

Hydrodynamic Characteristics of the Kort-Nozzle Propeller by Different Turbulence Models

Mehdi Chamanara, Hassan Ghassemi*

Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Iran

*Corresponding author: gasemi@aut.ac.ir

Abstract Kort-nozzle propellers are used for the tugs, trawlers and other vessels in which are working in heavy conditions. Thrust is generated by propeller and duct. In order to evaluate the propulsive performance of the Kort-nozzle propeller, a Reynolds-Averaged Navier Stokes (RANS) solver is employed. Kort-nozzle propeller is selected by Kort-nozzle propeller with 19A nozzle. The Kort-nozzle propeller is analyzed by three turbulence models of the $k-\varepsilon$ standard, $k-\omega$ SST and RSM. The numerical results are compared with experimental data. Hydrodynamic characteristics are presented and discussed.

Keywords: Kort-nozzle propeller, turbulence model, hydrodynamic analysis

Cite This Article: Mehdi Chamanara, and Hassan Ghassemi, "Hydrodynamic Characteristics of the Kort-Nozzle Propeller by Different Turbulence Models." *American Journal of Mechanical Engineering*, vol. 4, no. 5 (2016): 169-172. doi: 10.12691/ajme-4-5-1.

1. Introduction

In recent year, considerable efforts have been made in order to improve the propulsive efficiency of the propeller on the ships. One of this propulsion is called Kort-nozzle propeller. There are two types of duct, the first type is named as acceleration duct or Kort nozzle and the second type is deceleration duct. The efficiency of the Kort-nozzle propeller is therefore greater than of the open propeller. [1].

Computational fluid dynamics (CFD) have been highly used for the analysis of marine propellers. Due to the complex shape, flow turbulence, flow separation and the possibility of cavitation, the analysis of marine propellers is a difficult task; however some works have been done in the field of Kort-nozzle propeller. For example, Taketani et al. [2] presented the advanced design method of a Kort-nozzle propeller which has high bollard pull performance. A nozzle section shape and a propeller are newly designed by a parametric study of the numerical simulation to have higher performance than a conventional Kort-nozzle propeller. Tadeusz et al. [3] have completed design of Kort-nozzle propeller using the new computer systems. In this paper, the five different ducts performance are compared and the result show that in most cases only the 19A duct was considered and this duct was designed for low speed and high bollard pull conditions. Caldas et al. [4] presented CFD validation of different propeller ducts on open water condition. In this here, a controllable pitch propeller with different geometry and CFD calculation was done using the model RANSE. Celik et al. [5] presented of investigation of optimum duct geometry for a passenger ferry. The optimum duct geometry is investigated and effect of various duct sections on the performance of Kort-nozzle propeller is analyzed. Yu et al.

[6] presented of numerical analysis of Kort-nozzle propeller performance under open water test condition. Krzysztof et al. [7] presented the effect of duct shape on Kort-nozzle propeller thrust performance. The four different ducts are analyzed and a new geometry of the duct was designed as deeply modified geometry of Wartsila-Hr nozzle. Xueming et al. [8] analyzed the hydrodynamic performance of the Kort-nozzle propeller based on combination multi-block hybrid mesh and Reynolds stress model. In this paper, the Reynolds stress model (RSM) and $k-\varepsilon$ standard model are compared and show that the Reynolds stress model have better results than the $k-\varepsilon$ standard model. Recently, Majdfar et al carried out numerically on the ducted propeller using RANSE solver at various conditions [8,9,10,11]. Comparison of the accelerating and decelerating of the ducted propeller is presented by Razaghian & Ghassemi [12].

Various studies have been done on a Kort-nozzle propeller, but analyzing a Kort-nozzle propeller by using different turbulence models and comparing them with each other, studying the effects of propeller position along the duct on hydrodynamic characteristics have been less studied. In this paper, a Kort-nozzle propeller with 19A duct is used for CFD analysis. The Kort-nozzle propeller is analyzed by using three different turbulence models including $k-\varepsilon$ standard, $k-\omega$ SST and RSM and the results are compared with experimental data. Furthermore, the effects of the propeller position along the duct on hydrodynamic characteristics are investigated and the position in which the maximum thrust is produced will be determined.

2. Governing Equations

The governing equations of fluid flow are mass and momentum conservations, as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{V}) = 0 \quad (1)$$

$$\frac{\partial \bar{V}}{\partial t} + \bar{V} \cdot \nabla \bar{V} = -\frac{1}{\rho} \nabla (\bar{p} + \rho g z) + \nu \nabla^2 \bar{V}. \quad (2)$$

In many important engineering flows, we deal with rotating or swivel flows. Rotating flows are happened in turbo machinery, mixing tanks, marine propeller and a variety of other systems. Equations (1) and (2) are solved in stationary coordinate system but sometimes it is useful to be solved in moving coordinate system. When we are looking at moving parts from stationary coordinate system, the flow zones offer an unsteady problem, but with a choice of rotating reference frame around the moving parts, the problem convert to a steady problem. For simple problems, if we do not have any stationary zone, single rotating reference frame (SRF) method can be used. For complex geometries, using the SRF is not possible. In this case, the problem is divided into several zones and two methods of MPM and MRF can be applied. The MPM method, despite being more accurate and the inclusion of interactions between the stationary and moving zones, requires high computational time. In this paper, the MRF method is used, because not only needs low computational time but also have acceptable accuracy.

3. Modeling and Solving

In merchant practice, the ducts most commonly encountered are the 19A and 37 since they are both relatively easy to fabricate and have a number of desirable hydrodynamic features [13]. Kort-nozzle propeller is usually used for the analysis of Kort-nozzle propeller. Therefore, in this paper, a Kort-nozzle propeller with a 19A duct (which is an acceleration duct) is used for the analysis. The main data of Kort-nozzle propeller and 19A duct is shown in Table 1.

Table 1. Main data of Kort-nozzle propeller

Parameter	Value
Propeller diameter	D=300mm
Number of blades	Z=4
Pitch-diameter ratio	P/D=1.2
Expanded area ratio	EAR=0.70
Rotational velocity	n=750 rpm
Length of duct	0.5D
Tip clearance	(0.01D)

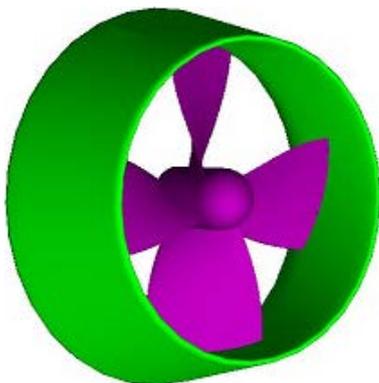


Figure 1. Three-dimensional of the Kort-nozzle propeller

Initial data of 19A duct was taken from Marine Propellers and propulsion book. In this book, no information are provided about the beginning and end data of the duct which must be in the form of curves. The Kort-nozzle propeller and 19A duct are shown in Figure 1.

The computational domain is required to be discretized to convert the partial differential equations into series of algebraic equations. The ICEM software is used for mesh generation. Figure 2 is shown computational domain of the Kort-nozzle propeller. The inlet was considered at a distance of 3D (where D is diameter of the propeller) from mid of the chord of the root section. Outlet is considered at a distance of 6D from same point at downstream. In radial direction, domain was considered up to a distance of 1.5D from the axis of the hub. Figure 3 shows grid over the duct propeller.

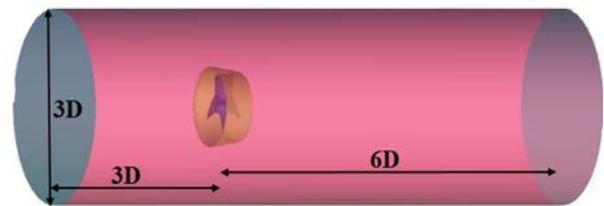


Figure 2. Computational domain

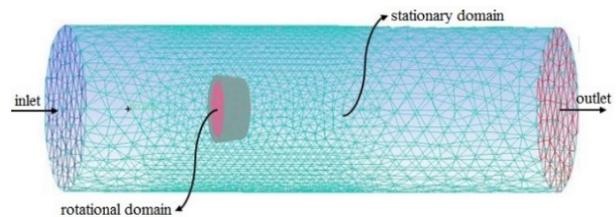


Figure 3. Boundary condition

As it mentioned, here we used MRF method. Therefore the computational domain divided into two zones. The zone around the propeller is a rotational area and the other is a stationary area. The rotational area velocity was assigned 750 rpm so that it is assumed constant. Therefore, according to advance coefficient formula, velocity inlet is variable in different advance coefficient. The inflow and outflow boundaries were set to Velocity Inlet and Pressure Outlet boundary conditions, respectively. The far field boundary was taken as wall.

4. Hydrodynamic Characteristics

The open water characteristics of a propeller are usually given in terms of the advance coefficient J , the thrust coefficient K_T , the torque coefficient K_Q and the open water efficiency η_o . Here, assuming constant rotational speed, the range of advance velocity (input velocity) values corresponding with advance coefficients of 0.2 to 0.8 is achieved. A complete computational solution for the flow was obtained using fluent software. The software estimated thrust and torque for different advance velocity. These were expressed in terms of K_T & K_Q which are defined as follows:

$$\text{Advance Coefficient} = J = \frac{V_A}{nD}$$

$$Efficiency = \eta = \frac{K_T}{K_Q} * \frac{J}{2\pi}$$

$$Torque\ Coefficient = K_Q = \frac{Q}{\rho n^2 D^5} \quad (3)$$

$$Thrust\ Coefficient = K_T = \frac{T}{\rho n^2 D^4}$$

The hydrodynamic characteristics of the Kort-nozzle propeller are analyzed by using the k- ω SST, k- ϵ standard and RSM turbulence models. Then, the results are compared with experimental data, as shown in Figure 4.

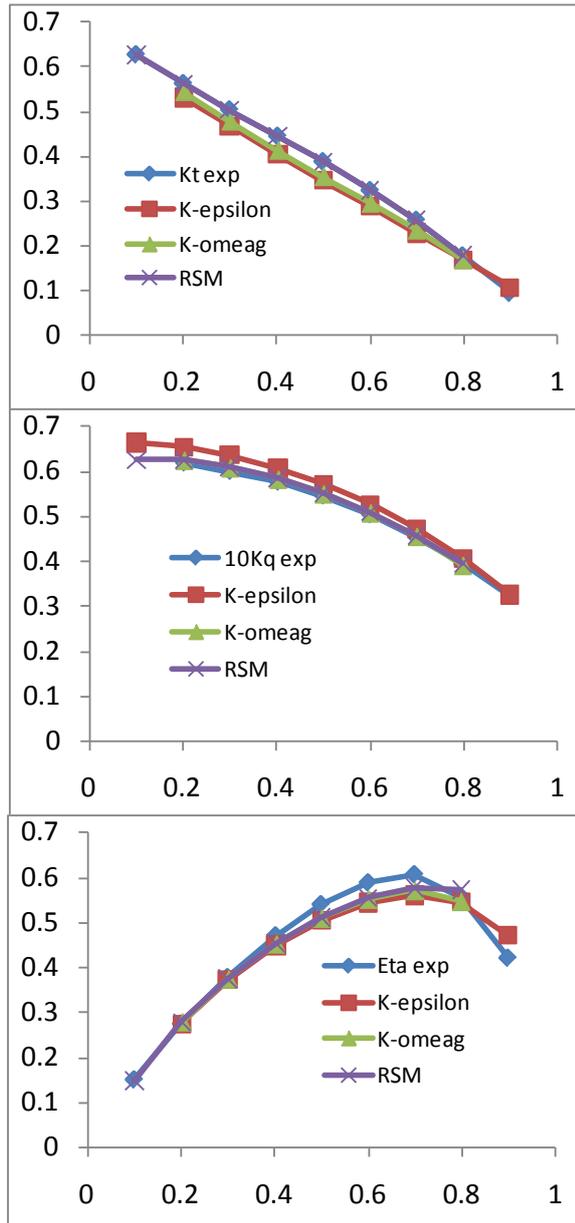


Figure 4. Comparison of open water characteristic with three turbulence models

Table 2 is given the relative error of calculated efficiency with experimental data. As can be seen, the relative error for k- ϵ standard model is smaller than the two other models at low advance coefficients. This means that the k- ϵ standard model has better results in heavy conditions. At high advance coefficients, the error percentage for the k- ω SST and RSM models is smaller

than the k- ϵ standard. Also, there is a little difference between the results of k- ω SST and RSM models at different advance coefficients.

Table 2. Relative error of calculated efficiency with experimental data

Relative error with RSM model	Relative error with k- ω SST model	Relative error with k- ϵ standard model
0.670697	0.187489	0.144466
0.685829	1.083604	1.544547
3.585964	3.873258	4.530412
5.192472	5.524279	6.583071
5.654494	6.055006	7.522543
4.73995	5.67678	7.557598
0.149689	1.805843	2.226791

5. Conclusions

In this study, a Kort-nozzle propeller with 19A duct was numerically analyzed using RANS solver. Based on the results, the following conclusions may be drawn:

- The Kort-nozzle propeller is analyzed with three models of k- ϵ standard, k- ω SST and RSM and the results were compared with experimental data. The error percentages showed that the RSM model has relatively lower error compared to other two models, but it requires a lot of time and high computational cost.
- Comparing the results of these three turbulent models are shown that the k- ϵ standard model has relatively better than the other two models in heavy conditions (lower J), while the k- ω SST model has relatively better in light condition (higher J).

References

- [1] Ghose J.P., Gokarn R.P.; "Basic Ship Propulsion"; Allied Publishers Pvt. Limited, 2004.
- [2] Taketani T., Kimura K., Ishii N., Matsuura M. and Tamura T. "Advanced Design of a ducted propeller with High Bollard Pull Performance", First International Symposium on Marine Propulsors smp'09, Trondheim, Norway, June 2009.
- [3] Tadeusz Koronowicz, Zbigniew Krzemianowski and Teresa Tuskowska; "A complete design of ducted propellers using the new computer system"; Polish Maritime Research, 2(60) Vol 16; pp. 34-39. 2009.
- [4] Caldas Alejandro, Meis Marcos and Sarasquete Adrián; "CFD validation of different propeller ducts on open water condition"; 13th Numerical Towing Tank Symposium; Duisburg/Germany; October 2010.
- [5] Celik F., Donguland A. Arikan Y.; "Investigation of optimum duct geometry for a passenger ferry"; IX HSMV Naples; May 2011.
- [6] Long Yu, Martin Greve, Markus Druckenbrod, Moustafa Abdel-Maksoud; "Numerical analysis of ducted propeller performance under open water test condition"; Springer, J Mar Sci Technol, 24 February 2013.
- [7] Krzysztof Szafran, Oleksandr Shcherbonos, Dariusz Ejmocki ; "Effects of Duct shape on ducted propeller thrust performance"; transactions of the institute of aviation; no. 4 (237), pp. 84-91, Warsaw 2014.
- [8] Xueming He, Hecai Zhao, Xuedong Chen, Zailei Luo, Yannan Miao ; "Hydrodynamic Performance Analysis of the ducted propeller Based on the Combination of Multi-Block Hybrid Mesh and Reynolds Stress Model"; Journal of Flow Control, Measurement & Visualization, 3, 67-74; April 2015 in Sci Res.

- [9] Majdfar S., Ghassemi H., Forouzan H. Hydrodynamic prediction of the ducted propeller by CFD solver, *J. of Mar Sci & Tech*, (Taiwan), accepted.
- [10] Majdfar S., Ghassemi H., Forouzan H. Hydrodynamic effects of the length and angle of the ducted propeller, *journal of ocean, Mechanical and Aerospace-Science and Engineering-*, Vol.25.
- [11] Majdfar S., Ghassemi H.. Calculations of the hydrodynamic characteristics of a ducted propeller operating in oblique flow, *Int. J of Technology*, in Press.
- [12] Razaghian A. H., Ghassemi H., Numerical analysis of the hydrodynamic performance of the accelerating and decelerating ducted propeller, *Scientific Journals of the Maritime University of Szczecin*, will appear Vol. 48, 2016.
- [13] Carlton J S “Marine Propellers and Propulsion”, Third Edition, 2012.