VI International Conference on Computational Methods in Marine Engineering
MARINE 2015
F. Salvatore, R. Broglia and R. Muscari (Eds)

COMPARISON OF UNSTRUCTURED GRID AND OVERSET GRID APPROACHES FOR FLOW COMPUTATIONS AROUND A SHIP WITH AN ENERGY-SAVING DUCT

TAKANORI HINO*, **MASATOSHI HIROTA**†, **NOBUYUKI HIRATA**‡ **AND KUNIHIDE OHASHI**‡

*,†Yokohama National University
Yokohama, Kanagawa, 240-8501 JAPAN
e-mail: hino@ynu.ac.jp, web page: http://www.ynu.ac.jp/

‡ National Maritime Research Institute
Mitaka, Tokyo, 181-0004 JAPAN
e-mail: hirata@nmri.go.jp, k-ohashi@nmri.go.jp, web page: http://www.nmri.go.jp

Key words: Overset Grid Method, Unsturucture Grid Method, Ship Flows, Energy-Saving Devices

Abstract. Two approaches for CFD simulations for a complex geometry of a ship with an energy-saving duct are compared. One method is an unstructured grid method in which the single block grid can cover the whole geometry and the other is an overset grid method which uses multiple grid blocks for each component of the complex geometry. The computed results are compared with each other and with the experimental data for the resistance and the self-propulsion factors together with the flow fields. Based on these comparisons, two approaches are assessed from the practical point of view.

1 INTRODUCTION

In the CFD applications of a marine engineering field, flow simulations for complex geometries such as a ship with various appendages including energy saving devices are demanded more than ever. Efforts required for grid generations of such a complex geometry are increasing and become a heavy burden to overcome.

Unstructured grid methods such as [1] have been developed with the intention to reduce the difficulties associated with grid generations around complex geometries due to their flexibility of the grid connections and cell shapes. Many commercial CFD codes also utilize this approach. On the other hand, another approach, an overset grid method, is attracting attentions of researchers and practitioners in the marine field. An overset grid approach which utilizes a set of overlapped grid blocks encompassing a flow domain is considered to be another efficient way to carry out a CFD analysis for complex geometries. Use of overset grid methods in marine hydrodynamics is increasing partly due to the availability of the overset grid libraries/tool kits such as [2].

In the present paper, two approaches are compared for flow computations of a ship with an energy-saving duct. A ship hull form used is a bulk carrier called Japan Bulk Carrier (JBC) [5] which has been designed together with a duct and this will be used as one of the new test cases at the coming CFD Workshop Tokyo 2015 [3].

The values of computed resistance and self-propulsion factors are compared with each other and with the experimental data. Also flow field data are compared and the accuracy of two approaches is examined. Through these comparisons, the advantages and disadvantages of two approaches are discussed from a practical point of view.

2 NAVIER-STOKES SOLVERS

In the present study, two approaches, an overset grid method and an unstructured grid method, are utilized for handling a complex geometry case of a ship hull with an energy saving device. Therefore, two Navier-Stokes solvers which are developed for each approach are used.

The first solver is called *NAGISA*[6], which is being developed at National Maritime Research Institute (NMRI) in Japan together with the overset grid assembler based on the Spline function[4]. *NAGISA* is a solver for incompressible Navier-Stokes Equations with an overset grid capability using structured grid blocks. The second one is *SURF*[1] which is an unstructured-grid-based incompressible Navier-Stokes Solver developed also at NMRI.

Both solvers are similar to each other. Spatial discretization is based on the finite volume method and an artificial compressibility scheme is adopted for velocity-pressure coupling. The multigrid method is used for the convergence acceleration.

Various turbulence models including one-equation models, two-equation models and an algebraic stress model are implemented in both solvers. The propeller model for self-propulsion simulations which is also common between two solvers is a body-force model based on the simplified propeller model for an infinitely bladed propeller.

The detail of the computational methods for each solver can be found in the references[6, 1].

3 SHIP MODEL

The ship hull concerned in the present study is a capesized bulk carrier called Japan Bulk Carrier (JBC)[5] which has been designed together with its energy saving duct for the validation of the CFD predictions of ship flows with energy saving devices. This hull is adopted as one of the test cases in the coming CFD Workshop Tokyo 2015[3]. The propeller is a conventional five-bladed propeller with the AU section. The geometries of a main hull and a duct are shown in Fig. 1 and the principal particulars of the ship and the propeller are listed in Tables 1 and 2.

The duct has a circular shape with the diameter of 0.55 D_p (the propeller diameter) and its section form is NACA4420 with the chord length of 0.3 D_p.

The ship model with the scale ratio of 1/40 (*i.e.* L_{PP}=7.0 m) is constructed and tested in the towing tank of National Maritime Research Institute, Japan. Also, the flow field measurement is carried out using the SPIV (Stereo Particle Image Velocimetry) system. Note that the ship model is not equipped with a rudder, since a rudder may interfere with the laser sheet of the SPIV.

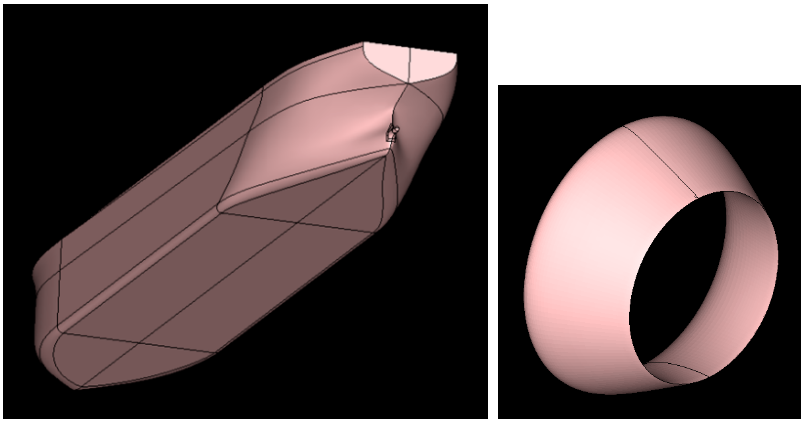


Figure 1: Geometry of a ship hull (left) and a duct (right).

Table 1: Principal particulars of a ship:JBC

L_{pp}[m]	L_{wl} [m]	B [m]	D [m]	d [m]	C_B
280.0	285.0	45.0	25.0	16.5	0.858

Table 2: Principal particulars of a propeller:MP 687

D_p[m]	Pitch Ratio	Boss Ratio	Expanded Area Ratio	Number of Blades	Blade Section
8.12	0.75	0.18	0.50	5	AU

4 COMPUTATIONAL CONDITIONS

All the computations are carried out with the model scale of the ship length L_{pp}=7.0m which is the same scale in the towing tank test. Reynolds number based on the ship length and the ship speed is set as 7.2484×10^6. The algebraic stress model is used as a turbulence model, since the stern flows are expected to have complex vortical structures.

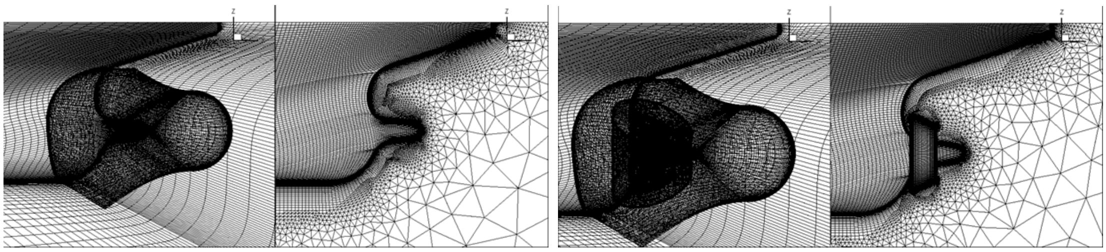
The free surface effects are ignored in the computations and a double model flow assumption is used, since the model tests are conducted at low Froude number Fr of 0.142.

Computational grids for each case are generated by *up_grid* system[4] for the overset grid approach and by *Pointwise* for the unstructured grid approach.

In case of the overset grid approach, total of three grid blocks are used, *i.e.* a main hull block of 2,150,400 cells, a stern tube block of 825,696 cells and a duct block of 752,640 cells. The total number of cells for a ship hull with a duct is 3,728,736 in both sides of a hull. In the case without a duct, simply a duct block is removed and the number of cells reduces to 2,976,096.

On the other hand, the unstructured grids are generated independently for a bare hull and a hull with a duct using hexahedra/tetrahedra hybrid grids. The number of cells are 4,421,292 cells for a bare hull and 4,546,226 for a hull with a duct. The grids near the stern are shown in Fig. 2.

The CPU time for the whole computations which include one towed simulation and one self propulsion simulation is one or two days in case of an overset grid method. In case of unstructured grid method, approximately ten days are required for the same computations. The overhead of an unstructured grid method appears to be large even though the total number of grid cells are larger than an overset grid method.



(a) (b) (c) (d)

Figure 2: Computational grids near the stern:(a) Overset without a duct, (b) Unstructured without a duct, (c) Overset with a duct, (d) Unstructured with a duct.

5 RESULTS AND DISCUSSIONS

5.1 Resistance and Self Propulsion Factors

First, computations are made for the towed condition in which propeller effects are not taken into account. Computed resistance components for cases with and without a duct is listed in Table 3. C_t is the total resistance coefficient normalized by $1/2\rho U^2 S$ where ρ is the water density, U is the ship speed and S is the wetted surface area. C_f and C_p are the frictional and the pressure resistance coefficients determined similarly. The abbreviations, OVS, UNS and EXP, are used to represent overset and unstructured approaches and experiment, respectively.

Also shown are the form factors $1 + K$ in the towing tank test and C_{FO} by the ITTC 1957 line.

The self propulsion simulations are performed following the towed computations. The propeller body force model is activated and the propeller rotation rate is determined in such a way that the self propulsion of a ship scale (the so-called ship point) is achieved.

Computed self propulsion factors, the thrust deduction $1-t$, the wake fraction $1-w_T$ and the relative rotational efficiency η_r, are listed in Table 4 together with the experimental values. Note that experimental values are based on the data at Fr= 0.142.

Table 3: Computed resistance coefficients

Cond.	Grid	C_t	C_f	C_p	$1+K$
w/o duct	OVS	4.103E-03	3.173E-03	9.302E-04	1.292
w duct	OVS	4.103E-03	3.165E-03	9.380E-04	1.292
w/o duct	UNS	4.403E-03	3.248E-03	1.155E-03	1.387
w duct	UNS	4.092E-03	3.036E-03	1.056E-03	1.289
w/o duct	EXP	—	C_{FO}=3.175E-03	—	1.314
w duct	EXP	—	C_{FO}=3.175E-03	—	1.305

Table 4: Computed self propulsion factors

Cond.	Grid	$1-t$	$1-w_T$	η_r
w/o duct	OVS	0.808	0.495	1.011
w duct	OVS	0.813	0.418	1.004
w/o duct	UNS	0.676	0.487	0.993
w duct	UNS	0.805	0.449	1.007
w/o duct	EXP	0.804	0.553	1.015
w duct	EXP	0.810	0.481	1.009

The total resistance coefficients are, in general, well predicted for the most cases except the unstructured grid computation without a duct which yields the considerably larger value than other cases. The experimental results of the form factor $1+K$ show a slight resistance decrease (less than 0.1%) due to a duct. In the overset cases, the total resistance is almost the same between with and without a duct and it appears to be reasonable since the difference of measured resistance is so small. The larger resistance predicted with the unstructured grid for a bare hull may be attributed to the quality of the computational grid. However, the unstructured grid of the same type for the case with a duct shows reasonable resistance value. Further study is required on this issue.

Variations of self propulsion factors due to the duct are also reasonable to some extent. While the thrust deduction coefficients are well predicted by the overset grid method, the predicted value of $1-t$ for the case without a duct by the unstructured grid method is quite low. This may be related to the higher prediction of the towed resistance in the same case. The tendency of the wake fractions is also predicted well, although the values of $1-w_T$ are under-predicted.

5.2 Velocity Distributions

The computed velocity distributions are compared with the experimental data[5] measured by the SPIV system.

Fig. 3 is the comparison of the axial velocity distributions in a propeller plane in case of the towed condition without a duct. The SPIV data and two computed results by the overset grid method and by the unstructured grid method are shown.

Since JBC has a full hull form, the so-called hook shape of the wake contour can be observed in all the plots. The algebraic stress turbulence model plays a certain role in these flow fields. In case of the unstructured grid method, the contour lines are distorted particularly in the outer part of the wake. This is due to the poor grid distribution as described below. It is observed that the wake contour of the unstructured grid method is asymmetric with respect to the center plane. The reason for this is unknown at present.

Fig. 4 is the same comparison as Fig.3 for the hull with a duct. The duct significantly affect the flow field and the U shaped low speed regions are present as the wake of the duct. Both numerical results reproduce this tendency quite well, although the distorted and asymmetric wake contours are observed again in the unstructured grid case.

Figs. 5 and 6 are the similar comparisons of the velocity fields in the section behind the propeller. Fig. 5 is for the bare hull and Fig.6 is for the hull with a duct. As described above, the propeller effect is taken into account by means of a body force method.

In Fig. 5, both computed wake distributions predict the acceleration of velocity in the propeller disc well compared with the experimental data. The difference of the detailed distributions may come from the difference of the grid density.

Comparison of Fig. 6 with Fig. 5 reveals that the acceleration due to the propeller is smaller in the case with the duct than without the duct in both computations. This corresponds to the energy-saving effects of the duct.

Fig.7 shows the comparison of the grid density among the cases considered. In case of the overset grid, the grid distributions are smooth and the sufficient number of grid points are placed in and around the propeller disc. On the other hand, in the unstructured grids computational cells are not smoothly distributed and the grids are rather coarse in the outer region, although the fine grids can be observed near the center of the propeller disc.

These differences of the grid quality between the overset grid and the unstructured grid approaches affects the predictions of the velocity fields as seen in the previous plots.

Although the unstructured grid generation seems relatively easy compared with the structured grid, it is rather difficult to control or improve the quality of the unstructured grid in the grid generation process. On the other hand, an overset grid method needs an extra care for the arrangement of the grid blocks and the overset grid assembly process sometimes fails to generate the inter-block connectivity data when the grid blocks are improperly overlapped.

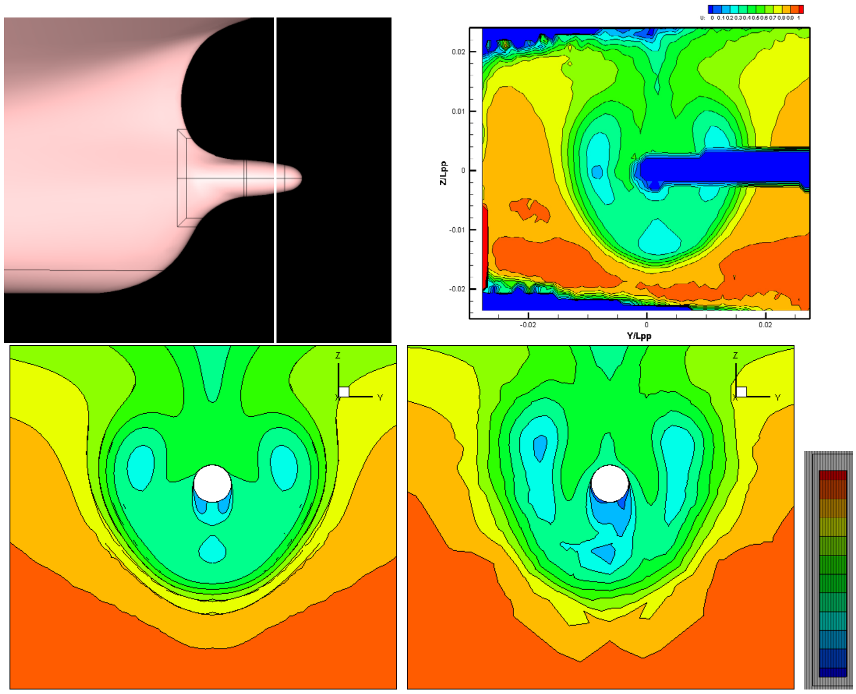


Figure 3: Axial velocity distributions in a propeller plane without a duct and without a propeller. Top right:EXP, bottom left:OVS bottom right: UNS.

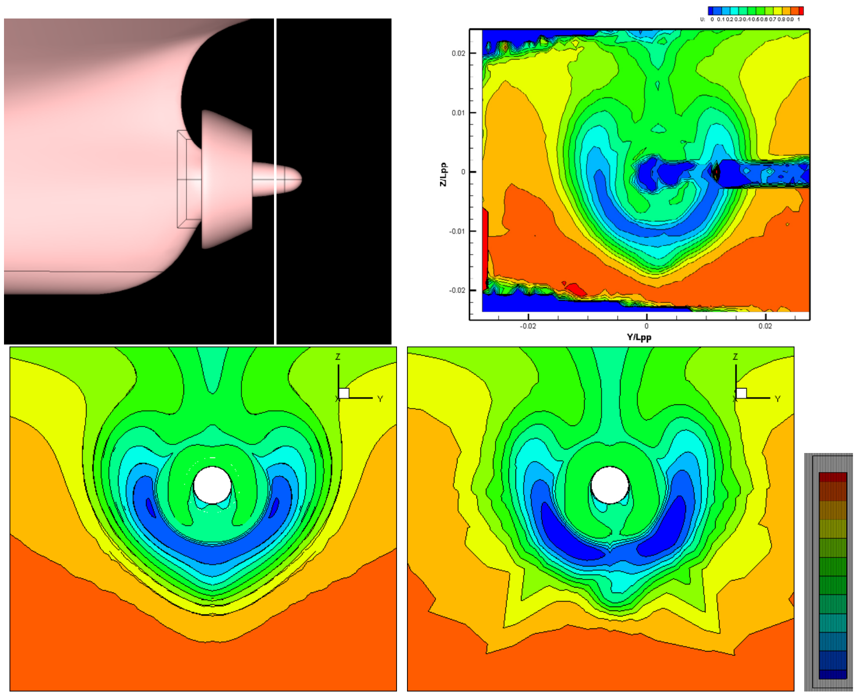


Figure 4 Axial velocity distributions in a propeller plane with a duct and without a propeller. Top right:EXP, bottom left:OVS bottom right: UNS.

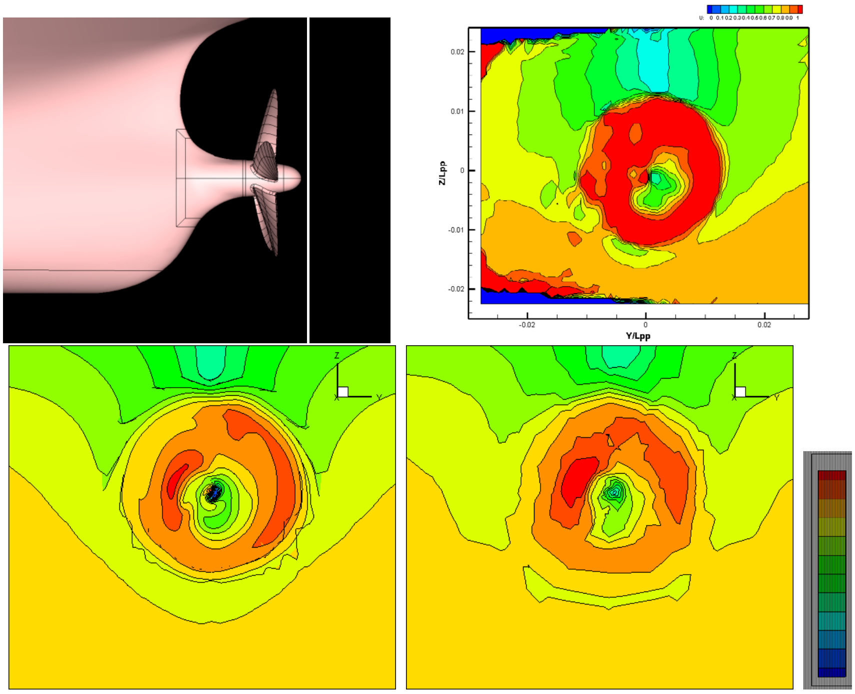


Figure 5: Axial velocity distributions in a propeller plane without a duct and with a propeller. Top right:EXP, bottom left:OVS bottom right: UNS.

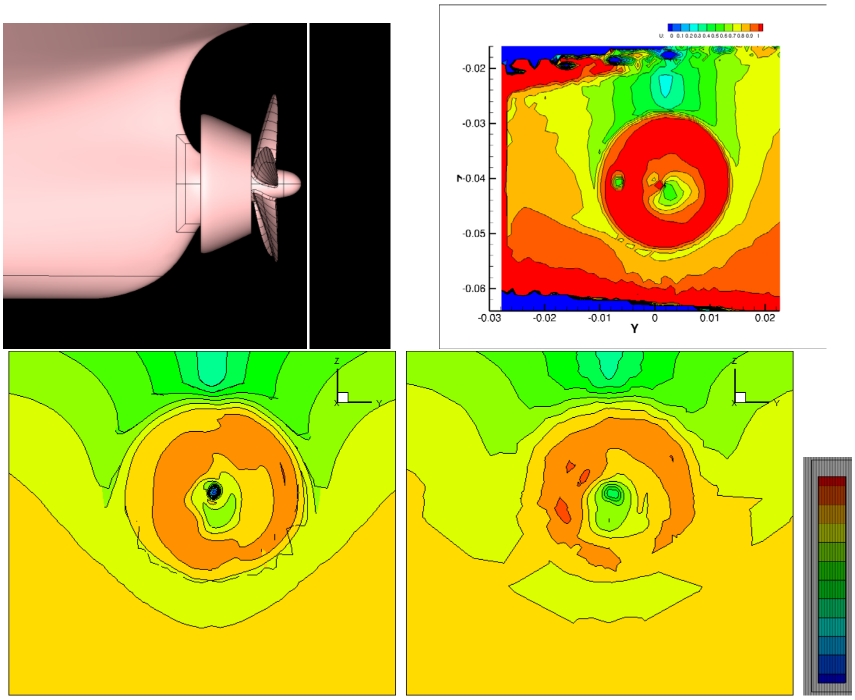


Figure 6: Axial velocity distributions in a propeller plane with a duct and with a propeller. Top right:EXP, bottom left:OVS bottom right: UNS.

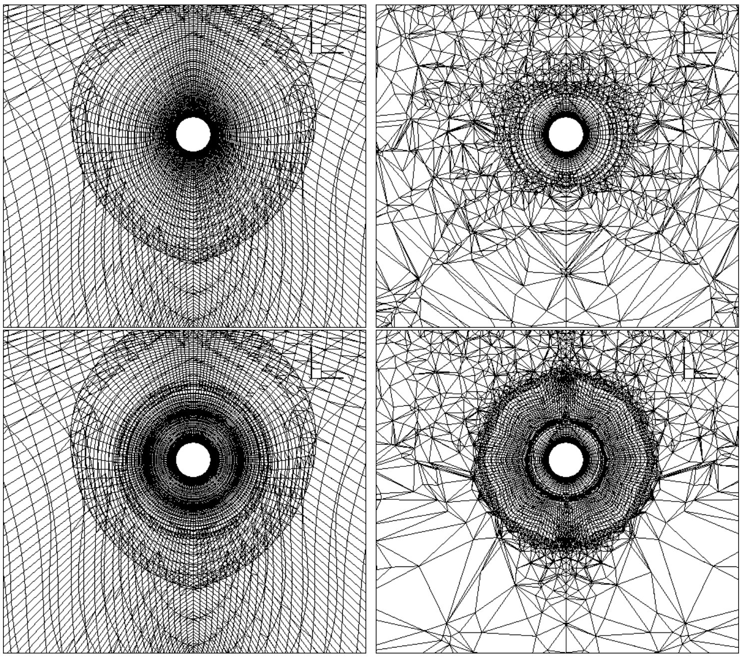


Figure 7: Comparison of computational grids in a propeller plane. Top left: OVS without a duct, top right: UNS without a duct bottom left:OVS with a duct and bottom right: UNS with a duct.

6 CONCLUSIONS

Conclusions of the present study can be summarized as follows:

- The resistance and self propulsion factors can be predicted well by either an overset grid or unstructured grid method, provided that the grid density is properly controlled in the region where the velocity variations are large.

- The grid resolutions and the density significantly affect the flow field predictions. An overset grid method has some advantages in generating grids with better quality.

- Although the grid generation of unstructured grid seems easy, the quality control is difficult. On the other hand, an overset grid generation and arrangement based on the structured grid blocks needs some expertise.

- In terms of CPU time, an overset grid method appears to have some advantages over an unstructured grid method, although this is highly case-dependent.

ACKNOWLEDGMENT

This research was carried out jointly by Japan Marine United Corporation, Kawasaki Heavy Industries, Ltd. National Maritime Research Institute, Mitsubishi Heavy Industries, Ltd., Mitsui Engineering & Shipbuilding, Co. Ltd., Osaka University, Ship Building

Research Centre of Japan Sumitomo Heavy Industries Marine & Engineering, Co. Ltd., Yokohama National University, and ClassNk as part of the ClassNK Joint R & D for Industry Program.

This research is also supported in part by JSPS KAKENHI Grant Number 25289315.

REFERENCES

[1] Hino, T. A 3D Unstructured Grid Method for Incompressible Viscous Flows, *J. of the Soc. Naval Archit. Japan* (1997) **182**:9–15.

[2] Noack, R.W. SUGGAR: A General Capability for Moving Body Overset Grid Assembly, *17th AIAA Computational Fluid Dynamic Conference*, Toronto, Ontario, Canada. Paper 2005-5117, (2005).

[3] Web site of CFD Workshop Tokyo 2015, http://www.t2015.nmri.go.jp

[4] Kobayashi, H. and Kodama, Y. Developing Spline Based Overset Grid Assembling Approach and Application to Unsteady Flow around a Moving Body, *VI International Conference on Computational Methods in Marine Engineering* (MARINE 2015), Rome, Italy, (2015)

[5] Hirata, N., Ooba, H., Hino, T. and Kanai, T. Detailed Flow Measurement around a Japan Bulk Carrier (JBC) with an Energy-Saving Circular Duct for Validation of CFD Codes, *VI International Conference on Computational Methods in Marine Engineering* (MARINE 2015), Rome, Italy, (2015)

[6] Ohashi, K., Hino, T., Hirata, N. and Kobayashi, H. Development of NS Solver with a Structured Overset Grid Method, (Japanese), *Proc. 28th CFD Symposium*, Tokyo, Japan, (2014).