Available online at www.sciencedirect.com

ScienceDirect

International Journal of Naval Architecture and Ocean Engineering 8 (2016) 110−116
http://www.journals.elsevier.com/international-journal-of-naval-architecture-and-ocean-engineering/

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

* Corresponding author.
E-mail address: luluhenry709@hotmail.com (L. Lu).
Peer review under responsibility of Society of Naval Architects of Korea.

http://dx.doi.org/10.1016/j.ijnaoe.2015.10.001

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:

$$\frac{\partial}{\partial x_i}(\rho \overline{u}_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \overline{u}_i) + \frac{\partial}{\partial x_j}(\rho \overline{u}_i \overline{u}_j) = \rho \overline{F}_i - \frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right)\right] \tag{2}$$

where $\overline{p}$ is the average pressure, μ is the molecular viscosity and $\rho \overline{u}_i \overline{u}_j$ 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 μ_t is used to model the turbulent Reynolds stresses.

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

where μ_t 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 $k-\varepsilon$ 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 $k-\varepsilon$ 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_∞ denotes the free stream velocity, n is the blade rotating velocity. The thrust coefficient $K_T = Thrust/(\rho_f n^2 D^4)$ and the torque coefficient $K_Q = Torque/(\rho_f n^2 D^5)$ 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 1
Parameters of the E779A propeller.

Propeller diameter (D)	P/D ratio	Skew angle	Rake	Blade area ratio	Hub diameter (D_H)
227.3 *mm*	1.1	4°48″	4°3″	0.689	45.53 *mm*