Numerical Study of Fillet Surface Effect on the Ducted Performance Using OpenFOAM

Aldias Bahatmaka1,*, Dong Joon kim2 and Hadi Pranggono3

1Interdisciplinary program of Marine Convergence Design, Pukyong National University, 48513 Busan, South Korea
2Naval Architecture and Marine Systems Engineering Department, Pukyong National University, 48513 Busan, South Korea
3Aquaculture Department, Pekalongan University, 51115 Pekalongan, Indonesia
aldis@pukyong.ac.kr

Abstract. In the present study, the effect of fillet surface on the performance of the ducted propeller using Reynolds-Average Navier Stokes (RANS) method is applied. The numerical simulations compare the propeller model with fillet on the surface and without fillet. OpenFOAM is used to simulate the performances and solve the problems in the open water propeller simulation. MRF approach is applied for this simulation. Kaplan model with the 19A nozzle is investigated by comparing the results with experimental data. The effect of fillet surface on the ducted propeller are presented and discussed in this study.

1. Introduction

The recent years, lot of numerical simulation to improve the performances of the ducted propeller. The nozzle of ducted propeller is used to improve the propeller efficiency on the ship. The ducted provides the extra thrust and minimize the cavitation. Other reason to use the ducted propeller to reduce underwater noise or protect the propeller damage. Generally, there are 2 types of ducted propeller. They are an acceleration duct or Kort nozzle and deceleration duct. The efficiency of the duct propeller is therefore greater than the open propeller [1].

The computational fluid dynamics (CFD) method is already growing up to solve the engineering problem and compared with experimental data. Several study in CFD for propellers have been conducted [2-5]. In other studies, Optimization for propeller have been investigated, such as using genetic algorithm in the propeller optimization [6], propeller shape optimization is also investigated [7]. Other references are conducting the improving propeller performances for the ducted propeller [8-12].

In the present study, OpenFOAM is used to simulate the performance of the ducted propeller. OpenFOAM is open-source CFD libraries which has ability to do numerical simulation. Several references have been validated the OpenFOAM simulation and the results were estimated and close to the experimental data [13-15]. Therefore, proving OpenFOAM is a powerful solution to solve the engineer problems, especially for the ducted propeller simulation.

In this research, MRF (Moving Reference Frame) method is applied to do the simulation. MRF is one of the meshing strategies in the numerical simulation. For the propeller simulation, MRF is having robust and closer to the experimental results than the AMI (Arbitrary Mesh interface), and already conducted in the OpenFOAM [16].
The fillet surface is one of important part to make the 3D model for engineer. The propeller design is complex model and should be investigate the effect of fillet surface between the blade propeller and the hub. This research shows the difference between model between fillet and without fillet on the ducted propeller. The result also compared the experimental data, to validate the approach method.

2. Geometry and conditions
The present study used the Kaplan Ka4-70 with the 19 nozzle. the main dimensions of the ducted propeller are listed in table 1.

Table 1. Main dimensions of the ducted propeller

| Parameters                  | Value          |
|-----------------------------|----------------|
| Diameter (D)                | 300.0 cm       |
| Number of blades            | 4              |
| Pitch of ratio              | 1.2            |
| Expanded area ratio         | 0.7            |
| Length of duct (LD)         | 0.5D           |
| Location of propeller       | 0.5LD          |
| Clearance between duct and propeller | 3 mm |

19A is used for the duct type. 19A is mostly used with Kaplan series. The duct section can be seen in figure 1.

Figure 1. The 19A duct section

Figure 2 shows the three dimensional of the ducted propeller. The model is created in Rhinoceros 3D program.

Figure 2. The 3D model of ducted propeller
3. Numerical approach
The numerical method predicts the flow around the propeller. These conservation equations are generally known as the Navier-Stokes equation and they are used in their incompressible form. Turbulence is modelled with the two-equation k-ω SST model. The simulations were conducted in two different methods, i.e., the steady-state computation that is based on the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm and PimpleDyMFoam method, which is an implementation of the PimpleFoam solver that allows the implementation of dynamic meshes.

3.1. Governing equations
In the Reynolds-Average Navier-Stokes equation, a large number of problems are considered in the open water test. Its main objective is to solve the average velocity field (thrust and torque) that depends on the mean flow characteristics. Velocity and pressure field can be expressed as:

\[ u = \bar{u} + u' \]  
\[ p = \bar{p} + p' \]

Where \( \bar{u} \) is value of velocity; \( \bar{p} \) is value of pressure. Then \( u' \) is the fluctuating part of velocity and \( p' \) is the fluctuating part of pressure, it can be seen in Eq. (1) and (2). The time-average continuity equation is substitution from velocity and pressure equation into the Navier-Stokes equation. It can be expressed as:

\[ \frac{\partial \bar{u}}{\partial t} + \nabla.(\bar{u} \bar{u}) - \nabla.(\nu \nabla \bar{u}) = -\nabla \bar{p} + \nabla.(\bar{u}' u') \]

Where \( \bar{u}' u' \) is the Reynold stress tensor in Eq. (3), it depends on the fluctuating variables of turbulence flow. The open water propeller characteristics were then conventionally presented in the form of the thrust and torque coefficient \( K_T \) detailed out in Eq. (4), \( K_Q \) in Eq. (5) denotes the term of the advanced coefficient, and \( J \) is elaborated through Eq. (6).

\[ K_T = \frac{T}{\rho n^2 D^3} \]  
\[ K_Q = \frac{Q}{\rho n^2 D^3} \]  
\[ J = \frac{V_A}{nD} \]

The propeller efficient is expressed by:

\[ \eta_0 = \frac{K_T}{K_Q} \frac{J}{2\pi} \]

Where \( \eta_0 \) and \( \pi \) shown in Eq. (7) denotes the efficiency of the propeller and motion in circle, which value is generally 3.14.

3.2. Boundary condition and method
Moving Reference Frame (MRF) is the steady-state solver employing the SimpleFoam as solver of the flow follows the blade rotation according to the Moving Reference Frame (MRF) method. The MRF method can be seen in figure 3.
The domain consists of an internal rotating cylinder enclosing the propeller and an external stationary cylinder with a radius of 10D. In addition, the inlet uniform boundary condition is located at 10D upstream of the propeller plane, and the outlet that is a constant pressure condition was located at 27D downstream of the propeller.

3.3. Benchmark Study
Validation of the numerical is needed for CFD. Therefore, in this study we compare the numerical simulation using MRF method with the experimental data. The comparison results are listed in table 2.

**Table 2.** Comparison with experimental data

| KT duct | KT duct | KT prop | KT prop | 10Kq | 10Kq | Efficiency | Efficiency |
|---------|---------|---------|---------|------|------|------------|------------|
| Exp.    | Exp.    | Exp.    | Exp.    | 10Kq| 10Kq| Exp.       | Exp.       |
| 0.200   | 0.169   | 0.162   | 0.531   | 0.562| 0.619| 0.655      | 0.275      |
| 0.300   | 0.127   | 0.127   | 0.468   | 0.504| 0.600| 0.636      | 0.372      |
| 0.400   | 0.088   | 0.084   | 0.404   | 0.445| 0.576| 0.606      | 0.450      |
| 0.500   | 0.052   | 0.052   | 0.346   | 0.389| 0.545| 0.572      | 0.505      |
| 0.600   | 0.020   | 0.028   | 0.287   | 0.325| 0.504| 0.526      | 0.544      |
| 0.700   | -0.011  | -0.015  | 0.229   | 0.258| 0.453| 0.474      | 0.561      |
| 0.800   | -0.047  | -0.050  | 0.170   | 0.179| 0.394| 0.408      | 0.545      |

**Figure 3.** The MRF method for propeller simulation [16]

**Figure 4.** Open water diagram, numerical simulation is compared to experimental data
From the Figure 4, it shows the numerical simulation is having close results to experimental data. It means, this method can be applied to produce the propeller performances.

4. Fillet surface on the propeller model
The Fillet surface is one of the method to make the three dimensional model. Mostly, fillet surface is using to complex model, such as propeller model. To make the smooth model is needed the technique and some parameters to produce the best model. In this study, we considered the fillet surface, and investigate the effect of flow around the blade and the hub of the propeller. Then, we compared the 2 model, between model with fillet surface and without fillet surface. For this research, the radius of fillet surface is constant, 1 mm. The detail model can be seen in figure 5 for the original model and figure 6 for modified with fillet surface.

5. Mesh convergence study
After conducted the modelling, starting the simulation with decide the mesh strategies. In this research, meshing is divided to be 3, e.g. Coarse, Medium and Fine Mesh. The difference of the meshes are listed in table 3.

| Results | Coarse (0.5M) | Medium (2M) | Fine (5M) |
|---------|--------------|-------------|-----------|
| \( T \) (N) | 130.870 | 217.070 | 219.922 |
| \( Q \) (Nm) | 6.585 | 9.942 | 14.942 |
Figure 7. Comparison of the meshing strategies
Figure 8. Comparison of thrust for all meshes

Figure 7 shows the contour distribution for all meshes and figure 8 presents the thrust differences for all meshes. From the Figures, it can be seen that Fine Mesh is relatively close to the original model and it is proved by the value of the thrust.

6. Results and discussions

The numerical result shows the difference between both models. The simulation is comparing the force and the torque. From the result, there is the little difference for the performances. In this stage, we calculated on the velocity of J=0.8. The detail results are presented in table 4.

| Results | Original model | Fillet surface | Differences (%) |
|---------|----------------|----------------|-----------------|
| T (N)   | 219.818        | 219.222        | 0.271           |
| Q (Nm)  | 14.988         | 14.942         | 0.315           |

From the table, it can be seen the difference between original model and modified model is just little bit, less than 1% for the force and torque. It does not have much effect to the ducted propeller performance.

The detail velocity distribution can be seen in figure 9 and figure 10 for pressure distribution. From all the figures comparison we can see that there are no striking differences between the original and fillet model for both velocity and pressure distributions.

(a). Velocity distribution of original model
Figure 9. Comparison of the velocity distribution

Figure 10. Comparison of the pressure distribution

7. Conclusions
The study has applied the RANS solver in OpenFOAM to analysis the effect of fillet surface for the Kaplan propeller on the ducted propeller. OpenFOAM can performed the numerical simulation for the
ducted propeller. MRF approach is applied for this simulation and the method can be used and the result is getting close to the experimental data. The simulations present the comparison between original model and modified model in CFD simulation. There is little bit difference effect from both model. The difference is not effecting to performance of the ducted propeller, because it just less than 1% of the differences. We assumed the fillet surface will just effect to cavitation performance, it’s because of the force and torque did not increase too much. Then, this research should be continued for the effect of fillet surface on the cavitation performance of the ducted propeller.

8. Acknowledgments
This paper was supported by the BK21 plus MADEC human resource development group of Pukyong National University, Busan, South Korea.

9. References
[1]. Ghose, J. P. and Gokarn, R 2004 Basic Ship Propulsion. (Allied Publishers Pvt. Limited).
[2]. Bahatmaka, A., Kim, D.-J., Chrismianto, D., J.D. S. & Prabowo, A. R 2017 Numerical investigation on the performance of ducted propeller. MATEC Web Conf. 138, 1–8
[3]. Santhosh Babu, K. P. and Padmanabhan, S 2017 Design of marine propeller blade with different blade sequences analyse the hydro formation under pressure hydrodynamic fill. ARPN J. Eng. Appl. Sci. 12, 4996–5002
[4]. Shen, Z., Wan, D. and Carrica, P. M 2014 RANS Simulations of Free Maneuvers with Moving Rudders and Propellers Using Overset Grids in OpenFOAM
[5]. Kukudzhanov, V. N 2013 Numerical continuum mechanics. 1, xviii + 429
[6]. Bahatmaka, A., Kim, D.-J., Chrismianto, D., Hai, N. & Prabowo, A. R 2017 Optimization of thrust propeller design for an ROV (Remotely Operated Vehicle) consideration by Genetic Algorithms. MATEC Web Conf. 138, 07003
[7]. Joung, T.-H., Sammut, K., He, F. and Lee, S.-K 2012 Shape optimization of an autonomous underwater vehicle with a ducted propeller using computational fluid dynamics analysis. Int. J. Nav. Archit. Ocean Eng. 4, 44–56
[8]. Willemsen, J. C 2013 Improving Potential Flow Predictions for Ducted Propellers. (UNIVERSITY TWENTE).
[9]. Chamanara, M. and Ghassemi 2016 H. Hydrodynamic Characteristics of the Kort-Nozzle Propeller by Different Turbulence Models. 4, 169–172
[10]. Ii, N. F 2014 The Nonlinear Actuator Disk Method as Applied to Open and Ducted Rotors
[11]. Elik, F. C., G Uner, M. and Ekinci, S 2010 An Approach to the Design of Ducted Propeller. Trans. B Mech. Eng. 17, 406–417
[12]. Yongle, D., Baowei, S. and Peng, W. 2015 Numerical investigation of tip clearance effects on the performance of ducted propeller. Int. J. Nav. Archit. Ocean Eng. 7, 795–804
[13]. Lloyd, T. P. P., Turnock, S. R. and Humphrey, V. F. F 2011 Unsteady CFD of a marine current turbine using OpenFOAM with generalised grid interface. Proc. 14th Numer. Towing Tank Symp. 1–6
[14]. Turunen, T., Siikonen, T., Lundberg, J. and Bensow, R 2014 Open-Water Computations of a Marine Propeller Using Openfoam. 11Th World Congr. Comput. Mech. 5Th Eur. Conf. Comput. Mech. 6Th Eur. Conf. Comput. Fluid Dyn. Vols Il - Iv 1123–1134
[15]. Bensow, R. E 2013 Simulation of unsteady propeller blade loads using OpenFOAM. 16th Numer. Towing Tank Symposium
[16]. Bahatmaka, A. and Kim, D 2018 The Influence of Meshing Strategies on the Propeller Simulation by CFD. 4, 78–85