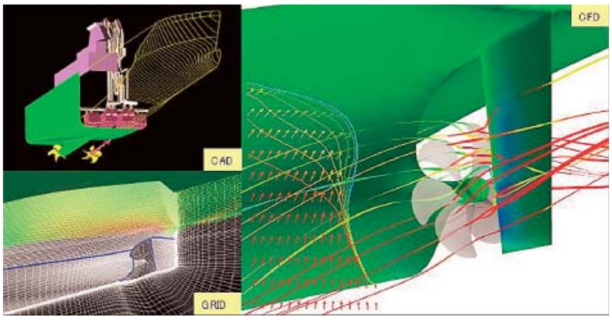


Ship Design Technology with CFD - Improvement of Propulsive Performance -

MAKOTO NISHIGAKI*1
KEI SATO*1
SATORU ISHIKAWA*1

With the progress of computational fluid dynamics (CFD), it is now being used more often as a performance prediction tool for hull design. In particular, combining computation codes capable of accurately interpreting free surfaces and the multi-block grid technique has enabled highly accurate estimations of the propulsion performance of ships with complex stern forms. Integrating this CFD technique with databases comprising CAD, model tests and experiments on actual ships, Mitsubishi Heavy Industries, Ltd. (MHI) has established a hull form design aid system that achieves better design. This report outlines this system and examples of applications. It also discusses an example of studying a combined propulsion system and a performance improvement device as a trial for applying CFD to complex stern forms.

1. Introduction

Advances in computational fluid dynamics (CFD) techniques have made accurate estimates of propulsive performance possible not only for automobiles, aircraft and rockets but also now for ships by combining with databases comprising data from model tests and experiments on actual ships. It has now begun to be employed to estimate propulsive performance at an early stage of ship design. Meanwhile, more sophisticated CAD techniques for ship design enable hull forms to be generated or modified in a short period of time. In addition, progress in parallel computers is also creating an environment which can deal effectively with a great number of computations. By integrating these three techniques, namely CFD, CAD and parallel computation, MHI has established a hull form design aid system capable of identifying high performance hull forms effectively in a short period of time from many candidates. The following sections describe some examples of applying this system to actual MHI hull designs.

2. Applying CFD techniques for developing hull forms

2.1 Hull form design aid system

By combining CAD and CFD techniques, MHI has established the hull form design aid system shown in **Fig.1**. First, as part of initial planning, a suitable parent hull form is chosen from the database according to the specification requirements. Based on this, hull form CAD is used to design the hull form. Next, the hull form is systematically modified according to the design strategy to create a number of hull forms. Calculations are carried out with these hull forms using the parallel computer, whose results are used to make a performance evaluation map. The designer uses the map to further develop the design for the optimum hull form. Each of these processes is linked with the databases of hull forms, model test results, CFD and results of sea trials of actual ships, making it easy to select a parent hull form, plan a design strategy and ensure accuracy in the performance evaluation. Employing the hull form design aid system has enabled hull forms of high performance to be determined and has drastically reduced the period of time required to develop a hull form.

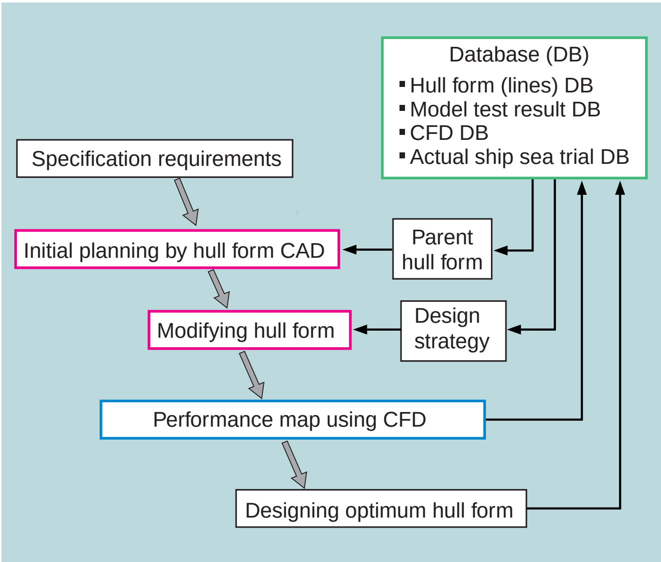


Fig. 1 Design aid system by hull form CAD and CFD
Combining hull form CAD and CFD with databases enables choosing hull forms with excellent performance.

*1 Nagasaki Research & Development Center, Technical Headquarters

2.2 CFD computation method

A CFD computation is a method of numerically solving the Navier-Stokes equation using a computational grid generated around an object. It can calculate the fluid forces exerted on an object and in the wake distribution around an object. To apply CFD to ships, it is necessary to consider the wave-making phenomenon on a free surface and the effect of the propeller at the stern. As a method for this purpose, MHI has developed a computational code, FS-MINTS (flow solver of Mitsubishi numerical tank system), that uses the multi-block grid technique to solve the RANS (Reynolds-averaged Navier-Stokes) equation[1-2].

2.3 Search for hull forms which reduce wave resistance

Normally, the factors governing the propulsion performance of ships are divided into hull resistance and propulsion efficiency. To reduce hull resistance, as shown in **Fig. 2**, hull form which has lower peak of wave profiles or pressure distribution is looked for. For this, CFD computation is conducted so that the hull form parameters affecting the hull resistance are selected and thoroughly changed to create a performance evaluation map as shown in **Fig. 3**. This in turn helps evaluate the direction of the hull improvements. Meanwhile, a method to automatically change the hull form and the grid around the hull has been developed and is used as shown in **Fig. 4** so that better hull forms can be obtained effectively. The development of these methods has enabled the design of hull forms with small hull resistance while effectively evaluating many kinds of hull form.

2.4 Search for hull forms with improved propulsion efficiency

Propulsion efficiency, which is the other factor which determines the propulsion performance of a ship, is evaluated as in **Fig. 5** by calculating the flow distribution around a ship with rudder in self-propelled condition (considering the propeller).

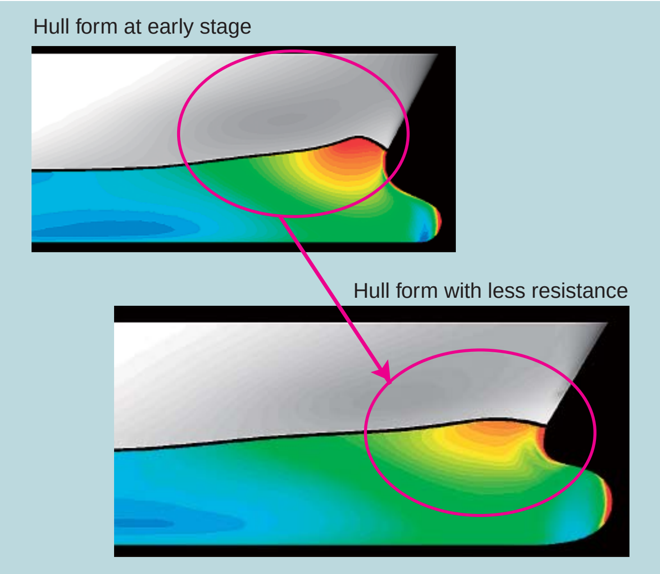


Fig. 2 Improving hull form for less resistance
CFD computation helps improve hull form to make waves in the fore region milder which in turn reduces wave resistance.

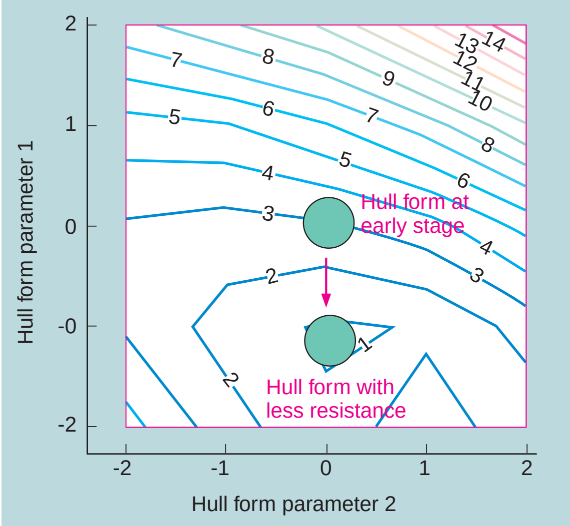


Fig. 3 Performance evaluation map
Results of computation with hull form parameters thoroughly modified are mapped to evaluate the direction of hull form improvements.

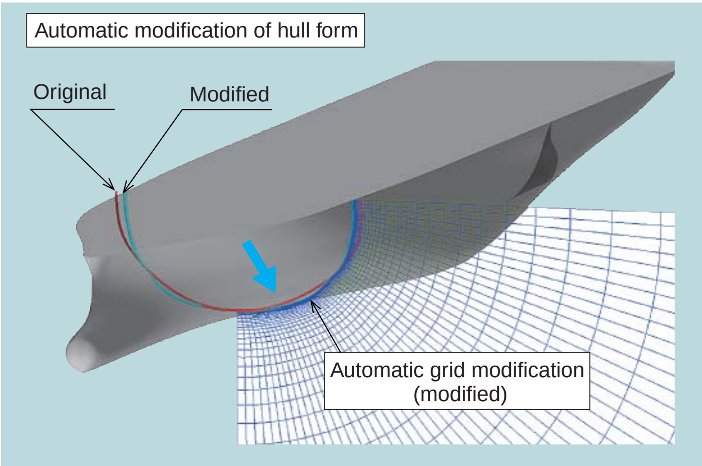


Fig. 4 Image of hull form automatic modification method
Computational grid around ship is modified following hull form modification.

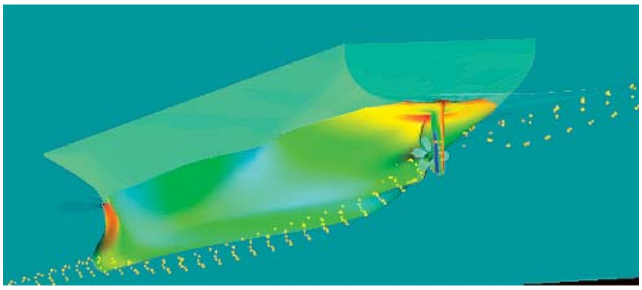


Fig. 5 Calculating flow distribution around ship with rudder in a state of self propulsion.
Computation shows ship with its propeller operating while thrust and resistance are in equilibrium, that is, in self-propelled condition.

Figure 6 shows an example where a hull form requiring low horsepower while considering the propulsion efficiency is determined. Each of the four hull forms A, B, C and D has a different stern profile. While hull form A generates the least resistance, hull form C requires the least horsepower when the consideration extends to the propulsion efficiency. This makes the problem complex because the hull form generating the least resistance is not necessarily the hull form requiring the least horsepower. Here, CFD accurately estimates these tendencies. Thus, in an earlier stage of design, a hull form requiring less horsepower can be designed while having CFD effectively evaluate the resistance performance and propulsion efficiency for many hull forms with a practically accurate level of estimation.

3. Trial application to sterns with devices of complex forms

3.1 Performance evaluation of CRP POD propulsion system

The multi-block grid technique enables CFD computation while accurately expressing the complex form of a stern. As an example, the following paragraphs describe how CFD is used to estimate the performance of a CRP POD propulsion system[3].

As shown in **Fig. 7**, a multi-block computational grid is created taking into account the hull and the POD main body. The computation is carried out using this grid with BFPs (body force panels) where the main propeller and the POD propeller operate in contra-rotation.

Figure 8 compares the thrust coefficients, K_T, and the torque coefficients, K_Q, of the main and POD propellers to those of the experimental values. As for the main propeller, the calculated results and the experimental data agree well with each other. Although the calculated value is smaller for the POD propeller, it is still sufficiently accurate for designing propellers of this ship if the value is corrected using the differences from the model test results. Thus, it is verified that with this method we can cope with the performance evaluation of ships equipped with a combined propulsion system like this.

3.2 Application to reaction fins

A reaction fin is a device installed ahead of a propeller to improve the propulsion efficiency by absorbing the rotating flow. As shown in **Fig. 9**, a grid correctly reflecting the form and the installation angle of the reaction fins is created by CFD computation.

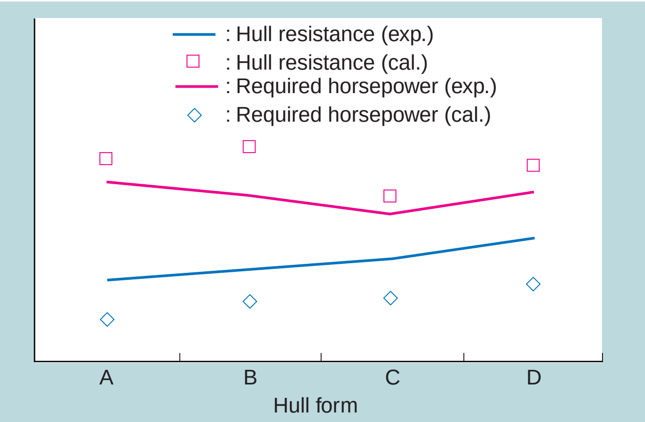


Fig. 6 Comparison of estimated results of required horsepower
Hull form A shows least resistance. Hull form C requires least horsepower. Computation accurately estimates these tendencies.

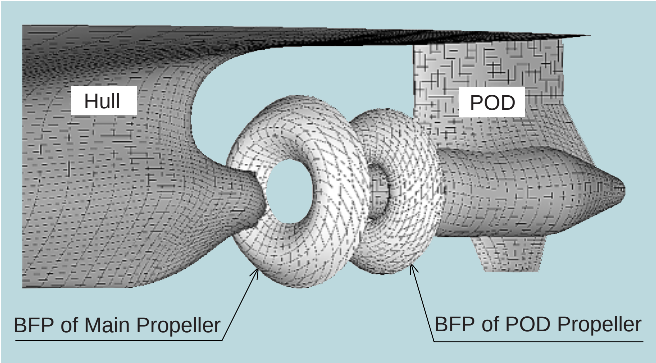


Fig. 7 CFD computation model for CRP POD propulsion system
Showing computation with ship in self-propelled condition while main and POD propellers are contra-rotating.

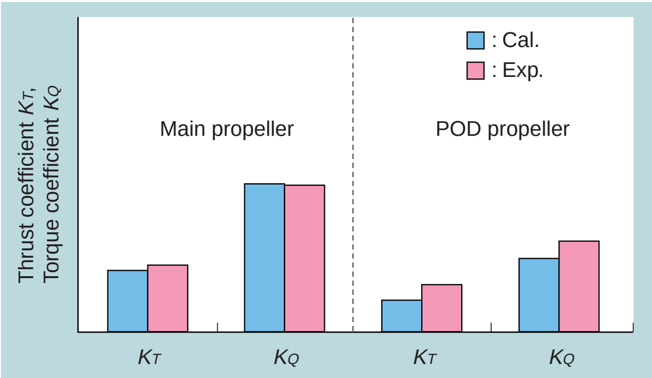


Fig. 8 Accuracy of estimating main and POD propeller thrust and torque coefficients
Calculated result accurately simulates experiment especially for main propeller.

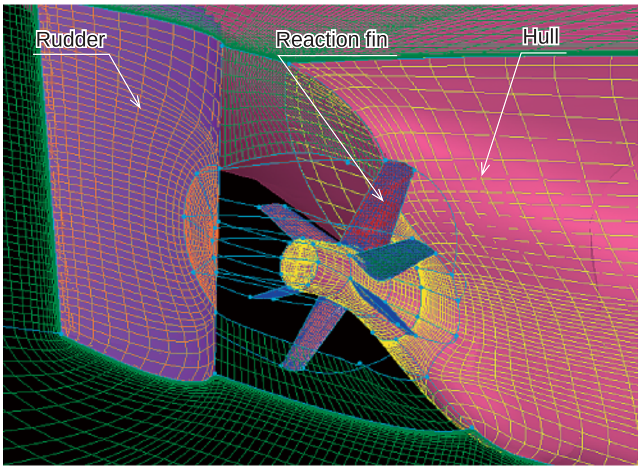
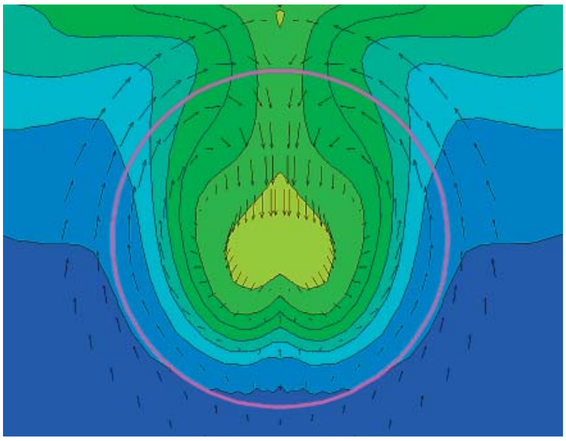
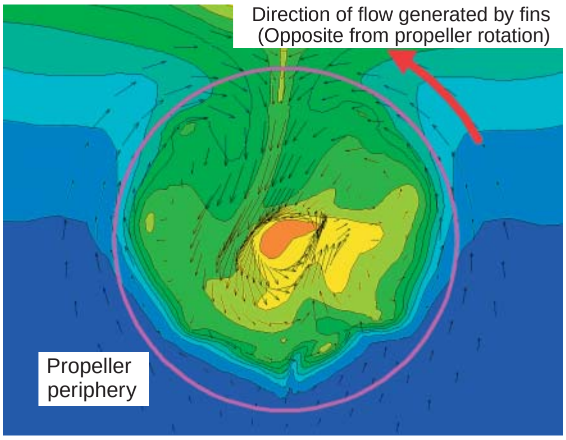


Fig. 9 Computational grid with reaction fins
Thorough use of multi-block grid technique enables accurate interpretation of fin forms and installation angles into models.



Without reaction fins



With reaction fins

Fig. 10 Estimating difference in wake distribution of flow towards propeller position with and without reaction fins
Reaction fins create rotating flows in direction opposite to the propeller rotation (clockwise).

Figure 10 shows the flow towards the propeller position with and without reaction fins. The fins create flows in the opposite direction to the propeller rotation. Furthermore, **Fig. 11** compares the horsepower required. The computation also indicates that installing reaction fins reduces the propulsive horsepower required. As suggested above, this method is also considered to be applicable for evaluating the performance improvements from devices having a complex form like this.

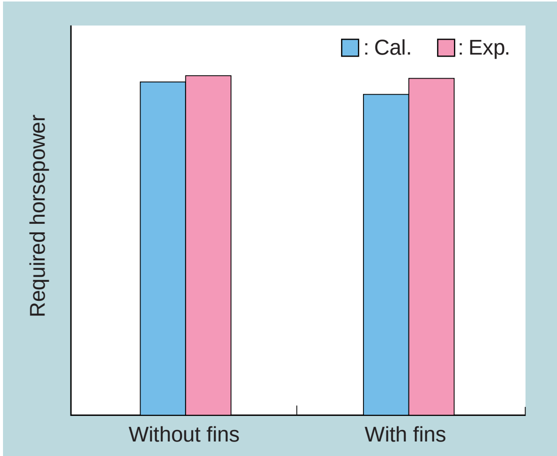


Fig. 11 Accuracy of estimating reduction in required horsepower with reaction fins
CFD can evaluate the tendency of reduction in horsepower required attributable to reaction fins.

4. Conclusion

As discussed above, MHI has developed a CFD-based design aid system that enables an effective and thorough search for hull forms that have excellent hull resistance and propulsive efficiency or, in other words, hull forms requiring less horsepower. Since this method is sufficiently accurate and efficient in computation when applied to hull form planning in an early stage of design, it is used for designing hull forms with excellent propulsive performance. Further, CFD has been applied extensively for examining the CRP POD propulsion system and for performance improvements using reaction fins. MHI is continuing to further improve its accuracy to enhance the practical use of computations that deal with complex stern forms.

References

(1) Takada, N., Computation of Ship Transom Stern Flow Using a Multiblock Grid Method, Journal of the Society of Naval Architects of Japan Vol. 190 (2001) p.13
(2) Takada, N. et al., Simulation of Free-Surface Viscous Flow around Practical Hull and Rudder with Propeller Effects, Journal of the Society of Naval Architects of Japan Vol. 192 (2002) p.63
(3) Oshima, A. et al., Hybrid CRP-POD Driven Fast ROPAX Ferry, Conference Proceedings, The Society of Naval Architects of Japan Vol. 5 (2005)


Makoto Nishigaki


Kei Sato


Satoru Ishikawa