CFD prediction and simulation of a pumpjet propulsor

Lin Lu a,*, Guang Pan a, Prasanta K. Sahoo b

a School of Marine Science and Technology, Northwestern Polytechnical University, Xi’an, 710072, China
b Department of Marine and Environmental Systems, Florida Institute of Technology, Melbourne, FL, USA

Received 2 January 2015; revised 17 June 2015; accepted 26 October 2015
Available online 19 January 2016

Abstract

In this study an attempt has been made to study the hydrodynamic performance of pumpjet propulsor. Numerical investigation based on the Reynolds Averaged Navier–Stokes (RANS) computational fluid dynamics (CFD) method has been carried out. The structured grid and SST k−ω turbulence model have been applied. The numerical simulations of open water performance of marine propeller E779A are carried out with different advance ratios to verify the numerical simulation method. Results show that the thrust and the torque are in good agreements with experimental data. The grid independent inspection is applied to verify accuracy of numerical simulation grid. The numerical predictions of hydrodynamic performance of pumpjet propulsor are carried out with different advance ratios. Results indicate that the rotor provides the main thrust of propulsor and the balance performance of propulsor is generally satisfactory. Additionally, the curve of propulsor efficiency is in good agreement with experimental data. Furthermore, the pressure distributions around rotor and stator blades are reasonable. Beyond that, the existence of tip clearance accounts for the appearance of tip vortex that leads to a further loss in efficiency and a probability of cavitation phenomenon.

Keywords: Pumpjet propulsor; Numerical investigation; Hydrodynamic performance; Computational fluid dynamics

1. Introduction

With the rapid development of computational methods in the last decade, computational fluid dynamics (CFD) has become more and more practical and been widely used in the studies of 3-D turbulent flows. Simultaneously, numerous numerical and experimental researches of the water jet propulsion, axial-flow pump and centrifugal pump were carried out. Some useful results about the velocity, pressure and hydraulic loss have been achieved. Park et al. (2005) presented the numerical analysis of a waterjet propulsion system to provide detail understanding of complicated three-dimensional viscous flow phenomena. The complicated viscous flow features of the waterjet are well understood by the present simulation. Li and Wang (2007) carried out a numerical investigation on an axial-flow pump equipped with an inducer. The pump performances are predicted and compared to the experimental measurements. Recommendations for future modifications and improvements for the pump design are also given. Gao et al. (2008) investigated the performance and three-dimensional flow fields in a waterjet pump. Overall performances by CFD simulation are in good agreement with the experimental results. In addition, the effects of a rear stator and different spacing between the rotor and the stator on the overall performance of the water-jet pump have also been investigated. Zhang et al. (2010) simulated the three-dimensional unsteady turbulent flows in axial-flow pumps based on Navier–Stokes solver embedded with k-ε RNG turbulence model and SIMPLEC algorithm. Numerical results show that the unsteady prediction results are more accurate than the steady results, and the maximum error encountered in unsteady prediction is only 4.54%. Their researches played an
important role for the further research of numerical simulation of pumpjet propulsor. However, present literature review suggests that the numerical simulation of hydrodynamic performance of pumpjet propulsor is few and far between. Though experimental studies of hydrodynamic performance can accurately reflect the variation of the flow field, experiments are time-consuming and cannot be carried out for some complex operating conditions. Ivanell (2001) described a CFD model of the pumpjet propulsor on a torpedo using FLUENT to verify its accuracy by comparing numerical simulation results with wind tunnel experiments. It can be concluded from the simulations that the result for propulsion force is about 10% higher when compared with measurements. On the other hand, the result for the resistance force is about 17% higher. Suryanarayana et al. (2010a,b) evaluated open water hydrodynamics and cavitation performance of the pumpjet propulsor on an axi-symmetric underwater body through CFD study. Results show that the stator can absorb the rotational energy of the rotor and reduce the radial component of wake flows leading to the increase of propulsor efficiency.

In this study, three-dimensional rotor-stator coupling flow fields in a pumpjet propulsor are investigated based on the RANS method. The SST $k$–$\omega$ turbulence model and structured grid has been used. The numerical simulation method and grid independence inspection is verified, and the CFD-predicted overall hydrodynamic performances of pumpjet propulsor are compared with experimental results by using ANSYS CFX. Additionally, the pressure distributions of rotor and stator are also studied at the same time.

### 2. Numerical simulation method

#### 2.1. Governing equations

The governing equations for the turbulent incompressible flow encountered in this research are the three-dimensional RANS equations for the conservation of mass and momentum, given as:

$$\frac{\partial}{\partial x_i} (\rho u_i) = 0$$

(1)

$$\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = \rho F_i - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]$$

(2)

where $p$ is the average pressure, $\mu$ is the molecular viscosity and $\rho u_i u_j$ is the Reynolds stress. To correctly account for turbulence, the Reynolds stresses are modeled in order to achieve the closure of Equation (2). An eddy viscosity $\mu_r$ is used to model the turbulent Reynolds stresses.

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

(3)

where $\mu_r$ is the turbulent viscosity and $k$ is the turbulent kinetic energy.

#### 2.2. Turbulence model

According to the existing study by Ji et al. (2010), the SST $k$–$\omega$ turbulence model is applied for closing the numerical simulation in this study. The SST $k$–$\omega$ turbulence model combines the advantages of stability of the near-wall $k$–$\omega$ turbulence model and independent of the external boundary $k$–$\epsilon$ turbulence model. The SST $k$–$\omega$ turbulence model has the following advantages specifically: it can adapt to a variety of physical phenomenon caused by the pressure gradient changes, and it can utilize the inner viscous layer combined with the wall function to accurately simulate the phenomenon of the boundary layer without the use of easier distortion viscous-attenuation function. When calculating, the solving program based on Reynolds number automatically invokes different turbulence models. In the low Reynolds number regions, the $k$–$\omega$ turbulence model is applied. While in the high Reynolds number regions, the $k$–$\epsilon$ turbulence model is adopted. Consequently, the SST $k$–$\omega$ turbulence model has better applicability in dealing with boundary problems of different Reynolds numbers.

### 3. Verification of numerical simulation method

In order to verify the accuracy of numerical simulation method, the steady flows over a skewed four-bladed marine propeller E779A have been studied. The non-dimensional geometry data of the E779A propeller is taken from Subhas et al. (2012) and presented in Table 1. E779A propeller has been widely tested for several years and reliable experimental data is available by Li and Grekula (2012). The computational domain and grid for E779A marine propeller is a 1/4 cylinder passage as shown in Figs. 1 and 2.

The advance ratio $J$ is defined as $J = U_\infty/(nD)$, where $U_\infty$ denotes the free stream velocity, $n$ is the blade rotating velocity. The thrust coefficient $K_T = Thrust/(\rho n^2 D^4)$ and the torque coefficient $K_Q = Torque/(\rho n^2 D^5)$ are defined, respectively. The numerical simulations of $K_T$ and $K_Q$ with different $J$ are investigated. The numerical results of $K_T$ and $K_Q$ are compared with the experimental data and shown in Fig. 3. As illustrated in Fig. 3, $K_T$ and $K_Q$ are decreasing with increasing of $J$. The numerical prediction results are in good agreement with the experimental results. Consequently, it is presumed

<table>
<thead>
<tr>
<th>Propeller diameter ($D$)</th>
<th>P/D ratio</th>
<th>Skew angle</th>
<th>Rake</th>
<th>Blade area ratio</th>
<th>Hub diameter ($D_h$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>227.3 mm</td>
<td>1.1</td>
<td>4°48'</td>
<td>4°3'</td>
<td>0.689</td>
<td>45.53 mm</td>
</tr>
</tbody>
</table>
that the numerical simulation method based on RANS and SST $k-\omega$ turbulence model would be applicable and reliable for open water performance simulation.

4. Computational model and grid

4.1. Model geometry and computational domain

In this study, the whole hydraulic passage of the pumpjet propulsor is selected as the computational geometry of numerical simulation as shown in Fig. 4. The diameter of pumpjet propulsor is $D_t = 0.324$ m. This pumpjet propulsor has 11 rotor blades, 9 stator blades, and the minimum clearance size between the rotor blade tip and the duct is 1 mm.

The computational domain and boundary conditions are shown in Fig. 5. The computational domain is a cylinder ($11D_t$ in length and $5D_t$ in diameter) surrounding the pumpjet propulsor, whose axis coincides with the symmetry axis of pumpjet propulsor. The inlet is located $4D_t$ from the front face of pumpjet propulsor, and the outlet is situated $6D_t$ from the end of propulsor. According to the structural characteristics of the pumpjet propulsor, the computational domain is divided into three parts: rotor domain, stator domain and external flow field domain. The rotor domain is a rotating domain, the other two domains are stationary domains. The rotor and stator domains are embedded in the external flow field domain. The interactions between the rotor domain and stator domain are solved by using the sliding mesh method.

4.2. Grid generation

The quality of computational grid directly affects the convergence and result of numerical analysis. The structured grid has the advantage of using less memory and is very favorable for the boundary layer calculation. Therefore, all the computational domains are filled with structured grids generated by ANSYS ICEM. According to the division of the computational domain, multi-block grid method is selected to generate high-quality structured grid. The grids around pumpjet propulsor adopt H hybrid grids. The propulsor blade surface is surrounded by O-hexahedral grids. The number of entire computational domain grids is approximately $1.5 \times 10^9$, including $7 \times 10^8$ rotor domain grids and $4.5 \times 10^8$ stator domain grids. The surface grids and three interfaces are formed as shown in Fig. 6, and Fig. 7 shows the rotor and stator blade surface grids.
4.3. Boundary condition

The setting of boundary condition plays a direct influence on the completion of numerical simulation. For computational domain boundary conditions in this study, the inlet boundary is set to normal speed, turbulence intensity is 5% as the default. The no-slip boundary condition is imposed on duct and blades. The free-slip wall boundary is imposed on the cylinder surface. The averaged static pressure is specified at the outlet. For the condition of steady, the “stage” and the “frozen rotor” can be selected for simulation in ANSYS CFX. In this study, the number of rotor and stator blade is relatively large (11 and 9), and the position of rotor and stator is fixed. Besides, according to the Help files of ANSYS CFX, the frozen rotor is more likely to converge and the “frozen rotor” is most useful as the circumferential variation of two frames. Therefore, the “frozen rotor” has been selected for the interface between the rotor domain and stator domain in this study. However, if the number of blade is small or under the transit condition, the selection of the solution of interface may be different and depend on the chosen rotor angle and the timestep, etc. In addition, the time derivatives are calculated using a second-order backward Euler algorithm, the spatial derivatives are calculated using a second-order upwind algorithm, and the finite volume method is selected for the discretization of equations.

5. Results and discussion

In the following sections, maintain $U_{\infty}$ equal to 25.72 ms$^{-1}$ and change $n$ from 25 rpm to 60 rpm to obtain different advance ratios. The thrust coefficients of rotor and stator are defined as $K_{Tr} = T_r/(\rho n^2 D^3)$ and $K_{Ts} = T_s/(\rho n^2 D^3)$. The torque coefficients of rotor and stator are defined as $K_{Qr} = Q_r/(\rho n^2 D^5)$ and $K_{Qs} = Q_s/(\rho n^2 D^5)$, where $T_r$ and $T_s$ represent the thrust of rotor and stator, respectively. $Q_r$ and $Q_s$ denote the torque of rotor and stator, respectively. The thrust coefficient and torque coefficient of pumpjet propulsor are defined as $K_T = K_{Tr} + K_{Ts}$ and $K_Q = K_{Qr}$, respectively. The pumpjet propulsor efficiency $\eta = (J \times K_T)/(2\pi \times K_Q)$ is defined.

5.1. Grid independent inspection

The grid independent inspection is applied in order to ensure the accuracy and precision of numerical simulation. The computational grid is regenerated by reducing the size of the first layer of wall grids. Table 2 shows the numerical simulation results of unrefined and refined grids with different advance ratios.

<table>
<thead>
<tr>
<th>$J$</th>
<th>$K_{T_r}$</th>
<th>$K_{T_s}$</th>
<th>$K_{Qr}$</th>
<th>$K_{Qs}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.72 Unrefined</td>
<td>0.9787</td>
<td>0.1827</td>
<td>0.4871</td>
<td>0.4786</td>
</tr>
<tr>
<td>Refined</td>
<td>0.9785</td>
<td>0.1826</td>
<td>0.4869</td>
<td>0.4785</td>
</tr>
<tr>
<td>2.23 Unrefined</td>
<td>0.9152</td>
<td>−0.0619</td>
<td>0.4711</td>
<td>0.4590</td>
</tr>
<tr>
<td>Refined</td>
<td>0.9152</td>
<td>−0.0616</td>
<td>0.4709</td>
<td>0.4588</td>
</tr>
<tr>
<td>3.17 Unrefined</td>
<td>0.7681</td>
<td>−0.5369</td>
<td>0.4331</td>
<td>0.4241</td>
</tr>
<tr>
<td>Refined</td>
<td>0.7677</td>
<td>−0.5367</td>
<td>0.4330</td>
<td>0.4240</td>
</tr>
</tbody>
</table>
results of hydrodynamic performance of propulsor with three
different advance ratios, \( J = 1.72, 2.23, 3.17 \). As shown in
Table 2, the errors of different coefficients are less than 1%
after the grids are refined. However, the refined grid results in
an increase in simulation time and a decline in grid quality.
Consequently, the undefined grid has been selected for nu-
merical simulation and analysis.

5.2. CFD results and discussion

The numerical investigation and experimental results of
propulsor efficiency are shown in Fig. 8. As can be seen from
Fig. 8, the pumpjet propulsor efficiency increases with the
increasing of advance ratio and then decreases, and achieves
the maximum at \( J = 2.03 \). The numerical simulation results
are in agreement with experimental data Liu et al., (2010).
However, the results of numerical simulation are slightly
larger than experimental data. That is mainly because the
circumstance of numerical simulation sets more idealistic.
The result of CFD prediction and experimental work shows
reasonably good correlation. Hence it may be presumed that
CFD modeling has been carried out with a reasonable degree
of accuracy and confidence.

The results of numerical simulation of pumpjet hydrody-
namic performance are summarized in Table 3. The pumpjet
hydrodynamic performance curves are shown in Fig. 9 by
using the data in Table 3. As can be seen from Fig. 9, the thrust
of rotor is much larger than stator when \( J \) is relative small,
which indicates that the main thrust of pumpjet propulsor is
derived from the rotor. The thrust of stator has a good linear
relation with \( J \). In addition, the thrust of stator changes into
resistance as \( J \) is larger than 2.23, which accounts for the loss
of propulsor efficiency. That is coincident with the efficiency
curve in Fig. 8. According to the trend of curves, the thrust and
torque of the rotor and stator decrease with the increase of the
advance ratio. Furthermore, there is almost no difference be-
tween the torque of rotor and stator, the errors are less than
8\%, which demonstrates that the balance performance of
pumpjet propulsor is generally satisfactory.

<table>
<thead>
<tr>
<th>( J )</th>
<th>( K_{Tr} )</th>
<th>( K_{Ts} )</th>
<th>( K_{Qr} )</th>
<th>( K_{Qs} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.32</td>
<td>1.0219</td>
<td>0.3814</td>
<td>0.4972</td>
<td>0.5007</td>
</tr>
<tr>
<td>1.39</td>
<td>1.0171</td>
<td>0.3423</td>
<td>0.4959</td>
<td>0.4987</td>
</tr>
<tr>
<td>1.49</td>
<td>1.0091</td>
<td>0.2974</td>
<td>0.4947</td>
<td>0.4932</td>
</tr>
<tr>
<td>1.59</td>
<td>0.9961</td>
<td>0.2695</td>
<td>0.4916</td>
<td>0.4857</td>
</tr>
<tr>
<td>1.72</td>
<td>0.9787</td>
<td>0.1827</td>
<td>0.4871</td>
<td>0.4786</td>
</tr>
<tr>
<td>1.86</td>
<td>0.9614</td>
<td>0.1187</td>
<td>0.4826</td>
<td>0.4715</td>
</tr>
<tr>
<td>2.03</td>
<td>0.9406</td>
<td>0.0364</td>
<td>0.4774</td>
<td>0.4641</td>
</tr>
<tr>
<td>2.23</td>
<td>0.9152</td>
<td>–0.0619</td>
<td>0.4711</td>
<td>0.4590</td>
</tr>
<tr>
<td>2.43</td>
<td>0.8832</td>
<td>–0.1849</td>
<td>0.4630</td>
<td>0.4503</td>
</tr>
<tr>
<td>2.81</td>
<td>0.8388</td>
<td>–0.3368</td>
<td>0.4520</td>
<td>0.4450</td>
</tr>
<tr>
<td>3.17</td>
<td>0.7681</td>
<td>–0.5369</td>
<td>0.4331</td>
<td>0.4241</td>
</tr>
</tbody>
</table>

Fig. 8 shows the velocity distribution of the pumpjet
propulsor at \( J = 2.38 \). From Fig. 10 we can see, the flow has
been obviously accelerated after it goes through the rotor and
stator blades. It is suggested that the stator increased the thrust
of propulsor and then accelerated the coming flow. Beyond
that, there is a greater velocity in the clearance between the
rotor blade tip and duct, which probably results in the occurrence of cavitation phenomenon in this blade region. Figs. 11 and 12 show the graphs of the pressure coefficients, defined as
\[
C_p = \frac{(p - p_{out})}{0.5 \rho_f U_\infty^2},
\]
of rotor and stator blades, respectively, which represent the pressure distribution along the blade chord direction at \(J = 2.23\) with different span. \(X\) denotes the distance from monitoring pressure point to the leading edge and \(C\) represents the chord length, where \(p\) is the local pressure, \(p_{out}\) is the outlet pressure. Due to the existence of the duct, the inlet velocity of propulsion has been reduced significantly, so that the rotor and stator can work in a relative high pressure environment which can delay the inception of cavitation and reduce the noise. Additionally, there is a great pressure gradient at the leading edge of the pressure side of the rotor and stator blades as shown in Figs. 11 and 12. In the middle of the blade surface, the pressure coefficients of the pressure side of the rotor and stator blades remain nearly constant. Furthermore, the pressure around the leading edge of rotor suction side is relatively low, which may cause the appearance of cavitation phenomenon. Simultaneously, there is a small clearance between the rotor blade tip and duct for pumpjet propulsor, and the pressure of pressure side of the rotor blade is higher than that of suction side, which finally accounts for the back-flow and the probability of emergence of tip vortex that are unfavorable factors for propulsion efficiency.

6. Conclusion

In this study, numerical investigation and analysis of steady flows around pumpjet propulsor have been presented. A structured grid based on RANS is applied. The SST \(k-\omega\) turbulence model and finite volume method have been employed. The hydrodynamic performance of pumpjet propulsor have been investigated.

A four-bladed skewed propeller E779A is selected for verification of numerical simulation method of open water performance. The numerical predictions of \(K_T\) and \(K_Q\) with different \(J\) are carried out. Results show that \(K_T\) and \(K_Q\) are decreasing with increasing of \(J\) and the numerical prediction results are in good agreement with the experimental results.

The computational results in CFD of pumpjet propulsor efficiency and experimental data shows reasonably good correlation. The computed results indicate that the efficiency of pumpjet propulsor increases with the increasing of \(J\) and then decrease, and achieves the maximum at \(J = 2.03\). In addition, the results of numerical simulation are slightly larger than experimental data because of the idealistic sets of numerical investigation.

Numerical investigation of pumpjet hydrodynamic performance has been presented and summarized. The thrust of rotor is much larger than stator, which indicates that the rotor provides the main thrust of pumpjet propulsor. Furthermore, there is basically no difference between the torque of rotor and stator, the errors are less than 8%, which demonstrates that the balance performance of pumpjet propulsor is generally satisfactory. The velocity distribution of the pumpjet propulsor shows that the flow has been obviously accelerated after it goes through the rotor and stator blades, which demonstrates that the stator increases the thrust of propulsor and then accelerates the coming flow.

The pressure distributions of the rotor and stator blades show that the pressure around the leading edge of rotor suction side is relatively low, which may cause the appearance of cavitation phenomenon. Simultaneously, there is a small clearance between the rotor blade tip and duct, so that the pressure of pressure side of the rotor blade is higher than that of suction side, which finally accounts for the back-flow and the emergence of tip vortex that are unfavorable for propulsion.

References
