1. INTRODUCTION

Motion of ships in turbulent flows is a complex fluid mechanics problem that must be analyzed by experimental and numerical techniques. Advances in random access memory (RAM) and powerful computer technologies enable engineers to use advanced computational fluid dynamics (CFD) techniques to resolve the marine engineering problems. Recently significant effort in the maritime industry has been spent to perform CFD analysis in designing efficiently ship hull forms. Accurate simulation of turbulent flows around ships has a central role in the optimal design. The main problem encountered is that there two different phases that should be considered, air and sea water. Existing turbulence models are generally proposed for single phase flows and may not represent the turbulence flow with two-phase flows [Senocak, 2005]. From the open literature it can be reported that ship hydrodynamics computations based on Navier-Stokes (N-S) solvers were initiated in the 1980s, and since then a number of useful codes have been developed [Sato et al., 1999]. In the recent years, CFD techniques have been incorporated into optimization procedures for hull configuration. In this way CFD simulation plays an important role in ship design, performance analysis and form optimization, etc. A flow-simulation method was developed to predict the performance of a sailing boat in unsteady motion on a free surface by Akimoto and Miyata, 2002. The sailing conditions of the boat are virtually realized by combining the simulations of water-flow and the motion of the boat. Skytt, 2004 has given the detailed information about how to get a CAD design to CFD meshes for ship geometries. Parolini and Quarteroni, 2003 and Gorski, 2002 reported recent innovative aspects of the numerical models used in CFD studies. Successful design and optimization of marine structures requires information about the flow problem on both integral and field quantity levels. Therefore, good insight in the flow problem is essential. Integral data, i.e. hydrodynamic forces and moments, is relatively easily measured in the towing tank, but when it comes to the field quantities, i.e. velocity, pressure and wave fields, extensive data sets are difficult and expensive to obtain experimentally. In order to provide a supplement to the experiment, CFD detailed information about the flow problems can be conducted.

This present study deals with RANS simulation of flow around a ship hull. During the analysis two different turbulence models are used, standard k-\(\varepsilon\) (SKE) and standard k-\(\omega\) (SKW) with standard wall functions as near-wall treatment. The main goal of this numerical study is to show the capability a general-purpose CFD code of FLUENT for design, analysis and feasibility of such a simulation for ship industry.

2. MATHEMATICAL MODEL

A CFD job involves three different activities; preprocessing, analysis the problem by using a solver and post processing to show results. In the preprocessing, the flow domain is discretized, i.e. the domain around the structure is divided into a number of small control volumes to solve the governing equations. In this current study the discretization is performed with GAMBIT preprocessor package of FLUENT which allows the user to generate the computational grid. Mainly hexahedral cells are used for obtaining a good resolution of the flow around the ship hull. Flow simulations are conducted with FLUENT. The code has been used commonly in fluid dynamics are through different turbulence models and can calculate both steady and unsteady solutions. The choice of scheme depends on the nature of the considered flow and the intended application of the results.
The typical output from a flow simulation consists of a set of field quantities: Velocity and pressure fields, streamlines and forces that acting on the hull. In the post processing stage, aforementioned flow information is extracted from the data set generated by the FLUENT and TECOPLANT is also used to visualization.

In this paper, three-dimensional, incompressible two-phase flow of air-water is considered. The governing equations for the fluid are the equation of continuity (Eq.1) and momentum balance (Eq.2) in Cartesian tensor notation.

\[
\frac{\partial p U_i}{\partial x_i} = 0
\]  

(1)

\[
- \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial U_j}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho u \mu_j \right) = 0
\]  

(2)

The standard k-ε model of Launder and Spalding, 1974 has been used extensively to simulate the turbulent flows. \( \vec{u} \mu \) correlation are calculated by eq. 3 and 4.

\[
\rho \frac{Dk}{Dt} = \frac{\sigma_k}{\sigma_k} \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_j} + G_k + \rho \epsilon - Y_M
\]  

(3)

\[
\rho \frac{D\epsilon}{Dt} = \frac{\sigma_k}{\sigma_k} \left( \mu + \mu_t \right) \frac{\partial \epsilon}{\partial x_j} + C_{\epsilon k} \frac{\epsilon}{k} (G_k + C_{\epsilon k} G_b) - C_{\epsilon \epsilon} \rho \frac{\epsilon^2}{k}
\]  

(4)

\( C_{\epsilon k}, C_{\epsilon e} \) and \( C_{\epsilon e} \) are set to equal to 1.44, 1.92 and 0.09, respectively. \( \sigma_k = 1.0 \) and \( \sigma_{\epsilon} = 1.3 \) are the turbulent Prandtl numbers for \( k \) and \( \epsilon \), respectively. \( G_k \) represents the generation of turbulent kinetic energy due to the mean velocity gradients. \( G_b \) is the generation of turbulent kinetic energy due to buoyancy. \( Y_M \) represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. In the near wall region, molecular viscosity affects the level of turbulence energy, so the standard k-ε model is not applicable in the vicinity of the solid walls. To handle the standard near wall flow, near-wall function which uses linear and logarithmic laws to describe the velocity profile near the wall is used.

The standard k-ω model is an empirical model based on model transport equations for the turbulence kinetic energy (k) and the specific dissipation rate (ω), which can also be thought of as the ratio of \( \epsilon \) to \( k \).

As the \( k-\omega \) model has been modified over the years, production terms have been added to both the k and \( \omega \) equations, which have improved the accuracy of the model for predicting free shear flows. The turbulence kinetic energy, k, and the specific dissipation rate, \( \omega \), are obtained from the following transport equations:

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k
\]  

(5)

and

\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_j} (\rho \omega u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega
\]  

(6)

\( G_\omega \) represents the generation of \( \omega \). \( \Gamma_k \) and \( \Gamma_\omega \) represent the effective diffusivity of \( k \) and \( \omega \), respectively, \( Y_k \) and \( Y_\omega \), represent the dissipation of \( k \) and \( \omega \) due to turbulence. All of the above terms are calculated as described below. \( S_k \) and \( S_\omega \) are user-defined source terms.

The effective diffusivities for the \( k-\omega \) model are given by

\[
\Gamma_k = \mu + \frac{\mu_t}{\sigma_k}
\]  

(7)

\[
\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega}
\]

where \( \sigma_k \) and \( \sigma_\omega \) are the turbulent Prandtl numbers for \( k \) and \( \omega \), respectively. The turbulent viscosity, \( \mu_t \), is computed by combining \( k \) and \( \omega \) as follows:

\[
\mu_t = \alpha \frac{\rho k}{\omega}
\]  

(8)

In FLUENT the governing equations are discretized using a first-order upwind interpolation scheme, and the discretized equations are solved using SIMPLE algorithm. The solution is considered converged when the normalized residuals of all the variables is lower than 10^-5.

Grid structure around the hull is shown in Fig.1 in which the mesh is clustered near the hull surfaces. As can be seen from the figure multi-block technique is used for generation of the grid. It must be confessed here that generate such a grid around the hull is time consuming and must be satisfied the requirements of a good description of the geometry.
No-slip conditions are imposed on the wetted part of the hull surface that means that all the velocity components on the surface are zero. At the inlet, a uniform flow is given and at the outlet, all variables are extrapolated. On the centerline boundary and the external boundary, the symmetry condition for all variables is implemented.

3. RESULTS AND MAIN FINDINGS
The main purpose of this study is to show the capability of the general-purpose CFD solver of FLUENT to use at naval architecture and marine industry. Some results obtained after three months running a three-parallel cluster are presented at this section.

Figure 2 presents the pressure contours on the hull surface. It displays higher pressure at the bow than at the stern and low pressure at the amidships. This is common for this type of ship form as reported by [Tingqiu et al., 20019]. However, near the aft perpendicular, the flow is relatively smooth with very low pressure.

It should be noted that, a relatively large region of low pressure parallel to the keel is developed, which causes the streamlines to diverge outwards from the keel and move upwards as shown in Fig.4-Fig.5. Above the low pressure region the pressure is seen to increase towards the rear of the hull, resulting in adverse pressure gradients in the streamwise direction.

Some pictures are showed in Fig.6. In this figures streamlines around the ship hull is presented. Wave pattern occurred at the bow of the hull can be seen especially in Fig.6b and in Fig.6c. Although not presented here, it must be noted here that a separation region occurs at the end of the hull. This is can be reproduced in Fig.7 which velocity contours are showed. Figures in Fig.7 also show the effect of two different turbulence models, SKE and SKW respectively. From these figures it is seen that there are some differences between velocity contours obtained from SKE and SKW.

The current study shows that the wave fields around the hull are not strongly dependent on the turbulence model, whereas the best performance is offered for the total drag coefficient by the SKW model (Table).

During calculation of the viscous pressure resistance, freesurface is taken constant, i.e. problem is considered as only one-phase (water). FLUENT uses VOF model to take into account the freesurface. VOF model assume two different fluids that do not get into each other. The transport equation for the volume fraction of the water (1 for completely filled, zero for completely empty cells) is solved (in addition to the usual conservation equations of mass and momentum). The method is more efficient than others methods such marker-and-cell scheme and can also be applied to breaking waves. However, the freesurface contour is not sharply defined and special techniques had to be developed to obtain an
accurate profile with reasonable numbers of cells [Bulgarelli et al., 2003].

Figure 5 Pressure distributions (left) and streamlines (right) on the hull from bottom view.

Figure 6 Pathlines colored by volume fraction obtained by SKE turbulence model.

Figure 7 Contours of velocity magnitude obtained from a) SKW and b) SKE.

Ship resistance can be identified as a reaction to the ship hull from water. From the experimental and numerical methodology, it can be said that there are three component of the ship resistance:

1. Wave resistance ($R_w$)
2. Viscous resistance ($R_F$)
3. Viscous pressure resistance ($R_{vp}$)

$$R_T = R_w + R_F + R_{vp}$$  \hspace{1cm} (9)

Total resistance and viscous pressure resistance comparisons are showed in Table 1. As can be seen, results obtained from SKW are in good agreement with experimental towing test data. It should be noted here that these CFD results are obtained from 638,150 hexahedral mesh elements and all the results including the experiment are calculated for only half-ship hull.

As aforementioned, in this numerical study, CFD code FLUENT is used. After the all runs the code calculate to different forces; viscous pressure and wave resistances. As it is well known, both viscous and wave resistances occur due the existence of pressure. During the pressure calculation it is difficult to discern these resistances from each other. Considering the double-model enable us to separate viscous pressure from the wave resistance. In this method freesurface is specified as
symmetry condition and by this approach, pressure resistance gives the viscous pressure value. The ratio of the viscous pressure to friction resistance gives form factor. In present study, all calculations of double-model are performed with SKE and SKW for 12 knots.

Table 1 Total and viscous resistances comparisons.

<table>
<thead>
<tr>
<th></th>
<th>Total Resistance (kN)</th>
<th>Viscous Resistance (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>24.3</td>
<td>13,500</td>
</tr>
<tr>
<td>SKE</td>
<td>18</td>
<td>14,686</td>
</tr>
<tr>
<td>SKW</td>
<td>26.1</td>
<td>13,781</td>
</tr>
</tbody>
</table>

4. CONCLUSION

In this work, we have presented some of the most recent results on numerical fluid dynamic modeling obtained by using CFD code FLUENT. We have highlighted the importance that CFD analysis is achieving in the ship resistance calculation process. The effects of turbulence models are investigated for two different model, standard k-epsilon and standard k-omega, respectively and it is concluded that standard k-omega turbulence model is in good agreement with experimental towing test data.

5. REFERENCES


