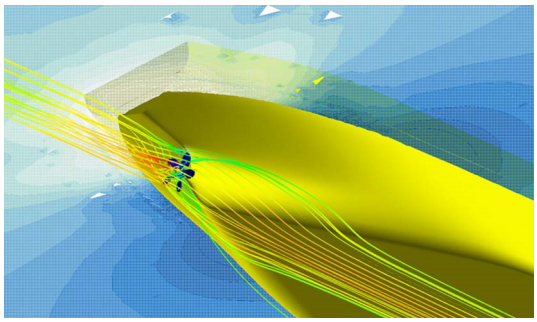
Development of Technology to Estimate the Flow Field around Ship Hull Considering Wave Making and Propeller Rotating Effects



MAKOTO KAWABUCHI*1 MASAYA KUBOTA*1

SATORU ISHIKAWA*2

As can be seen from the application of energy efficiency design index (EEDI)-related regulations the enhancement of propulsive performance in the design phase of merchant ships have been increasing in recent years. Mitsubishi Heavy Industries, Ltd. (MHI) develops hull forms precisely and efficiently using enhanced CFD technologies. The company improved the estimation accuracy of flow fields around the vessel by eliminating simplification of models as much as possible, by reproducing conditions close to the actual phenomenon. This paper presents the developed CFD tool and examples of its application to product development.

1. Introduction

The common practice in evaluating the propulsive performance of a merchant vessel is to separate it into the hull resistance, the self-propulsion factor. Self-propulsion factor is the interference effect between the hull and propeller and the propeller open water characteristics. These parameters can be obtained by resistance tests, self-propulsion tests and propeller open-water tests, respectively. These performance parameters vary while interacting with each other according to changes in the shape of the hull, the appendages and the propeller. It is necessary for the development of a vessel shape superior in energy-saving performance to accurately evaluate the resistance, the self-propulsion factor, and the propeller open water characteristics to determine whether the performance is good or bad.

Improvement in the estimation accuracy of numerical parameters related to propulsive performance requires enhancement in the estimation accuracy of flow fields. Accordingly, MHI utilized an ultrahigh-speed mesh generator[1] to precisely reproduce the shape of an analysis object and developed large-scale analysis technology using meshes with the mesh density required for the precise reproduction of the phenomenon occurring around the hull.

We developed an analysis technique with higher versatility and higher estimation accuracy of flow fields compared to the conventional analysis technique[2] by improving the self-propulsion analysis technique

2. Establishment of analysis technique

2.1 Summary of conventional technique

The conventional analysis technique focuses on the analysis time and the improvement of the stability and uses, particularly in the estimation of self-propulsion factors. The simplification of the analysis model regarding the two phenomena of a free surface effect and a propeller rotation effect as shown in **Table 1**. Specifically, a two phase flow model that assumes the free surface effect to be small and gives a vertical symmetric condition to the boundary corresponding to a still water interface is adopted. The propeller model simulates the flow generated from the propeller by providing the effect of the propeller rotation and the suction flow calculated by the UQCM (unsteady quasi-continuous method) as a body force into the disc region defined in the calculating area.

*1 Fluid Dynamics Research Department, Research & Innovation Center
*2 Senior Manager, Research & Innovation Center

Table 1 Comparison of analysis techniques

	Conventional CFD	Latest CFD
Treatment of stern shape	Complex shape is reproduced precisely	Complex shape is reproduced precisely
Grid type	Unstructured hexahedral grid	Unstructured hexahedral grid
Number of grids	Several million to 10 million	25 million to 50 million
Propeller model in self-propulsion calculation	Body force model based on UQCM	Sliding mesh based on actual propeller geometry
Handling of free surface in self-navigation calculation	Still water interface	Interface capturing (VOF)
Turbulence model	SST k-ω (two equations)	SST k-ω (two equations)
Number of parallel CPUs	16 to 64	64 to 256

2.2 Improvement of analysis technique

We tried to improve the analysis accuracy by precisely modeling phenomena that are simplified by the conventional techniques. In other words, the analysis of a free surface using a VOF (volume of fluid) model in self-propulsion analysis, reproduction of the blade shape and a model for rotating the propeller shape using sliding mesh methods are adopted. **Figure 1** shows a mesh arrangement of the analysis grid around the propeller for a Japan Bulk Carrier (hereinafter referred to as JBC), which is a bulk carrier for which the vessel shape and tank test results are publically available. It is known that three-dimensional separation of vortexes occur at the stern of a full ship such as a bulk carrier. The mesh is subdivided in order to capture the inception of the vortex separation and suppress the numerical attenuation of the vortexes. The mesh refinement is provided for the area covering distances from the central part of the ship (midship) to the stern end in the longitudinal direction, half of the vessel along the width and from the Base Line of the vessel to the still water interface in the height direction.

Figure 2 shows the isosurface of the second invariant of the velocity gradient tensor.[3] This figure indicates that the inception of vortexes occurs in the refined mesh area and that the vortexes flow into the propeller surface without attenuation.

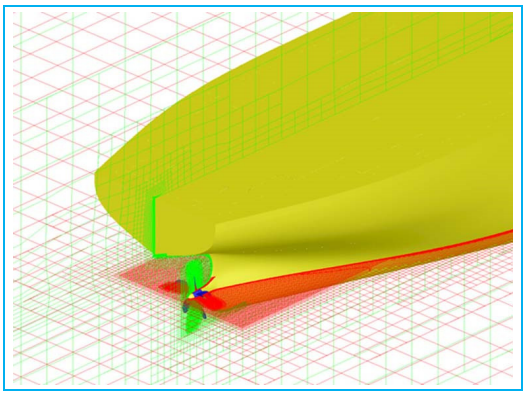


Figure 1 Mesh arrangement around JBC stern

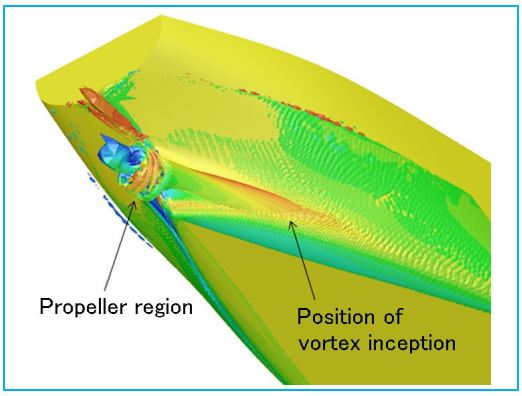


Figure 2 Isosurface of the second invariant of the velocity gradient tensor for JBC

Figure 3 compares the flow rate contours in the direction of the main flow field to the propeller surface without the propeller between the flow field PIV (Particle Image Velocimetry) measurement results in the conventional analysis technique and the improved analysis technique for JBC. This figure indicates that the improved analysis results are closer to the experimental values than the conventional analysis results, such as the distribution profile of the low-speed region shown around the width direction coordinate of -0.01 and the height direction coordinate of -0.04, and the distribution of the low-speed region shown around the width direction coordinate of -0.01 to 0 and the height direction coordinate of -0.05 to -0.04.

Figure 4 compares the estimation results of self-propulsion factors in the analysis of JBC. The self-propulsion factor parameters, 1-t (where t is the thrust deduction factor(Note 1)), 1-wm (where wm is the wake fraction t(Note 2)), and er (relative rotative efficiency(Note 3)) are compared. This figure indicates that the estimation accuracy of self-propulsion factors has increased significantly in comparison with the conventional technique due to the improvement of the flow field estimation accuracy and the enhancement of the propeller model.

Note 1: The ratio of difference between the propeller thrust in the self-propulsion of a ship at a certain speed and the hull resistances occurring in the towing of the same ship at the same speed against the propeller thrust

Note 2: The ratio of difference between the inflow velocity to the propeller and the ship advancing speed against the ship advancing speed

Note 3: The ratio of the propeller efficiency behind the ship, which is the efficiency of the propeller operating in a non-uniform flow field at the stern against the propeller open-water characteristics , which is the efficiency of the propeller operating in a uniform flow

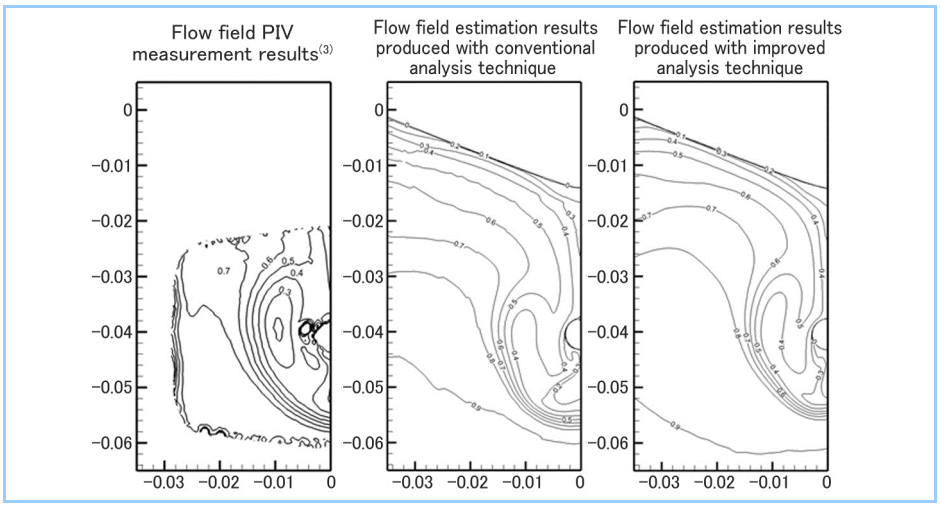


Figure 3 Comparison of wake fraction in the direction of main flow on propeller surface

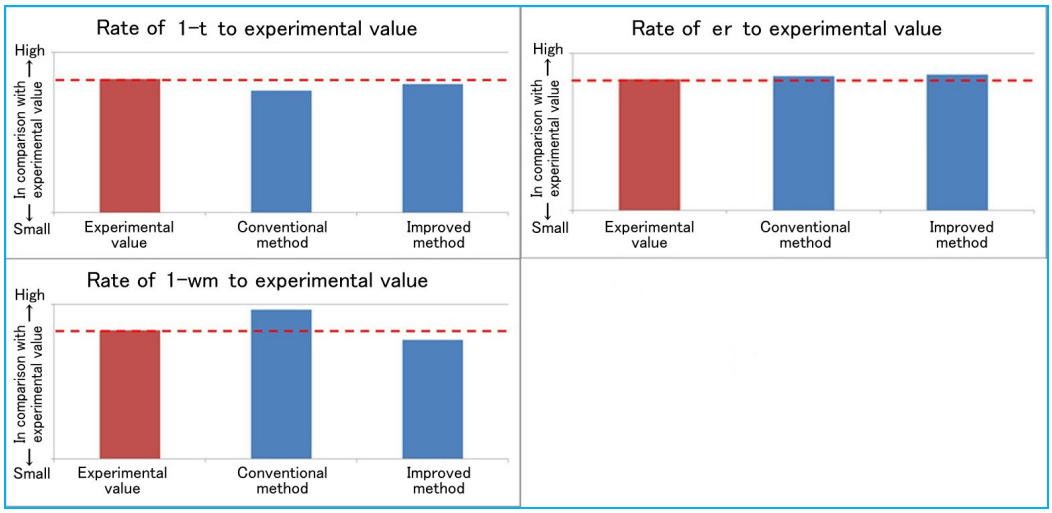


Figure 4 Comparison of self-propulsion factor obtained from experiments and CFD(new and old models)

3. Application examples of analysis technique

MHI utilizes the analysis technique for which the self-propulsion factor analysis accuracy has been enhanced in comparison with the conventional techniques for the development of a hull form and additional appendages around the propeller. This chapter presents two examples.

3.1 Improvement of shape of rudder, propeller and energy-saving device

MHI uses highly accurate flow field estimation results to apply a new analysis technique to improve the shape of the propeller, rudder and energy-saving devices. The accuracy of the estimation is strongly influenced by the flow generated by the propeller for designing the cross sectional shape and the installation angle of the rudder and energy-saving devices.

Figure 5 shows unsteady integrative analysis situations where the shapes of the rudder, energy-saving device and a propeller are reproduced. Based on the highly accurate analysis of interference between wakes from the hull and flows at the energy-saving device, the propeller and the rudder our development does not aim the individual optimization for maximizing each effect, but overall optimization of the propulsion system including the hull, device, propeller and rudder. As an example of such improvement, **Figure 6** shows an analysis case where the blade angle of an

energy-saving device was changed. Before the improvement, there was separation at the suction side of the blade that was assumed to cause the increased device resistance. After the improvement, however, no separation occurs on the suction side or the pressure side of the blade at the same time low resistance and a higher device impact can be expected.

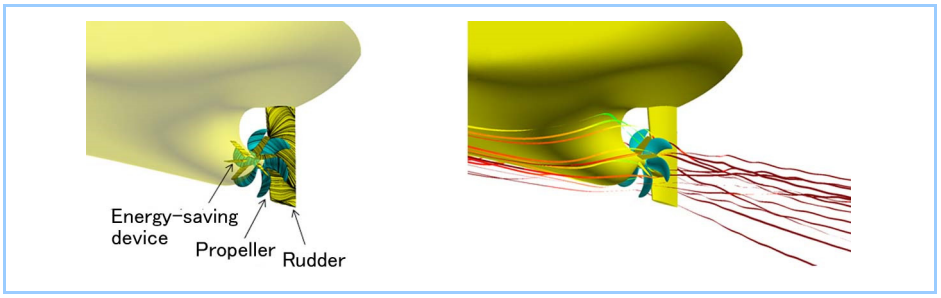


Figure 5 Integrative analysis situation of rudder, energy-saving device and propeller
Left figure: Arrangement of energy-saving device, propeller rudder assembly and limiting flow line of object surface
Right figure: Indication of flow line within the space at the stern and a check of the flow field improvement effect of the energy-saving device

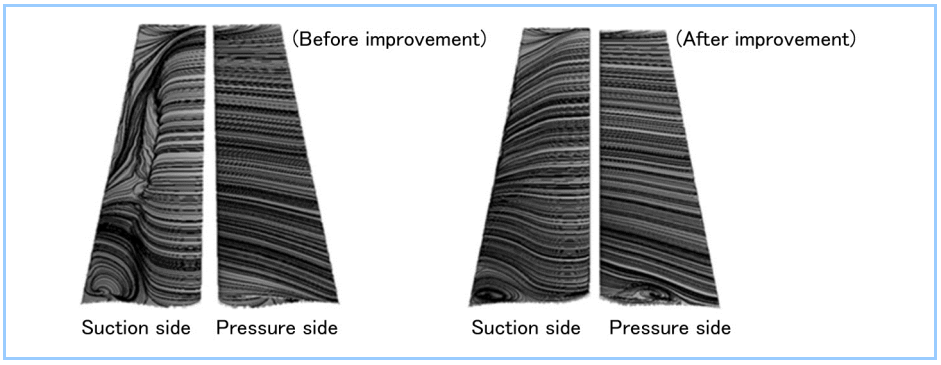


Figure 6 Comparison of flow on energy-saving device blade surface before and after changing the blade angle

3.2 Consideration of submergence at transom stern

Due to the precise reproduction of the stern shape and the increased grid resolution close to the free surface wave making at the stern, which is important for the reduction of hull resistance can be improved using CFD analysis.

For ensuring the stability of container carriers and other ship types that require higher stability performance at a relatively high speed, it is necessary to enlarge the water plane area. **Figure 7** shows an example case where wave making was improved by lowering the transom stern height. After the improvement the transom stern is set at a lower height, the wave upsurge just behind the transom stern is reduced and a broader water plane area can be kept.

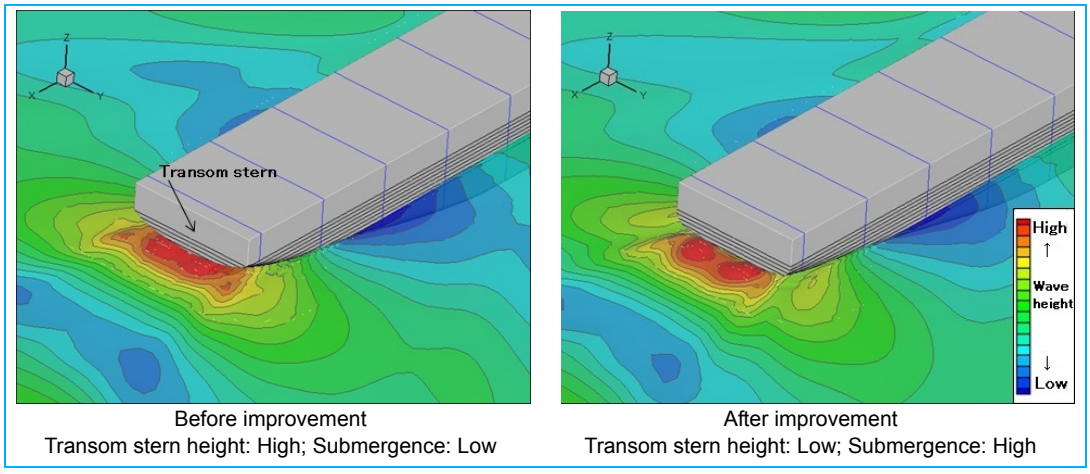


Figure 7 Change in wave height by changing height of transom stern
The stern wave height can be controlled by adjusting the height (submergence) and the sectional shape of the transom stern.

4. Conclusion

MHI improved the analysis accuracy considering the free surface effect and propeller rotation effect in the self-propulsion condition of a ship by using an analysis model that reproduces the actual flow field accurately. In addition to enhancement in the accuracy of the optimum design of the hull itself, efficient improvement in the shape of appendages such as the rudder or energy-saving devices around the propeller is also made possible. This leads to the reduction of the number of tank tests. It also became possible to consider the unsteadiness caused by the rotation of the propeller. In the future, we will utilize these technologies to develop devices and rudders with greater energy-saving effects in terms of robustness against changes in the flow field.

References

1. Tanaka, M. et al., Product Innovation by applying the Latest Technology of Numerical Simulations, Mitsubishi Juko Giho Vol. 48 No. 1 (2011)
2. Nishigaki, M. et al., Application of Advanced CFD Technology to Energy-Saving Hull form Development, Mitsubishi Heavy Industries Technical Review Vol. 50 No. 2 (2013)
3. Hino, T. et al, FLOW SIMULATIONS OF JAPAN BULK CARRIER USING OVERSET GRID APPROACH, Tokyo 2015 A Workshop on CFD in Ship Hydrodynamics, Proceedings, Volume III, (2015), P. III-443–III-447