column gap for Paired-Column Semi-Submersible



小内外柱间距对柱式平台涡激运动压力云图

Pressure contour at half-draft plane for small gap PC Semi



大内外柱间距对柱式平台涡激运动压力云图

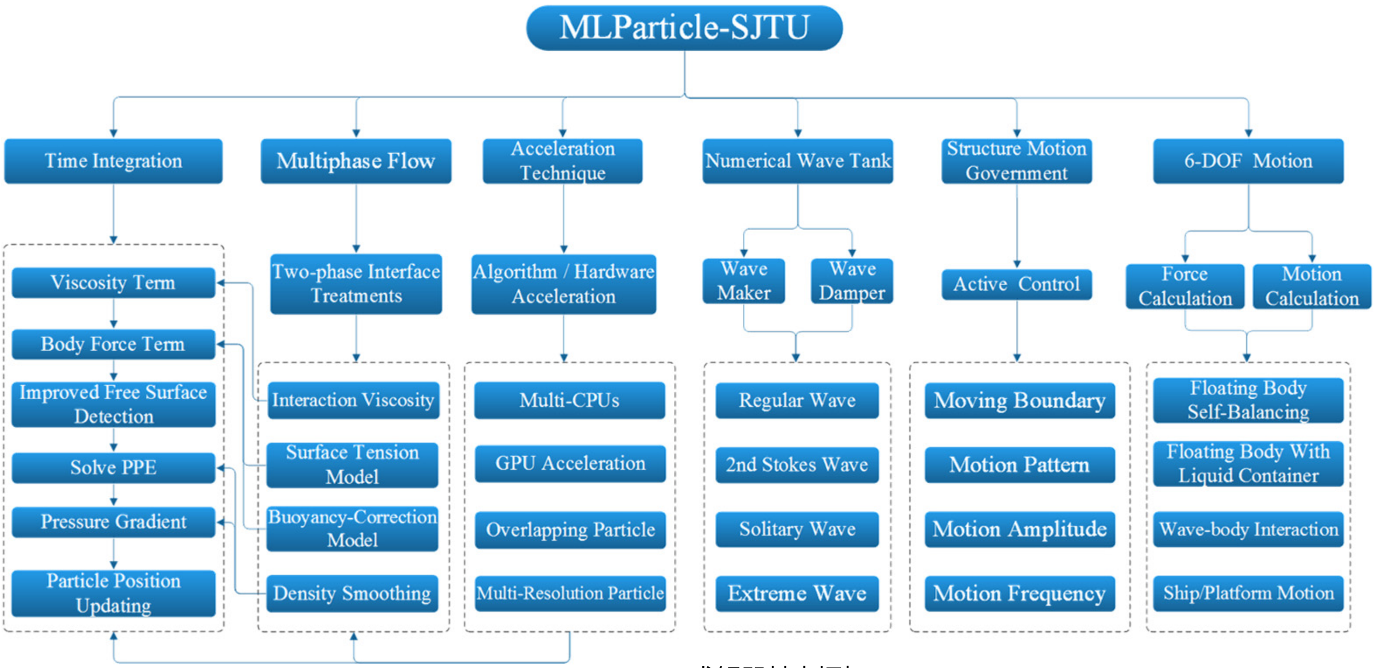
Pressure contour at half-draft plane for large gap PC Semi

船舶与海洋工程无网格粒子法求解器 MLParticle-SJTU

Mesh-less Particle Solver for Marine Hydrodynamics MLParticle-SJTU

上海交通大学CMHL研究中心基于改进的移动粒子半隐式方法，自主开发了船舶与海洋工程无网格粒子法求解器软件MLParticle-SJTU，包括粒子积分及运动、数值波浪水池、六自由度运动、加速计算、多相流等多个模块。求解器通过改进核函数、自由面粒子判别方法和压力泊松方程源项表达式等途径，集成多CPU并行、GPU并行、重叠粒子及多分辨率粒子等加速技术，抑制了压力震荡问题，极大提高了计算效率，能有效模拟液舱晃荡、溃坝流、物体入水、涌潮波、船舶在波浪上的运动、甲板上浪、多相流等复杂剧烈流动问题。

CMHL of SJTU developed an in-house solver MLParticle-SJTU based on improved MPS method, including particle integral, numerical wave tank, 6-DOF motion, acceleration techniques and multiphase flow modules. By modifying kernel function, free surface detection and source term of pressure Poisson equation, the improved MPS method can suppress pressure oscillation effectively. And the calculation efficiency of solver is improved dramatically by applying acceleration techniques such as multi-CPU parallel, GPU parallel, overlapping particle technique and multi-resolution particle technique. Therefore, MLParticle-SJTU solver can simulate various violent flow problems including liquid sloshing, dam-break flows, water entry, tidal wave, ship motion in waves, green water, multiphase flows, etc.



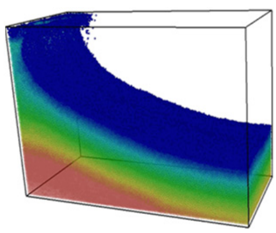
MLParticle-SJTU求解器基本框架
The framework of MLParticle-SJTU solver

液舱晃荡

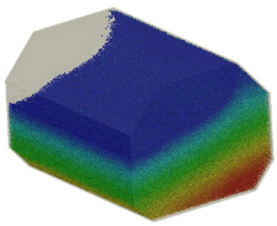
Liquid sloshing

MLParticle-SJTU求解器可对各液舱类型，包括方型液舱、LNG液舱的晃荡问题进行模拟，还可对带环形隔板或十字隔板的液舱晃荡问题进行模拟。

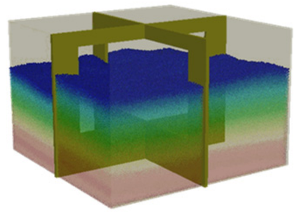
MLParticle-SJTU solver can simulate the liquid sloshing in different tanks, such as rectangular tanks, LNG tanks as well as tanks with ring baffles and cross-shape baffles.



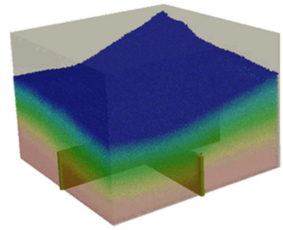
方型液舱晃荡
Liquid sloshing in rectangular tanks



LNG液舱晃荡
Liquid sloshing in LNG tanks



带环形隔板的液舱晃荡
Liquid sloshing in tanks with ring baffles



带十字隔板的液舱晃荡
Liquid sloshing in tanks with cross-shape baffles

MLParticle-SJTU求解器在溃坝流和物体入水问题上的应用

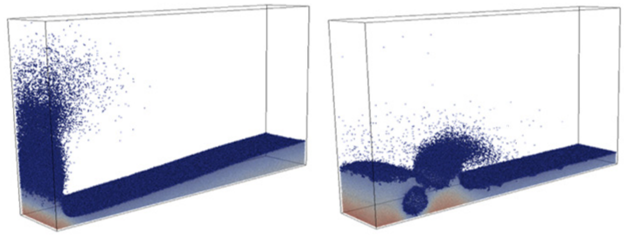
Applications of MLParticle-SJTU to Dam-break Flows and Water Entry Problems

溃坝流

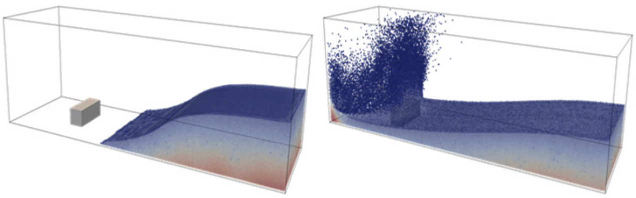
Dam-break flows

溃坝流问题研究对于水库下游防灾减灾有着重要意义，MLParticle-SJTU求解器可模拟溃坝过程中的剧烈砰击流动现象。

The MLParticle-SJTU solver can simulate the violent impact phenomena in the process of dam-break flows, which are of great importance to the prevention and reduction of disaster in the downstream reservoir.



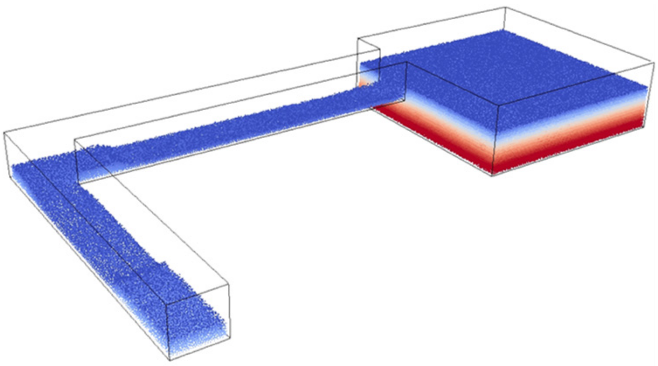
溃坝流砰击壁面
Dam-break flow impacting onto the wall



溃坝流砰击障碍物
Dam-break flow impacting onto an obstacle

MLParticle-SJTU求解器可模拟溃坝流在细长河道中的行进过程，能准确地预报溃坝流的到达时间和波高峰值，对于溃坝问题具有重要工程意义。

MLParticle-SJTU solver can simulate the process of the dam-break flow in the long and narrow channel, and can accurately predict the arrival time and wave peak value of the dam-break flow, which are of great engineering significance to the dam-break problems.



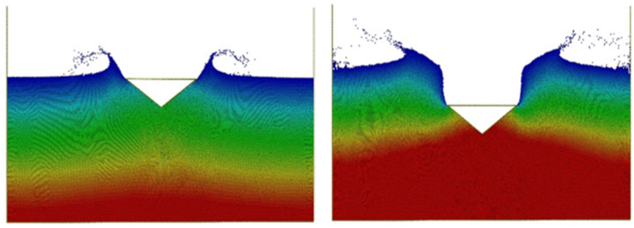
细长河道溃坝流
Dam-break flow in long and narrow channel

物体入水

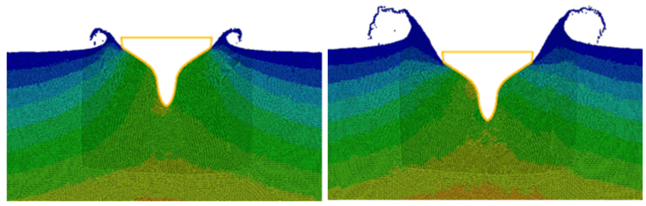
Water entry problems

MLParticle-SJTU求解器可模拟不同剖面物体的入水问题，得到入水过程自由面的剧烈变形，并能准确预测物体受到的砰击压力和运动状态。

MLParticle-SJTU solver can simulate the water entry problems of the objects with different section shapes, and obtain the nonlinear deformation of the free surface during the water entry process. Besides, the pressure and motion of the objects can also be accurately predicted.



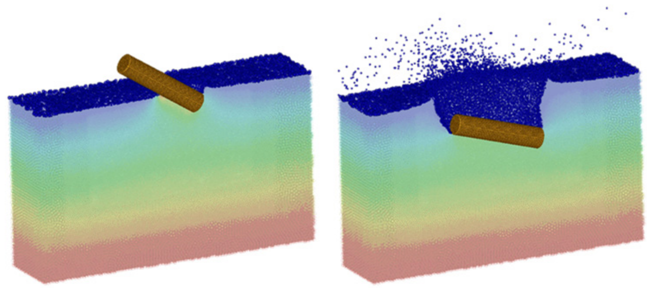
楔形物体入水
Water entry of a wedge object



船型剖面入水
Water entry of a ship section

应用六自由度运动模块，求解器还可以对带倾角的物体入水问题进行模拟，得到多自由度下的运动轨迹与实验数据吻合。

Appling the 6-DOF motion module, the solver can also simulate the water entry of an object with an inclined angle. The multi-DOF motion trajectory of the object shows good agreement with the experimental data.



带倾角圆柱入水
Water entry of an inclined cylinder

MLParticle-SJTU求解器在波物相互作用问题上的应用

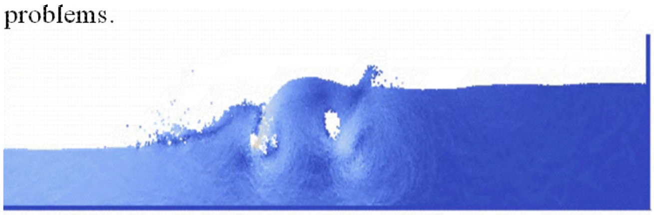
Applications of MLParticle-SJTU to Wave-structure Interaction Problems

涌潮波与波浪爬坡

Tidal waves and wave run-up

MLParticle-SJTU求解器可模拟涌潮波演化过程，也可有效模拟波浪爬坡过程中出现的波浪破碎现象。应用重叠粒子和多分辨率粒子技术，还可以实现大区域和多尺度复杂流动问题的模拟。

MLParticle-SJTU solver can simulate the evolution of tidal waves, as well as breaking waves in the process of wave climbing. By applying overlapping particle technique and multi-resolution particle technique, the solver can simulate the large-scale complicated flow problems.



波破碎涌潮
Breaking tidal wave

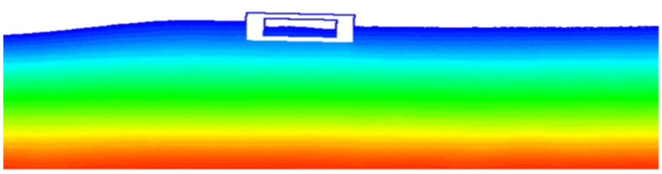
波浪在斜坡上破碎
Wave breaking on the slope

波浪作用下的液舱晃荡

Wave-induced liquid sloshing

MLParticle-SJTU求解器可对波浪-浮体-晃荡波的耦合作用进行分析，模拟浮体内外流场的流动细节，波浪的传播，舱内自由液面的演化过程等，为真实的带液舱船舶安全运行提供数据支撑。

MLParticle-SJTU solver can be used to analyze the coupling effect of wave, floating body and sloshing wave. And it can simulate the inside and outside flow field, the spreading process of the waves, evolution of free surface inside the tank, and provide useful data for safety operation of ships with liquid tanks.



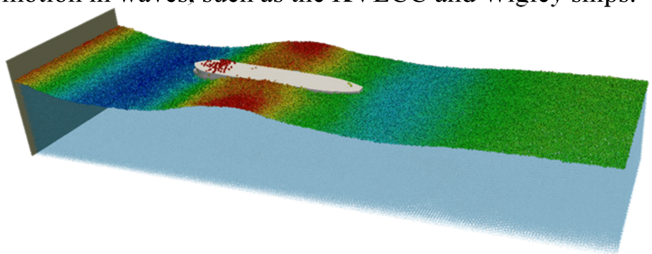
波浪上带液舱浮体的晃荡
Wave-induced liquid sloshing in the floating body

船舶在波浪上的运动

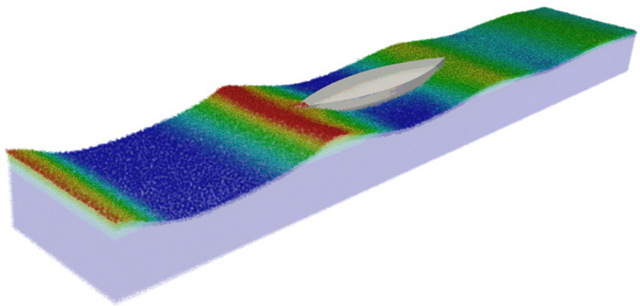
Ship motion in waves

MLParticle-SJTU还具备数值造波、消波、六自由度运动功能，成功模拟了波浪作用下不同船型的运动，包括KVLCC船和Wigley船。

With the help of numerical piston-type wavemaker, MLParticle-SJTU solver can be used to simulate ship motion in waves, such as the KVLCC and Wigley ships.



KVLCC船在波浪上的运动
Motion of KVLCC ship in waves



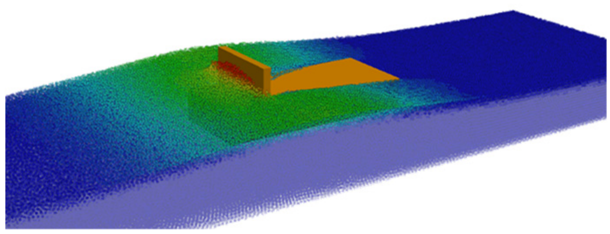
Wigley船在波浪上的运动
Motion of Wigley ship in waves

甲板上浪

Green water

甲板上浪现象发生时，波浪可能对船舶上层结构造成严重的破坏。MLParticle-SJTU求解器可对FPSO船的甲板上浪现象进行模拟，并精确预报上层建筑所受的砰击压力。

When the green water occurs, the waves may cause severe damage to the superstructure of ship. MLParticle-SJTU solver can simulate the green water phenomena on the FPSOs and accurately forecast the slamming pressure acting on the superstructure.



甲板上浪
Green water simulation

MLParticle-SJTU求解器在多相流问题上的应用

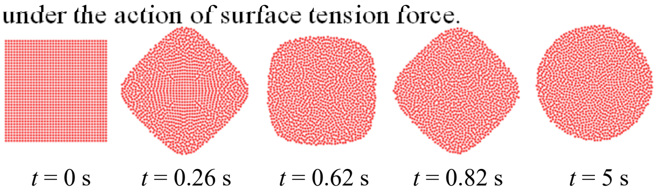
Applications of MLParticle-SJTU to Multiphase Flows

液滴变形

Deformation of droplet

通过引入一系列相界面处理方法，MLParticle-SJTU求解器实现了对复杂多相流问题的数值模拟。其中，通过引入表面张力模型，成功模拟了方形液滴在表面张力作用下逐渐变圆的过程。

Through introducing a series of interface treatments, MLParticle-SJTU solver has realized the numerical simulation of complex multiphase flows. Specially, a surface tension model is introduced to simulate the deformation of the square-droplet into a round shape under the action of surface tension force.



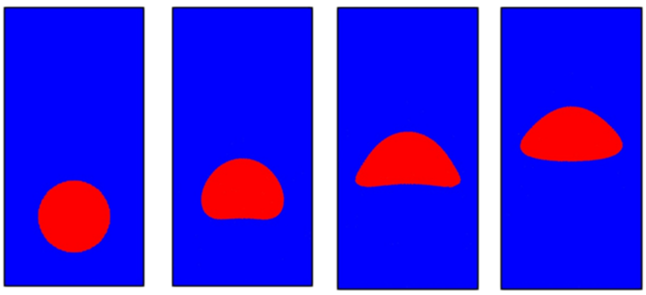
方形液滴变形
Deformation of the square-droplet

气泡上升

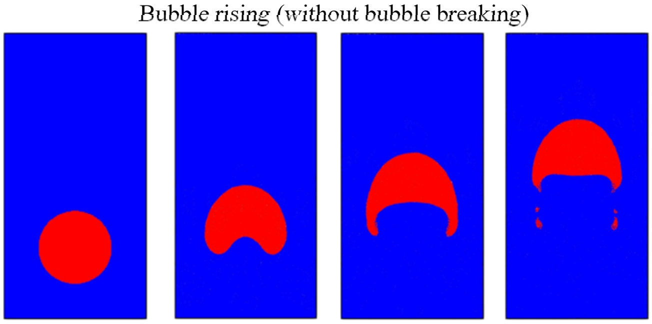
Bubble rising

利用MLParticle-SJTU求解器的多相流模块，可模拟气泡在液体中的上升过程，并能较好地捕捉气泡和液体交界面的剧烈形态变化。

The multiphase module of MLParticle-SJTU solver can be used to simulate the rising process of bubbles in liquid, and the sharp morphological changes of interface between bubbles and liquid can be well captured.



气泡上升（无气泡破碎）
Bubble rising (without bubble breaking)



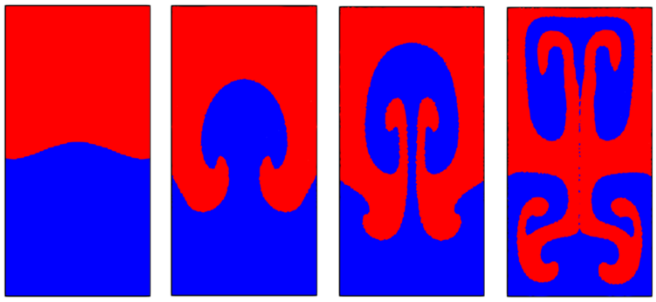
气泡上升（有气泡破碎）
Bubble rising (with bubble breaking)

瑞利-泰勒不稳定性

Rayleigh-Taylor instability

瑞利-泰勒不稳定问题是指当密度大的流体处于密度小的流体之上时产生的一种交界面不稳定现象。利用MLParticle-SJTU求解器的多相流模块可对该现象进行数值模拟。

Rayleigh-Taylor instability is an interface instability that occurs when a denser fluid is above a lighter one. This phenomenon can be numerically simulated by using the multiphase module of MLParticle-SJTU solver.



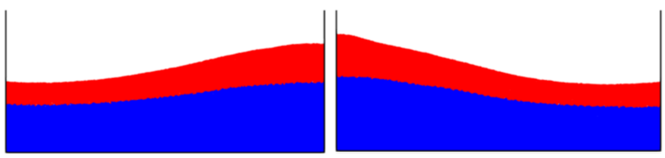
瑞利-泰勒不稳定性
Rayleigh-Taylor instability

多层不同液体晃荡

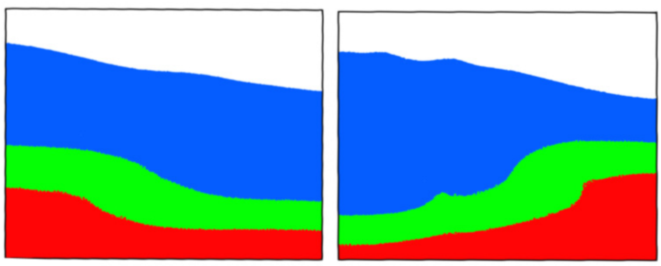
Sloshing flows of the layered fluid

多相流模块还可对多层不同液体的液舱晃荡进行模拟。该现象广泛存在于FPSO和采油平台等海洋结构物上的油水分离器中，对分离效率有重要影响。

Multiphase module can also be used to simulate sloshing flows of the layered fluid. This phenomenon exists widely in the oil-water separators installed on FPSOs and oil platforms, and has significant influence on separation efficiency.



双层不同液体晃荡
Two-layer-liquid sloshing



三层不同液体晃荡
Three-layer-liquid sloshing

MLParticle-SJTU求解器加速技术的应用

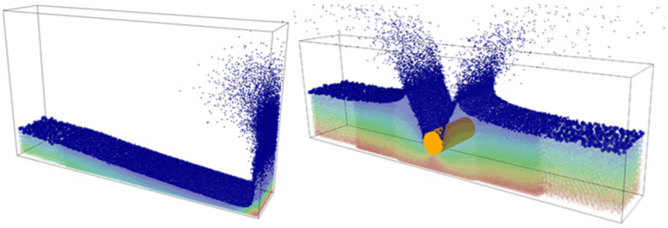
Applications of Acceleration Techniques of MLParticle-SJTU

多分辨率粒子技术

Multi-resolution particle technique

多分辨率粒子加速技术采用在流场中布置不同尺寸粒子的方式，实现了重点关注区域流场的局部细化，从而降低了数值计算中需要的粒子数量，提高了计算效率。该技术已成功应用于三维入水问题、溃坝流动问题。

Depending on the arrangement of particles with different sizes in the flow field, multi-resolution particle technology realizes a high resolution of particles in important areas, so as to reduce the number of particles necessary for numerical accuracy, and improve the calculation efficiency. The technology has been applied to the problems of 3D water entry and dam-break flow.



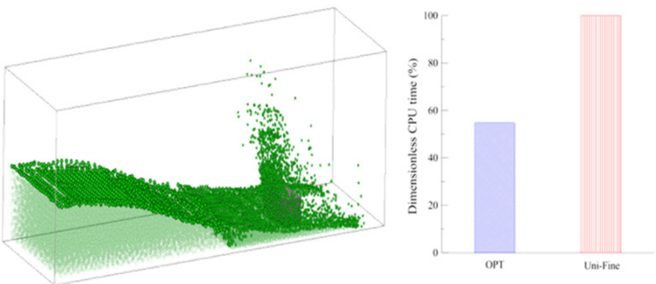
基于多分辨率粒子技术的溃坝流和物体入水
Water entry and dam-break flow based on multi-resolution particle technique

重叠粒子技术

Overlapping particle technique

重叠粒子加速技术采用大尺度的粒子填充于整个流动区域，而用小尺度的粒子布置在局部重点关注的区域，能够做到在保证流场计算精度的同时，降低仿真时间。

Overlapping particle technology disperses the whole calculation domain with coarse particles and distribute extra fine particles in important areas. With overlapping particle technique, the simulation time can be reduced while calculation accuracy is ensured.



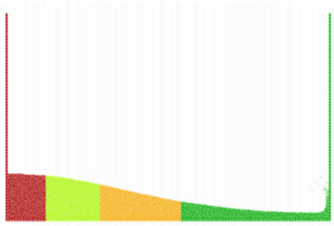
基于重叠粒子技术的带障碍物溃坝流
Dam-break flow with an obstacle based on overlapping particle technique

多CPU并行技术

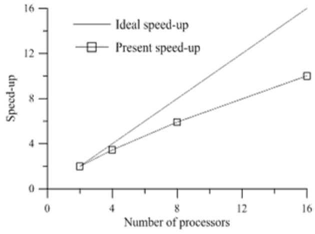
Multi-CPU parallel technique

为了提高计算效率，模拟大规模复杂流动问题，开发了采用动态负载平衡方法的多CPU并行加速技术，16核CPU并行加速比可以达到10倍。

In order to improve the calculation efficiency in large-scale complex flow problems, the multi-CPU parallel acceleration technique is developed based on dynamic load balancing. By using 16 CPU cores, the speed-up ratio can reach 10 times compared with the serial computing.



基于多CPU并行技术的溃坝流
Dam-break flow based on multi-CPU parallel technique



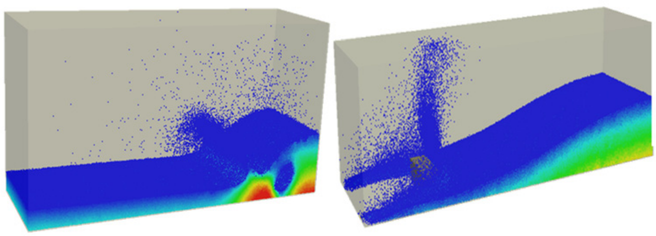
不同CPU核数的加速比
Speed-up ratio with different CPU cores

GPU并行技术

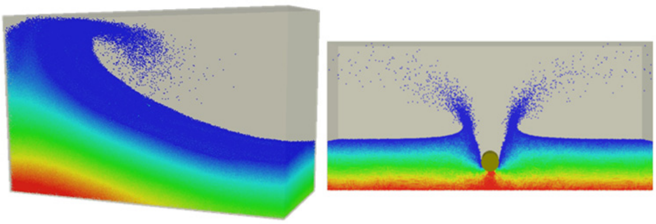
GPU parallel technique

结合GPU显卡拥有众多计算单元和高浮点计算能力的特点，求解器还采用GPU并行加速技术，大幅提高MPS方法计算效率，计算速度可提高30倍。

GPU owns numerous calculation units and high floating-point computation ability. In MLParticle-SJTU solver, the GPU parallel acceleration technique is developed and can increase calculation efficiency by 30 times compared with CPU computing.



基于GPU并行技术的溃坝流
Dam-break flow based on GPU parallel technique



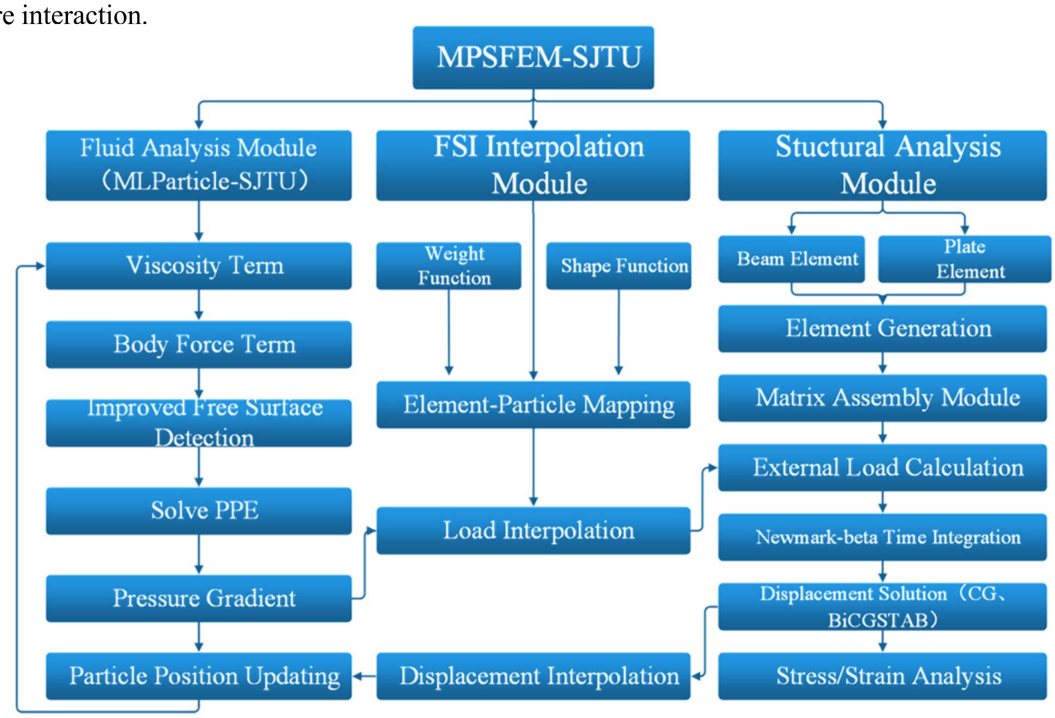
基于GPU并行技术的液舱晃荡和圆柱入水
Liquid sloshing and water entry based on GPU parallel technique

船海流固耦合粒子法-有限元求解器 MPSFEM-SJTU

FSI Solver Based on Mesh-less Particle and Finite Element Method MPSFEM-SJTU

上海交通大学CMHL研究中心在自主开发的无网格粒子法求解器MLParticle-SJTU的基础上，扩展开发了有限元法的结构场求解模块以及流-固界面数据插值传递模块，形成了船舶与海洋工程流固耦合粒子法-有限元求解器软件MPSFEM-SJTU。该求解器结合了MPS方法在剧烈自由面流动问题模拟的优势，以及FEM方法在结构动力学响应分析中的可靠性，可应用于船舶与海洋工程中多种典型流固耦合问题的数值仿真，例如溃坝流与弹性结构物的相互作用、弹性液舱内的晃荡、波浪与弹性结构物的相互作用等问题。

CMHL of SJTU implemented the FEM module and interfacial interpolation module on the basis of MLParticle-SJTU solver, to develop the fluid-structure interaction solver MPSFEM-SJTU. This solver combines advantages of MPS method while dealing with the problems of violent flow and structural deformation with stabilities of FEM while coping with the structural dynamic response. So it can be applied to the FSI problems in the field of naval architecture and ocean engineering, such as the dam-break flows interacting with the elastic structures, liquid sloshing in an elastic tank and wave-structure interaction.



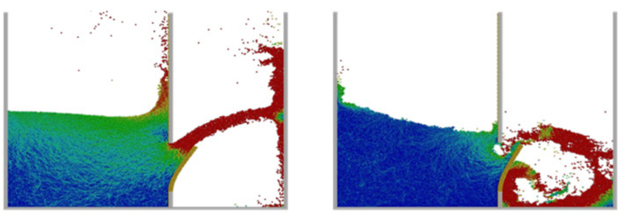
MPSFEM-SJTU求解器框架
The framework of MPSFEM-SJTU

溃坝流固耦合

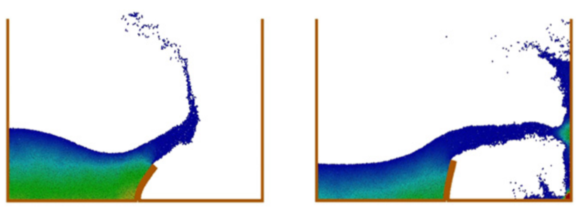
Dam-break fluid-structure interaction

当遭受溃坝流时，结构物可能会产生显著的变形甚至破坏。MPSFEM-SJTU求解器能够很好地模拟溃坝流对弹性结构物的砰击作用，得到的弹性结构物变形及自由面演化均与文献结果吻合良好。

While encountering dam-break flow, the elastic structures may produce considerable deformation or damage. The MPSFEM-SJTU solver can be employed to simulate the interaction between dam-break flows and the elastic structures. The deformation of the elastic structure and evolution of free surface are in good agreement with those of the references.



溃坝流对弹性窗体的砰击
Dam-break flow interacting with elastic window



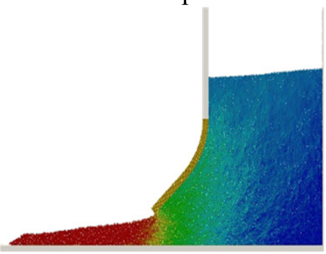
溃坝流对弹性障碍物的砰击
Dam-break flow interacting with elastic obstacle

MPSFEM-SJTU在溃坝流及液舱晃荡流固耦合问题上的应用

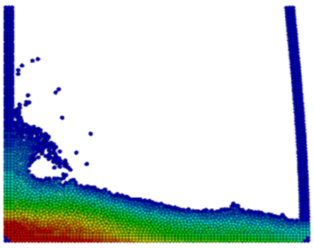
Applications of MPSFEM-SJTU to Dam-break Flows and Liquid Sloshing Fluid-structure Interaction Problems

利用该求解器，可以模拟泄洪流与弹性闸门之间的耦合作用问题。计算的结构运动响应、水槽液深变化和实验结果基本一致。求解器还可以模拟溃坝流对水槽侧壁的砰击作用，得到自由面形状与已发表数据一致。

By applying MPSFEM-SJTU solver, dam-break flow interaction with elastic gate is simulated. Dynamic responses of the structures and the evolution of water level in the tank present an agreement with the experiment. This solver can also simulate the dam-break flow with elastic wall. The shape of free surface is consistent with published data.



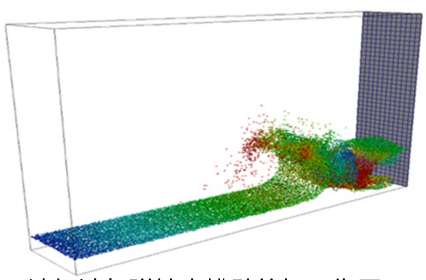
泄洪流与弹性闸门的相互作用
Dam-break flow interacting with elastic gate



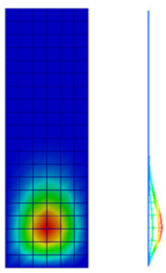
溃坝流对弹性侧壁的砰击作用
Dam-break flow interacting with elastic wall

在三维溃坝流对弹性壁面的砰击研究中，壁面变形具有显著的三维特征，呈现椭球形，并且由于壁面变形吸能作用，流体沿壁面爬升高度显著减小。

In study of three-dimensional dam-break flow with elastic wall, the deformation of elastic wall presents remarkable three-dimensional characteristics of ellipsoid. Besides, the height of jet flow is slightly lower due to the energy dissipation during the deformation of the lateral wall.



溃坝波与弹性水槽壁的相互作用
3-D dam-break flow with elastic wall



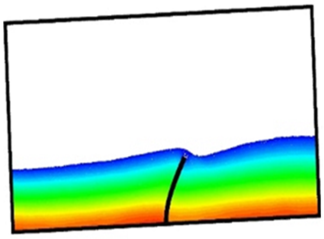
壁面椭球变形
Ellipsoid deformation of elastic wall

液舱晃荡流固耦合

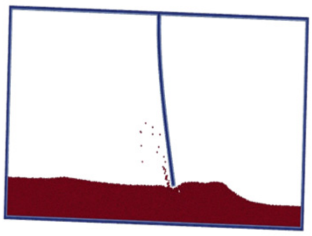
Liquid sloshing fluid-structure interaction

MPSFEM-SJTU求解器还可以应用于模拟液舱晃荡中的流固耦合问题。采用该求解器，模拟带底部弹性隔板的液舱晃荡，得到的隔板顶端横向位移与实验结果十分吻合。此外，还可以模拟带顶部弹性隔板的液舱晃荡，以及低充液率弹性液舱侧壁砰击和高充液率液舱顶部砰击等问题。

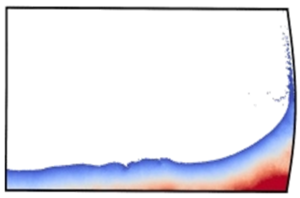
MPSFEM-SJTU solver can also simulate the liquid sloshing in an elastic tank. By applying the solver, the liquid sloshing with bottom elastic baffle can be simulated, and the obtained displacement at the tip of baffle is in good agreement with the experiment. In addition, the liquid sloshing with top elastic baffle, and liquid sloshing in the tanks of low filling ratio and high filling ratio can also be simulated.



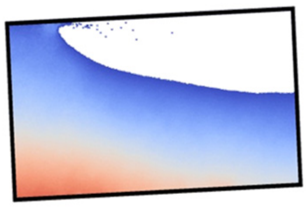
带底部弹性隔板的液舱晃荡
Liquid sloshing with bottom elastic baffle



带顶部弹性隔板的液舱晃荡
Liquid sloshing with top elastic baffle



低充液率弹性液舱侧壁砰击
Lateral impact in the tank of low filling ratio



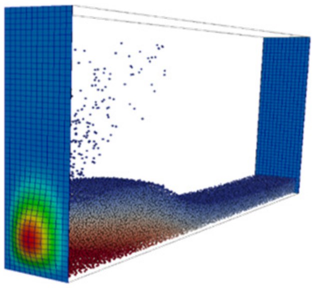
高充水率液弹性舱顶部砰击
Roof impact in the tank of high filling ratio

对不同激励下的三维液舱晃荡流固耦合进行数值模拟，得到了弹性舱壁在晃荡波周期性砰击作用下产生的振动、变形，以及模拟液体在壁面上的爬升、飞溅、坠落等过程。

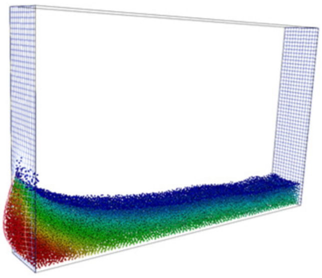
In the study of three-dimensional liquid sloshing, sloshing flow in elastic tank under different excitations is simulated using the MPSFEM-STJU solver. The vibration and deformation of the elastic wall can be clearly investigated under the periodic impact, and the evolution of free surface on the wall such as climbing, splashing and falling can also be obtained.

MPSFEM-SJTU在波物作用流固耦合问题上的应用

Applications of MPSFEM-SJTU to Wave-structure Interaction Problems



横荡激励下带弹性舱壁的液舱晃荡
Liquid sloshing with elastic wall under sway excitation



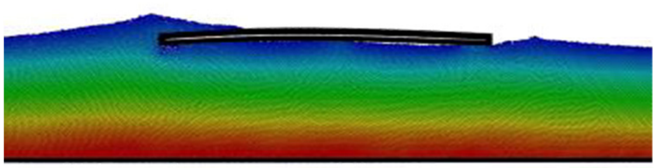
横摇激励下带弹性舱壁的液舱晃荡
Liquid sloshing with elastic wall under roll excitation

波物作用流固耦合

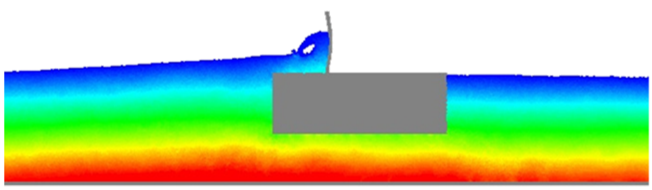
Wave-structure interaction

波物作用流固耦合是船舶与海洋工程中的常见问题，MPSFEM-SJTU求解器可以很好地模拟波浪砰击对弹性结构物的影响。采用MPSFEM-SJTU求解器，模拟孤立波对水平弹性板的砰击和甲板上浪对竖直弹性板的砰击，可以得到结构物变形和载荷分布，对船海实际工程问题具有一定指导意义。

Interaction between wave and structure is a hot issue in the field of naval architecture and ocean engineering, MPSFEM-SJTU solver can simulate the wave slamming onto the elastic structure well. By applying the solver, the impact induced by solitary wave slamming onto the flexible plate and the impact induced by the green-water slamming onto the superstructure can be simulated. The deformation of the structure and the pressure distribution are of great significance to the practical engineering problems.



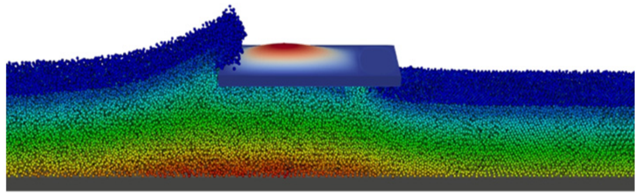
孤立波砰击水平弹性板
Solitary wave slamming onto the horizontal flexible plate



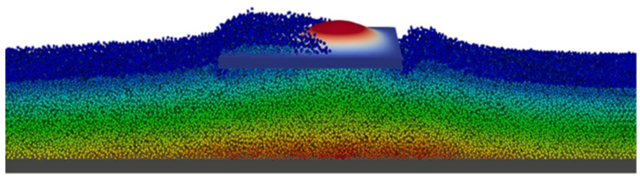
甲板上浪砰击竖直弹性板
Green water slamming onto the vertical flexible plate

该求解器还可模拟孤立波对三维水平弹性板的砰击，从甲板上浪、上托力峰值、砰击压力分布和流体速度场等方面研究结构物弹性对砰击的影响。通过观察不同时刻下板的位移和压力云图，发现砰击过程呈现出明显的三维特征。

The solitary wave impacting onto a three-dimensional flexible plate can also be simulated using the MPSFEM-SJTU solver. Some special characteristics regarding evolutions of free surface, peak of uplift force, distribution of impact pressures, velocity field of fluid and dynamic responses of the structures are presented to study the effects of flexibility of the structure. Remarkable three-dimensional characteristics can be seen by investigating the distribution of pressure and displacement on the flexible plate.

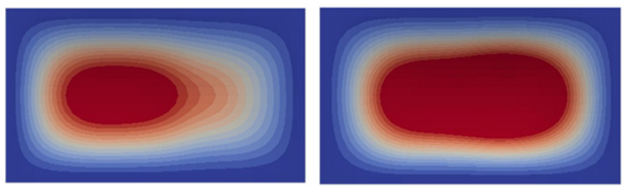


t =1.58 s



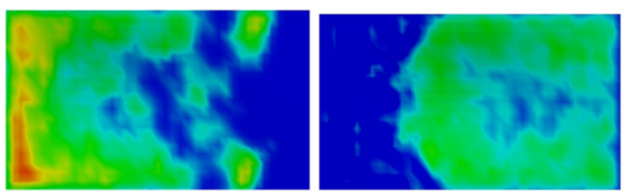
t =1.72 s

孤立波砰击三维水平弹性板
Solitary wave slamming onto a 3-D flexible plate



t =1.58 s　　*t* =1.72 s

不同时刻下板的位移云图
The displacement distribution on the flexible plate



t =1.58 s　　*t* =1.72 s

不同时刻下板的压力云图
The pressure distribution on the flexible plate