NUMERICAL PREDICTION OF FLOW FIELD AROUND A SHIP DRIFTING WITH

NUMERICAL PREDICTION OF FLOW FIELD AROUND A SHIP DRIFTING WITH VARIOUS ANGLES

Osman Md. Amin

Ph.D. Candidate

Graduate School of Osaka University

Achieved B.Sc. (Eng.) from Bangladesh University of Engineering and Technology (BUET). After finishing M.Sc. from the Ship Design Laboratory of Osaka University, currently is affiliated to the same laboratory as a Ph.D. candidate (Last year). Research work is related to the implementation of computational fluid dynamics in the prediction of manoeuvring characteristics of ships.

Kazuhiko Hasegawa

Professor

Osaka University

PhD at Osaka University (OU) in 1982. Present affiliation after 7 years in Hiroshima University. Visiting researcher/professor-ship at University of Twente, Strathclyde University, Pusan National University and Ecole Centrale de Nantes. Contributed as Deputy Editor of JMST, a member of Technical Committee on Marine Application, IFAC, Vice President of the Japan Society of Naval Architecture and Ocean Engineering (JASNAOE) etc.

Osman Md. Amin (Graduate School of Osaka University, Japan)

Kajuhiko Hasegawa (Graduate School of Osaka University, Japan)

Abstract

Theoretical manoeuvrability prediction has been one of important topics of research for several decades. The capability to take account of viscous diffusion of flow into calculation, which has a large influence in yaw motion of ships, made Reynolds Averaged Navier-Stokes (RANS) simulation very much popular in calculating detailed pressure and shear force distributions around drifting ships. The objective of this research is to simulate the wake distribution at the propeller plane of a manoeuvring ship in steady motion using pure unstructured grid. An in-house code using unstructured grid based RANS solver has been developed to investigate the behavior of ships in manoeuvring motion. For unstructured grid the oscillations in result caused by the adoption of second order differencing scheme have been minimized through the implementation of a slope limiter algorithm in discretizing the diffusion term of the Navier-Stokes equation. Two different turbulence models have been implemented to observe the influence of those models in simulating vortex shed in the wake which is crucial in estimating nominal wake at the propeller plane. Validation of the code was carried out by comparing experimental and computation data on force and moment coefficients induced upon a steady drifting tanker ship.

1. INTRODUCTION

Manoeuvrability of a ship largely depends upon the forces and moment acting upon the hull and rudder. Wake behind the ship determines the inflow velocity towards rudder from propeller, which in turn affects the normal force induced upon the rudder. This issue has been investigated by many researchers for decades through the usage of either experiment or numerical simulation. In effect, several mathematical models[1,2] have gotten developed to take account of these physical phenomena in simulating the manoeuvring behavior of ships. Numerical analysis has been another field which took over this challenge to measure the manoeuvrability indices as defined by the mathematical models in evaluating the manoeuvrability characteristics of ships. As potential theory still lacks the versatility to consider the viscosity and flow separation effects in the calculation, Reynold's Averaged Navier Stokes (RANS) simulation has gained its popularity in calculating detailed pressure and shear force distributions around a manoeuvring ship.

Since several of the institutes[3,4] are working for last couple of decades on the development of robust RANS code to accurately predict the forces and moments acting on the ship, it has been a concern whether the existing codes are really practical enough to be a supplementary tool along with experiment. Several of the factors influence the reliability of RANS codes. Different turbulence models, various types of grid topology, no. of node/cells in the domain of grid, numerical/modeling errors, computational cost etc. are the various aspects which still prohibit any of the RANS code to be versatile enough to be applicable for the manoeuvrability prediction of all types of ships. Usage of parallel computing has increased the ability to overcome some of the limitations as mentioned before, which in an effect rendered researchers the option to predict wake behind ship along with the forces and moment with considerable accuracy. Although still the grid dependency of the results are still at times problematic for most of the robust RANS codes.

In this context, the objective of the authors' is to develop a RANS code suitable to be applicable to any types of grids (structured/unstructured) and carry out the manoeuvrability prediction using that robust code. This paper basically is concerned about the applicability of different turbulence models on pure unstructured grid (comprised of tetrahedral/prismatic elements) in the evaluation of wake distribution at the propeller plane of a tanker. Empiricism involved with the turbulence models along with dependencies of them on the grid distribution around the propeller plane had proved to be a great challenge for carrying out RANS simulations, where most of the reasonable results are being achieved through the usage of very fine and orderly distributed grids[5].

Ease of generation of unstructured grid instigated earlier researchers to go for RANS simulation using those grids, where application of large number of cells in the grid (around 9-10 Million) is considered to be reliable one in simulating the flow field, as has been addressed by Burg et al[6]. Similar conclusions were drawn by Fathi et al[7], suggesting inherent lack of predictability of forces and moment acting on a ship hull due to the usage of unstructured grid. These analyses inspired the authors to go for developing a RANS code on the basis of unstructured topology of the grid that comprises the space around a hull.

In this paper, the Authors' approach is to establish an unstructured grid based RANS solver to predict the hull forces and moment acting on a drifting tanker, with nominal accuracy in calculating the wake behind the ship. A double hull model has been implemented in the manoeuvring simulation instead of considering the free surface flow.

2. NUMERICAL FORMUALTION OF COMPUTATIONAL METHOD

The governing equations for RANS simulation of incompressible flow around a body can be expressed by the following two equations in differential form, which as a group are called Reynolds Averaged Navier-Stokes equations:

Continuity equation:

$$\frac{\partial u_j}{\partial x_j} = 0 \tag{1}$$

Momentum equation:

$$\rho\frac{\partial u_i}{\partial t} + \rho\frac{\partial}{\partial x_j}\left(u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} - \rho\overline{u_i' u_j'}\right) \tag{2}$$

where, stress tensor $\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$ (3)

In these equations, u_j represents the velocity components, x_j are the Cartesian coordinates, p is the pressure, μ is the dynamic viscosity and $\rho\overline{u_i' u_j'}$ is termed as the Reynolds stress. The quantities without bar are considered to have mean values.

The closure of the Reynolds stress is achieved by considering it to be expressed as the following equation in most of the turbulence models,

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\rho\delta_{ij}k \tag{4}$$

Here, k is the turbulent kinetic energy and μ_t is the eddy viscosity.

Finite volume method for collocated arrangement of unstructured grid has been adopted in this analysis to discretize the convection and diffusion terms of the Navier-Stokes equation. The whole domain has been discretized into a substantial number of control volumes (CV). For spatial discretization the steady state continuity and momentum equations are considered to be as,

$$\int_S \rho v \cdot \boldsymbol{n} dS = 0 \tag{5}$$

$$\int_S \rho \boldsymbol{v}\boldsymbol{v} \cdot \boldsymbol{n} dS = \int_S \mu\, grad\boldsymbol{v} \cdot \boldsymbol{n} dS + \int_\Omega \rho b d\Omega \tag{6}$$

These integral equations are applied to each CV and the integral quantities are evaluated at different points within the elemental volumes. The variable values are calculated at those points through linear interpolation of nodal values, except for the CV centers. No overlapping among the control volumes is assured by defining each cell face to belong to both cells to which it is common.

Gamma differencing scheme[8], which is a local blending of first order upwind difference scheme and second order central difference scheme, has been used to satisfy the boundedness and second order accuracy in discretizing the convection term of Navier-Stokes equation. For second order differencing scheme the appearance of oscillations in solution process can be limited by the application of least square scheme as proposed by Michalak et al[9]. This scheme consists of finding a weight factor for each control volume, which will limit the gradient for the linear algebraic equation solver.

Three time level method[10], which is an implicit time advancing scheme, has been implemented for the unsteady flow simulation. This scheme is more suitable than Crank-Nicolson schemes in a sense of producing less oscillatory solutions when the time steps are relatively small.

The pressure field around the ship has been measured through the solution of Pressure-Poisson equation using PISO[11] (pressure implicit with splitting of operators) algorithm.

An asymmetric version of the Bi-conjugate Gradient method[12] along with ILUT preconditioning[13] has been used to solve the set of algebraic equations.

Most of the turbulence models for RANS simulation are based on the eddy viscosity approximation of Reynolds stress term[14]. Two different two equation turbulence models have been used to evaluate the wake distribution at the propeller plane in this analyses. To take account of the effect of viscous sub-layer of the boundary layer in calculation, the turbulent quantities near the wall have been calculated using the wall functions[15]. This application reduces the computational cost which otherwise would have incurred upon through the use of very fine grid near the wall.

Same differencing scheme, as has been used for discretizing convective and diffusive terms of Navier-Stokes equation, is used to discretize both turbulence kinetic energy and energy dissipation equations.

3. SIMULATING CONDITIONS FOR SUBJECT SHIP

As a test case, the authors considered to simulate steady drifting motion of a tanker called KVLCC2M (Fig.1) in unbounded fluid for which the experimental data are available in publications[16]. The specifications of the model for which the data are available are given in Table 1.

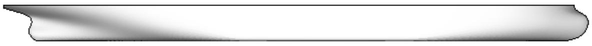


Fig. 1 KVLCC2M hull

Table 1: Specifications of the ship and model

	Principle Particulars		
Item	Symbol	Unit	Value
Length between perpendiculars	Lpp	m	4.97
Breadth(molded)	B	m	0.9008
Draft(molded)	D	m	0.3231
Wetted surface area without appendages	S_w	m^2	6.5597
Centre of Buoyancy from midship (+forward)	l_{cb}	m	3.50
Block coefficient	C_B	-	0.8098

3.1 Computational Domain and Grid Generation

The computational domain is taken to be of the shape of a basin (Fig. 2), where the coordinate system is considered to be Cartesian in nature. The grid topology is taken to be unstructured in nature. Octree algorithm has been used to generate unstructured grids for the whole domain by a non-commercial software. A refinement box has been included to the domain to discretize the grids finely around the hull (Fig. 3). The grids near the surface of the hull have been discretized into 3D prism cells to make the normal of the grid faces perpendicular to the surface.

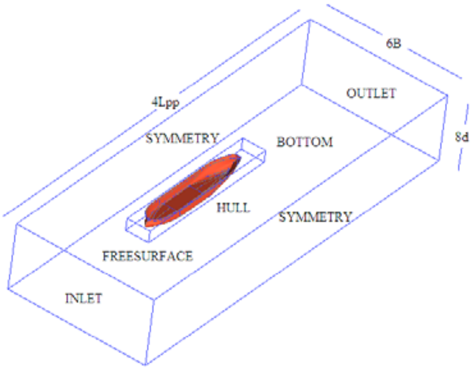


Fig. 2 The domain of grid

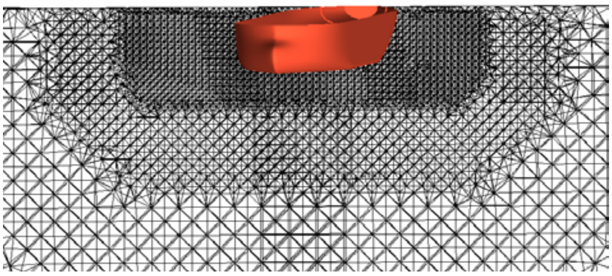


Fig. 3 Refinement of the grid around the hull

3.2 Boundary Conditions

To implement the no-slip condition on the impermeable wall, the normal viscous forces are considered to be zero. The shear stress is calculated in case of finite volume method through the use of velocity gradients parallel to the wall. For k-ε model all the turbulence quantities except, the energy dissipation term, ε are considered zero at the wall. For k-ω model the specific dissipation rate is expressed in terms of a singular solution to be calculated at the first cell center from the wall[14]. On the basis of the 'law of the wall' a wall function approach in the determination of energy dissipation near the wall has been utilized, where dissipation and turbulence production terms in the turbulence model have specific expression to be calculated at the first grid near the wall.

Symmetry conditions are applied to the side walls, where the shear stress is considered to be zero while the normal stress is not. The normal stress is calculated from the velocity gradient normal to the plane. At the inlet the velocity components correspond to the Reynolds number used during the experiments. At downstream of the ship the flow is calculated through second order linear extrapolation along the grid lines from the interior to the outlet. The bottom is considered to have no vertical component of velocity. The free surface modeling hasn't been considered in our simulation. So, instead of the free surface boundary condition a mirror image has been applied. In this case the velocity gradients normal to the surface has been termed as zero along with the normal velocity.

Table 2. Comparison between computation and experiment data on force and moment coefficients

	Computations (k-ε turbulence model)			Computations (k-ω turbulence model)			Experiments by NMRI		
Beta $(^0)$	C_X	C_Y	C_N	C_X	C_Y	C_N	C_X	C_Y	C_N
0	-0.0167	-0.0001	-0.0001	-0.0164	-0.0003	-0.0002	-0.0176	0	0
3	-0.0168	0.0102	0.0056	-0.0166	0.01	0.0058	-0.0178	0.0126	0.0061
6	-0.0166	0.0248	0.0115	-0.0164	0.0251	0.0108	-0.0177	0.0256	0.0139
12	-0.0169	0.065	0.024	-0.0167	0.06	0.0229	-0.0175	0.0708	0.0254

4. STEADY MANOEUVRING SIMULATION

Several steady drifting cases have been simulated to validate the predictability of the developed code. The reference axes is defined with origin at mid-ship, positive x-axis directed forward, positive y-axis towards starboard and the z-axis directed downward. The non-dimensional drag ($C_x = F_x / (0.5\rho U^2 L_{pp} d)$), lateral force ($C_y = F_y / (0.5\rho U^2 L_{pp} d)$) and yaw moment ($C_N = M_z / (0.5\rho U^2 L_{pp} d)$) for different drifting cases are being compared in Table 2. The simulating conditions for different drifting cases are given in Table 3.

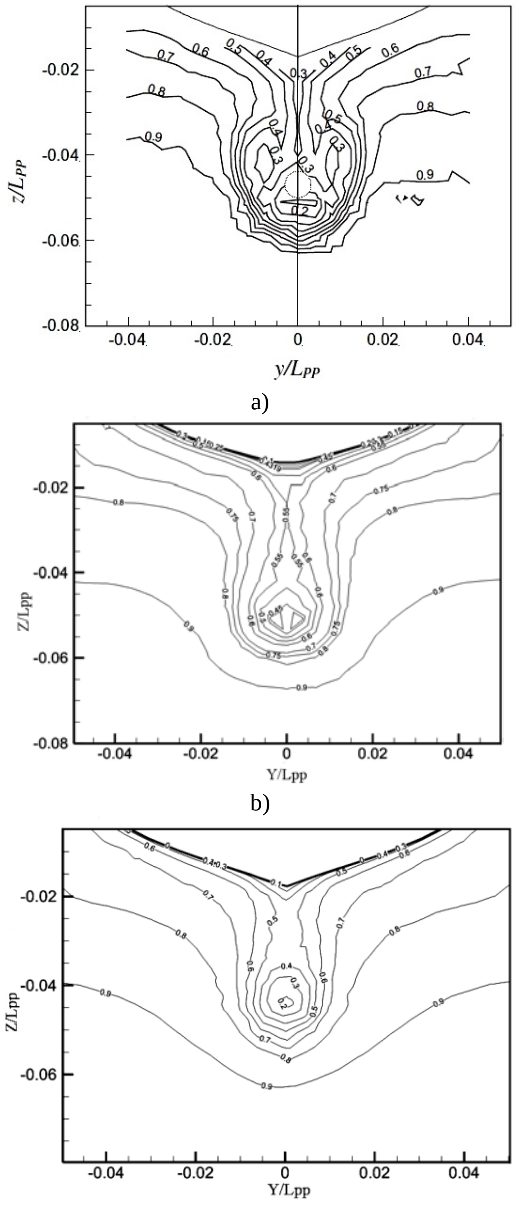
Table 3. Simulating condition for different test cases

Drift angle, (^{0})	Reynolds No.	Mesh Size(elements)
0	4.011e6	862737
3	3.945e6	967467
6	3.967e6	1467409
12	4.00e6	1677043

Usage of unstructured grid has an inherent disadvantage in the application of differencing schemes, which largely influences the outcome of the simulation. And since the turbulence models are also very much sensitive to the grid distribution, the developed code was run at first for $\beta = 0^0$ with very low no. of unstructured grids to verify it's effectiveness in simulating the wake. Fig. 4 shows the axial velocity distribution at the propeller plane for $\beta = 0^0$, where experiment data shows pronounced vortex shed from both sides of the centre plane. The simulated velocity field using k-ε turbulence model although predicts a hook shaped vortex generated at the centre of the plane but the production of turbulence energy wasn't sufficient enough to let the vortex grow out on both sides. On the other hand the application of k-ω model although predicts better velocity distribution at the centre of the plane but completely looses it ability to generated any form of vortices.

These results although didn't prohibit the prediction of forces and moment coefficients with considerable accuracy (Table 2), where the computed data shows an average error of less than 8% in value as compared to the experimental data. This is probably due to the fact that the boundary layer entirely lies within the fine prism layers (fig. 5) distributed around the wall, whereas the grids beyond boundary layers significantly varies in aspect ratio and orientation which in effect didn't hinder the proper calculation of friction velocity at the wall.

As the no. of cells in the grid increases from case to case the wake distribution gets much more elaborate and starts to replicate the actual flow pattern as can be seen from Fig.6. For $\beta = 12^0$ the average distribution of axial velocity resembles that of the experiment data. Both of the turbulence models seem to predict very much similar pattern, whereas the bilge vortex shed at the



a)

b)

c)

Fig. 4 Axial Velocity Field contours in propeller plane for $\beta = 0^0$ (experiment: a, simulated: b(k-ε model), simulated: c(k-ω model)

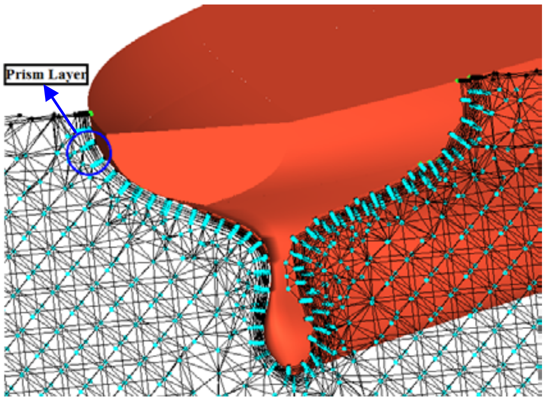
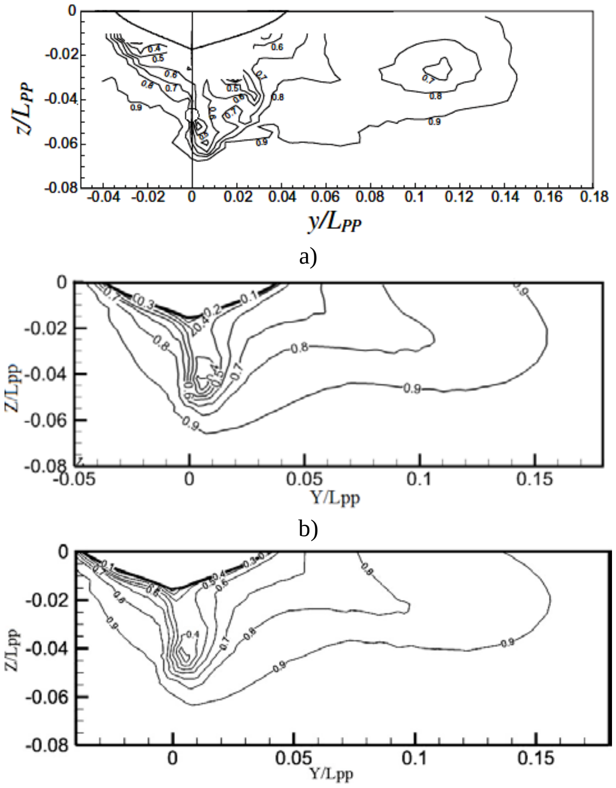


Fig. 5 Prism layer and node distribution around hull (nodes are represented by blue dots)

starboard side doesn't get simulated by none of the turbulence models.



a)

b)

c)

Fig. 6 Axial Velocity Field contours in propeller plane for $\beta = 12^0$ (experiment: a, simulated: b(k-ε model), simulated: c(k-ω model)

Though Gamma differencing scheme takes account of finding the directionality in discretizing the integral equations, the large cluster of cells (Fig.5) originated from each of the nodes in the grid limits the applicability of this blended differencing scheme.

The pressure distribution as measured using the in-house code manifests reasonable resemblance to the experiment data as can be seen from Fig. 7, where for $\beta = 12^0$ the distribution of surface pressure coefficient ($C_P = (P / 0.5 * \rho * U^2)$) has been plotted for the bottom of the ship along with the computation data. From qualitative as well as quantitative point of view the pressure distribution along the ship hull can be predicted with considerable accuracy using pure unstructured grid. The pattern of simulated distribution shows a bit higher estimation of negative pressure at stern of the hull, whereas both the turbulence models predict almost similar pattern of surface pressure distribution.

5. CONCLUSIONS

The development of a robust RANS solver largely depends upon the grid topology, where unstructured grid provides a significant amount of challenge in calculating accurately the flow field at the wake. On this premise, the flow solver that has been developed in this research work seem to verify the versatility of the code in evaluating motion behavior of ships. The conclusions which can be drawn from the above discussions are:

1) Despite the involvement of non-directionality in the

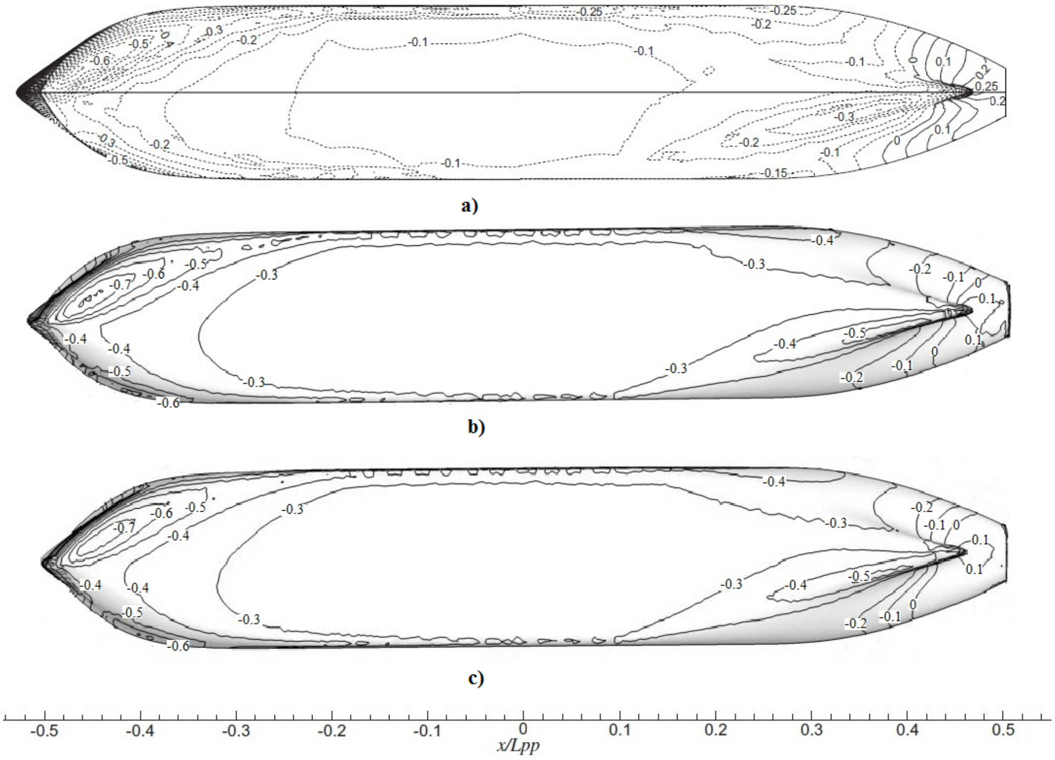


Fig. 7 Distribution of surface pressure coefficient for $\beta = 12^0$ (for bottom of hull); a) experiment, b) simulation (k-ω model), c) simulation (k-ε model)

discretization process of unstructured grid, the developed flow solver did provide a second order accurate result for flow simulation around a tanker.

2) The reliability of the code has been verified from the reasonable estimation ability of the forces and moment acting on the steady drifting ship.

3) Both the turbulence models used in this analysis has showed that the grid orientation around the propeller plane hugely influences the flow field estimation.

4) Although the average distribution of nominal wake can be estimated reasonably well by the implication of unstructured grid, usage of structured/unstructured hexagonal girds would be implemented later on for the proper simulation of shed vortices behind the ship.

6. REFERENCES

1) Abkowitz, M.A.: Lectures on ship hydrodynamics - steering and maneuverability, Hydro-Org Aerodynamisk Laboratorium, report Hy-5, Lyngby, Denmark, 1964, pp.113.
2) Ogawa, A. and Kasai, H.,: On the mathematical model of manoeuvring motion of ships, International Shipbuilding Progress, Vol.25, Issue No. 292, 1978.
3) Paterson, E.G., Wilson, R.V., and Stern, F.: General-purpose parallel unsteady RANS ship hydrodynamics code: CFDSHIP-IOWA, The University of Iowa, Iowa Institute of Hydraulic Research, IIHR Report No 432, 2003.
4) M. Hoekstra and L. Eca,: PARNASSOS: an efficient method for ship stern flow calculation, Third Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, Japan, pp.331-357.
5) Hino, T.: Navier-Stokes computations of ship flow on unstructured grids, Proceedings of the 22nd Symposium on Naval Hydrodynamics, 1998.
6) Burg, C.O.E., Sreenivas, K., Hyams, D.G. and Mitchell, B.: Unstructured nonlinear free surface simulations for the fully appended DTMB model 5415 series hull including rotating propulsors, 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan, 2002, pp.192-210.
7) Fathi, F., Kaij, C., and Koop, A.: Predicting loads on a large carrier with CFD, Proceedings of the 29th International Conference on Ocean, Offshore and Arctic Engineering, Shanghai, China, 2010.
8) Jasak, H., Weller, H.G. and Gossman, A.D.: High resolution NVD differencing scheme for arbitrarily unstructured meshes, International Journal of Numerical Methods in Fluids, Vol.31, 1999, pp.431-449.
9) Michalak, K. and Gooch, C.O.: Limiters for unstructured high-order accurate solutions of the Euler equations, AIAA, 46th Aerospace Sciences Meeting, 2008.
10) Shvedov, A.S.: A three-time-level explicit difference scheme of the second order of accuracy for parabolic equations, Mathematical Notes, Vol.60, No.5, 1996, pp.562-568.
11) Issa, R.I. : Solution of implicitly discretized fluid flow equations by operator-splitting, Journal of computational physics, Vol.62, 1986, pp.40-65.
12) Vorst, V.D. and Sonneveld, P.: CGSTAB, a more smoothly converging variant of CGS, Technological Report 90-50, Delft University of Technology.
13) Chapman, A., Saad, Y., and Wington, L.: High order ILU preconditioners for CFD problems, International Journal for numerical methods in fluids, vol.33, pp. 767-788, 2000.
14) Wilcox, D.C.,: Turbulence modeling for CFD, DCW industries Inc., 2006.
15) Launder, B.E., and Spalding, D.B.: The numerical computation of turbulent flows, Computer methods in applied mechanics and engineering, vol.3, pp.269-289, 1974.
16) K. Kume, K., Hasegawa, J., Tsukada, Y., Fujisawa, J., Fukasawa, R., and Hinatsu, M.: Measurements of hydrodynamic forces, surface pressure, and wake for obliquely towed tanker model and uncertainty analysis for CFD validation, Journal of Marine Science and Technology, 2006, Vol.11, pp.65-75.