Shanghai Jiao Tong University

School of Naval Architecture, Ocean & Civil Engineering

State Key Laboratory of Ocean Engineering

Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration

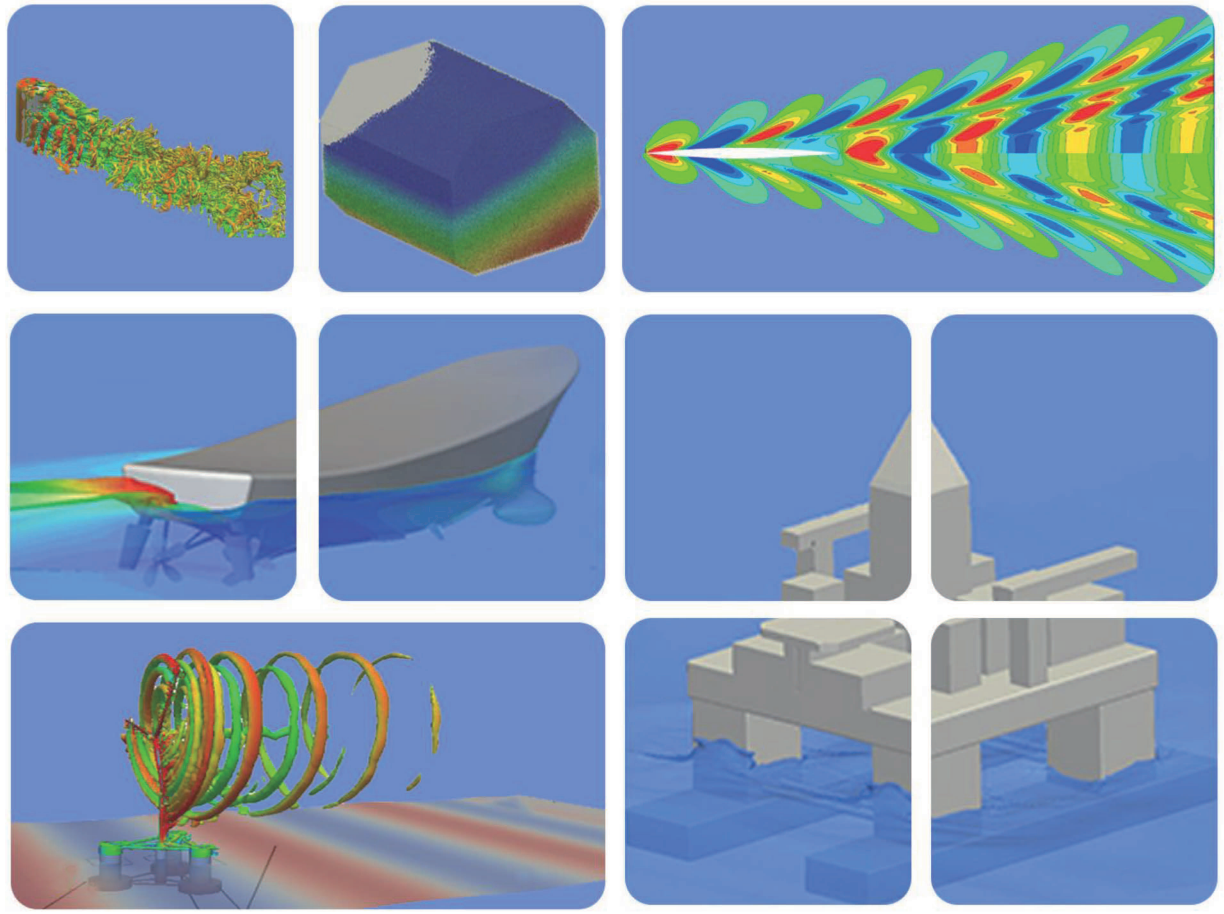
COMPUTATIONAL MARINE HYDRODYNAMICS LAB
SHANGHAI JIAO TONG UNIVERSITY

CONTACT US

Dongchuan Road 800, Minhang District, Shanghai
Postcode：200240
Tel：021-34207971
Tax：021-34207991
Email: dcwan@sjtu.edu.cn
Website：http://dcwan.sjtu.edu.cn

Printed in Dec. 2017

Integrated CFD Solvers for Ship and Ocean Engineering



http://dcwan.sjtu.edu.cn

COMPUTATIONAL MARINE HYDRODYNAMICS LAB
SHANGHAI JIAO TONG UNIVERSITY

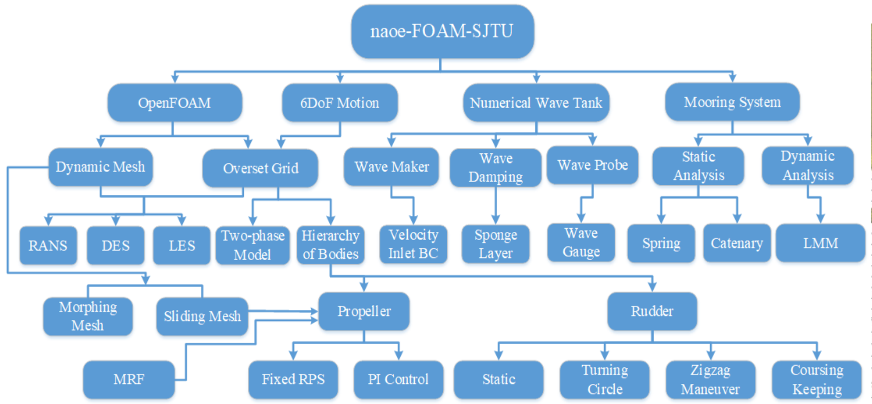
Integrated CFD Solvers for Ship and Ocean Engineering

Aiming at the requirement of ship and offshore structure intelligent designs, computational marine hydrodynamics lab (CMHL) developed the integrated simulation software system for naval architecture and ocean engineering problems. The system includes seven hydrodynamic solvers: the CFD solver naoe-FOAM-SJTU, solver of deep-sea riser vortex induced vibration viv-FOAM-SJTU, solver of floating offshore wind turbine FOWT-UALM-SJTU, solver of hull form optimization OPTShip-SJTU, solver of hydrodynamic meshless particel method MLParticle-SJTU and fluid-structure interaction solver MPSFEM-SJTU.

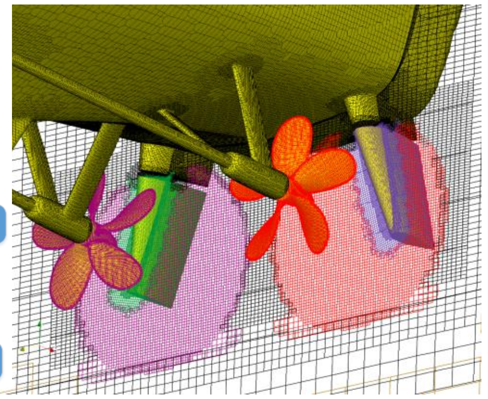
With the advantages of open source code and the professional development, the integrated simulation software system can run standalone or on high-performance parallel cluster. It can be applied to many complex hydrodynamic problems, such as ship resistance, propulsion, sea-keeping and ship maneuvering under complex sea states, multi-objects optimization for ship hulls, vortex induced vibrations of deep-sea flexible risers, vortex induced motions of large scale ocean platforms, performance of floating wind turbine with fully coupled aero-hydrodynamics and mooring systems, slamming on deck of ship, dam break flows, liquid sloshing in tanks and fluid structure interactions, etc. It can realize the numerical tests about the comprehensive hydrodynamic performances of ocean structures, provide reliable and effective software for the optimal design and development of ship and ocean structures.

Naval Architecture and Ocean Engineering CFD Solver (naoe-FOAM-SJTU)

CMHL developed naoe-FOAM-SJTU solver based on the open source platform OpenFOAM. The solver consists of 6DOF motion of ship and ocean structures, numerical wave tank and mooring system modules. By introducing overset grid technique into OpenFOAM, the solver is capable of achieving numerical simulation of the large amplitude motion and complex motion of ship hull-propeller-rudder system.

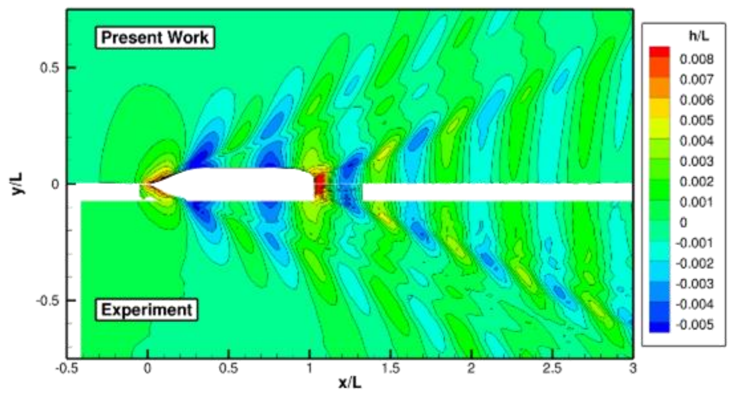


Module framework of naoe-FOAM-SJTU

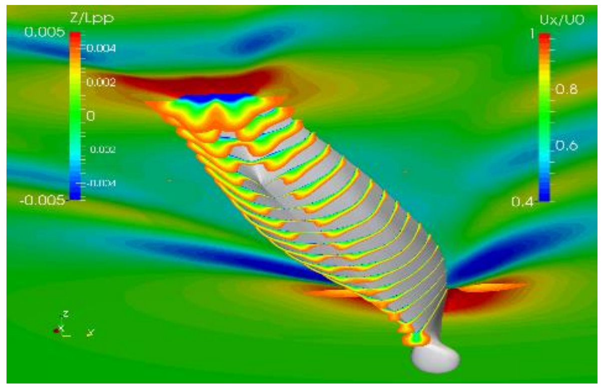

Overset grid around hull-propeller-rudder

Ship resistance

The solver can accurately predict the ship resistance, with error less than 3%. The predicted flow field information, such as wave pattern, boundary layer distribution and wake field, are consistent with the experiment.



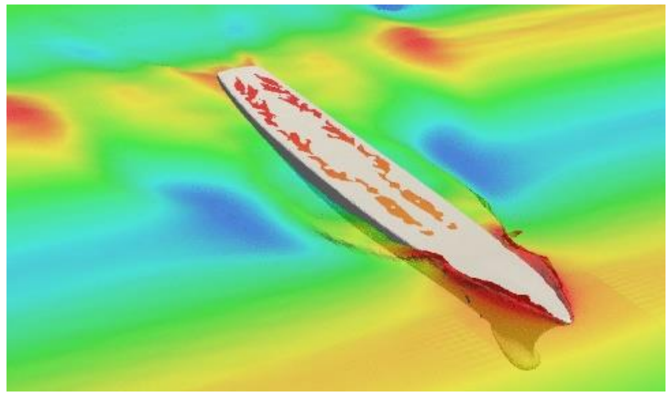
Comparison of wave pattern with measurement

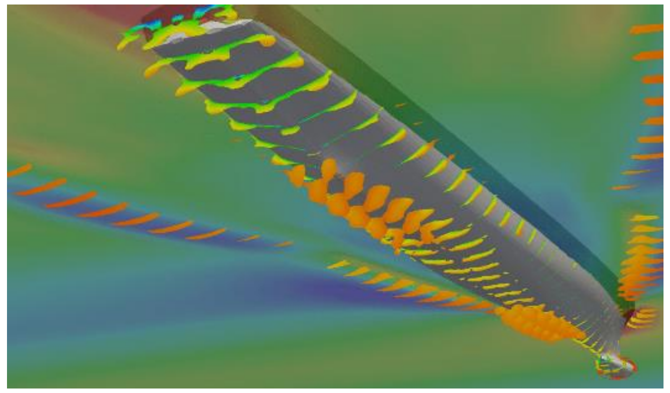


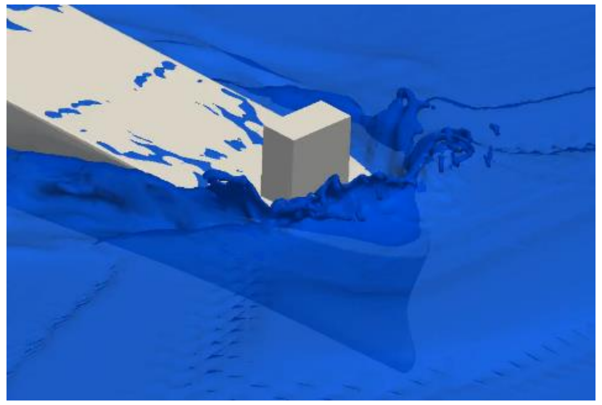
Boundary layer distribution

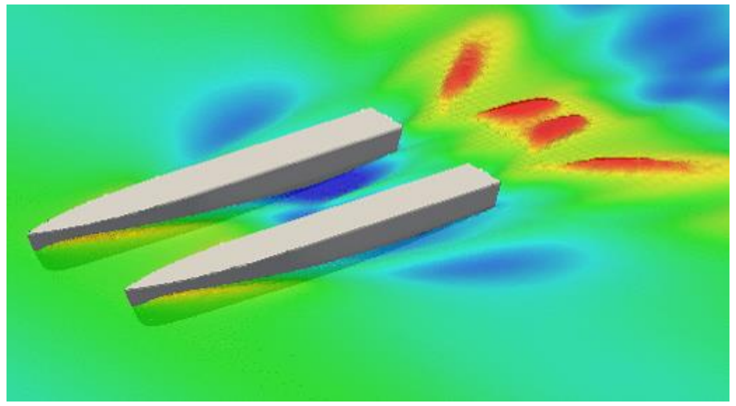
Seakeeping performance

Combined with 6DOF motion module and numerical wave tank, the solver can simulate the ship motion in waves, green water and slamming. The error of ship motion and wave added resistance is less than 5%.

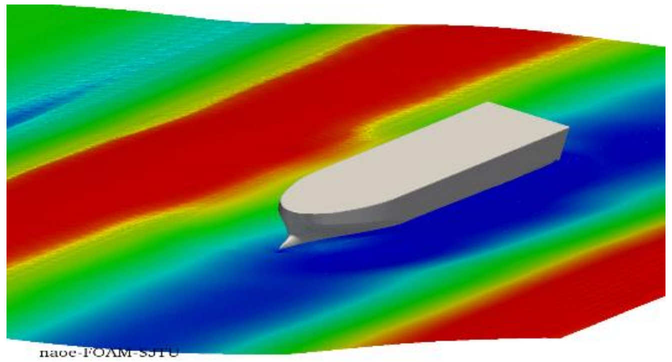

Sea-keeping and added resistance of DTMB5415

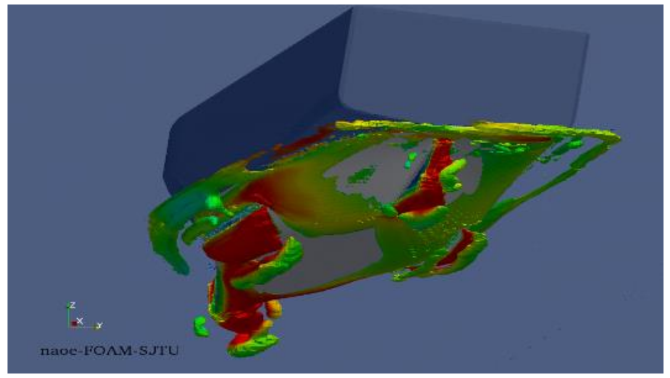

Vorticity distribution around the hull of DTMB5415


Green water of FPSO


Wave added resistance of catamaran

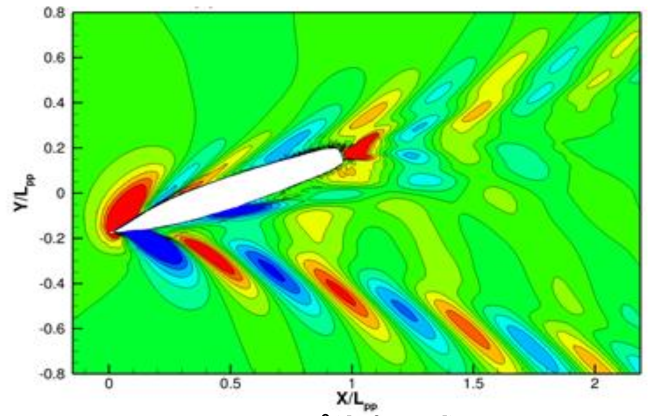
The simulation of forced roll motion and ship motion response in waves can be achieved via the solver. The predicted errors of roll damping and motion response are within 10%.


Roll motion in waves

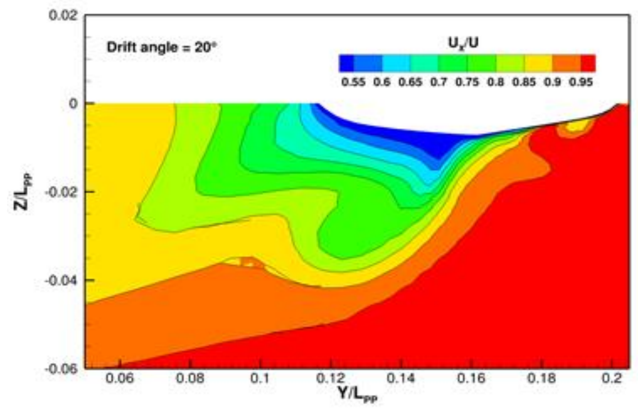


Vorticity distribution of the hull with bilge keel

With controlling ship 6DOF motion, complicated maneuvering tests, such as oblique towing test, pure sway and yaw tests, can be simulated. Through the regression analysis of maneuvering mathematical model, the errors of predicted hydrodynamic derivative are less than 10%.



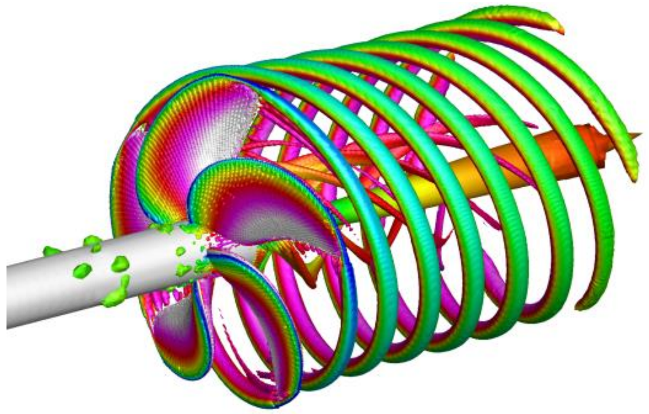
Wave pattern, 20°(drift angel), DTMB5415



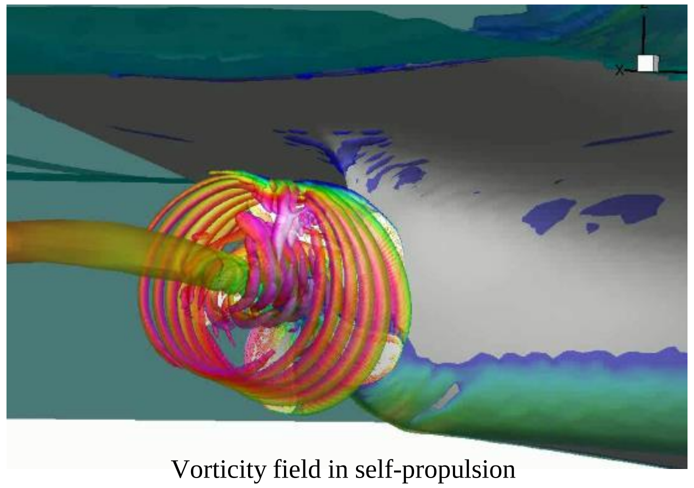
Wake field, 20°(drift angel), DTMB5415

Ship propulsion and maneuvering

By introducing overset grid method, the solver can predict the open water performance of propeller with the error within 3%. In the simulation of self-propulsion, the rotation speed of propeller automatically matches via PI controller, and the error of self-propulsion point is within 4%.

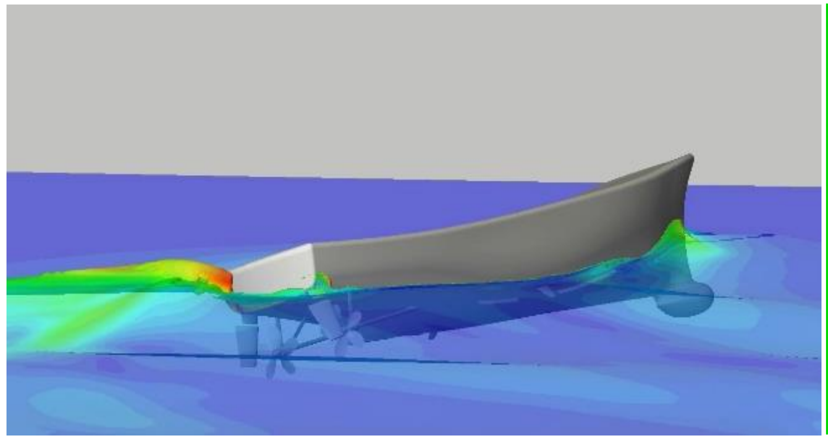


Vorticity field of propeller in open water

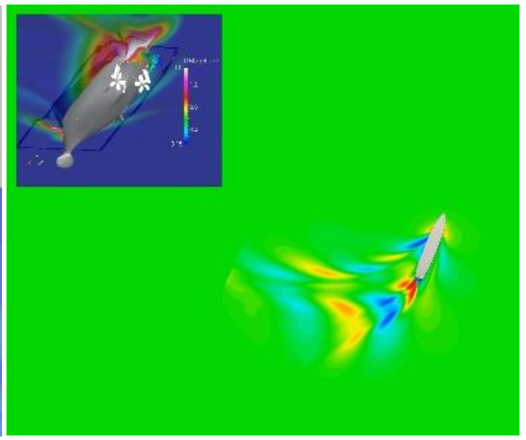


Vorticity field in self-propulsion

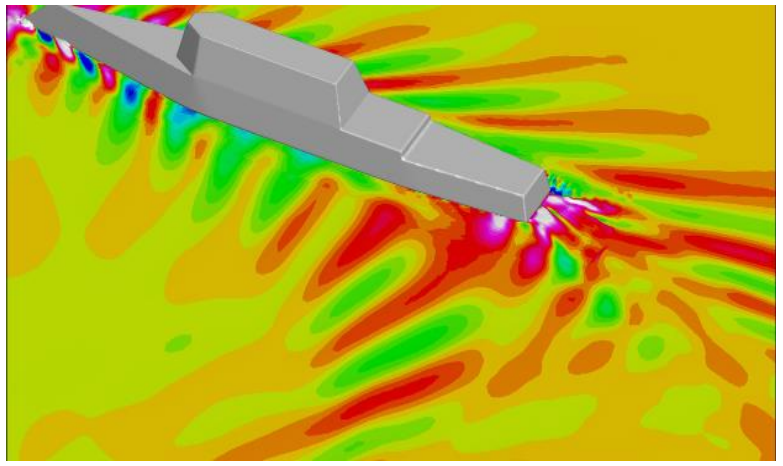
Combined with overset grid method, the maneuvering control module is developed to predict the free running ship maneuverability with direct rotating propellers and turning rudders.



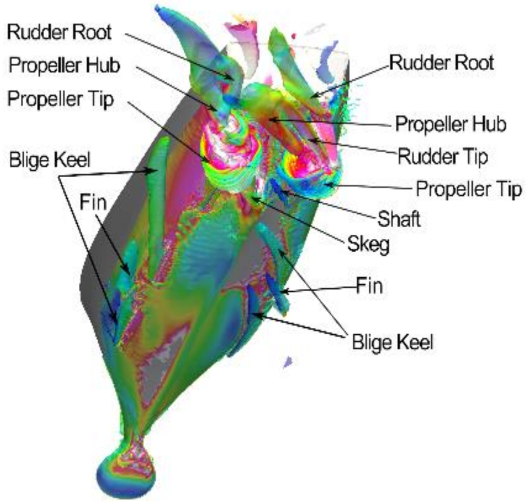
Zigzag maneuver in calm water



Turning circle maneuver in calm water



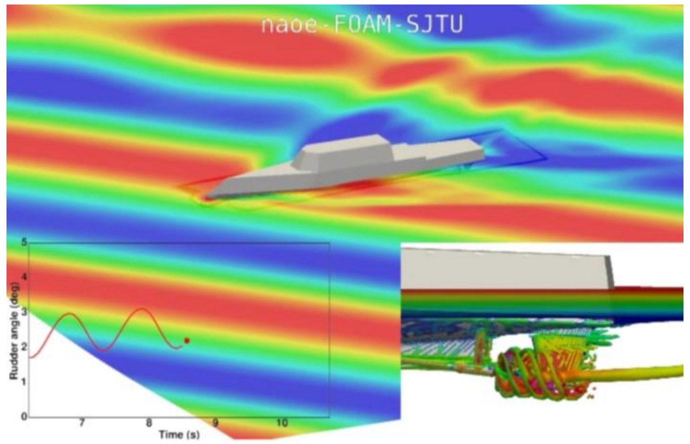
Wave pattern of turning circle maneuver in calm water



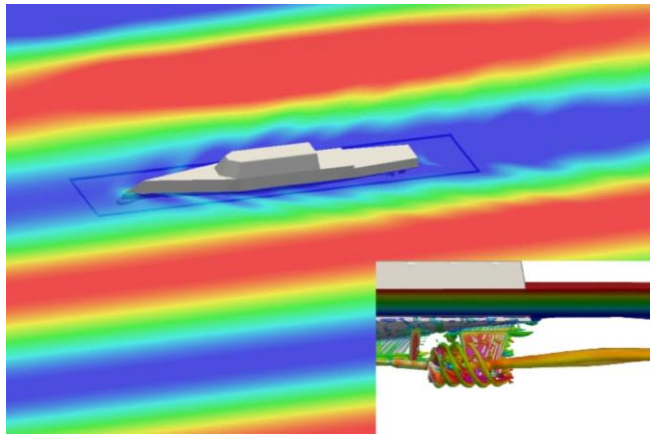
Vortical structures of turning circle maneuver

Ship maneuvering in waves

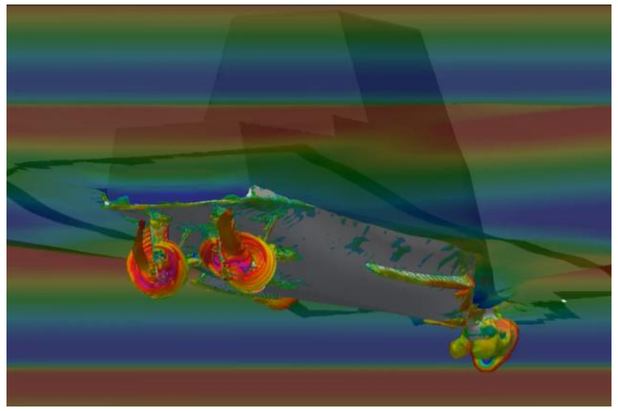
naoe-FOAM-SJTU solver can also directly simulate the free running ship maneuver in waves. So far, free running ship maneuvering simulations, such as course-keeping, zigzag and turning circle maneuver in complex waves are achieved. The numerical predictions of the ship maneuvering parameters are in good agreement with the test results. And it is proved that the present solver is an effective tool to predict ship maneuvering in waves.



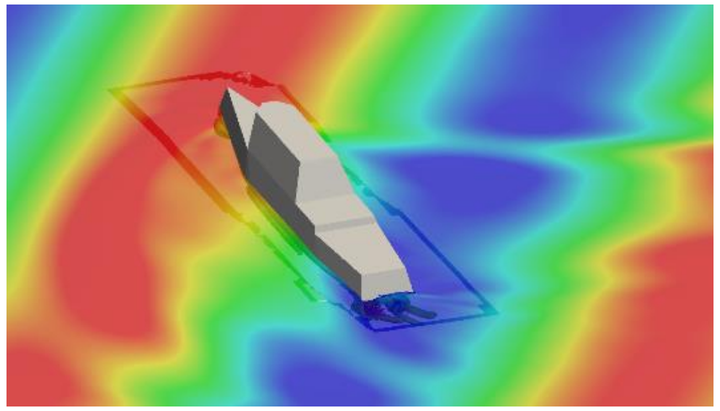
Course-keeping in bow quartering wave



Course-keeping in beam wave



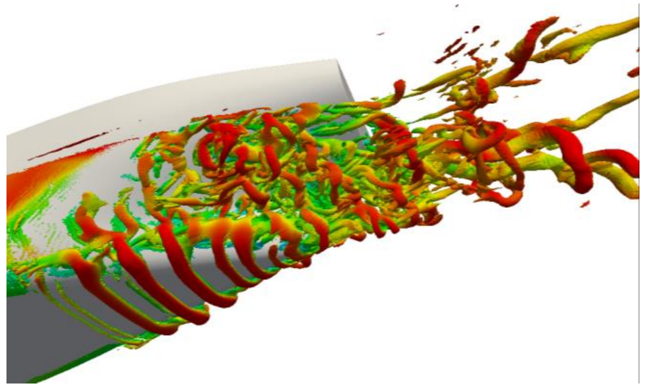
Zigzag maneuver in waves



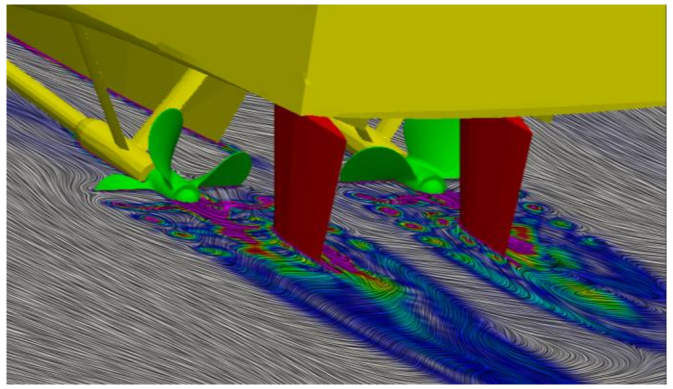
Turning circle maneuver in waves

Complex flow field

Supported by high performance computing, the solver can give detailed flow field information, such as complex free surface deformation in high Reynolds number and turbulent flow.

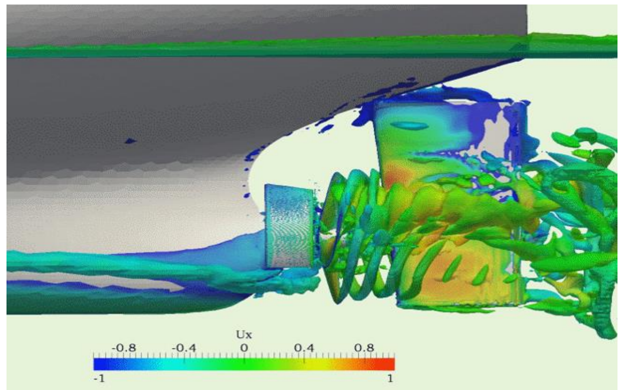


Refined vorticity field of JBC stern

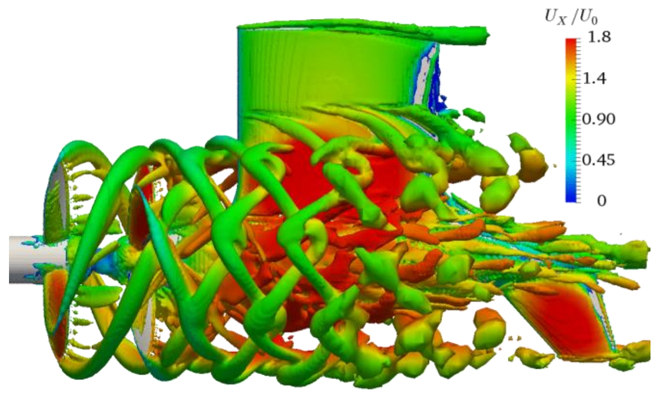


Wake field around the propeller and rudder

The naoe-FOAM-SJTU solver can predict the effect and hydrodynamic characteristics of energy saving devices, such as the pre-duct, counter-rotating propeller and pod propeller.



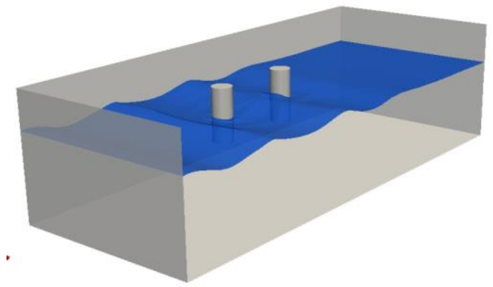
Refine vorticity field with pre-duct



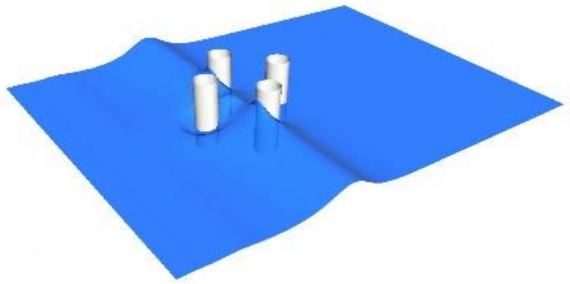
Vorticity field of pod propeller

Wave run-up

The naoe-FOAM-SJTU solver can also study practical problems in offshore engineering. It can simulate the wave run-up on fixed structures, and the predicted error of wave loads is no more than 5%.



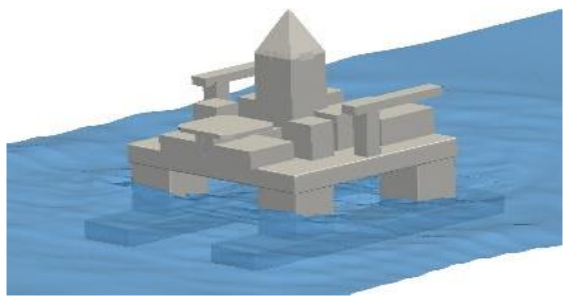
Wave run-up on two cylinders in regular wave



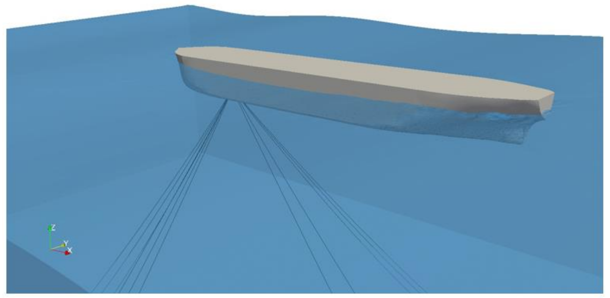
Wave run-up on multi cylinders in solitary wave

Motion response of floating platforms

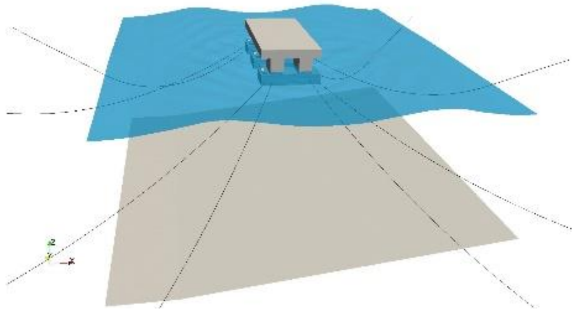
Combined with the 6 DoF motion module and mooring system module, the solver can simulate the motion responses of 981 semi-submersible platforms, single-point mooring FPSO, floating wharfs and multi-vessel arranged side by side in complex wave environment, and predict the wave loads on ocean structures.

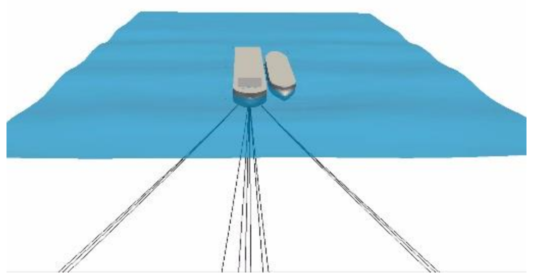


Motion response of 981 semi-submersible in waves



Motion response of single-point mooring FPSO

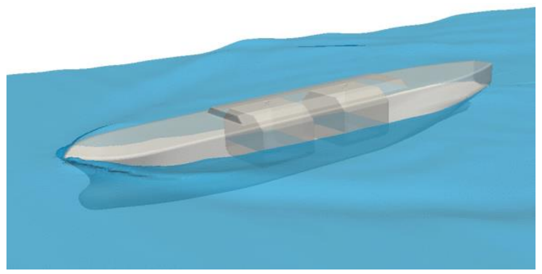




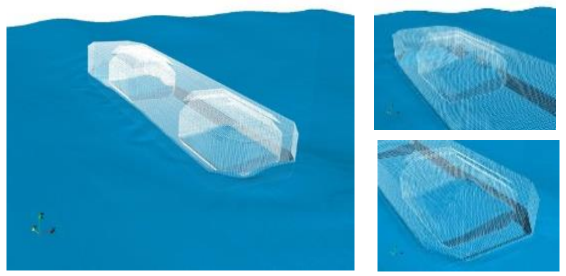
Motion response of single-point mooring FLNG berthing system

Liquid sloshing

Using coupling method for solving the internal and external flow field, the solver can simulate the motion response of the LNG ship coupled with sloshing tanks in waves. The forecast error of the slamming pressure on the tank wall is within 5%.



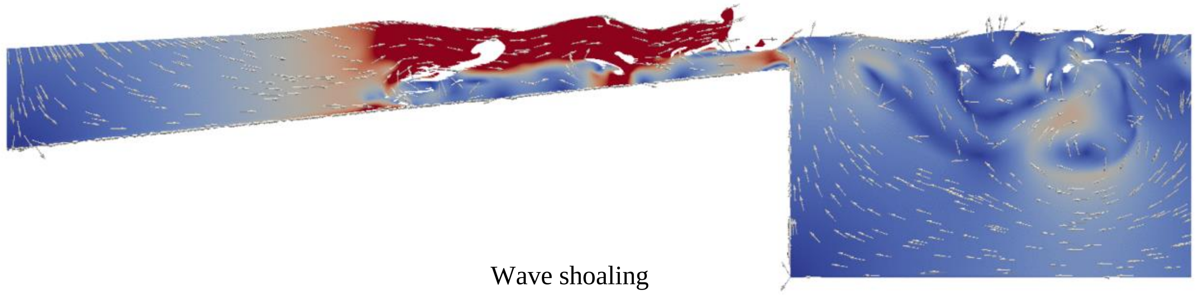
Motion response of LNG ship in head waves



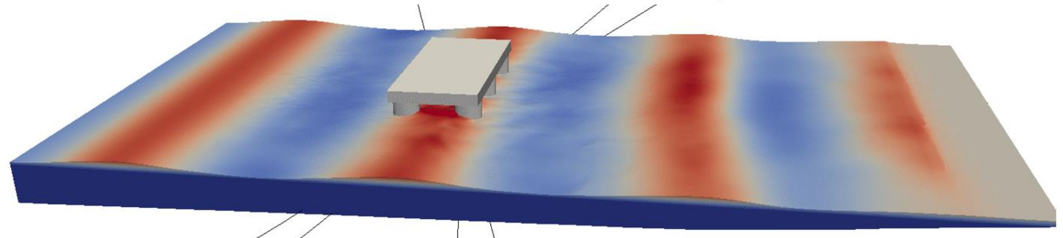
Numerical simulation of FLNG with sloshing tank

Wave shoaling

The solver can simulate the wave evolution in shallow waters, and capture wave run-up and wave breaking phenomena. The effect of shallow water topography on the motion of floating platform can also be simulated.



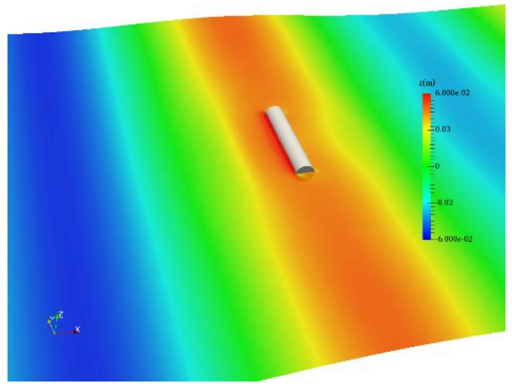
Wave shoaling



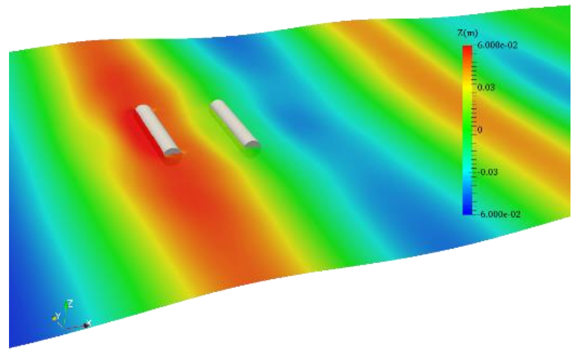
Motion response of floating platform in submerged terrain

Motion response of wave energy converter

The solver can simulate the motion response of wave energy converter device in waves. It can capture the dynamic pressure distribution on the surface of the floater and the wave diffraction around the floater. The error of predicted motion is less than 3%.



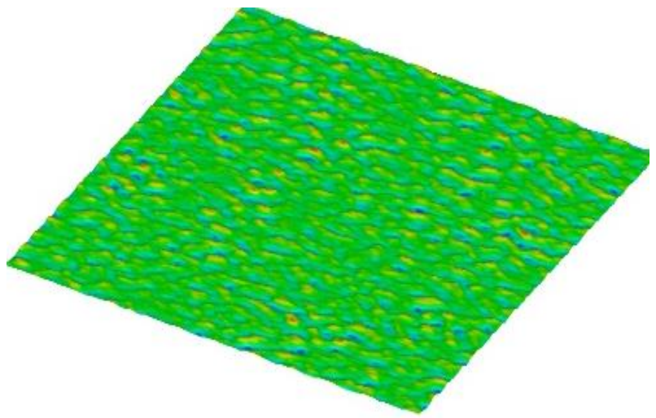
Wave energy converter (one floater) in waves



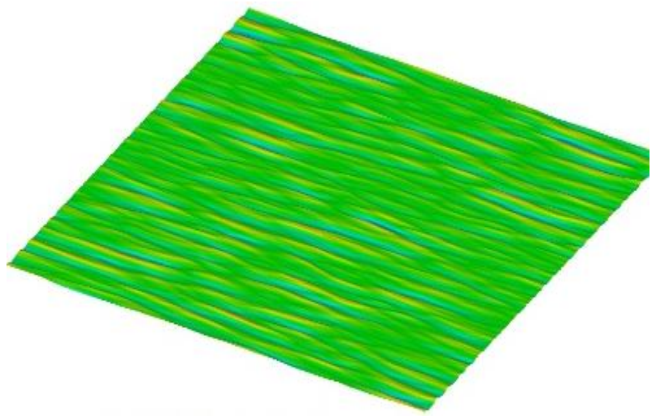
Wave energy converter (two floaters) in waves

Simulation of nonlinear wave evolution

Combined with the open source program HOS-ocean based on the high-order spectral method, the naoe-FOAM-SJTU solver can also simulate the nonlinear wave evolution in large areas for a long time.



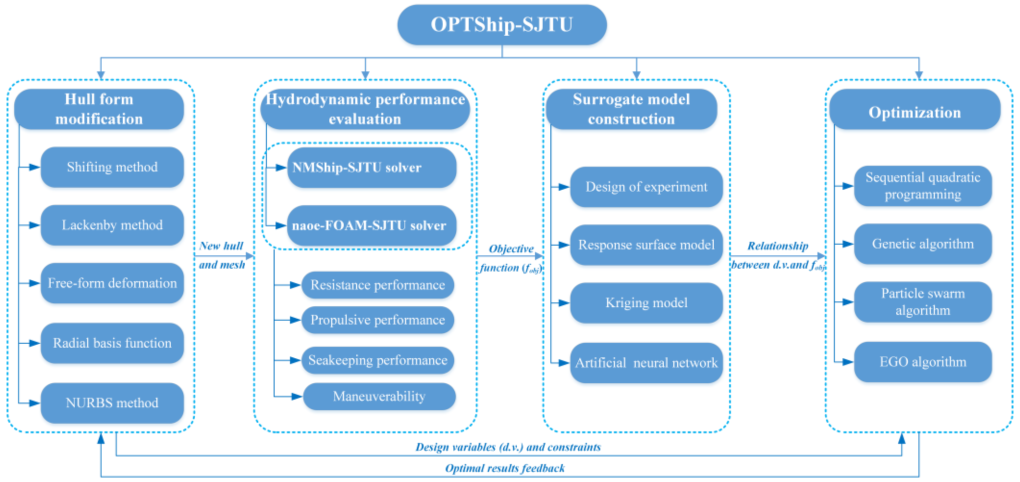
Simulation of wave evolution in large area
(directionality parameter =90)



Simulation of wave evolution in large area
(directionality angle parameter =30)

Ship hull optimization design solver (OPTShip-SJTU)

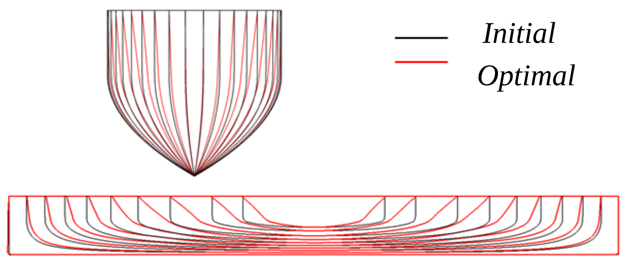
CMHL developed the OPTShip-SJTU solver based on C++ language for the ship hull form optimization. The solver integrates with a hull surface modification module, a hydrodynamic performance evaluation module, a surrogate module and an optimization module, which can achieve the ship hull form optimization 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.



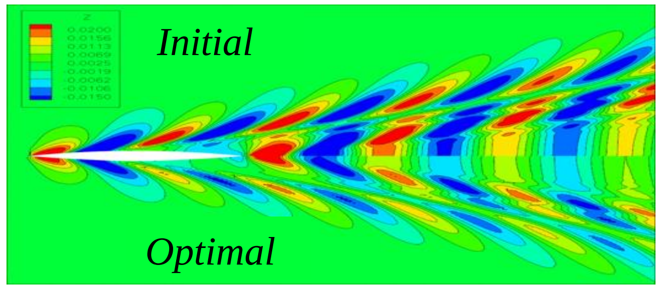
Module framework of OPTShip-SJTU

Single-objective optimization design of wave drag for Wigley hull

The solver can be applied to the single-objective optimization of Wigley hull for the wave drag at Fr=0.3. The optimal hull is obtained and the wave drag coefficient has been reduced by 47.1%. Comparing free surface elevations between the original and optimal hull , the wave height is remarkably lower.



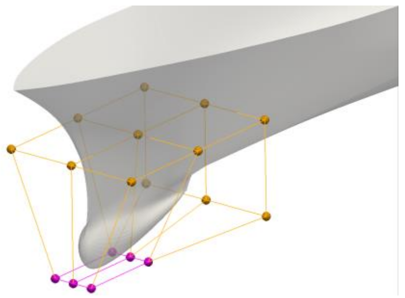
Comparisons of body plans and sheer plans between the original hull and the optimal hull



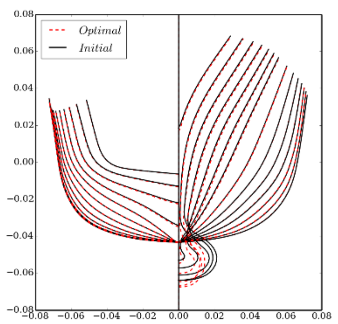
Comparisons of wave patterns between the original hull and the optimal hull

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

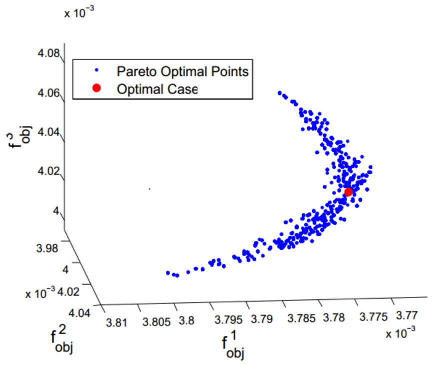
The solver can be used to achieve the multi-objective optimization of DTMB 5415 hull for the total drag at three speeds. a series of optimal hulls are obtained with reduced total drags at three speeds. One optimal hull is selected to further analyze. Through the comparison of free surface elevations between the original and optimal hulls, it can be seen that the wave height around the optimal hull is lower. And the pressure at the bulbous bow of the optimal hull is smaller from their pressure distributions.



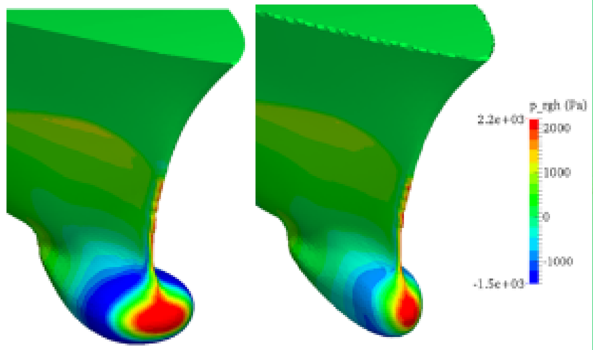
Ship bulbous bow modification based on FFD method



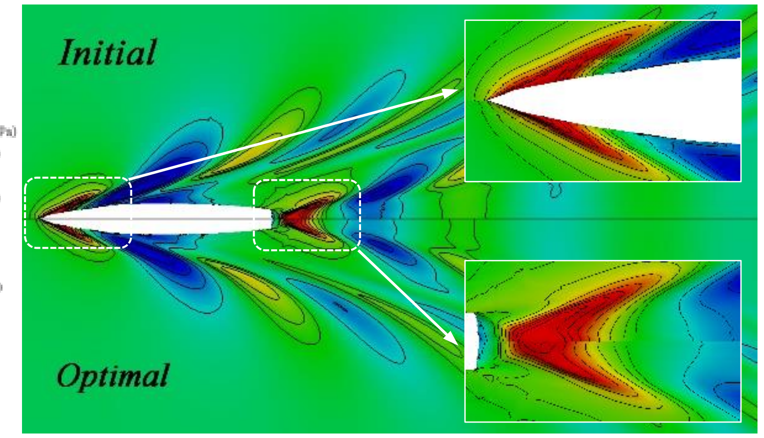
Comparisons of body plans between the original hull and the optimal hull



Optimal Pareto set



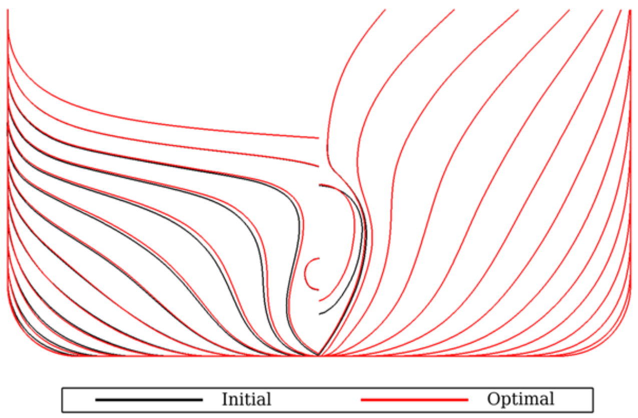
Comparisons of pressure distributions on bulbous bow between the initial hull and the optimal hull



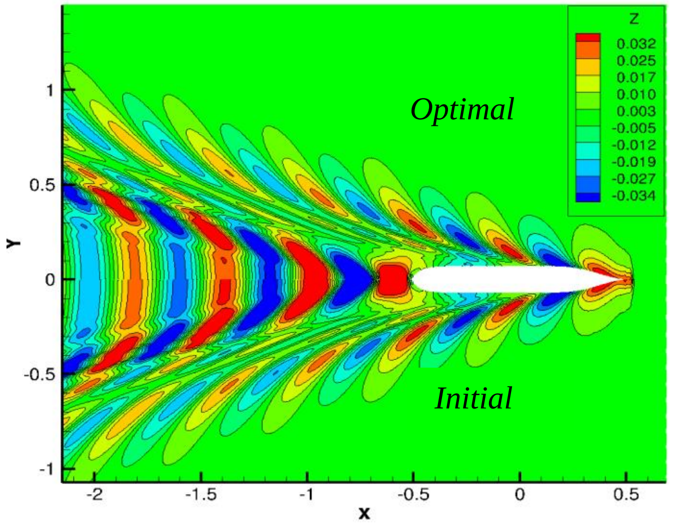
Comparisons of wave patterns between the original hull and the optimal hull

Comprehensive optimization design of drag and wake performance for KCS hull

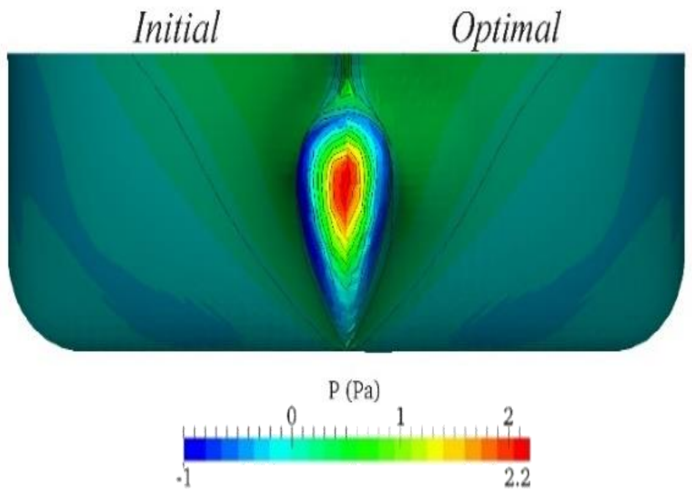
The solver can also realize the comprehensive optimization of KCS hull in terms of total drag and wake performance at the speed of Fr=0.26. The optimal hull has a drag reduction of 3.62% and wake performance improvement of 2.8%. Through the comparison of wave patterns, the wave height of the optimal hull is reduced. The pressure of the optimal hull is smaller than that of the initial hull. The velocity field at the propeller disk becomes more uniform, so the wake performance is improved.



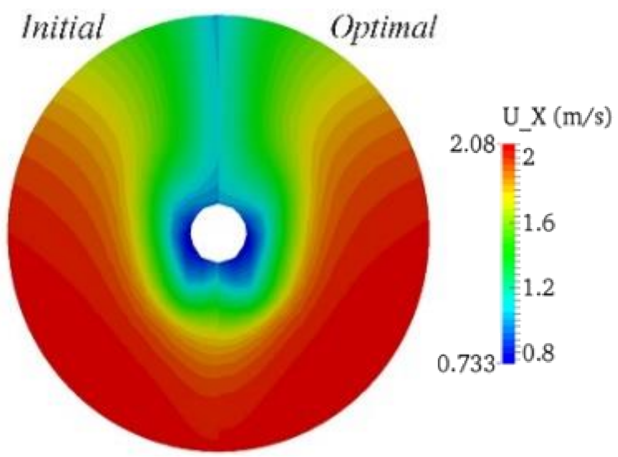
Comparisons of body plans between the original hull and the optimal hull



Comparisons of wave patterns between the original hull and the optimal hull



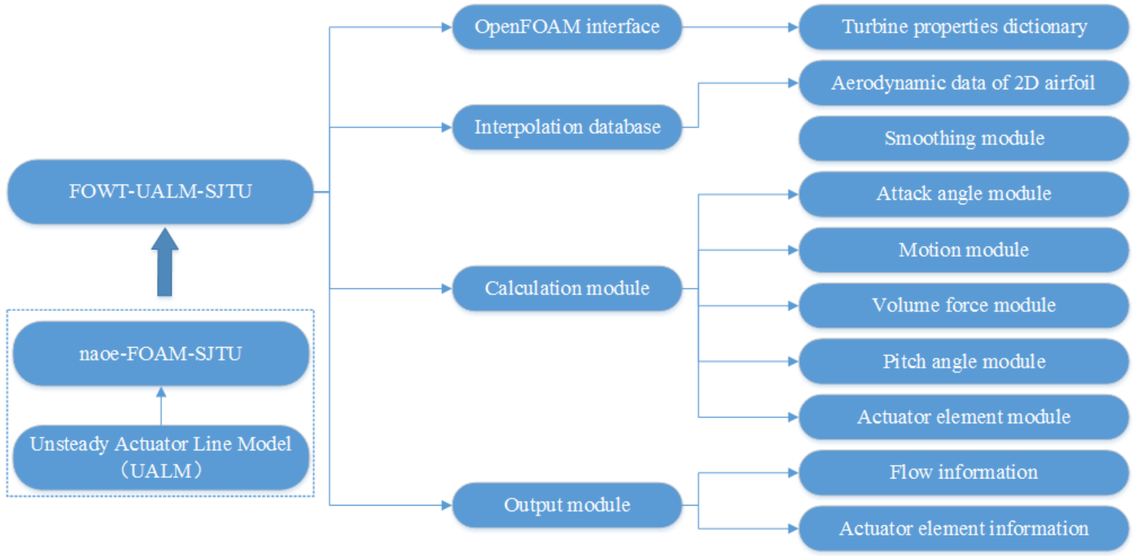
Comparisons of pressure distributions on bulbous bow between the initial hull and the optimal hull



Comparisons of the wake at the disk between the initial hull and the optimal hull

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

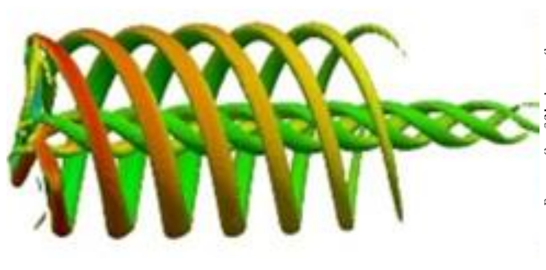
CMHL have developed the FOWT-UALM-SJTU solver to achieve fully coupled aero-hydrodynamic simulations of floating offshore wind turbines (FOWTs) based on the unsteady actuator line model (UALM). FOWT-UALM-SJTU solver includes OpenFOAM interface, interpolation database, calculation module and output module. 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.



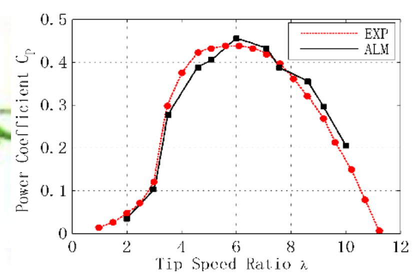
Module framework of FOWT-UALM-SJTU

Numerical simulations of a single wind turbine

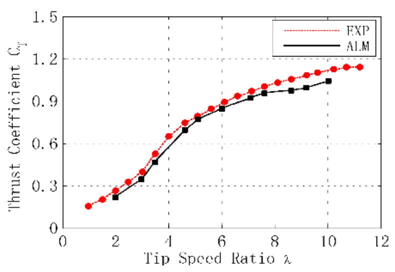
With unsteady actuator line model (UALM), accurate aerodynamic simulations of a single wind turbine are achieved, where the error of aerodynamic loads prediction is within 5%. Moreover, the unsteady aerodynamic simulations of FOWT with prescribed periodic surge/pitch motions are conducted.



Vortex structure

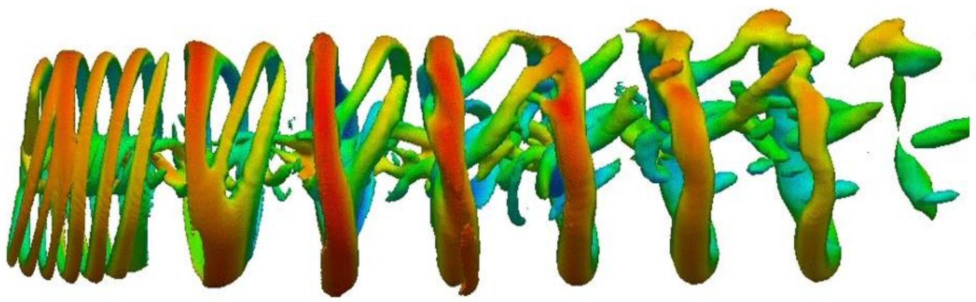


Power Coefficient

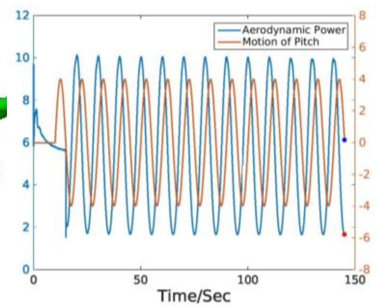


Thrust Coefficient

Numerical simulations for a single wind turbine compared to ‘Blind Test 1’



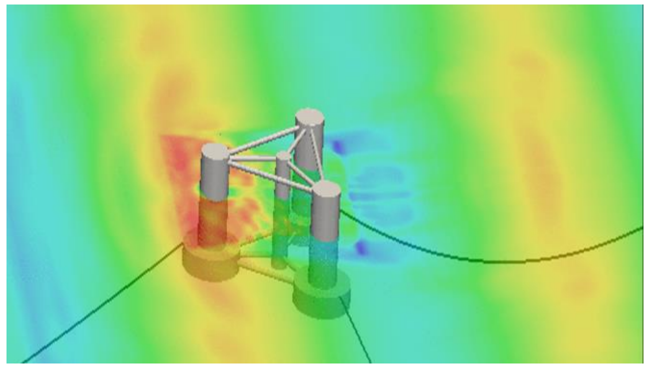
Wake vortex of NREL-5MW wind turbine with periodic pitch motion



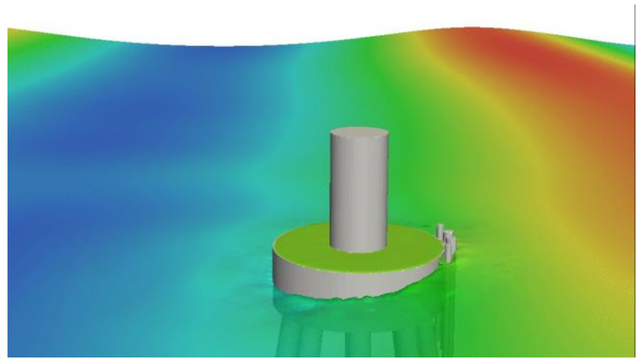
Pitch motion and power output

Hydrodynamic simulations of platforms

FOWT-UALM-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.



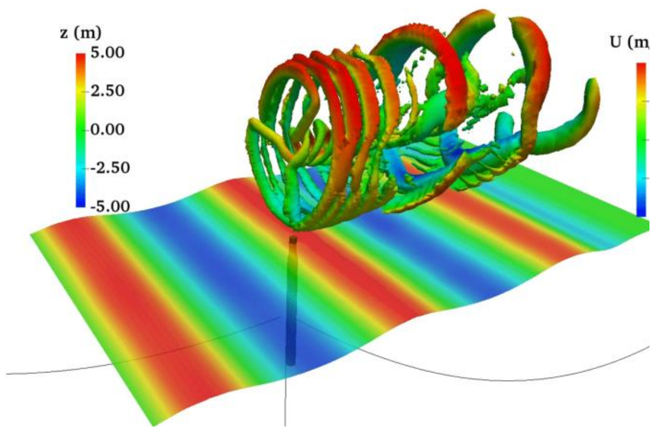
Hydrodynamic prediction of floating platform



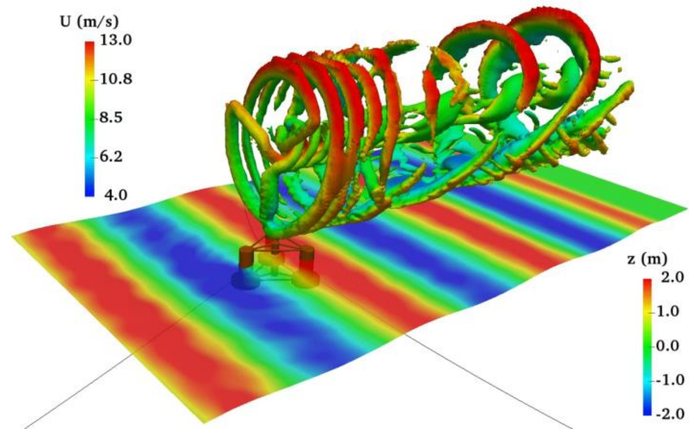
Hydrodynamic prediction of fixed platform

Fully coupled aero-hydrodynamic simulations of FOWTs

FOWT-UALM-SJTU solver can achieve coupled aero-hydrodynamic simulations of different FOWTs, and the performance of wind turbine-floating platform-mooring system can also be predicted.



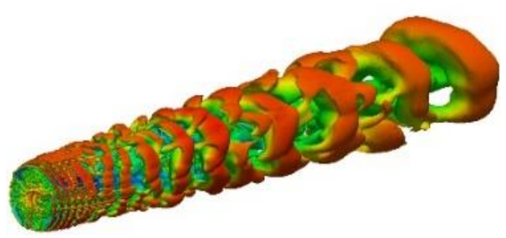
Fully coupled simulations for spar type FOWTs



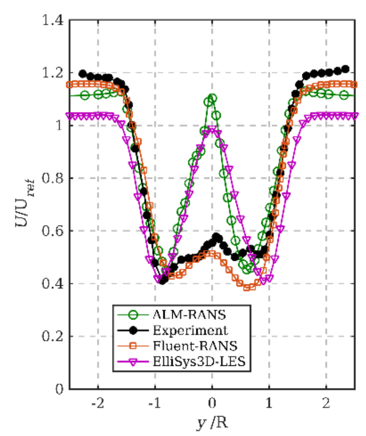
Fully coupled simulations for semi-submersible FOWTs

Aerodynamic simulations of multiple wind turbines

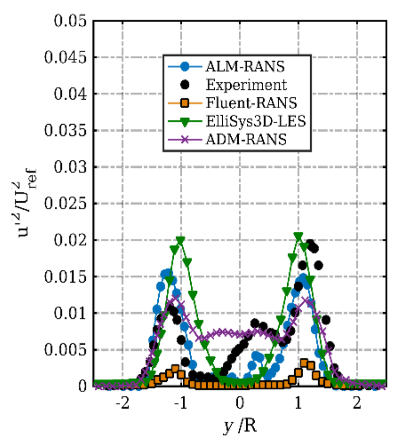
Based on the actuator line model, the numerical aerodynamic simulations of two in-line model wind turbines have been conducted. The aerodynamic loads, flow characteristics and vortex structure can be obtained, and the error of aerodynamic loads prediction is within 10% .



Vortex structure of two in-line model wind turbines



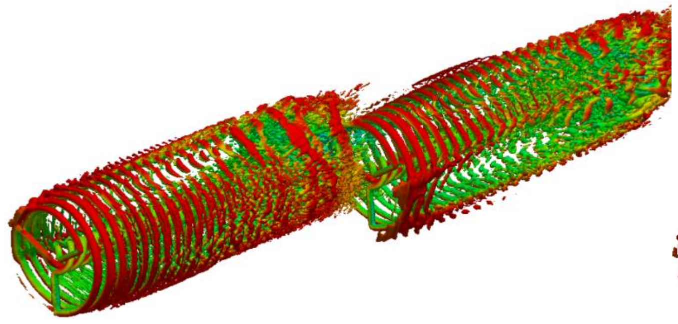
Mean velocity profiles in wake flow field



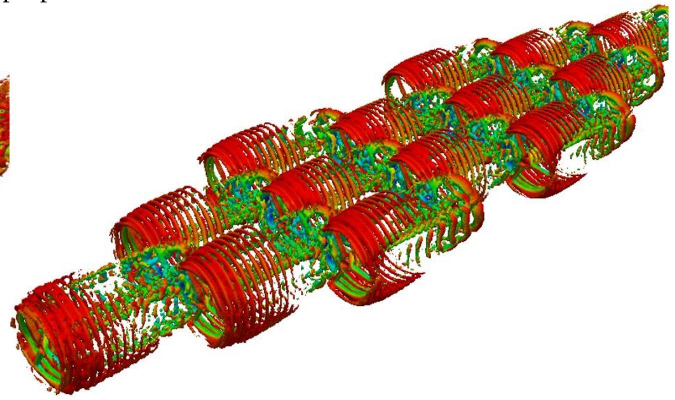
Mean turbulent stress profile

Numerical simulation for Two in-line model wind turbines compared to ‘blind test 2’

The FOWT-UALM-SJTU solver is capable of numerical simulations for complex flow exited in the large wind farm containing multiple wind turbines in array layout. The significant wake interference phenomenon is observed and aerodynamic loads of each wind turbine can be predicted. By analyzing the complex wake characteristics, reasonable advices for the optimizing research of wind farm layout are proposed.



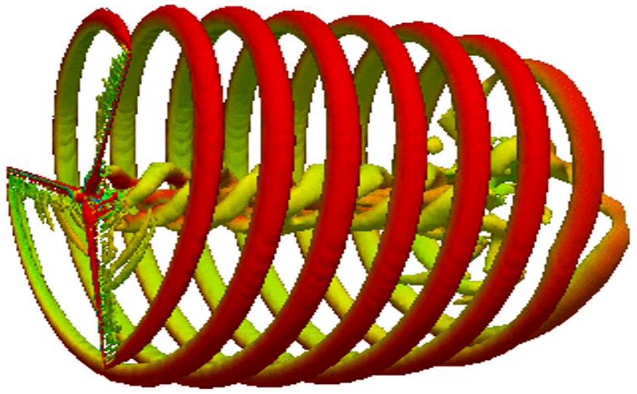
Vortex structure of two offset model wind turbines



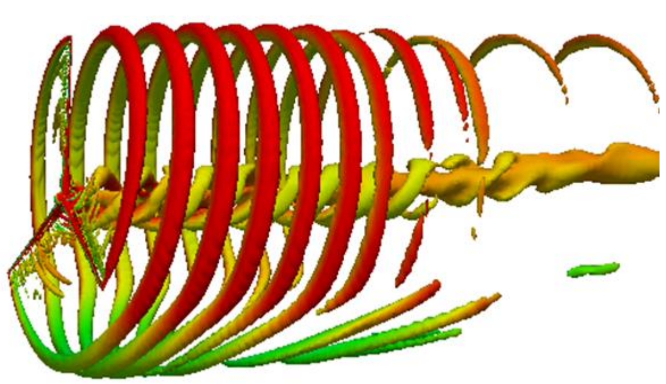
Vortex structure of fourteen wind turbines in array layout

Numerical simulations based on the overset grid technology

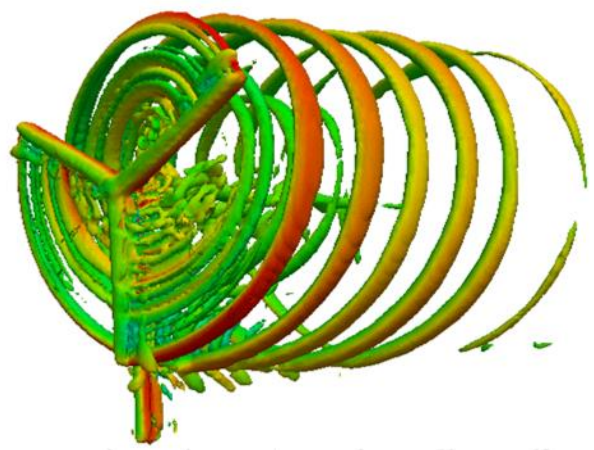
Besides, more accurate aerodynamic simulations of wind turbines have been conducted based on the overset grid technology, where the influence of wind shear effect and tower shadow effect are taken into consideration. Furthermore, the fully coupled aero-hydrodynamic simulations of the floating offshore wind turbines under wind-wave conditions have been achieved.



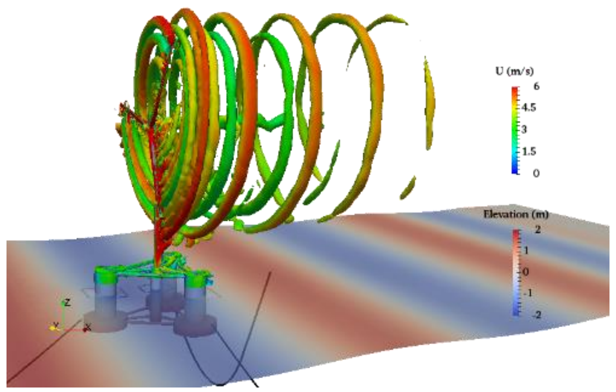
Aerodynamic simulations of NREL-5MW wind turbine in uniform wind



Aerodynamic simulations of NREL-5MW wind turbine in shear wind



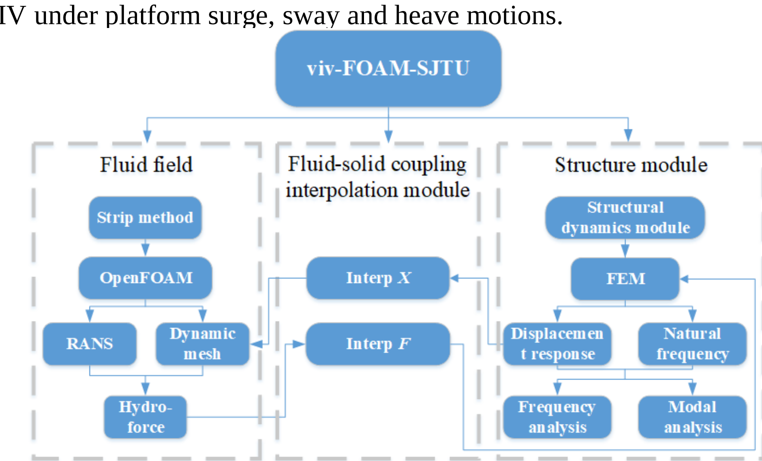
Aerodynamic simulations of NREL-5MW wind turbine with tower shadow effect

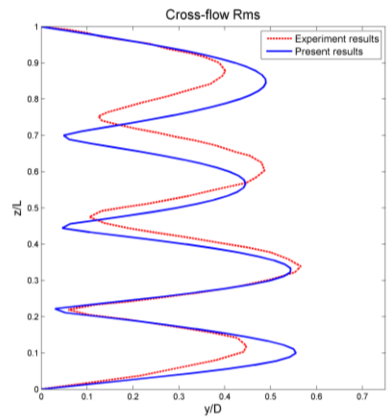


Fully coupled aero-hydrodynamic simulations of the OC4 Phase II floating offshore wind turbine system

CFD solver for VIV of deep-sea risers (viv-FOAM-SJTU)

CMHL focuses on the vortex-induced vibration of deep-sea risers and have developed viv-FOAM-SJTU solver based on OpenFOAM. Strip method, FVM and RANS are adopted to simulate the flow field to acquire hydrodynamics. The Eulerian-Bernoulli beam model and the 3-D FEM are used to simulate the riser model. Interpolation is used to transmit data realizing coupling calculations of the mesh deformation and the structural dynamic response. After continuous development, viv-FOAM-SJTU solver can perform numerical simulations of single or multiple risers vortex-induced vibration under different flow field profiles including uniform flow and shear flow, and can simulate VIV under platform surge, sway and heave motions.

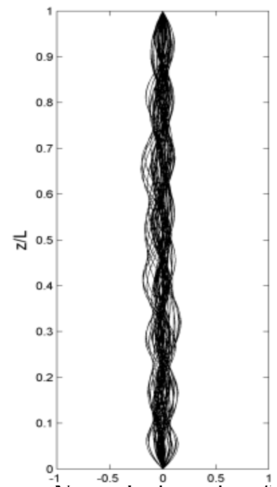




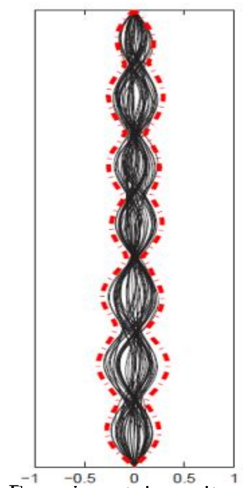
Module framework of viv-FOAM-SJTU

VIV of a single riser under the stepped flow

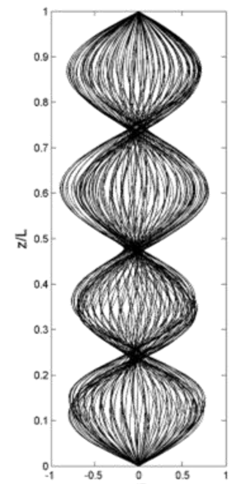
The viv-FOAM-SJTU solver can simulate vortex induced vibration of a riser under stepped flow and accurately calculate the dynamic response of the riser. Spatial-temporal images and displacement envelopes are in good agreement with the experiment with a maximum error less than 5%.



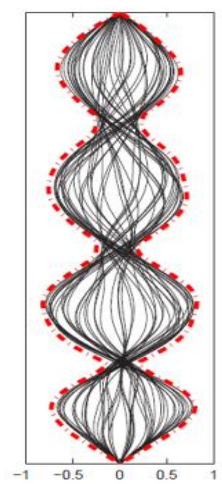
Numerical results of in-line vibration



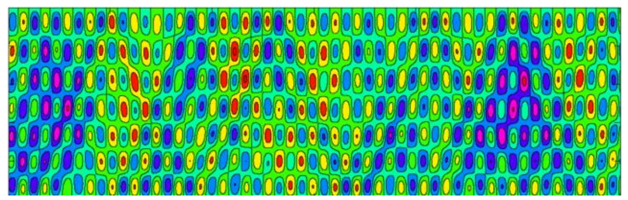
Experimental results of in-line vibration



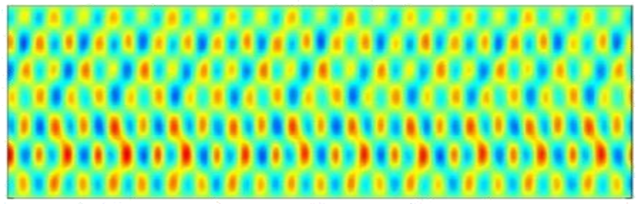
Numerical results of cross-flow vibration



Experimental results of cross-flow vibration

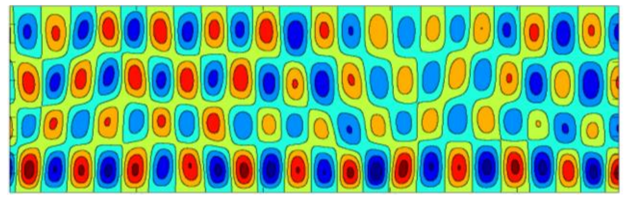


Numerical results

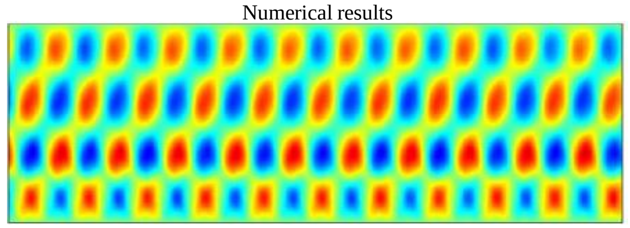


Experimental results

Spatial-temporal image of the in-line vibration



Numerical results

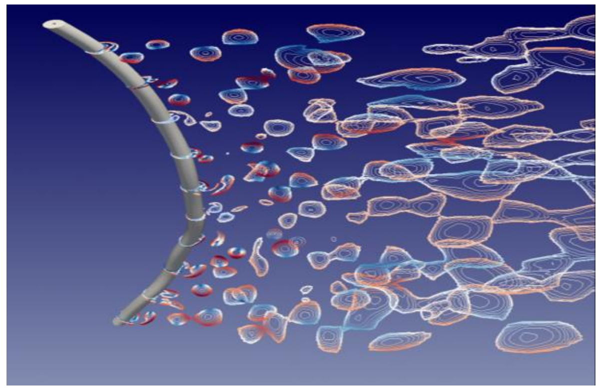


Experimental results

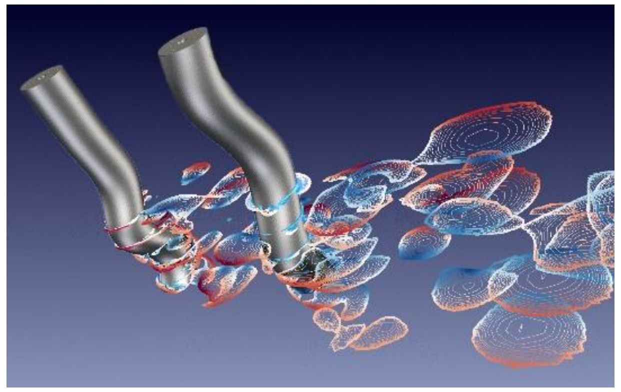
Spatial-temporal image of the cross-flow vibration

VIV of single and multiple risers under the uniform flow

The solver can perform numerical simulations of VIV of single and multiple risers under the uniform flow.



VIV of a single riser

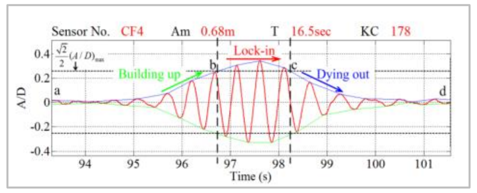


VIV of tandem risers

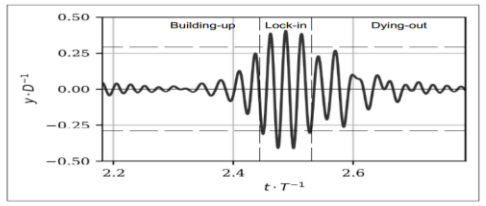
VIV of risers under the oscillatory flow

The solver can simulate VIV of risers under the oscillatory flow. The typical process of "build-up--lock-in--die-out" under the oscillatory flow is accurately. The lock-in interval is in good agreement with the experimental results.

VIV of the riser under the oscillatory flow



Model experiment results

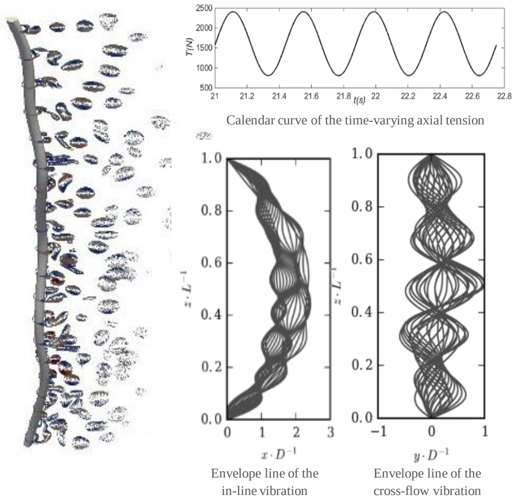


Numerical results

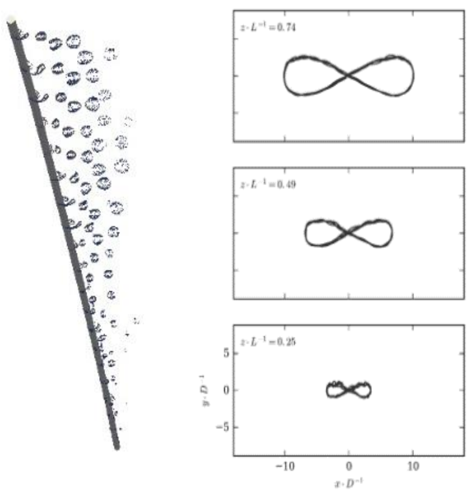
Calendar curve of the cross-flow vibration

VIV of risers with platform motions

The solver can simulate the vortex-induced vibration of a riser with platform motions. Surge/sway motions of the platform are simulated by applying a support excitation to the riser model Heave motions of the platform is simulated by applying a time-varying axial tension to the riser.



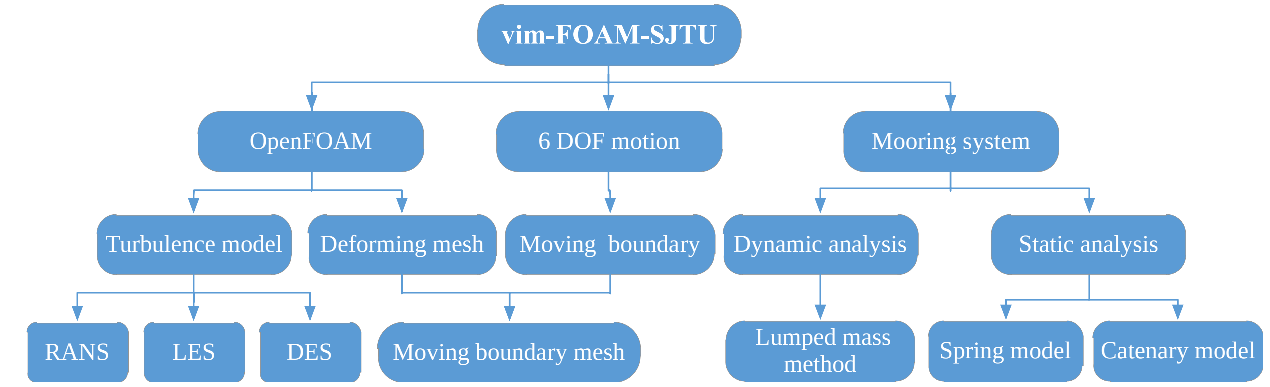
VIV of a riser under heave motions of the platform



VIV of a riser under sway motions of the platform

Vortex-induced motion solver for offshore platform (vim-FOAM-SJTU)

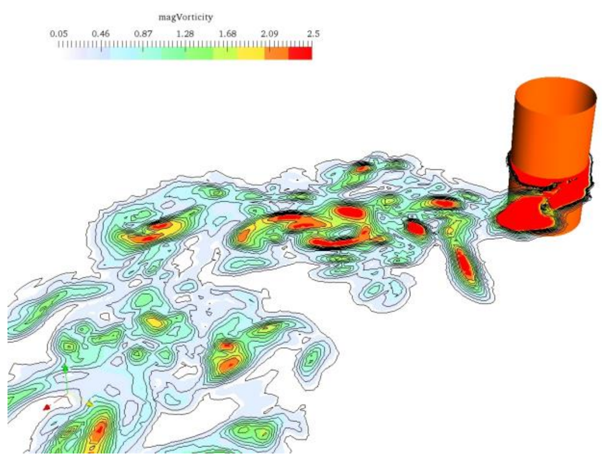
CMHL has independently developed the fluid structure interaction solver (vim-FOAM-SJTU) based on OpenFOAM which simulates the vortex-induced motion of the offshore platform. The detached eddy simulation (DES) method is applied to deal with the three-dimensional flow separation problem at high Reynolds number. Combined with the six degrees of freedom motion theory and the moving boundary spring mesh technique, the vortex-induced motion of offshore platform can be simulated. The prediction of the vortex-induced motion can be applied for floating offshore platforms such as buoyancy cans, Spars and semi-submersible platforms.



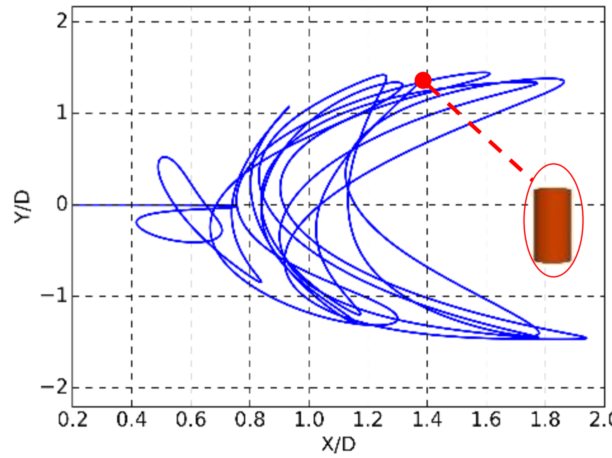
Module framework of vim-FOAM-SJTU

VIM of Spar platform

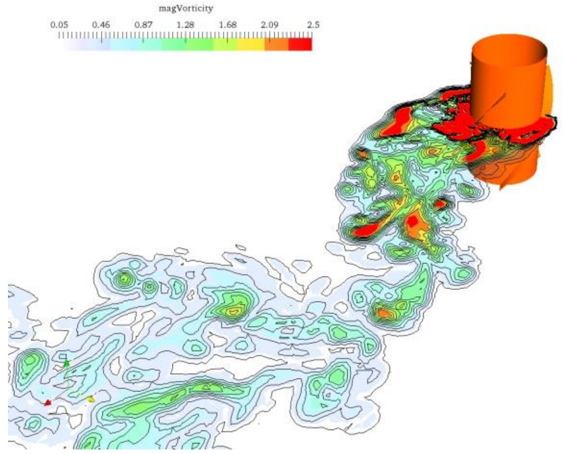
vim-FOAM-SJTU can be used to predict the vortex-induced motion of the Spar platform, and the vortex reduction effect of the helical side plate on the cylinder surface. The helical strakes can significantly reduce the inline and crossflow motion response.



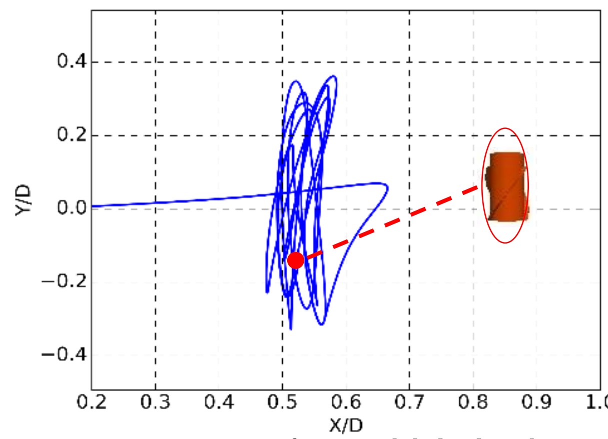
Vortex structure of Spar without helical strakes



VIM trajectory of Spar without helical strakes



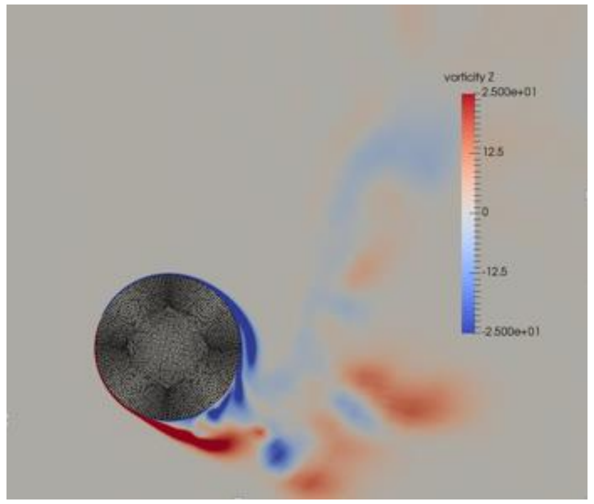
Vortex structure of Spar with helical strakes



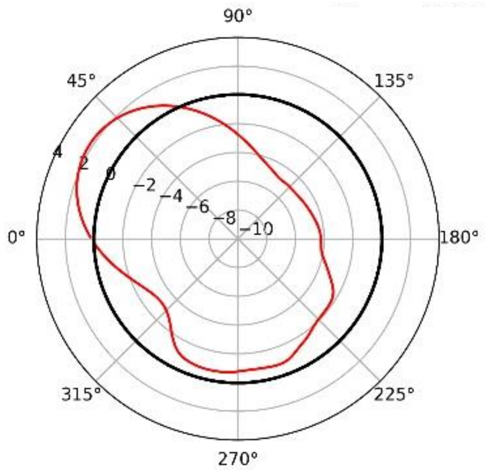
VIM trajectory of Spar with helical strakes

VIM of a rotating buoyancy can

vim-FOAM-SJTU can simulate the vortex-induced motion of a cylindrical buoyancy can with rotating motion.



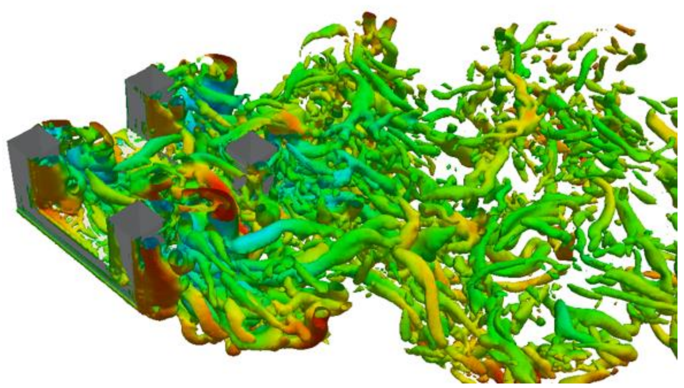
VIM around a rotating buoyancy can



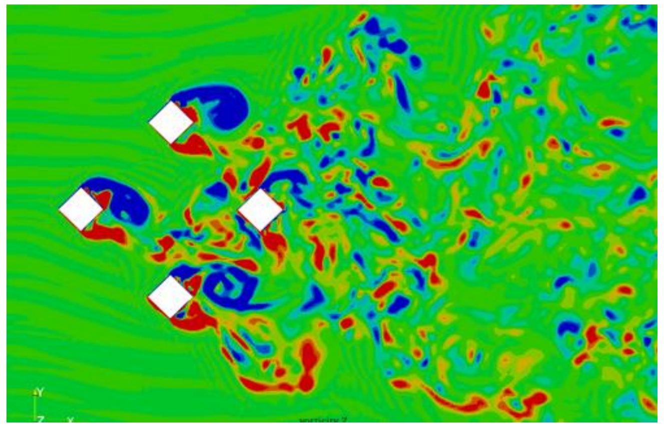
Circumferential distribution of the pressure coefficient C_p

VIM of semi-submersible platform

vim-FOAM-SJTU can also simulate the vortex-induced motion of multi-column semi-submersible platform and vortex characteristics.



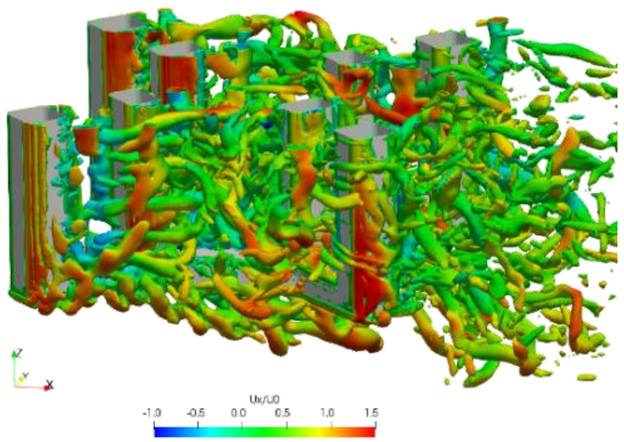
Vortex structure of semi-submersible platform



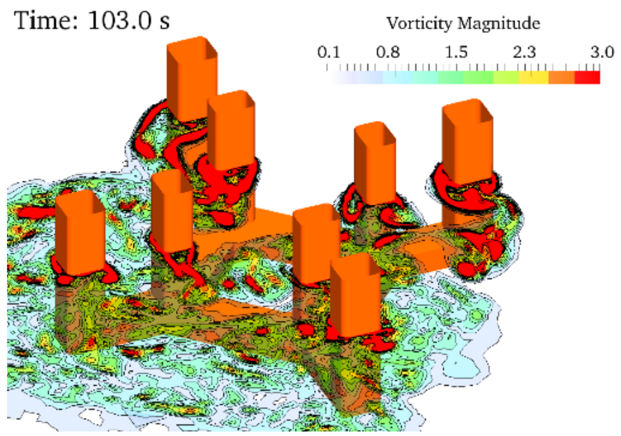
Vorticity slice of VIM around semi-submersible

VIM of paired-column semi-submersible platform

vim-FOAM-SJTU can simulate the vortex induced motion of the paired-column semi-submersible platform and predict its response characteristics under different conditions.



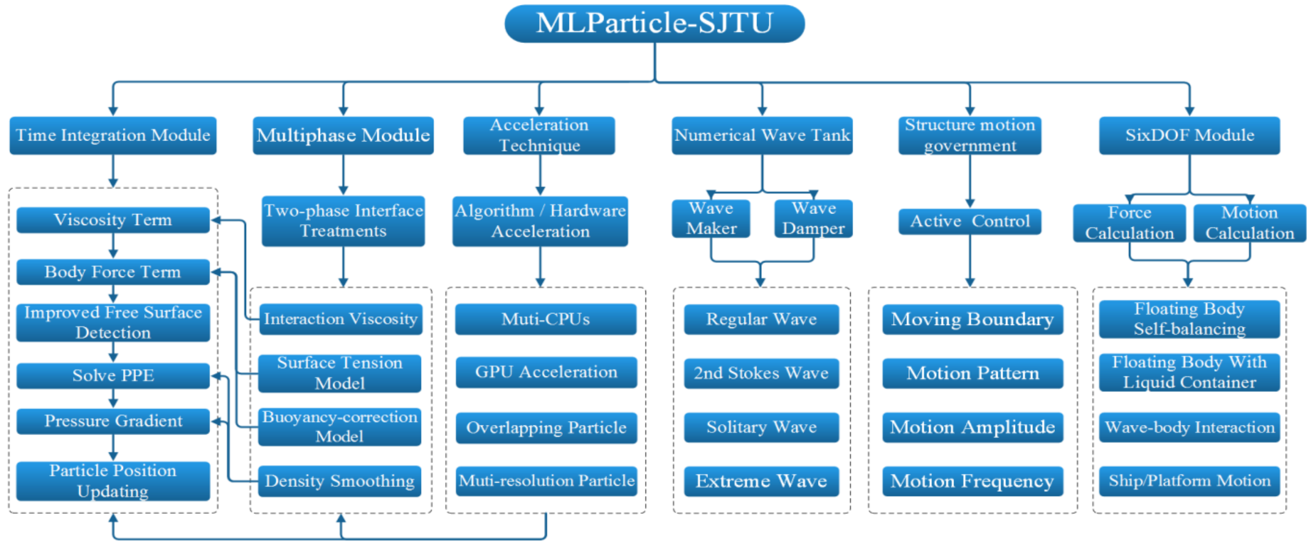
Vortex structure around the paired-column semi-submersible platform



Vortex structure on the z/L=0.5 plane around the paired-column semi-submersible platform

Mesh-less particle solver (MLParticle-SJTU)

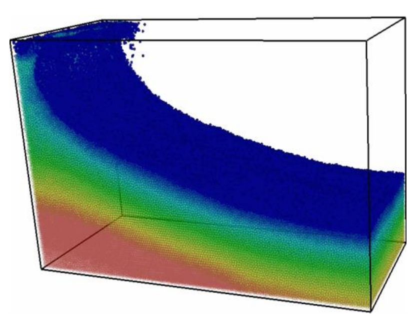
CMHL developed an in-house solver MLParticle-SJTU based on improved MPS method, including particle integral, numerical wave tank, six degrees of freedom movement, acceleration techniques and multiphase flow. By modifying kernel function, free surface detection and source term of pressure Poisson equation,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



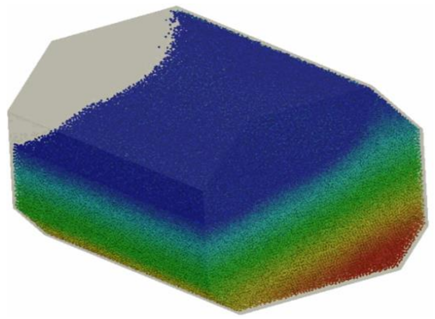
Module framework of MLParticle-SJTU

Liquid sloshing

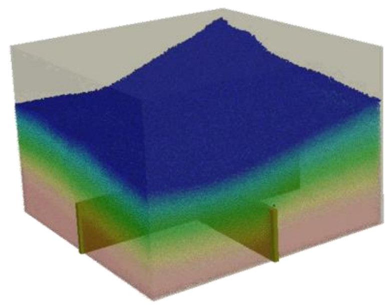
MLParticle-SJTU can simulate the liquid sloshing in different tanks, such as rectangular tanks, LNG tanks and baffled tanks.



Liquid sloshing in rectangular tanks



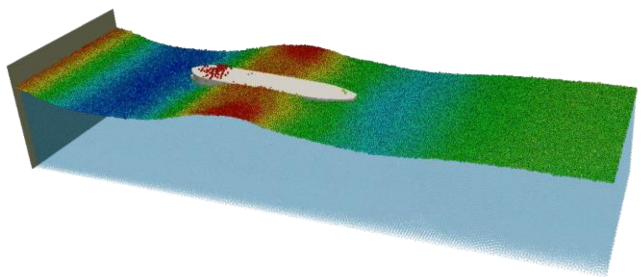
Liquid sloshing in LNG tanks



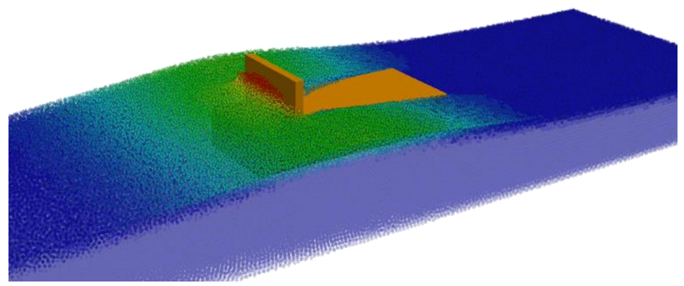
Liquid sloshing in baffled tanks

Ship motion in waves

With the help of numerical piston-type wavemaker, MLParticle-SJTU can be used to simulate ship motion in waves, green water and slamming on the superstructure induced by the green water.



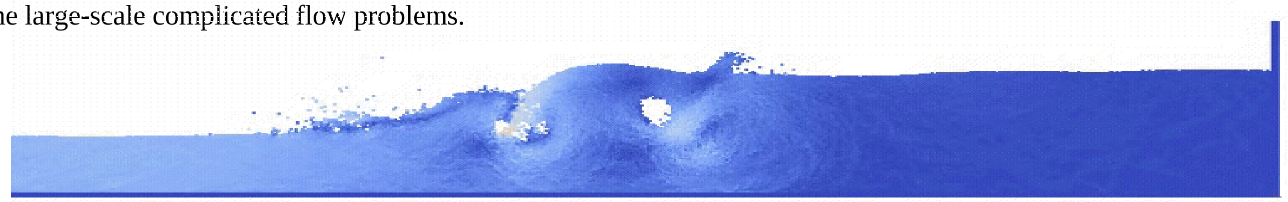
Motions of KVLCC in waves



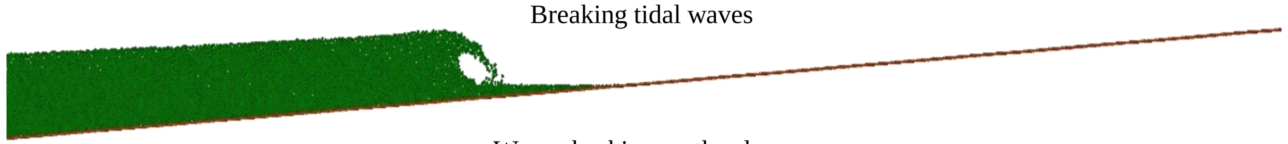
Slamming on the superstructure induced by the green water

Tidal waves and waves climbing uphill

MLParticle-SJTU can simulate the evolution of tidal wave, as well as breaking wave 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.



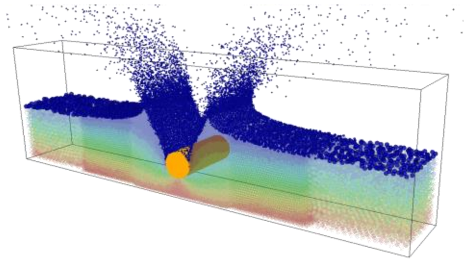
Breaking tidal waves



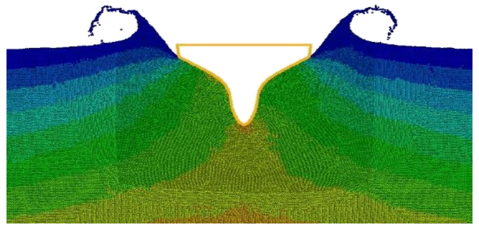
Waves broking on the slope

Acceleration techniques

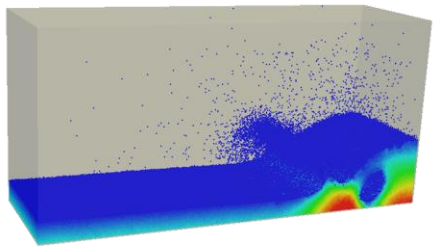
Some used acceleration techniques such as overlapping particle technique, multi-resolution particle technique, multi-CPU and GPU can dramatically improve the calculation efficiency of MLParticle-SJTU. The solver can simulate many large-scale problems such as water entry, dam break, liquid sloshing and so on. The computing efficiency can be increased by 30 times.



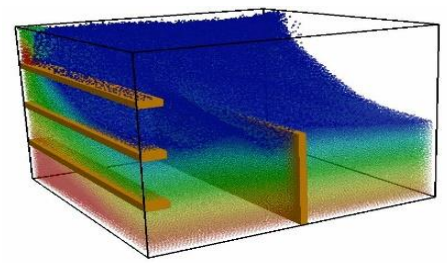
Multi-resolution particle technique for water entry of cylinder



Multi-resolution particle technique for water entry of ship section



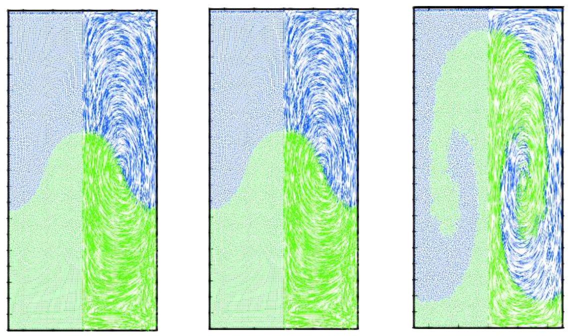
GPU acceleration technique for dam-break flow



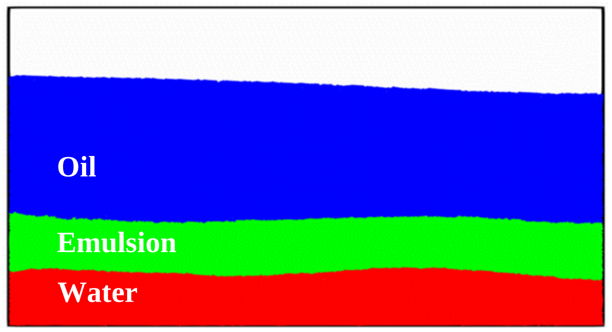
GPU acceleration technique for liquid sloshing

Multiphase flow

By introducing special treatments for the interface between different phases, MLParticle-SJTU successfully implements the numerical simulations of complicated multiphase problems, including Rayleigh-Taylor instability and multi-liquid-layer sloshing in oil-water separators.



Rayleigh-Taylor instability



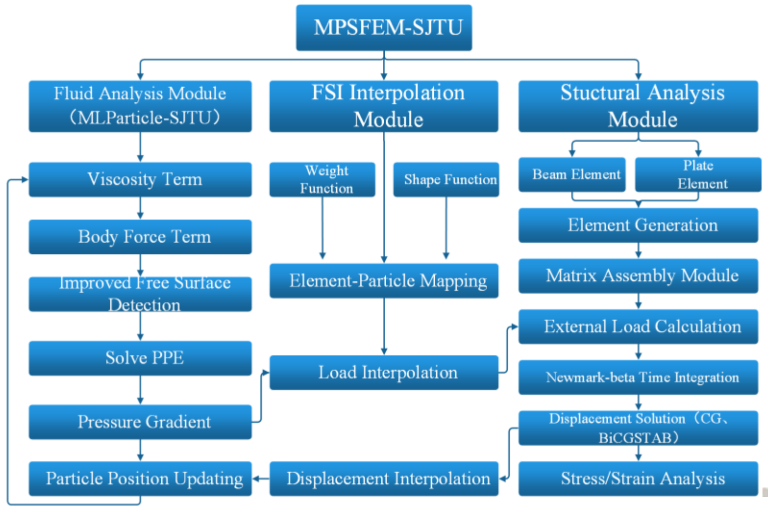
Multiliquid-layer sloshing in oil-water separators

Fluid-structure interaction solver based mesh-less particle and finite element method (MPSFEM-SJTU)

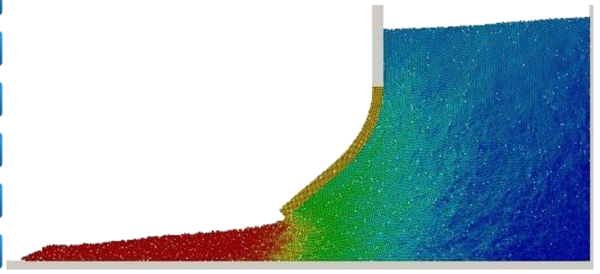
The CMHL center of SJTU implemented the FEM module and interfacial interpolation module on the basis of MLParticle-SJTU, to develop the fluid-structure interaction solver MPSFEM-SJTU. This solver demonstrates advantages while dealing with the problems of voilent flow and structural deformation. It can be applied to the FSI problems in the field of naval architecture and ocean engineering, such as the dam-break flow interacting with the elastic structure and liquid sloshing in an elastic tank.

Dam-break fluid-structure interactions

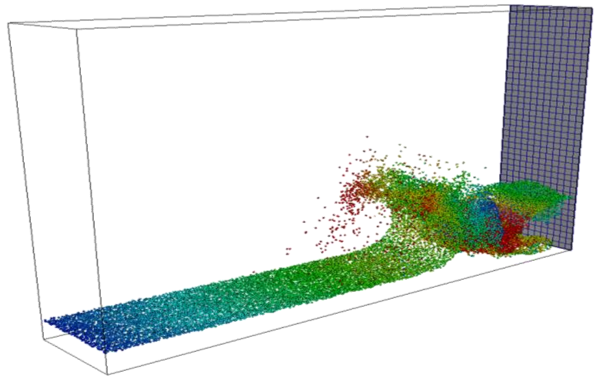
While encountering dam-break flow, the elastic structures may produce considerable deformation or damage. The MPSFEM-SJTU solver can be used to simulate the interaction between dam-break flow and the elastic structures.



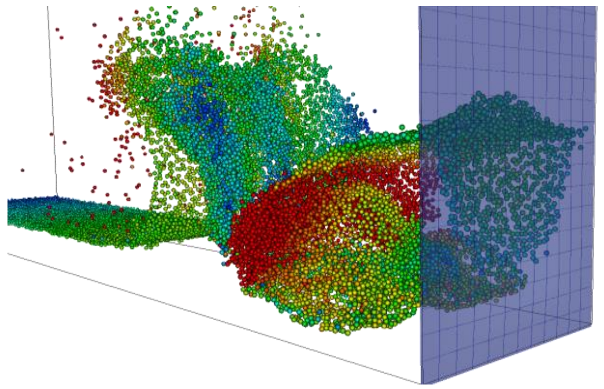
Frame diagram of MPSFEM-SJTU



Interaction between dam-break flow and elastic gate



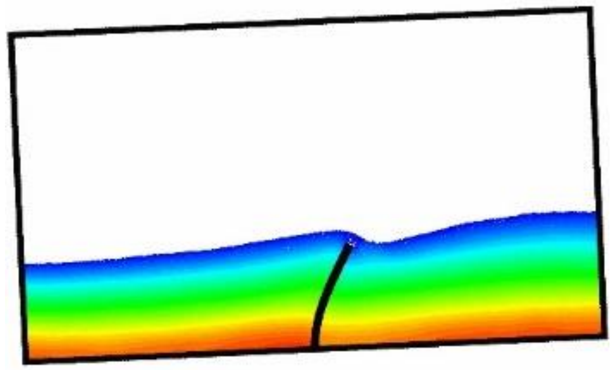
Interplay between dam-break flow and elastic wall



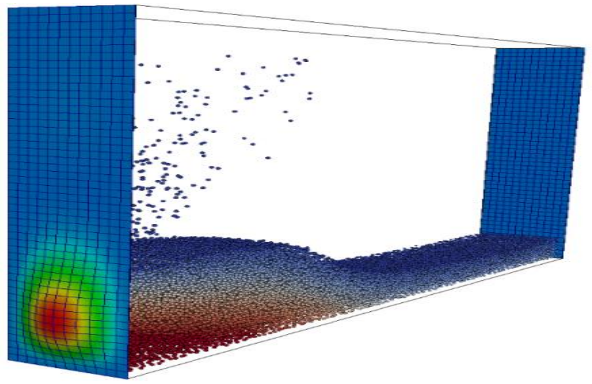
Interplay between dam-break flow and elastic wall (Partially enlarged details)

Liquid sloshing fluid-structure interactions

Liquid sloshing is a common problem in the field of naval architecture and ocean engineering. MPSFEM-SJTU solver can simulate the liquid sloshing in an elastic tank, such as 2D liquid sloshing in a tank with elastic baffle and 3D liquid sloshing in a tank with elastic side walls.



2D liquid sloshing in a tank with elastic baffle



3D liquid sloshing in a tank with elastic side walls