Numerical Prediction of Ship's Self-Propulsion Parameter by Using CFD Method

N.T.N. Hoa
Ho Chi Minh City University of Transport, Ho Chi Minh, Vietnam

ABSTRACT: This paper reports the results of numerical simulations of ship self-propulsion using the computational fluid dynamics (CFD) method. The sliding mesh method is utilized to model the actual propeller working behind the ship. In addition, the volume of fluid method was applied to accurately track and solve the free surface. Some important factors such as mesh generation, time step, turbulence model that can affect the accuracy of the obtained simulation results are discussed in this research. The Benchmark Japanese Bulk Carrier vessel was used in this study as the case study. The numerical obtained results are compared with measured data to verify and validate the numerical results.

1 INTRODUCTION

The prediction of a ship’s self-propulsion parameters is a challenging task in ship hydrodynamics in general and in the ship power estimation in particular due to the accuracy in evaluating this parameter will effect on accurate power estimation.

Among various methods to evaluate the ship’s self-propulsion, the CFD method is the most commonly used due to its accuracy and computational time [1-4]. Therefore, this paper aims to evaluate the ship’s self-propulsion based on the CFD method.

Nowadays, according to CFD method, there are two different approaches for predicting ship’s self-propulsion point, which consists of using the actual propeller and using virtual disk instead of the actual propeller, some previous research works using these approaches are reported in the literatures [3-19]. Although, the advantage of virtual disk method is simple and faster in predicting the ship’s self-propulsion, it is unable to provide detailed information about flow around the propeller.

However, using actual propeller method can provide us all the information about flow field in the wake regions efficiently and reliably. Therefore, this study used actual propeller method to evaluate the ship’s self-propulsion parameters.

Previous research has employed the actual propeller method to evaluate the self-propulsion characteristics of a ship. Tu T.N. et al [3] utilized the CFD method to investigate the interaction between the hull of the ship and propeller, as well as the propulsive coefficients for the actual propeller. The simulation results showed good agreement with measured data. Castro, A.M., et al. [10] used actual propeller method to evaluate the ship’s self-propulsion parameters for containership at full-scale. The obtained simulation results are agreed well with the available data. In the research of Sun, W., et al. [20] actual propeller method was used to performed self-propulsion simulation. The numerical results obtained for ship self-propulsion in full-scale shows good agreement with the measured data. The previous studies have played a vital role in predicting
propulsive coefficients, and this study uses the actual propeller method to simulate the self-propulsion of the JBC ship in model-scale. The sliding mesh method was applied to model actual propeller located behind the ship.

2 MATERIAL AND METHOD

2.1 Flow model

The Reynolds-Averaged Navier-Stokes Equations (RANSE) is amended with the force \( F_0 \). This represents the propeller acting on the fluid as given in Equation 1.

\[
\frac{\partial (\rho \tilde{u}_i)}{\partial x_j} = \left( \frac{\partial^2 \tilde{u}_i}{\partial x_j^2} - \frac{\partial \tilde{u}_i}{\partial x_j} \right) = \frac{\partial \tilde{p}}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + F_i
\]

where \( \rho \) is the fluid density, \( \mu \) is the dynamic viscosity, \( \tau_{ij} \) is the Reynolds stress, \( \rho \) presents the mean pressure, \( \tilde{u}_i \) presents the averaged Cartesian components.

The RANSE are defined as follows:

\[
\frac{\partial (\rho \tilde{u}_i)}{\partial x_j} = 0
\]

where \( x \) and \( \tilde{u}_i \) are the position and velocity vector, \( \rho \) is the fluid density, \( \rho u_{ij} \) is the Reynolds stress tensor, \( \tilde{p} \) is the mean pressure, \( t \) is the time and \( \tilde{u}_i \) is the mean viscous stress tensor.

\[
\tau_{ij} = \mu \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right)
\]

where \( \mu \) is the dynamic viscosity.

2.2 Turbulence model

The Realizable k- \( \varepsilon \) two-layer model is one of the turbulence models that calculates the eddy viscosity by solving equations for \( k \) and \( \varepsilon \). This model is designed to accurately predict the turbulent flow in a wide range of applications.

\[
\mu_t = \rho C_f f_\nu k T
\]

where \( f_\nu \) is a damping function, \( T \) is a turbulent time scale and \( C_f \) is a model coefficient.

Eqn. (6) determines the turbulent time scale as follow:

\[
T = T_
u
\]

The transport equations for \( k \) and the \( \varepsilon \) are given as follows:

\[
\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho \tilde{u}_i k) = \nabla \cdot ( \mu (\nabla k + (\nabla \tilde{u}_i)^T) ) + \rho f_k + \rho \varepsilon - S_k
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \tilde{u}_i \varepsilon) = \nabla \cdot ( \mu (\nabla \varepsilon + (\nabla \tilde{u}_i)^T) ) + \rho f_\varepsilon + (\varepsilon + \mu \nabla \cdot \tilde{u}_i) - S_\varepsilon
\]

Production terms \( P_k \) and \( P_\varepsilon \) are given by Eqn. (9) as follow:

\[
P_k = f_k G_1 + G_2 \gamma_k \quad P_\varepsilon = f_\varepsilon S_k + C_{\varepsilon} G_3
\]

The damping functions is given by Eqn. (10) as follow:

\[
f_k = \frac{k}{k + \sqrt{\alpha}}
\]

\[
f_\varepsilon = \frac{1}{C_{\varepsilon} \left( 4 + \sqrt{6} \cos \left( \frac{1}{3} \cos^{-1} \left( \frac{\sqrt{6} - S_\varepsilon^2}{2S_\varepsilon} \right) \right) \right)} \frac{k}{\varepsilon + S_\varepsilon + \nu \cdot \nu}
\]

3 NUMERICAL SIMULATION

3.1 Case study

The vessel used as a case study in this study is JBC vessel. This vessel was developed by the Japanese National Maritime Research Institute. The simulation is conducted at a model-scale of \( \lambda = 40 \), so it allows us to carry out a direct comparison with experimental data. The hull form and propeller specifications of the JBC vessel are listed in Tables 1 and 2. For further visualization, Figures 1 and 2 provide views of the ship and its propeller. The towing tank measured data for JBC are available in [21, 22], providing a reliable source for comparison with the simulation results.

Table 1. JBC ship parameters

<table>
<thead>
<tr>
<th>Descriptions</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ship length</td>
<td>L</td>
<td>7.000</td>
</tr>
<tr>
<td>Ship breadth</td>
<td>B</td>
<td>1.1250</td>
</tr>
<tr>
<td>Design ship draft</td>
<td>T</td>
<td>0.4125</td>
</tr>
<tr>
<td>Volume displacement</td>
<td>V</td>
<td>2.787</td>
</tr>
<tr>
<td>Froude number</td>
<td>Fr</td>
<td>0.1420</td>
</tr>
</tbody>
</table>

Figure 1. The geometry of JBC vessel

Table 2. Propeller parameters

<table>
<thead>
<tr>
<th>Descriptions</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diameter of propeller</td>
<td>Dp</td>
<td>0.203</td>
</tr>
<tr>
<td>Angle of rake</td>
<td>( \Theta ) [deg.]</td>
<td>5</td>
</tr>
<tr>
<td>Expanded area ratio</td>
<td>AE/A0</td>
<td>0.500</td>
</tr>
<tr>
<td>Boss ratio</td>
<td>Dv/Dv</td>
<td>0.18</td>
</tr>
<tr>
<td>Pitch ratio</td>
<td>P0/P0</td>
<td>0.750</td>
</tr>
<tr>
<td>Number of blades</td>
<td>Z</td>
<td>5</td>
</tr>
<tr>
<td>Direction of rotation</td>
<td>-</td>
<td>Clockwise</td>
</tr>
</tbody>
</table>
propeller to be attached to the ship’s hull. An essential factor affecting the level of accuracy of the numerical results is the selection of the time-step size. To predict self-propulsion, a time-step size was chosen that results in the propeller rotating approximately 0.5 to 1.5 degrees per time step [23].

Table 3. Setup for physics model

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Setting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver</td>
<td>3D, implicit unsteady</td>
</tr>
<tr>
<td>Turbulence model</td>
<td>Realizable $k$–$\varepsilon$ two layer</td>
</tr>
<tr>
<td>Multiphase model</td>
<td>The volume of fluid</td>
</tr>
<tr>
<td>Temporal discretization</td>
<td>First-order</td>
</tr>
<tr>
<td>Wall treatment</td>
<td>all wall $y^+$ treatment</td>
</tr>
</tbody>
</table>

The numerical results are significantly affected by the mesh generation. In this research, a trimmed cell mesher was utilized to generate meshes for both stationary and cylindrical rotating sub-regions. The grid was refined at the free surface to accurately capture the Kelvin wave. Additionally, local volume grid refinements were implemented around the propeller, ship stern and ship bow regions, and the rotating sub-region to improve the resolution of the simulations. To accurately capture the interactions between the ship hull and the propeller, the trailing and leading edges of the propeller were subjected to additional refinement. Prism layers were also applied to resolve the boundary layer. The mesh consisted of a total of 8.7 million cells. Figure 4 displays the mesh generation results.

Figure 3. The calculated domain for self-propulsion prediction

The type of boundary condition was setup as follows [3, 4]: The top, inlet, and bottom boundaries are classified as velocity inlets, while the outlet is subject to pressure conditions. The side boundaries are set as symmetry planes. Additionally, the boundary conditions are set for the ship hull surface and propeller are no-slip wall.

Table-3 presents the physics model settings used in this study. The Volume of Fluid method was utilized for tracking and solving the free surface, and the Realizable $k$–$\varepsilon$ turbulence model is chosen to close RANSE due to its proven accuracy in previous studies [24]. The vessel was permitted to move with heave and pitch motions. The propeller’s rotation was introduced using the DFBI model, which enables the

4 RESULT AND DISCUSSION

In this study, the self-propulsion point was defined as the point at which the propeller thrust is equal to the resistance of the ship. However, in model scale simulations, it is necessary to take into account the Skin Friction Correction Force (SFC) that accounts for
the variation in skin friction coefficients between the model scale and the full-scale ship. Ignoring this correction can lead to inaccurate results [25]

\[ T = R_{\text{TSFPR}} - SFC \]  

(11)

The SFC value used in this study was 18.2 N based on measured data [21, 22]. Since it is challenging to determine the self-propulsion in one run, so normally, two constant speed runs were carried out with two propeller revolution rates \( n = 7.80 \) and \( n = 8.00 \) rps. The linear interpolation method was used to determine the self-propulsion point. The time step was set at 3.5.10-4s.

The Table-4 in this study displays the results of the resistance and thrust as a function of the propeller's rotation rate. The self-propulsion point was identified at a rotation rate of 7.85 revolutions per second, as depicted in Figure 5. The comparison between the numerical results (CFD) and the measured data (EFD) is presented in Table 7. The results show a good agreement between the two datasets. The difference between the numerical results and the measured data was found to be 1.57%, 2.84% and 0.64% for resistance of the ship, thrust of propeller and self-propulsion point, respectively. Figure 6 presents a time history of resistance of the ship and propeller thrust at a rotation rate of 7.8 revolutions per second. The oscillations of propeller thrust are five times the rotational frequency due to the effect of ship hull form.

Table 4. Numerical obtained results

<table>
<thead>
<tr>
<th>( n ) [rps]</th>
<th>( R_{\text{TSFPR}} )-SFC [N]</th>
<th>( T ) [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.80</td>
<td>22.85</td>
<td>22.55</td>
</tr>
<tr>
<td>8.00</td>
<td>23.85</td>
<td>24.75</td>
</tr>
</tbody>
</table>

Figure 5. Defining the self-propulsion point procedure

Detailed flow characteristics around the ship hull and propeller in the self-propulsion simulation were also investigated. The figures illustrating these flow characteristics are presented in Figures from 7 to 12, respectively.

Table 5. Computed Self-propulsion point in comparison with measured data

<table>
<thead>
<tr>
<th>Parameters</th>
<th>EFD [22]</th>
<th>CFD</th>
<th>E%D</th>
</tr>
</thead>
<tbody>
<tr>
<td>( R_{\text{TSFPR}} ) [N]</td>
<td>40.760</td>
<td>41.39</td>
<td>1.57</td>
</tr>
<tr>
<td>( T ) [N]</td>
<td>22.560</td>
<td>23.19</td>
<td>2.84</td>
</tr>
<tr>
<td>Self-propulsion point ( n )</td>
<td>7.800</td>
<td>7.85</td>
<td>0.64</td>
</tr>
</tbody>
</table>

Figure 6. Time histories of resistance of the ship and thrust at \( n=7.8 \) rps.

Figure 7. Wave-elevation at \( n=7.8 \) rps.

Figure 8. Water free surface at \( n=7.8 \) rps.

Figure 9. Velocity distribution in symmetry plane at \( n=7.8 \) rps.

Figure-10. Dynamic pressure distribution on the blades surface of propeller at \( n=7.8 \) rps.
resistance of the ship, thrust and self-propulsion point, respectively. Subsequent investigations will focus on enhancing the accuracy of the simulations by exploring various alternatives such as adjusting the grid generation process, increasing the mesh size, and using different turbulence models.

ACKNOWLEDGMENT

I acknowledge the support of time and facilities form Ho Chi Minh City University of Transport for this study.

REFERENCES


