HEXAGON

Case Study Brochure

Cradle CFD

Marine and Ocean Engineering

Wind Resistance on a Sea Carrier Transporting Wood Chips

Case Study for Sanoyas Holdings Corporation

SC/Tetra is used to evaluate the wind resistance on a sea carrier transporting wood chips and predict effects caused by fitting-out equipment on the deck

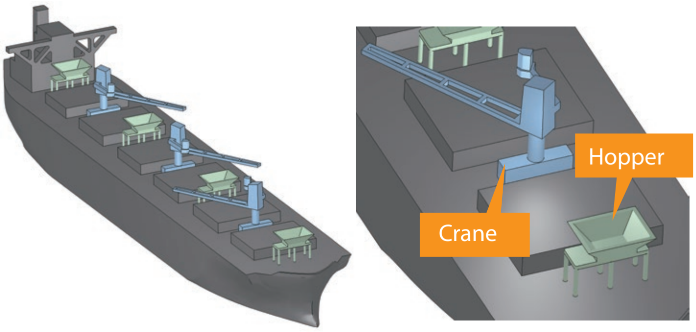
Reducing Wind Resistance on Chip Carriers

One of the problems that sea carriers loaded with lightweight wood chip face is slow vessel speed during stormy weather.

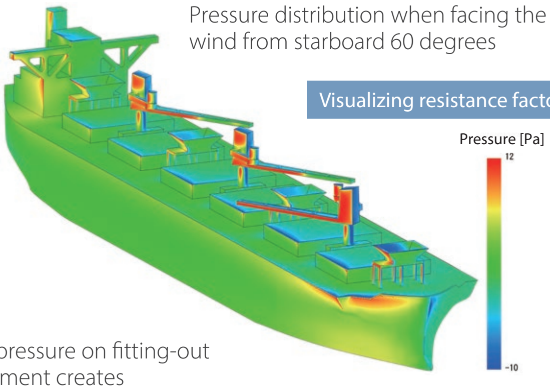
The wind pressure acting on the carriers is known to be strongly affected by the fitting-out equipment on the deck. The shape and appropriate allocation layout of the equipment are critical for minimizing the wind resistance.

In this case study, CFD was used to calculate the wind pressure resistance acting on a wood chip sea carrier. Evaluations included analyzing the wind resistance due to changes in wind direction, and analyzing how the wind resistance changes with and without fitting-out equipment.

Model of wood chip sea carrier



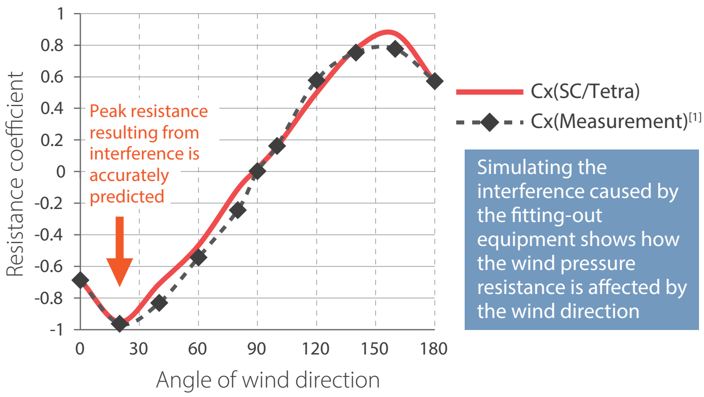
Surface pressure distribution



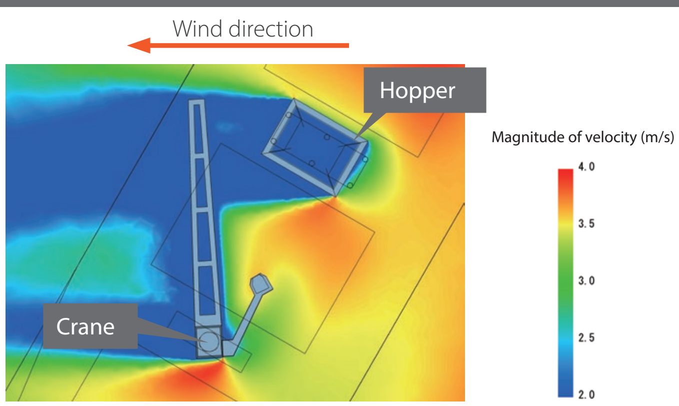
High pressure on fitting-out equipment creates substantial wind resistance

Comparison between analysis results and measurement

Wind pressure resistance affected by changes in wind direction angle



Interference between fitting-out equipment

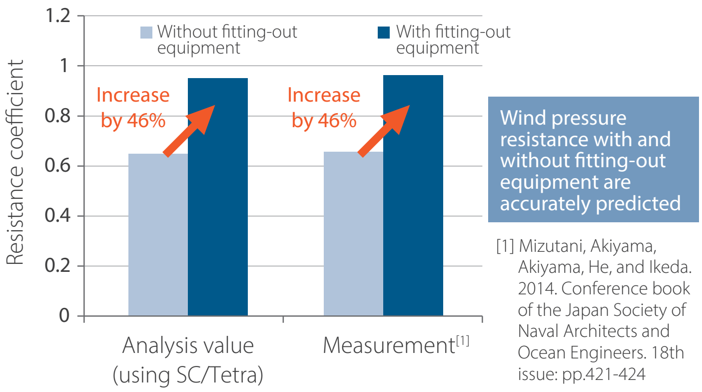


Wind resistance is reduced by locating the end of the crane downstream of the hopper, where the wind velocity is lower.

Changes in wind pressure resistance

With and without fitting-out equipment

(* wind resistance when wind direction angle is 20 degrees)



[1] Mizutani, Akiyama, Akiyama, He, and Ikeda. 2014. Conference book of the Japan Society of Naval Architects and Ocean Engineers. 18th issue: pp.421-424

Customer Comments

Wind resistance on wood chip sea carriers was analyzed using SC/Tetra. Analysis results agreed extremely well with model test results. CFD can be used to accurately calculate wind pressure resistance and validate the optimal layout and shape of the fitting-out equipment. This will reduce the time and cost of developing more fuel-efficient chip carriers.

Tank Test Simulation of Blunt Ship (Towing Condition) 1/2

Using SC/Tetra to perform tank test simulation of a blunt ship and to examine the effectiveness of Energy Saving Devices (ESD)

Analysis Objectives

Tank tests of ship models play vital roles in enhancing the propulsive performance of ships and the development of ESD, which have been actively developed as they have significant effects on ships' energy efficiency. In this case study, SC/Tetra was used to perform tank test simulation of a ship in towing condition. The simulation is targeted at a blunt ship where bilge vortices, which are key factors in CFD estimation of the propulsive performance of ships, are generated prominently.

Overset grid function of SC/Tetra was used to examine the effectiveness of ESD in towing test condition.

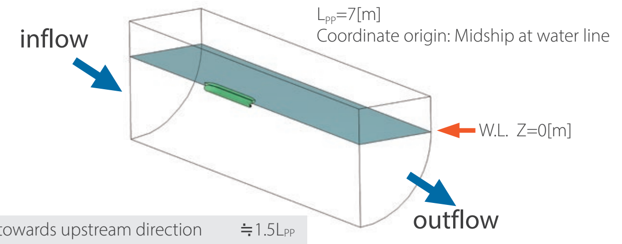
Principal particulars of model ship*[1]

*[1] JAPAN Bulk Carrier (JBC)

		Model scale
Length between perpendiculars	L_{PP} [m]	7.0
Length of waterline	L_{WL} [m]	7.125
Maximum beam of waterline	B_{WL} [m]	1.125
Depth	D [m]	0.625
Draft	T [m]	0.4125
Wetted surface area w/o ESD	$S_{0\_w/oESD}$ [m²]	0.2494
Wetted surface area with ESD	S_{0_wESD} [m²]	0.2504
Service speed	Fn	0.142
	U [m/s]	1.179
	Re	$7.46 \cdot 10^6$

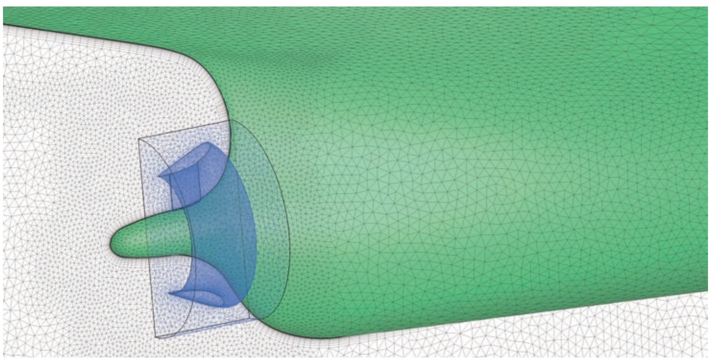
* Reference: Tokyo 2015 A Workshop on CFD in Ship Hydrodynamics
http://www.nmri.go.jp/institutes/fluid_performance_evaluation/cfd_rd/cfdws15/index.html

Analysis domain



From F.P. towards upstream direction	≒1.5L_{PP}
From A.P. towards downstream direction	≒2.5L_{PP}
Width and depth	≒L_{PP}
Above water line	=3[m]

Considering ESD using overset grids



Overset grids allocated around ESD

Analysis details*

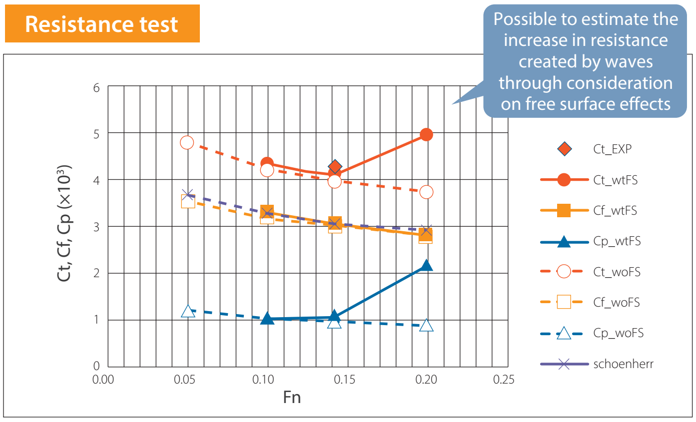
* Half model is used

Ship speed change test [without ESD]
- Without free surface (Double model)
- With free surface

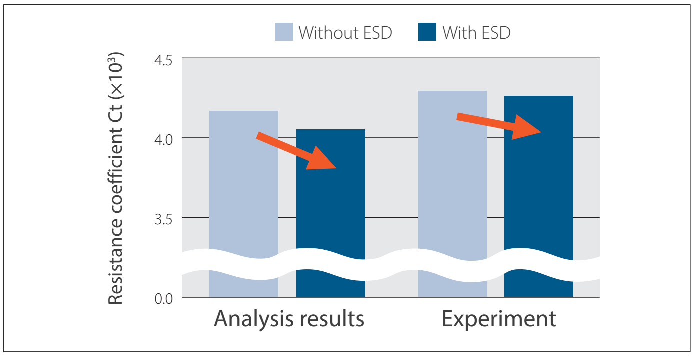
Comparison with experiment
- Resistance coefficients with/without ESD at design speed
- Wake distribution with/without ESD at design speed
- Wave height distribution without ESD at design speed

Analysis results

Resistance test

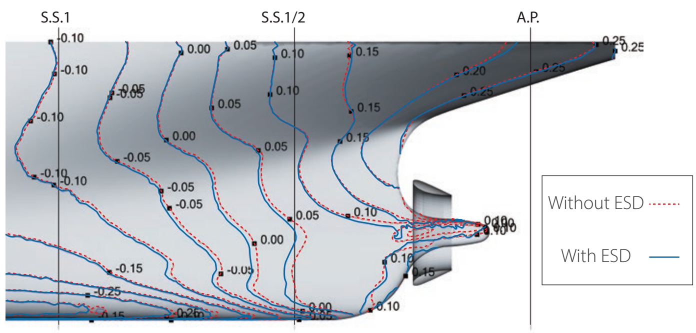


Resistance coefficients



Possible to estimate the tendency that hull resistance is reduced by attaching ESD

Pressure distribution on hull surface



Comparison of pressure distributions near aft part
[$Cp=P/(0.5 \cdot \rho \cdot U^2)$]
Pressure recovers near aft part by attaching ESD

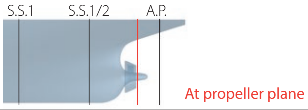
SC/Tetra

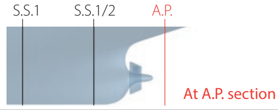
Tank Test Simulation of Blunt Ship (Towing Condition) 2/2

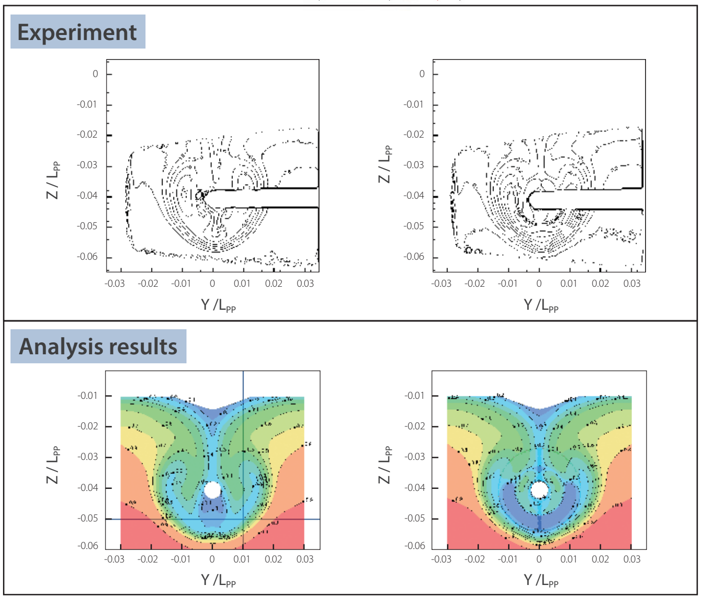
Using SC/Tetra to perform tank test simulation of a blunt ship and to examine the effectiveness of Energy Saving Devices (ESD)

Analysis results

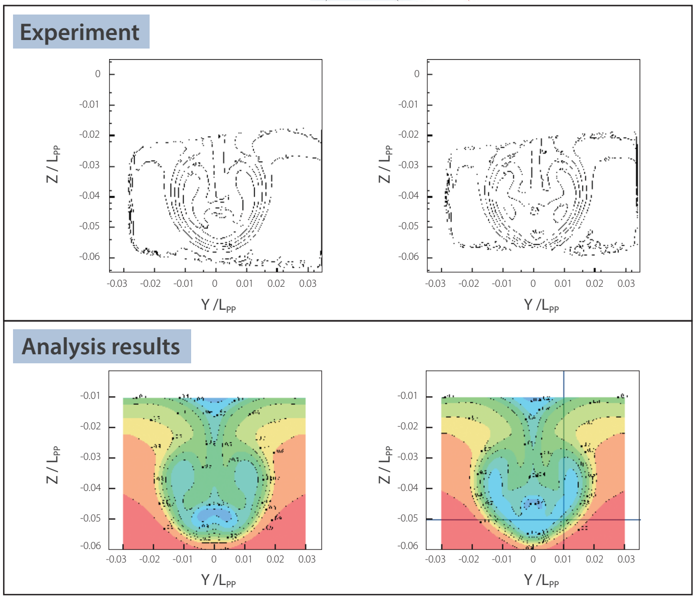
Wake distribution





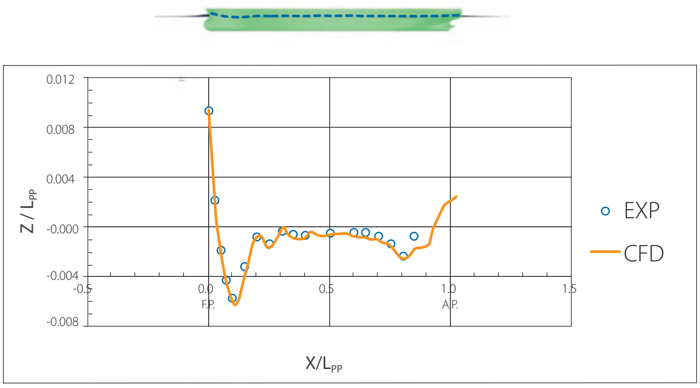


Without ESD With ESD

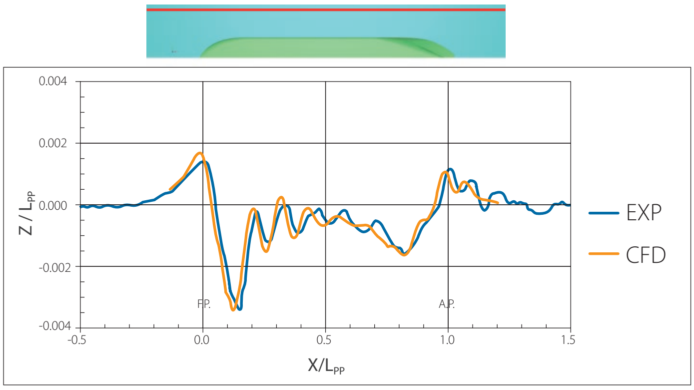


Without ESD With ESD

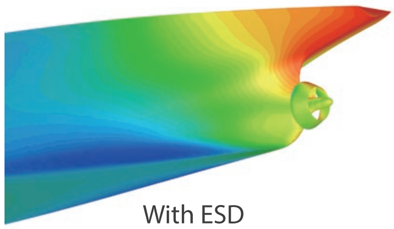
Wave profiles on hull surface



Longitudinal wave height distribution (Y/Lpp=0.19)

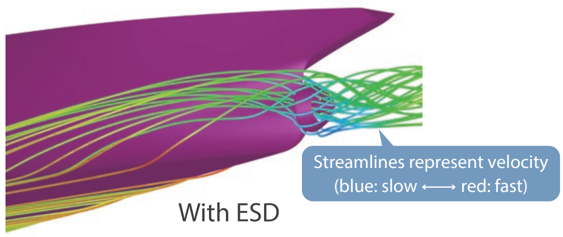


Surface pressure distribution near aft part



With ESD

Streamlines in 3D near aft part



With ESD

Notes

- The case study of a blunt ship showed that it is possible to reproduce bilge vortices near aft part, which are key factors when considering the propulsive performance of ships.
- The case study also showed SC/Tetra's overset grid function is effective for examining the effect of ESD and evaluating whether ESD help improve energy efficiency.
- Further evaluation of the ship in self-propulsion condition is possible by additionally considering rotation of an actual propeller or by applying the simplified propeller model based on the infinitely bladed propeller theory.

Tank Test Simulation of Blunt Ship (Self-Propulsion Condition)

Using SC/Tetra to simulate self-propulsion test and verify analysis results with experiment

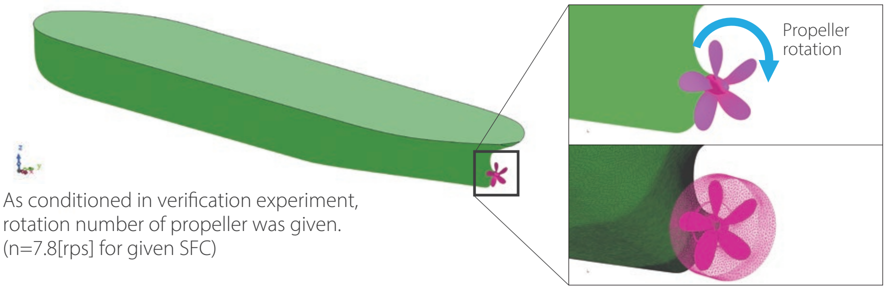
Analysis Purpose

Self-propulsion test that accounts for propeller operation is very important for the propulsive performance of ships.

SC/Tetra was used to simulate self-propulsion test and compare analysis and experiment results. Propeller influence was considered by an actual propeller rotation.

Analysis target was a blunt ship, which was same in towing condition. In this section, free surface was ignored (double model was employed).

Analysis details and meshing around propeller

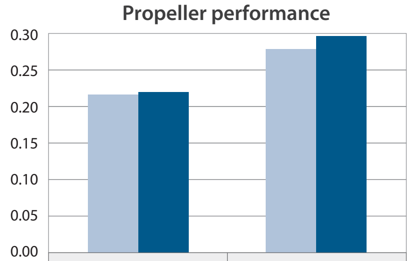


As conditioned in verification experiment, rotation number of propeller was given. (n=7.8[rps] for given SFC)

Reference: Tokyo 2015 A Workshop on CFD in Ship Hydrodynamics
http://www.nmri.go.jp/institutes/fluid_performance_evaluation/cfd_rd/cfdws15/index.html

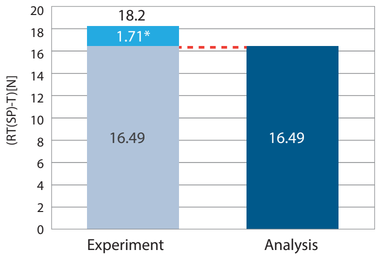
Analysis results (self-propulsion parameter)

Propeller performance



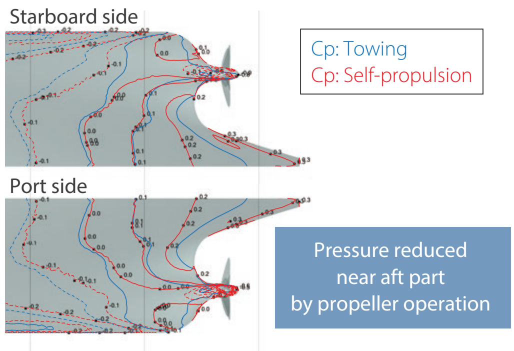
	K_T	$10K_Q$
Experiment	0.217	0.279
Analysis	0.219	0.297

Hull resistance in self-propulsion condition

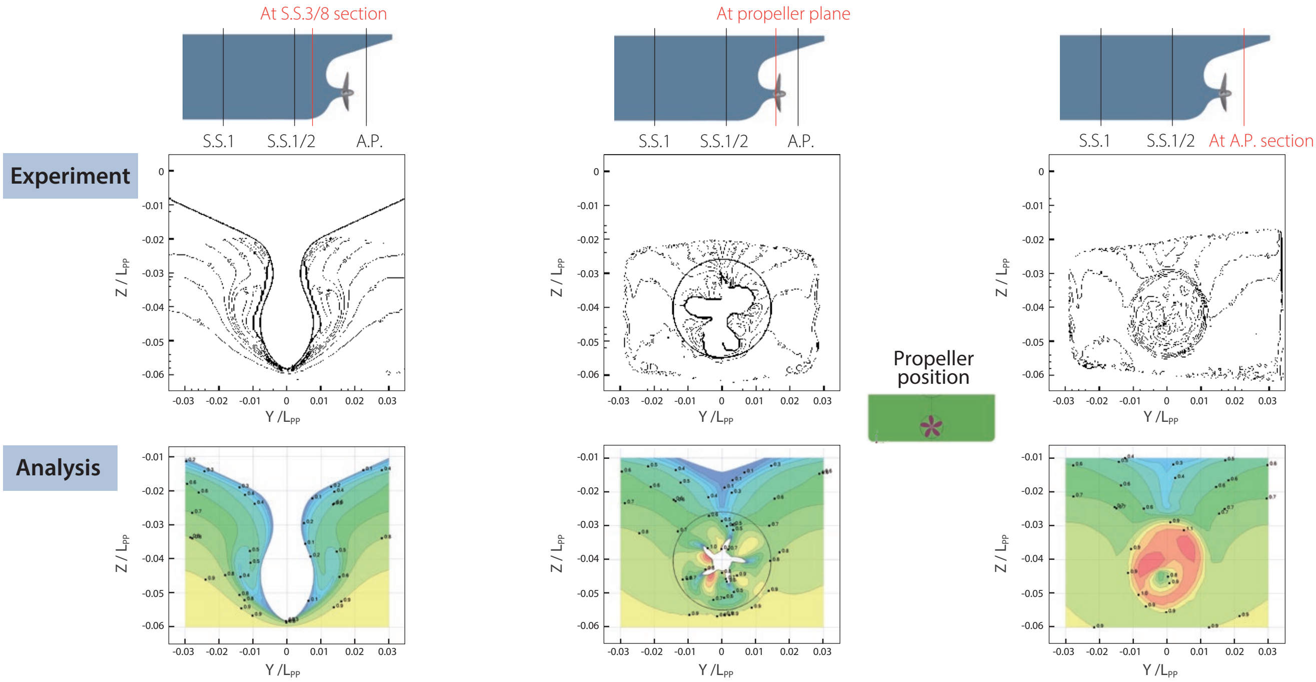


* Wave resistance is calculated as follows by towing test (RTwtFS − RTwoFS)[N]=1.71[N]

Analysis results (pressure distribution)



Analysis results (wake distribution)



Notes

- SC/Tetra was used to simulate self-propulsion test. Propeller influence was considered by actual shape rotation.
- Estimated propeller performance parameters and hull resistance in self-propulsion condition showed good agreement with results of model tests.
- Simplified propeller model based on infinitely bladed propeller theory that requires small calculation load can be applied to simulate self-propulsion condition, and either approach is applicable.

SC/Tetra

CFD Application of Simplified Propeller Model to Simulate Propeller/Rudder Interactions

Case Study of SC/Tetra

SC/Tetra and infinitely bladed propeller theory are used to analyze propeller and rudder performance

Propeller-Rudder Interaction

Interactions between the hull, propeller and rudder are critical considerations when examining the propulsion performance of a vessel. Analysis methods that rotate a realistically shaped propeller are becoming more practical. However, a high calculation load is still the main bottleneck.

To address this, a simplified model based on the infinitely bladed propeller theory [1-3] was used with SC/Tetra. This propeller model has been verified in a variety of applications and reduces the calculation load. In this case study, the simplified propeller model was used to simulate the performance of a propeller and the associated propeller-rudder interactions. Analysis results were compared with test measurements.

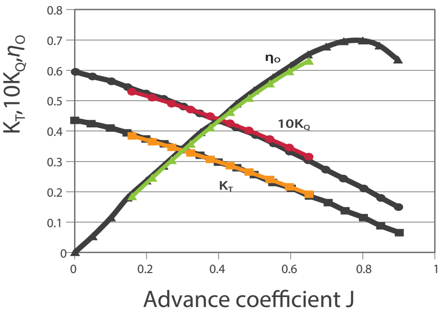
[1] Kuniharu Nakatake. 1967. Report of the West-Japan Society of Naval Architects, 34th volume: p25-36

[2] Fumio Moriyama.1979. Report of the Japan Ship Technology Research Association. 16th volume, 6th issue: p361-376

[3] Takero Tamada, Jun Ando. 2015. Conference book of the Japan Society of Naval Architects and Ocean Engineers. 21st issue: p555-558

Infinitely bladed propeller theory

Individual performance of propeller

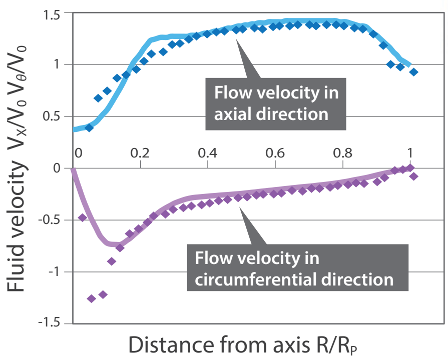


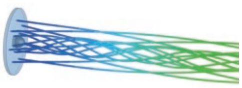
Comparison between measurement and calculated values of thrust and torque coefficients

K_T (Measurement[4])
$10K_Q$ (Measurement[4])
η_O (Measurement[4])
K_T (SC/Tetra)
$10K_Q$ (SC/Tetra)
η_O (SC/Tetra)

Performance of propeller can be predicted with high accuracy

Slipstream of propeller





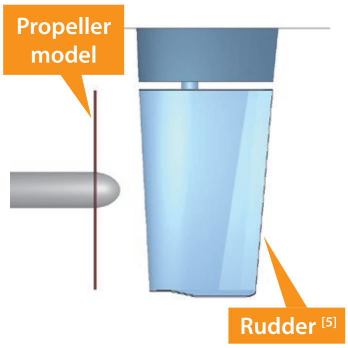
V_x/V_0(Measurement[4])
V_θ/V_0(Measurement[4])
V_x/V_0(SC/Tetra)
V_θ/V_0(SC/Tetra)

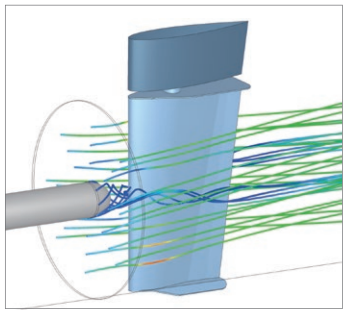
Slipstream of propeller can be simulated with small calculation load

Fluid velocity at 1.446 R_P behind propeller (R_P: propeller radius)

[4] Kazuyuki Ouchi, Masahiro Tamashima, Toshio Kawasaki, Koizuka Hajime. 1989. Journal of the Society of Naval Architects of Japan. 165th issue: pp.43-53

Analyzing propeller-rudder interaction

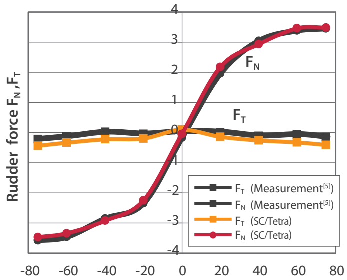




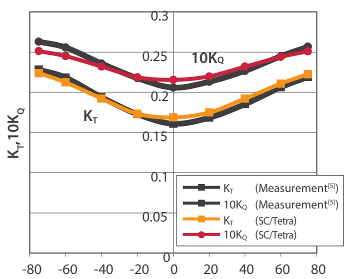
Comparison between analysis and test results

Performance of propeller and rudder during interaction

Rudder normal force F_N and rudder drag F_T



Thrust coefficient K_T and torque coefficient of $10K_Q$



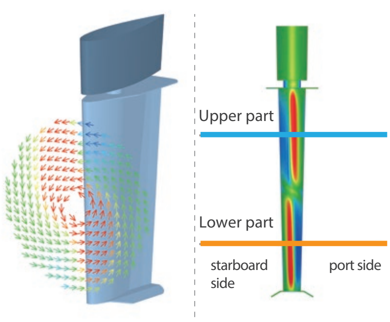
Effects of propeller-rudder interaction can be accurately assessed

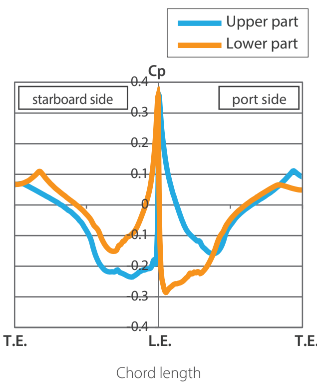
[5] Yukio Tomita, Takayuki Wakabayashi. 2001. Fune no kagaku. 54th volume: p58-61

Rotational flow near front edge of rudder and surface pressure distribution

Rotational flow near front edge of rudder

Surface pressure distribution on rudder





Can be utilized for design improvements

Notes

Propeller-rudder interactions were analyzed with a low calculation load by applying the infinitely bladed propeller theory to SC/Tetra. Analysis results closely agreed with measurements. The next step is to include the interactions with the hull. This will establish the simplified propeller model as a valuable tool for accurately estimating vessel propulsion performance using SC/Tetra.

Estimation of Marine Propeller Performance in Open Water

Case Study of SC/Tetra

Using SC/Tetra to estimate marine propeller performance for the boundary layer transition phenomenon

Estimation of Propeller Performance in Open Water

Most of a ship's propulsive power is provided by a propeller. Enhancing propeller efficiency, even by small percentage points, can lead to significant environmental and economic benefits. These potential benefits can include massive reductions in carbon dioxide emissions and major improvements in fuel efficiency.

Developing a highly efficient propeller requires conducting an accurate performance estimation during development. In this case study, the suitability of using a CFD simulation tool to evaluate propeller effectiveness was performed by comparing analysis results with experimental measurements[1].

[1] Fujiyama et al., Turbomachinery, 40th volume, pp.212-217, 2012 (in Japanese)

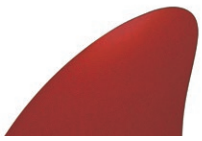
Analysis model

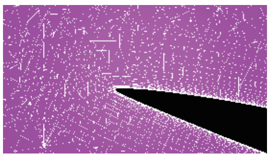
High skew propeller of Seiunmaru

Seiun-Maru high skew propeller
Model: HSP-II (MP No.220)
Number of blades: 5
Diameter: 220 [mm]
Rotation speed: 12 [rps]

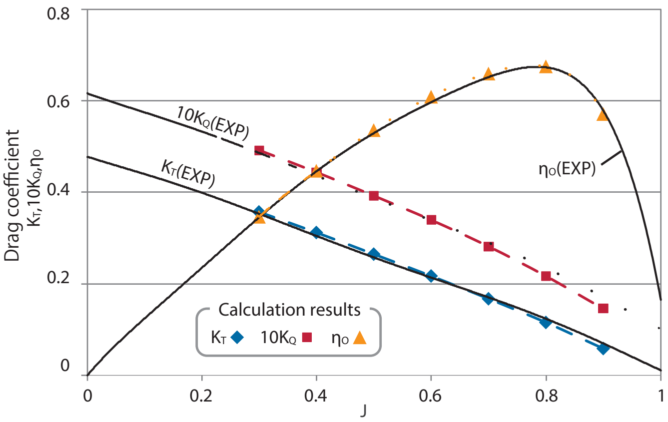
Analysis Mesh

Number of mesh elements: 55 million
Boundary layer elements: First layer $5 \cdot 10^{-7}$ [m], 30 layers





Analysis results of propeller performance in open water



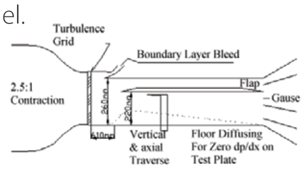
Advance ratio	$J = U/nD$
Thrust coefficient	$K_T = T/\rho n^2 D^4$
Torque coefficient	$K_Q = Q/\rho n^2 D^5$
Efficiency	$\eta_O = J \cdot K_T / 2\pi \cdot K_Q$

Analysis results agreed well with measurement[4]

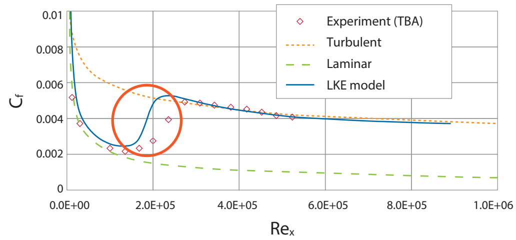
LKE k-k_L-ω model

LKE k-k_L-ω model[2], which accounts for laminar-turbulent transition, was used for the turbulence model.

Laminar-turbulent transition of the flat plate boundary layer[3]

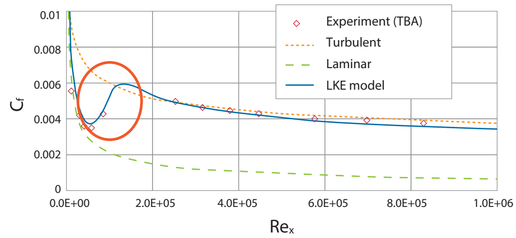


T3A



◯ = Position of transition is accurately predicted

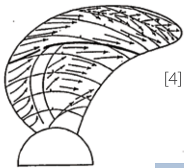
T3B



[2] Walters, D.K., et. al., ASME J. of Fluids Engineering, 130, 121401, 2008
[3] Coupland, J., ERCOFTAC Special Interest Group on Laminar to Turbulent Transition and Retransition,

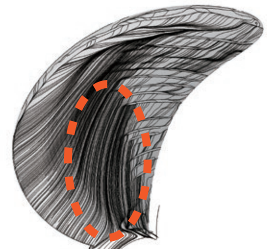
Flow on propeller blade surface

Experiment [4]



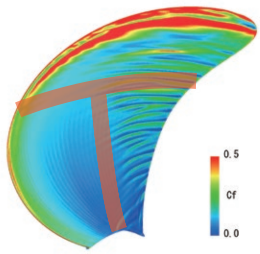
[4] Japan Ship Technology Research Association, Research on propulsion capability of propeller and estimation methods of noise characteristics. 1986 (in Japanese)

Analysis results



Changes in limiting streamlines on blade surfac were captured

Distribution of wall friction coefficient



Changes in streamline positions and increasing wall friction coefficient along the blade surface, due to boundary layer turbulent transition, agreed with experiment

Notes

By using SC/Tetra and applying a turbulence model that accounted for boundary layer transition, an accurate simulation was achieved for a marine propeller operating in open water. This confirms that CFD can be used for both propeller conceptual and detailed design evaluations. Conducting these studies as part of the propeller design and development processes can lead to more efficient propellers.

SC/Tetra

Predicting Marine Propeller Cavitation

Case Study of SC/Tetra

Using SC/Tetra to predict propeller cavitation including tip vortex region

Cavitation Flow Analysis

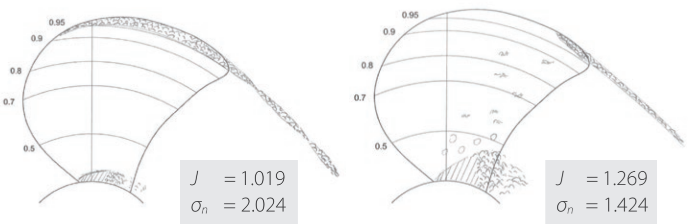
Cavitation in fluid machinery causes device degradation, vibration, and erosion. CFD can be used to predict the extent of cavitation during the propeller design and development phases, which reduces design cycle time and cost.

In this case study, CFD was used to simulate cavitation in a marine propeller, focusing especially on tip vortex cavitation. Analysis results and experimental measurements were compared and evaluated.

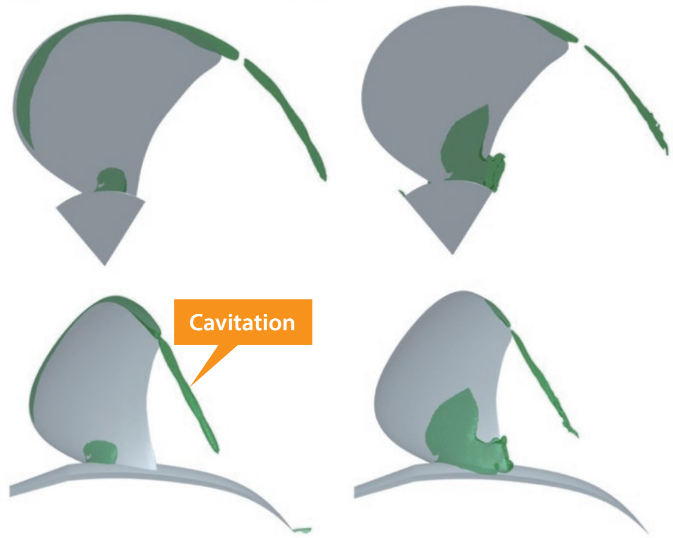
[1] Fujiyama, K. et al, smp'11 Workshop on Cavitation and Propeller Performance, 2011

Predicting the extent of cavitation

Measurement



Analysis results



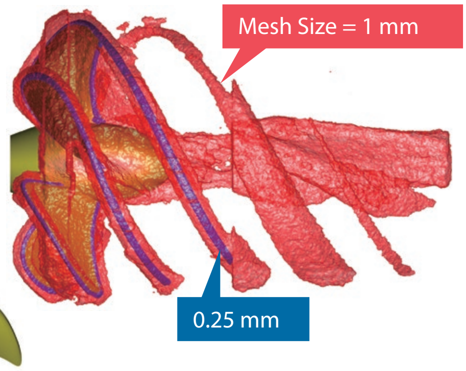
Case 1	K_T	Case 2
0.3750	(Thrust values)	0.2064
+0.67%	Versus test values	-3.59%

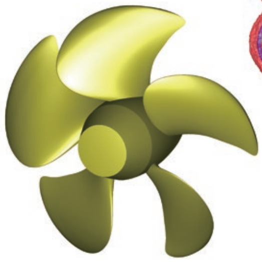
The extent of cavitation and thrust values are accurately estimated

Mesh generation using mesh adaptation analysis

Potsdam Propeller Test Case (PPTC)

Target propeller
Model: VP1304
Number of blades: 5
Diameter: 250 [mm]
Rotation number: 25 [rps]



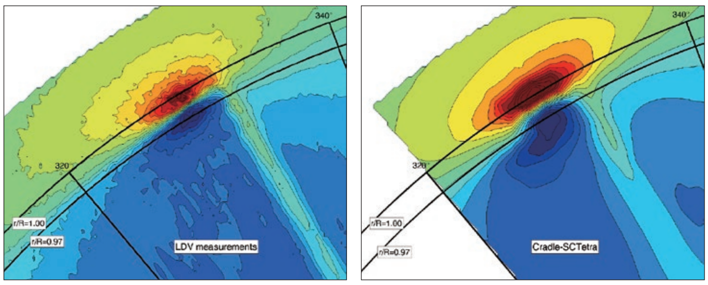


Mesh elements in tip vortex region are automatically refined by using mesh adaptation analysis

Tip vortex resolution

Vortices around propeller

Propeller velocity distribution at 0.1D downstream



Measurement using LDV | Analysis results

Velocity distribution of blade tip vortex are accurately predicted

Notes

SC/Tetra was used to accurately predict both the extent of cavitation around a marine propeller and the changes in thrust associated with the cavitation. Using mesh adaptation analysis to generate fine mesh elements, SC/Tetra accurately simulated local phenomena, such as tip vortex cavitation.

SC/Tetra

Using SC/Tetra to Estimate Ship Hull Pressure Fluctuation

Case study for cavitation flow analysis

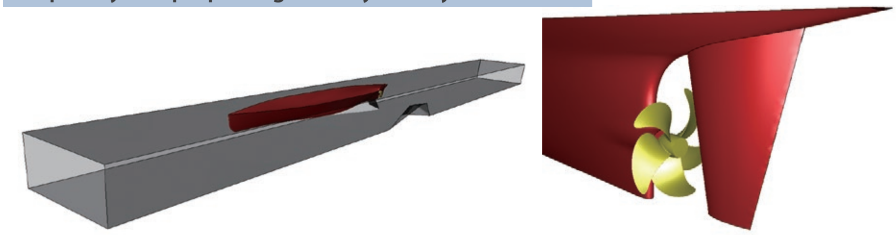
SC/Tetra is used to simulate transient cavitation around a ship propeller and estimate the resultant ship hull pressure fluctuation

Estimating Ship Hull Pressure Fluctuation

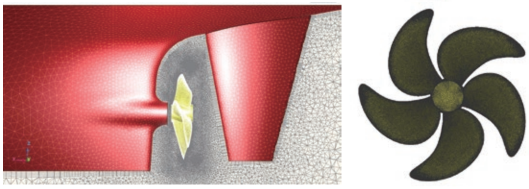
Transient cavitation around a ship propeller is caused by non-uniform flow in the wake of the ship body. Because this results in increased ship vibration, noise, and erosion, predicting cavitation during the ship design phase is essential.

Referencing cavitation flow test conditions for a ship model[1], SC/Tetra was used to evaluate propeller transient cavitation and verify the accuracy of computationally estimating the resultant ship hull pressure fluctuation[2].

Ship body and propeller geometry – analysis overview



Analysis target
Ship body: Seiunmaru
Propeller: HSP-II, CP-II
Scale: 1/16.293
Analysis condition: Ship propeller speed, 163rpm for both propellers

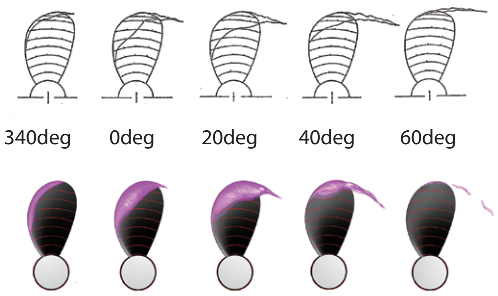


[1]Kurobe, Y., et al., "Measurement of Cavity Volume and Pressure Fluctuation on a Model of the Training Ship" 'SEIUNMARU' with Reference to Full Scale Measurement (in Japanese)", SRI Report, 1983

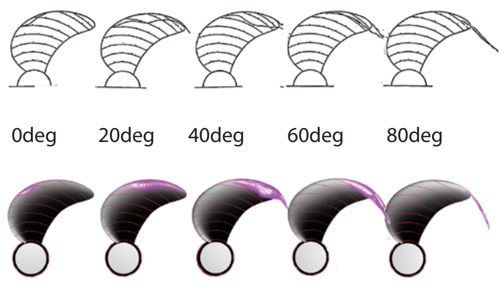
[2]Fujiyama, K., "Investigation of Ship Hull Pressure Fluctuation induced by Cavitation on Propeller using Computational Fluid Dynamics", Proc. of the 17th Cavitation Symposium, 2014

Comparison of cavitation patterns

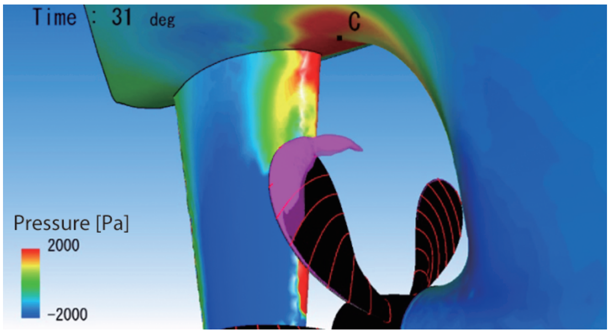
CP-II 163rpm condition

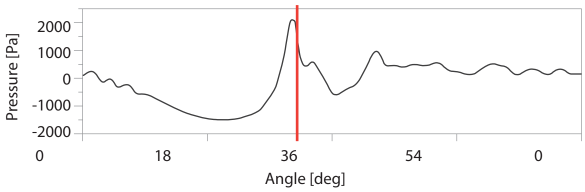


HSP-II 163rpm condition



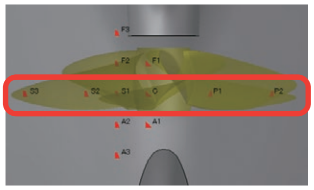
Analysis of pressure fluctuation

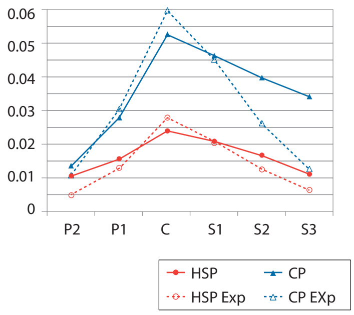




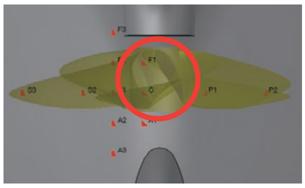
Comparison of pressure amplitude fluctuation between analysis and experiment

Primary blade passing frequency component

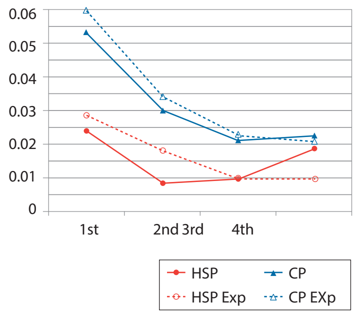




Amplitude of pressure fluctuation*



* directly above propeller



Notes

SC/Tetra was used to predict transient cavitation around a ship propeller and the subsequent induced pressure fluctuation on the ship hull with high accuracy. With this confidence, these analyses can be performed during the design phase for new ships to optimize ship body and propeller geometries.

SC/Tetra

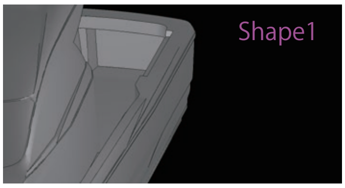
Evaluation of Small Cruising Vessel Posture using Free Surface Analysis

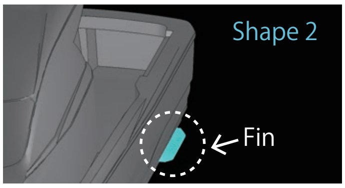
Case Study for Yamaha Motor Co., Ltd.

Analyzing posture of a cruising ship using free surface and dynamical functions

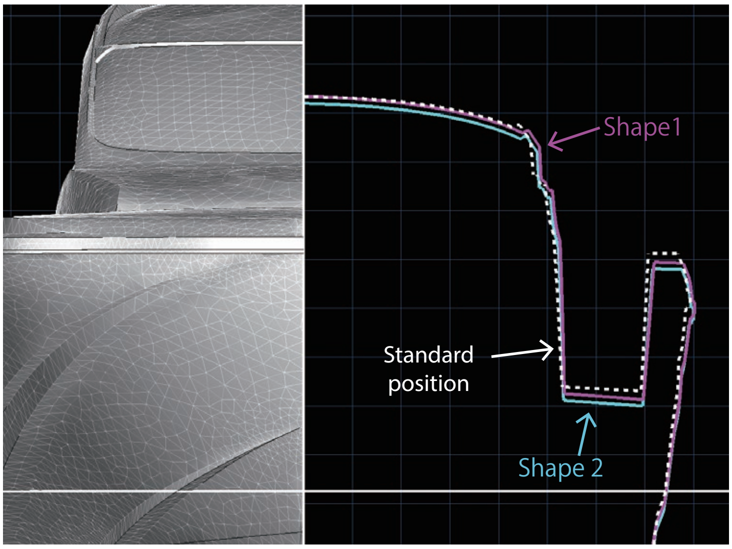
Analysis Objectives

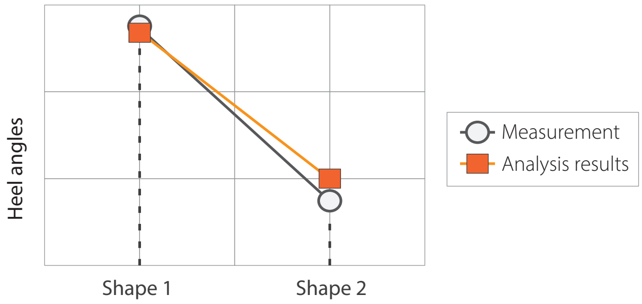
Analyses were performed to evaluate ship stability while cruising. The VOF (Volume of Fluid) method was used to simulate free surface motion and the dynamical function was used to calculate ship movement. To evaluate ship posture, two ship configurations were considered: one without a fin (shape 1) and the other with a fin on each side of the vessel (shape 2). Heel (sideward inclination) of the ship was compared for the two configurations as the ballast was moved away from the center of gravity.





Comparison between Analysis Results and Measurement

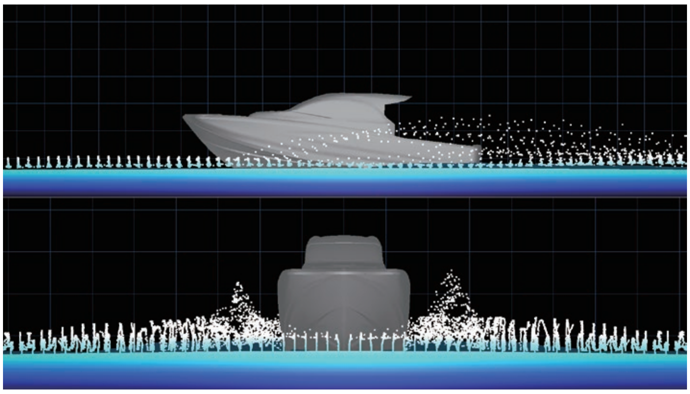




The figure on the left visually compares analysis results for heel when moving the ballast. The standard position shown in the figure represents the position of the ship when the heel angle is 0 [deg.]. The graph above compares the measured and calculated heel angles. These analysis results included stabilizing fins and confirmed the effectiveness of the fins on the vessel posture. The calculated results correlated well with measurements.

Using particles to simulate water splash

The VOF (Volume of Fluid) method, is often used to simulate free surface movement by the transport of the volume fraction of the fluid. But the VOF alone was not well suited for simulating water splash onto the ship surface. For this study, mass particles were used to simulate the water splash created by waves. The figure show particles and the interface from two directions for a VOF value = 0.5. Particles successfully simulate water splash that cannot be captured by the VOF method alone.



Customer Comments

SC/Tetra was used to simulate the posture of a small cruising vessel and to evaluate ship stability. Analysis results illustrated the effectiveness of fins attached to the sides of the ship. Calculated heel angles correlated well with measurements. Future analyses could be performed with trim.

Predicting the Proportion of Discharged Air from an Aeration Tank

Case Study for TAIKO KIKAI INDUSTRIES CO., LTD.

Evaluating gas-liquid two-phase flows using a dispersed multi-phase flow analysis function

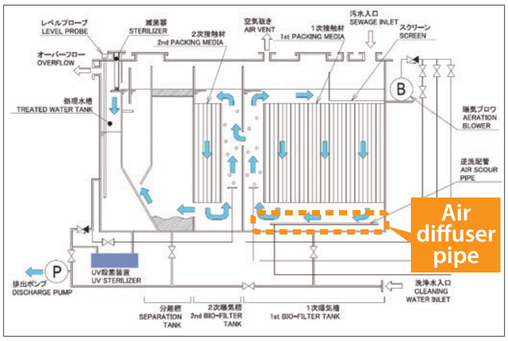
Analysis Purpose

Seaworthy marine sewage treatment equipment must be compact and highly efficient. Vital for improving the performance of the device is evening out the amount of aeration from the air diffusion pipes as shown in the diagram. In this study, the dispersed multi-phase flow analysis function in SC/Tetra was used to predict the gas-liquid two-phase flows, and to evaluate the distribution and total amount of aeration. These results were used to optimize the shape of the air diffusion pipes.

The Taiko Ships Clean "SBH Series"

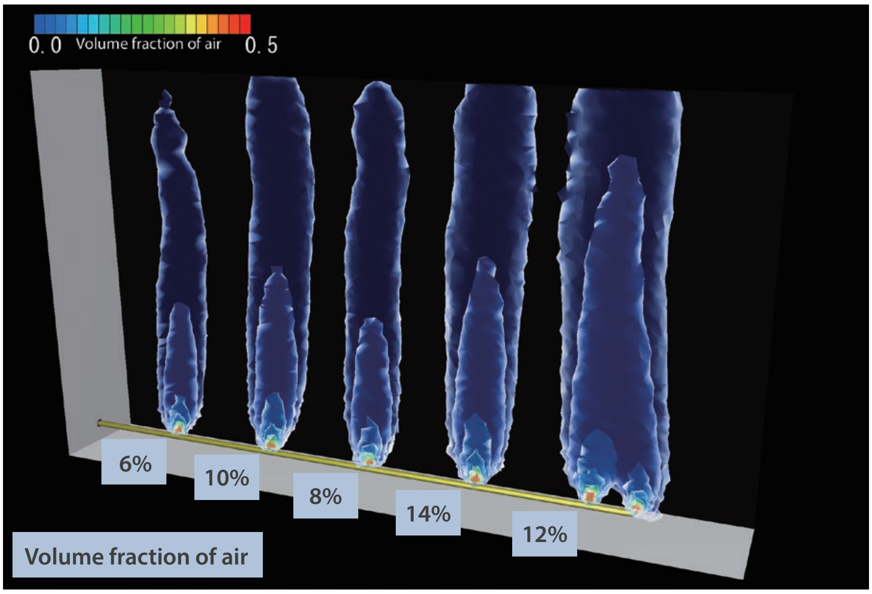
Items	SBH-15	SBH-25	SBH-40	SBH-65
Average of sewage volume (L/day)	900	1500	2400	3900
Peak of sewage volume (L/h x time/day)	94×1	156×1	250×1	406×1
BOD load (g/day)	202.5	337.5	540	877.5
Blower air flow (m^3/min)	0.1	0.255	0.40	0.59
Discharge pump capacity (m^3/h)	4 (60Hz)		3 (50Hz)	

Standard specification



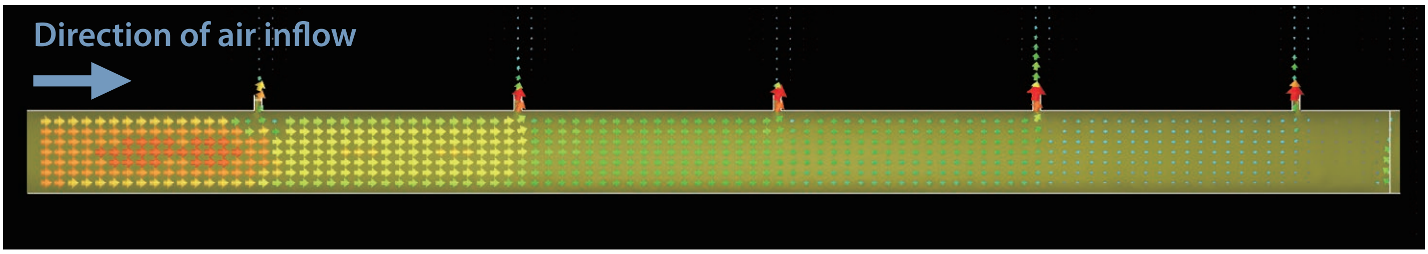
Schematics

Analysis example



The diagram shows analysis results for the distribution of air released from the air diffusion pipes for a specific inflow condition. Traditional thinking said the amount of released air would be greater near the air supply. But the analysis results contradicted this once the air system was fully filled. This was thought to be due to the diameter and layout of the air ejection holes affecting the velocity distribution within the air diffusion pipes. This, in turn, affected the distribution of the air from the pipes.

The calculation results correlated well with experimental measurements for the validation case. Additional analyses were performed to determine the optimal shape of the air diffusion pipes.



Customer Comments

Using SC/Tetra enabled us to design the air diffusion pipes for a marine sewage plant without having to perform water tank model tests. Test results for the actual device showed that the air was evenly aerated. This confirmed the value and effectiveness of using SC/Tetra during the design and development phases.

SC/Tetra

Evaluation of Marine Diesel Engine Coolability

Case Study for DAIHATSU DIESEL MFG.CO.,LTD

Using SC/Tetra to analyze the cooling water jacket for marine diesel engine and to evaluate coolability

Diesel Engine DE-18

Product image

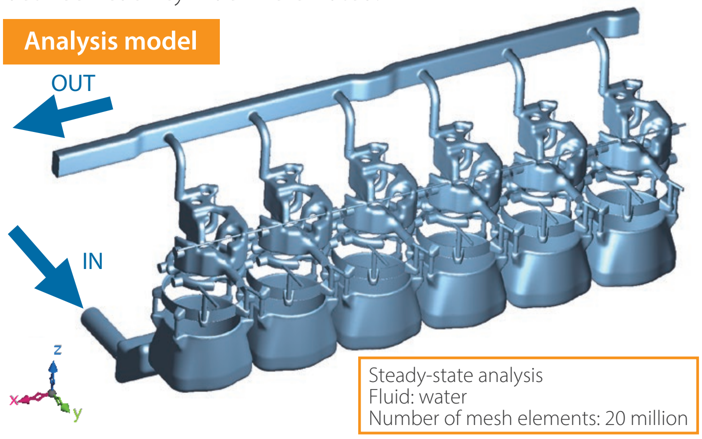
The Daihatsu Diesel DE-18 is an economically efficient, next-generation environmentally friendly diesel engine. Having complied with IMO Tier II exhaust emissions requirements while realizing the regulations will continue to tighten in the future, the DE-18 achieves energy-efficiency and low maintenance-costs. The DE-18 takes full advantage of Daihatsu Diesel's proficient experience in developing highly reliable and durable diesel engines.

Design of the cooling water jacket for a diesel engine is a vital part of engine development. Engineers need to generate sufficient cooling effectiveness through highly complicated cooling water passages near the cylinder head. Because the engine is large, prototype tests were extremely difficult and expensive to perform. As an alternative, computational analyses were performed to evaluate product performance.

Comparison between cylinders

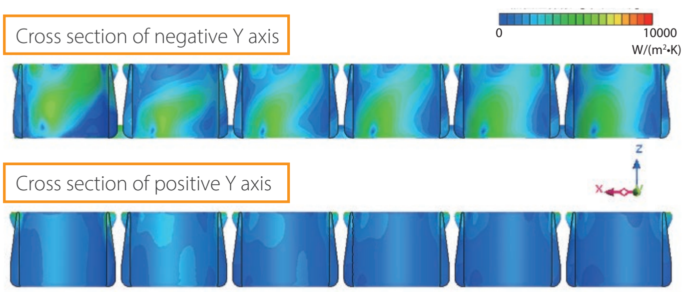
Analyses were performed using an all-cylinder model. Differences between each cylinder were noted.

Analysis model



Flow path model of water jacket

Cross section of cylinder inner wall



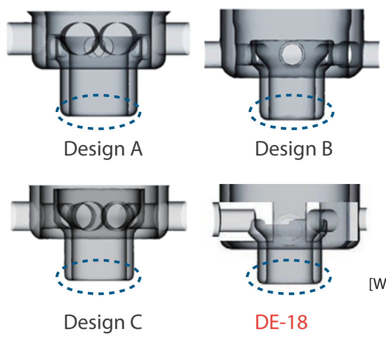
Distribution of heat transfer coefficient on inner wall of cylinder

Deviation in heat transfer distribution between cylinders is small. Predicted values satisfy the requirement.

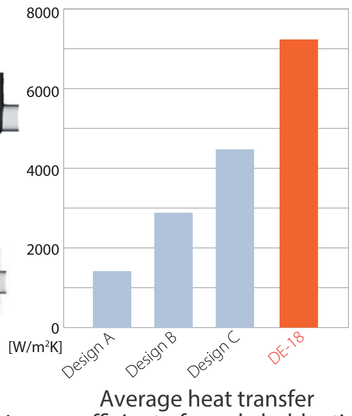
Comparison between different designs

Analysis results were compared between different designs of components, which are heated during the operation.

Tip of nozzle holder

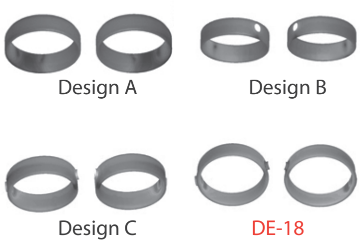


Nozzle holder shapes and holder tip



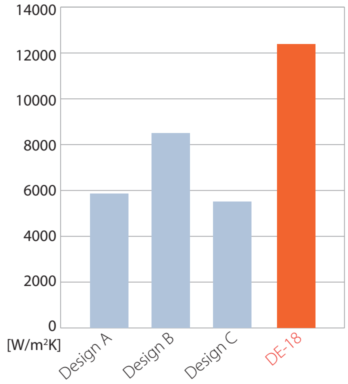
Average heat transfer coefficient of nozzle holder tip

Exhaust valve seat



Valve seat shapes

* Different shapes and upstream cooling paths for each type of valve seat



Average heat transfer coefficient of valve seat

Comparison shows that the coolability of the DE-18 is higher than any other designs.

Customer Comments

Applying SC/Tetra to design the water jacket for a marine diesel engine significantly contributed to enhancing coolability and reducing development costs. The large size of a marine engine makes it difficult to perform trial and error tests using the actual product. During development of the DE-18, only one prototype was used. The design was successfully iterated using simulations to predict performance. As a result, both the time needed for the design phase and prototyping costs were drastically reduced.

SC/Tetra

Assessing the Capability of Water Lens Turbine for Tidal Power Generation

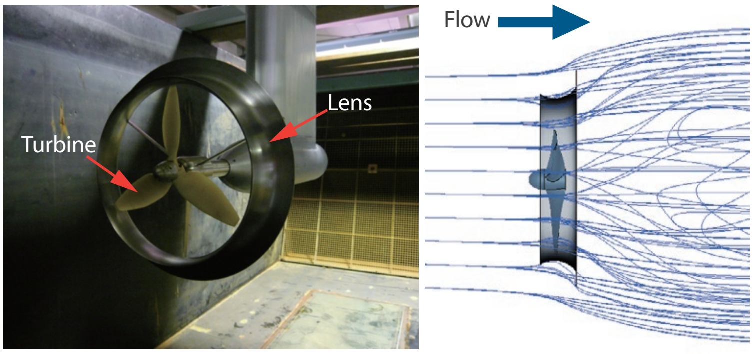
Case Study for Kyushu University

SC/Tetra shows that the power coefficient is significantly increased by attaching a lens to a water turbine

Background

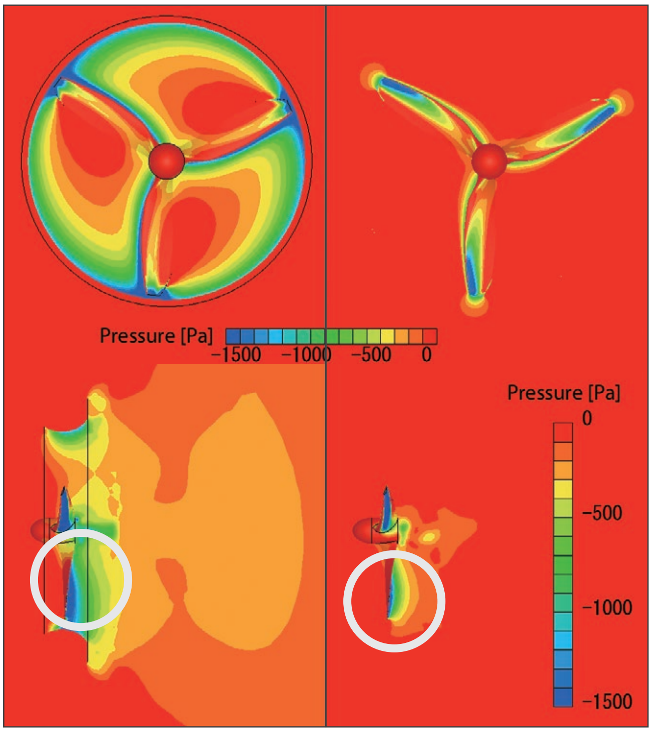
Tidal power generation is considered as the most promising approach for marine renewable energy development. Various researches have been made on its energy efficiency.

The "wind lens" turbine, developed by Professor Ohya from Kyushu University, has a ring-shaped diffuser along the outer edge of its blades and is known to be effective in improving energy efficiency. Using this technology, the "water lens" turbine was developed to create the same effects in tidal power generation. In this case study, analyses were performed to evaluate performance of a water turbine with and without the lens and to compare flow distributions and power coefficients.



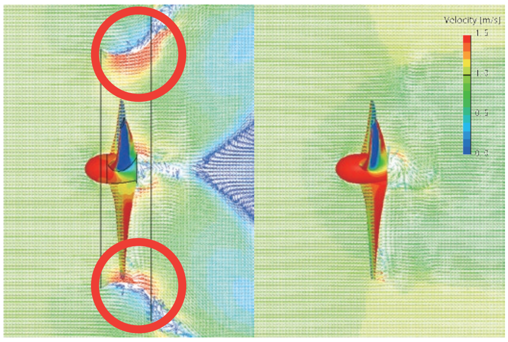
Simulation results

Pressure distribution



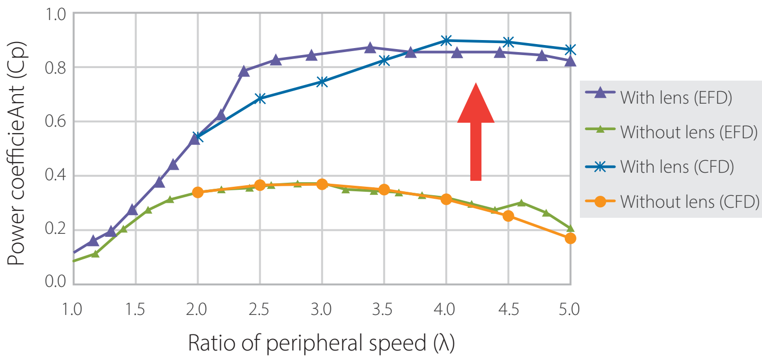
With lens, pressure difference across the turbine in the flow direction is greater

Velocity distribution



With lens, flow velocity becomes higher due to the diffuser effect

Comparison between analysis and experiment results



Both analysis and experiment results show that the power coefficients increase by more than double when the lens is attached

Customer Comments

Moving element function of SC/Tetra was used to analyze turbine models with and without the lens. SC/Tetra enabled visualization and verification of the difference caused by effects of the lens diffuser in flow velocity and pressure distribution. Power coefficients, used as indices for power generating efficiency of turbines, were estimated by CFD calculations. Comparison between the cases with and without the lens showed that the power coefficient value was more than doubled with the lens. The experiment result showed a similar pattern. This validates that CFD can be used to assess capability of the water lens turbine effectively.

Contact us here

Hexagon is a global leader in digital reality solutions, combining sensor, software and autonomous technologies. We are putting data to work to boost efficiency, productivity, quality and safety across industrial, manufacturing, infrastructure, public sector, and mobility applications.

Our technologies are shaping production and people-related ecosystems to become increasingly connected and autonomous – ensuring a scalable, sustainable future.

Hexagon's Manufacturing Intelligence division provides solutions that use data from design and engineering, production and metrology to make manufacturing smarter. For more information, visit hexagonmi.com.

Learn more about Hexagon (Nasdaq Stockholm: HEXA B) at hexagon.com and follow us @HexagonAB.