

Journal of Maritime Research, Vol. V. No. 3, pp. 67-78, 2008 Copyright © 2008. SEECMAR

Printed in Santander (Spain). All rights reserved ISSN: 1697-4840

NUMERICAL STUDY OF THE FLOW FIELD AROUND A SHIP HULL INCLUDING PROPELLER EFFECTS

A.M. Tocu¹ and M. Amoraritei²

Received 23 June 2008; received in revised form 7 July 2008; accepted 3 December 2008

ABSTRACT

Although extensive research concerning the flow around ship hull has been carried out in the past decades it is still difficult to calculate the flow around the hull including the propeller. It is well known that the flow behind the ship is affected by the propeller, and the hydrodynamic performances of the propeller are dependent on the incoming flow velocity. In the present paper the viscous flow around an existing vessel is computed including the propeller action. This analysis combines the numerical investigation of flow around the ship with propeller theory to simulate the hull-propeller interaction. The computations are performed using Shipflow code and in-house codes for propeller design. The designed propeller geometry is specified in the flow module in order to obtain the thrust and torque coefficients. The minimum iteration number and grid density are carefully chosen to reduce the computational effort required. The velocity field behind the ship is recalculated into an effective wake and given to the propeller code that calculates the propeller load. Once the load is known it is transferred to the RANS solver to simulate the propeller action. Knowledge on the propeller behavior and the impact of improvements in hull and propeller geometries introduce valuable new perspectives for hull design.

Key words: Potential flow, viscous flow, RANSE, $k - \omega$ SST turbulence model, propeller, wake.

¹ Ana Maria Tocu, PhD Student, University of Galati, Faculty of Naval Architecture, (ana.tocu@ugal.ro), Domneasca Street, 800008 Galati, Romania. ² Mihaela Amoraritei, Lecturer, University of Galati, Faculty of Naval Architecture (mamor@ugal.ro), Domneasca Street, 800008 Galati, Romania.

INTRODUCTION

In recent years, due to the increased computers capacity as well as to the reduced time spent on running the practical calculation of the flow around a ship, the interaction between the ship hull and its propeller seems to be a very interesting topic. Previously studies have focused upon reducing hull resistance while neglecting the effects of the propeller and the interaction between the ship hull and the propeller. Nowadays, the interaction between the propulsor and ship stern flow became the subject of many investigations. Some previous studies focused on observing the propeller action under fully wetted condition, in order to compare the pressures on the hull with and without propeller effects, but this is mainly limited by the high demand of accuracy in the CFD codes and by the large computational effort. It is very important to know how to investigate, locate and even eliminate the influence of the possible errors, such as turbulence modeling errors, integral and interpolation errors, flow limiter errors, grid and geometry errors, iterative errors and other errors yet unnoticed. Therefore, a careful analysis and validation is required.

As a first step, some calculations using the coupled potential flow-boundary layer method have been carried out. Then, various viscous flow simulations based on Reynolds-Averaged-Navier-Stokes (RANS) approach have been performed with and without propeller effects. The RANS computations include the propeller action by applying the body force method. The method considers the thrust and the torque of propeller as a field of forces which can be added to the body force terms in the RANS equations. The propeller forces are calculated using a simple force actuator disk or a lifting line method with specified effective inflow.

MATHEMATICAL MODEL

An alternative approach to the experimental measurements is to use CFD to predict the fluid velocity distribution by solving the fundamental equations of motion using numerical methods. The fundamental equations for fluid flow, which describe the conservation of fluid mass and momentum, are the equation of continuity and the Navier-Stokes equations. In practice, it is impossible to solve these equations directly since the fastest supercomputers with largest memory available falls short of the required performance by many orders of magnitude. However, if the fundamental equations are averaged over a period of time, the computer requirements for the resolution of the flow features are eased enormously. The time-averaging process introduces new variables into the equations that are known as the Reynolds stresses. To close the set of equations, i.e. to have as many equations as unknowns, turbulence models are introduced to express the Reynolds stresses.

Two-equation turbulence models represent the largest class of turbulence models used in engineering CFD. The models generally consist in a coupled transport equations for the turbulent kinetic energy k and its rate of dissipation. Since the

standard $k - \varepsilon$ model often does not produce satisfactory results, some other turbulence models have been developed. One of them is the $k - \omega$ model that in some ways performs better then the $k - \varepsilon$ model. A new shear stress transport $k - \omega$ model, with remarkable advantages when compared with $k - \varepsilon$ and previous $k - \omega$ models, was proposed by Menter. The model is based on the assumption that the principal shear-stress is proportional to the turbulent kinetic energy, which is introduced into the definition of the eddy-viscosity. In the $k - \omega$ SST model the $k - \omega$ model, is used near the wall and the $k - \varepsilon$ model, transformed to resemble the $k - \omega$ model, is used outside of this region. The different sets of coefficients and the additional cross-diffusion term from the transformed $k - \varepsilon$ model are combined by blending or switching functions F_i in an intermediate region. With these approaches, it is possible to solve all equations without the use of wall functions, and this is itself a significant improvement.

Governing equations and turbulence modeling

In Cartesian tensor notation form, using the Reynolds decomposition, the unsteady incompressible RANS and continuity equation are written as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\left(u_j u_i + u_j u_i^{'}\right)}{\partial x_j} = \overline{R_i} - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\upsilon \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right)$$
(2)

where x_i represents the Cartesian coordinates, $x_i = (x, y, z)$, u_i are the time averaged velocity components in Cartesian directions, $U_i = u_i + u'_i$, u'_i , are the time fluctuating velocity components in Cartesian directions, R_i is the volume force, P is the instantaneous pressure, $P_i = p + p'$, p is the time average pressure, p' is the fluctuating pressure, ρ is the density, v is the kinematic viscosity, $v = \mu/\rho$, and μ is the dynamic viscosity. Using the Boussinesq approximation, the Reynolds stress tensor $\rho \ \overline{u'_i u'_j}$ can be written in the following form:

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

where δ_{ij} is Kronecker delta and is the turbulent kinetic energy. The new Reynoldsaveraged equations are written as:

$$\frac{\partial u_i}{\partial t} + \frac{\left(u_j u_i + u_j' u_i'\right)}{\partial x_j} = \overline{R_i} - \frac{1}{\rho} \frac{\partial p}{\partial x_i} - \frac{2}{3} \frac{\partial k}{\partial x_i} + \frac{\partial}{\partial x_j} \left(v_E \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j} \right) \right)$$
(4)

where v_E is the effective kinematic viscosity defined as $v_E = v + v_T$. The Reynolds stresses are related to the mean rate of strain through an isotropic eddy viscosity $v_T = \mu_T / \rho$, which is calculated using Menters combination of $k - \omega$ and $k - \varepsilon$ turbulence models:

$$\frac{\partial k}{\partial t} + \frac{\partial \left(u_{j} k \right)}{\partial x_{j}} = -\overline{u_{i} u_{j}} \frac{\partial u_{i}}{\partial x_{j}} - \beta^{*} k \omega + \frac{\partial}{\partial x_{j}} \left(\left(\upsilon + \sigma_{k} \upsilon_{r} \right) \frac{\partial k}{\partial x_{j}} \right)$$
(5)

$$\frac{\partial\omega}{\partial t} + \frac{\partial(u_j\omega)}{\partial x_j} = -\frac{\gamma}{v_T} \overline{u_i'u_j'} \frac{\partial u_i}{\partial x_j} - \beta^* \omega^2 + \frac{\partial}{\partial x_j} \left(\left(\upsilon + \sigma_\omega \upsilon_T \right) \frac{\partial\omega}{\partial x_j} \right) + 2\sigma_{\omega_2} \frac{1 - F_1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial\omega}{\partial x_j}$$
(6)

Here β^* , γ , σ_k , σ_{ω_2} are the modeling coefficients for $k - \omega$ equations and F_1 is the switching function for handling the change between the ω and ε equations.

Boundary conditions

The boundary conditions within the computational domain are requiring the no-slip conditions on the hull surface as follows: zero velocity, a Neumann condition

for the pressure, and a Dirichlet conditions for k and ω . In the symmetry plane, zero-gradient Neumann conditions are imposed for all the variables. At the upstream, k, ω and the velocity are supposed to be constant, whereas the pressure is extrapolated with zerogradient. At the downstream, the velocity, k and ω are extrapolated with zero-gradient, while the dynamic pressure has the zero value.



Figure 1. Solution domain

STRATEGY

The geometry of the ship is defined by an offset file based entirely on the lines plan depicted in Figure 2. The offset file has been prepared in order to discretize the hull for the numerical computations. For computing waves, wave resistance, lift and induced resistance, a potential flow method has been used for various Reynolds numbers. This potential solving module provided also the input to a boundary layer method, which predicts transition and boundary layer parameters on the forward half of the ship. For predicting the viscous flow, a RANS code with boundary conditions defined by the potential flow results and the boundary layer parameters was used. The free surface is obtained as potential-flow solution and is kept fixed for the solution of the RANS equations. The viscous computations were performed on a LPG carrier having the main particulars tabulated in Table1, with and without propeller. The propeller model calculates the body-forces based on the effective wake field, which means that the propeller solver, which is implemented in the CFD code, is running interactively with the RANS solver. The viscous module specifies the body forces in the cells of an additional cylindrical grid that covers the location of the propeller. The body forces are distributed between the hub and the maximum propeller diameter and also in the axial direction to avoid abrupt changes and concentration of introduced forces. The propeller can be modeled as an actuator disk with prescribed thrust and torque, or using the lifting line theory. The propeller position, detailed geometry and advance ratio are specified for the lifting line method,

-		
Length overall	125.36	[m]
Length between perpendiculars	117.9	[m]
Breadth moulded	20.5	[m]

7.64

16

[m]

[knots]

Table 1. Main particulars of the ship.

Figure 2. Lines plan.

Draught

Ship speed



while if a force actuator disk model is used, the detailed geometry is not necessary.

Along the whole hull a single structured 3D numerical grid was created with clustering of cells in near the bow and stern regions. By using suitable parameters for viscous computations, the 3D overlapping grid generator provided a grid having 283x32x60 grid nodes in the longitudinal, transverse and normal direction, respectively, and the inner surface fitted to the hull (see Figure 3). Taking into account that the solver does not use wall laws, another clustering of cells was needed closest to the hull surface where the height of the cells should be very thin. For generating this single block grid, the program computes the outer boundaries, controlling permanently the orthogonality functions. Interpolation weights are computed and saved for all interpolation points at the edge of each grid. To avoid mismatch, special techniques are used to handle the interpolation between the extremely thin cells close to boundaries where the no-slip condition is to be applied. The detailed grid topology generated around the hull extremities is depicted in Figure3. To make the picture more readable, only some slices in the grid were displayed.



Figure 3. Three dimensional grid generated for the viscous calculations. The detailed aft (left)and the fore parts of the hull (right).

After the viscous calculations of the flow around the ship without propulsor, an additional cylindrical component grid (having 12x16x16 grid nodes) was added to simulate the operating propeller. Figure 4 shows the circular grid fitted behind the ship to create both symmetric and three dimensional effect of the propeller in order to compute the trust and torque.



Figure 4. Cylindrical grid for simulating the propeller.



RESULTS AND DISCUSSIONS

Useful CFD results are provided by the potential flow method which can be extensively used for the optimization of hull shapes. The optimization is normally based on the predicted waves and hull pressure distribution that is why an analysis of the simulated wave profiles has been performed. The results can be seen in Figure 5, which depicts the wave profile non-linearly computed for the service speed. Another set of simulations was performed for a speed range between 14 and 18 knots for a better understanding on how the velocity affects the wave patterns on the hull. As expected, the more the speed increases, the higher the stern wave crest becomes. For validation purposes, a comparison between the total hull resistance calculated with the Holtrop-Mennen method and the one computed with Shipflow is further brought into focus in Figure 6.



Having various input data (such as ship resistance and propulsive factors), a set of series propellers have been designed. In the propeller design process, for the estimation of the propulsive performances, it is customary to do first stock propeller simulation. By doing that, one could get a quick idea about the effective wake fraction, thrust and assessment of power before designing the final ship propeller geometry. At this initial stage of the propeller design, series propellers were used (Wageningen B-series propeller) in the numerical and experimental simulations. In the first case, with results from the Holtrop-Mennen method, a B-series propeller was calculated in order to obtain the optimum propeller diameter. Having the main characteristics, some simulations including an actuator disk were performed. The computations provided an effective wake that was used in re-designing the propeller for the next simulation case. Although quite accurate propeller characteristics were obtained, as a last step, a wake-adapted propeller was designed using one of the inhouse codes based on the lifting-line theory (case3).

Prop	eller characteristics	Case1	Case2	Case3	Experiment
K _T	thrust coefficient	0.193	0.197	0.226	0.206
K_Q	torque coefficient	0.032	0.033	0.037	0.034
η_0	efficiency	0.637	0.624	0.639	0.623
P/D	pitch ratio	10.007	0.984	0.971	1.05
A_e/K_0	expanded area ratio	0.55	0.55	0.53	0.55
D	propeller diameter	5.4	5.4	5.3	5.4
W	wake	0.268	0.319	0.319	0.290

Table 2. Measured and calculated propeller quantities.



Figure 7. Main characteristics of the B-series propeller used in Case 2. Propeller geometry (left) and open-water characteristics (right).

Simulations based on RANSE approach were carried out to obtain a base reference for comparing the flow around the hull with and without propeller effect. Developing thrust and accelerating the flow in which it works, the propeller leads to different hydrodynamic characteristics in open water and behind the ship. Between nominal and effective wake a serious distinction must be made. The nominal wake is the wake behind the hull without propeller. The wake velocities with the propeller operating behind the ship and developing thrust is called effective wake. As seen in Figure 8, the propeller thrust creates strongly axial flow acceleration behind the propeller comparing with the bare hull conditions.



Figure 8. Axial velocity contours for nominal (left) and effective wake (right).



The numerical solution reveals a rather complex flow field in the stern region where the velocity distribution and propeller loading reflects changes in the flow field.

Figure 9. Wake contours behind the ship hull.

The wake contours in the propeller disk are quite well predicted, exhibiting islands of low axial velocity and a fairly pronounced "hook" as proven in Figure 9,. Many researchers in viscous flow CFD directed their jobs towards improving the ability to predict these features. On closer inspection it turns out that the wake

hooks, which are present for large classes of ships, are caused by the bilge vortices, generated at the bilge and hitting the propeller plane inside the propeller disk. An accurate calculation of these vortices and then a better prediction of the hooks are determined by the advanced turbulence model. Experience shows that such predictions are mostly sufficiently accurate for design purposes.

CONCLUDING REMARKS

Summarizing the results, non-linear free-surface potential flow around the hull was successfully computed and ship resistance was also determined through the numerical simulations. Based on the RANSE approach various simulations were carried out to compute the flow around the ship hull with and without an operating propeller (applying the body force method). A Wageningen B-Series propeller has been employed to estimate the performances of the given hull. Taking into account the efforts related in modeling the propeller, the method appears to be useful in connection with studies of propeller-hull related flow problems. In general, quite accurate propeller characteristics were obtained. In the prediction of the wake details, turbulence modeling was the key issue. Thanks to the $k - \omega$ SST turbulence model, seems like the RANS method predicted integrated quantities of the wake reasonably well. In the future, another wake validation will be performed by comparing numerical and experimental simulations but using the final optimum propeller.

ACKNOWLEDGEMENTS

The present work could not be possible without the financial support of the Romanian National University Research Council (CNCSIS) trough the Grant Code TD_180/2007.

REFERENCES

- Larsson, L., Regnström, B., Broberg, L., Li, D., Janson, C.E. (1999): Failures, Fantasies, and Feats in the Theoretical/Numerical Prediction of Ship Performance, *Twenty-Second Symposium on Naval Hydrodynamics*, pp.11-30.
- Lungu, A., (2007): Free-surface turbulent flow around an LPG ship hull, *Proceedings of the* 3rd Workshop on Vortex Dominated Flows Achievements and Open Problems, Timisoara, Romania, pp.61-68.
- Menter, F.R., (1993): Zonal Two Equation $k-\omega$ Turbulence Models for Aerodynamic Flows, 24th Fluid Dynamics Conference, Orlando, AIAA paper 93-2906.
- Lee, SK., Chen, H.C (2005): The Influence of Propeller/Hull Interaction on Propeller Induced Cavitating Pressure, ISOPE Conference, Korea, pp.25-33.
- Zhang, D.H., Broberg, L., Larsson, L., Dyne, G. (1992): A Method for Computing Stern Flows with an Operating Propeller, *Transactions, Royal Institution of Naval Architects*, *Vol. 134*.
- Stern, F., Kim, H.T., Zhang, D.H., Toda, Y., Kerwin, J., Jessup, S. (1994): Computation of viscous flow around propeller-body configurations: Series 60, C_B=0.6 ship model, *Jour*nal of Ship Research, Vol. 38, No. 2.

*** Flowtech International AB, XCHAP - theoretical manual, 2006.