Numerical prediction of cavitation performance for rim driven thruster

[1]Li-wei Zhang; [1]Zi-ru Li; [1,*]Wei He; [1] Ling-yu Zhu

[1]*Wuhan University of Technology, Wuhan, Hubei, China*

Abstract

The open-water characteristics and cavitation behavior of a rim-driven thruster (RDT) are examined by using the Reynolds-averaged Navier-Stokes (RANS) method. The total thrust coefficients and torque coefficients calculated by RANS solver with the SST k-ω turbulence model show relatively better agreement with the experimental data than other two-equation turbulence models. The cavitation behavior of the RDT with a cavitation number of σ_n=3.0 in uniform and non-uniform flow are then separately explored by using the tow-phase RANS method with the SST k-ω turbulence model and the Schnerr-sauer cavitation model, as well as the sliding mesh technique. It can be observed that a sheet cavitation occurs on the suction side of the blades, occupying almost 35% blade area, and kind of cloudy structures are captured to be shed from the sheet cavity for the RDT in uniform flow. Whereas, the sheet cavitation is observed to take up more than 50% blade area for the RDT in a 'Triple Peak' wake field.

Keywords: rim-driven thruster; Cavitation performance; Non-uniform flow; Schnerr-sauer model

Introduction

The rim-driven thruster (RDT) is a relatively new marine propulsion device that uses a motor in its casing to drive a propeller by its rim[1]. The well-known advantages of the rim driven thruster are its low noise emission, vibration generation and space requirement. In addition, cavitation, especially for the annoying tip vortex cavitation, that often occurs when the propeller rotates at a high speed will be alleviated and even eliminated.

Recent years, RDT has been investigated by many researchers. Song et al. (2015) applied Reynolds-averaged Navier-Stokes (RANS) method with moving reference frame (MRF) model to investigate the open-water performance of the RDT by using the steady solver without experimental validation[2]. Huyer et al. (2010) used the commercial software FLUENT to calculate and discuss the influence of the duct geometry, front stator and rotor on the open-water performance of the RDT[3]. Dubas et al. (2015) calculated the influence of the gap between the stator and the blade on the torque in OpenFoam by using the transient solver with experiment validation[4]. It was found that the SST k-ω turbulence model is more suitable to capture the flow separation at low advanced velocities while the RNG k-ε turbulence model is better at high advanced velocities. Yang et al. (2017) analyzed the cavitation performance of the RDT in an uniform flow with a cavitation number of σ_n=3.0 and found that the tip-vortex cavitation that often occurs at the tip of the traditional shaft driven thrusters was not observed around the tip of the blades due to the special configuration of the RDT[5]. However, the propulsion system often operates behind the ship with a non-uniform wake in practical engineering. This would then rationalize further investigation of the cavitation behavior of the RDT in non-uniform flow, which would benefit the subsequent research of the vibration and noise characteristics by analysis of the hydro-acoustic noise field.

The purposes of the current study are to predict the open water performance and study the cavitation behavior of a RDT in uniform and non-uniform flow using computational fluid dynamics method. Firstly, the open-water characteristics of a RDT are calculated by using the RANS method and MRF model. Secondly, the cavitation performance of the RDT is predicted by using the unsteady two-phase RANS solver and the Schnerr-sauer cavitation model together with the sliding mesh technique. Then, a 'Triple Peak' wake field is applied at the inlet boundary for the simulation of the cavitation behavior of the RDT that operates in a non-uniform flow. Finally, the characteristics of the wake field are explored by analysis of the pressure fluctuations at some distance behind the RDT.

Case Description

Rim-driven Thruster Geometry

The rim-driven thruster that is investigated in the current study is composed by a duct and an embedded rotor with seven blades. Part of the geometry data of the RDT is listed in Table 1. The open-water experiment of the RDT was conducted at the towing tank of Wuhan University of Technology. A hub and shaft were fitted to reduce the processing

*Corresponding Author, Wei He: hwcuduca@163.com

difficulties and test unsteadiness. Thus, a hub is added to the three-dimensional RDT model in the numerical simulation of the open-water performance for proper validation against the experimental observations.

Items	Nomenclature	Unit	Value
Diameter	D	[m]	0.2
Design pitch ratio/R=0.7	$P_{0.7}/D$	[-]	-
Area ratio	A_E/A_0	[-]	0.7
Number of blades	Z	[-]	7
Direction of rotation		[-]	right

Table 1. Geometry data of RIM-driven thruster model

Computational Settings

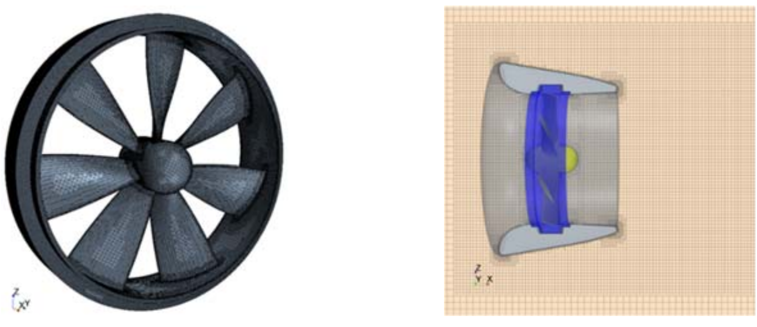
A cylindrical domain with the RDT model along its central axis has been established for the numerical simulation in non-cavitating and cavitating conditions. According to our previous studies[5,7], the radius of the overall cylindrical domain is set to be 4D with an extension of 8D ahead of the thruster's mid plane and 10D behind the thruster's mid plane by considering both of the computation time and resource. As two coordinate systems are involved in the computation of the hydrodynamic performance of the RDT, the whole domain are divided into a rotating region and a static region. The information exchange in the fluid filed is achieved by the interface between the two regions.
For the investigation of the open-water performance of the RDT, the number of revolution of the thruster is fixed to n=1000rpm with advance coefficients varying from 0.1 to 0.5. The density and dynamic viscosity of the water is set to be the water properties at 20 degree. The inlet boundary is set as velocity inlet condition with the inlet velocities corresponding to various advance coefficients. No-slip wall is applied to the overall configuration of the RDT and the outer boundary of the cylindrical domain. The outlet boundary is set as pressure outlet with no assigned pressure. Whereas for the numerical simulation of the cavitation behavior of the RDT, the advance coefficient is set as the design point with a value of J=0.3. And the pressure at the outlet should be specified according to the cavitation number when the cavitation performance of the RDT is examined.

Meshing

Mesh generation is performed using the trimmed mesher, which provides a robust and efficient method of producing high-quality mesh for the blade surface where large curvature variation occurs. Comparison of results calculated by three grids that have been refined substantially with similar topology is conducted to study the grid sensitivity for the calculation of the open-water performance. The detailed information of these three grids are shown in Table 2. The mesh around the rim part and the seven blades is shown in the figure 1 (a). The leading and trailing edge of the blades are refined due to the characteristics of high curvature and small thickness, as shown in figure 1 (b). Owing to the tiny gap between the rim part and the duct part with around 1mm thickness in the model scale, eight prism layers are applied to improve the flow solution near the duct.

Grid	Total cells	Level	Y^+max	Prism layers
G1	2.4 million	Coarse	101.2	5
G2	4.5 million	Medium	76.3	8
G3	8.9 million	Fine	38.6	8

Table 2. Grid features of the rim-driven thruster



(a) Mesh of the RIM and blades (b) The refined mesh region
Figure 1. Details of the medium grid G2 of the RDT

*Corresponding Author, Wei He: hwcuduca@163.com

Results and Discussions

Open-water Characteristics

Validation of the numerical open-water characteristics of the RDT against the experimental data from Wuhan University of Technology has been performed by using the RANS method with the two-equation turbulence models and a moving reference frame. Firstly, the sensitivity of the grid densities is examined on three geometrically similar girds with substantially refined levels (as shown in Table 2). In Figure 2. , the thrust and torque coefficients that are predicted by the three grids are compared with the experimental measurement. It can be observed that the benefit from the refinement of grids in the current study is very small for the grids G2 and G3. Thus, comparison of results that are calculated by different two-equation turbulence models has been made by using the grid in medium level G2 with consideration of accuracy and computational resource.

The total thrust coefficients and torque coefficients calculated by the SST k-ω turbulence model with the advance velocities varying from 0.1 to 0.5 show relatively better agreement with the experimental data than other two-equation turbulence models. Figure 3 shows the open-water characteristics obtained by the medium grid G2 with the SST k-ω turbulence model. The calculated torque coefficients K_Q match well with the experimental measurement with errors less than 2.5%, whereas the thrust coefficients K_T at low advance ratios are underestimated with errors within 6.7%, which are acceptable for the engineering application. By comparison with our previous work[5], the differences between the predicted thrust coefficients and the experimental data decrease around 1% by the increment of the number of the prism layers from five to eight, but with a reduction of total grid number from 4.8 million to 4.5 million. It is suggested that the sufficient resolution in tiny gap between the duct and the rim has more important influence than the total grid number on the prediction of the hydrodynamic performance of the RDT.

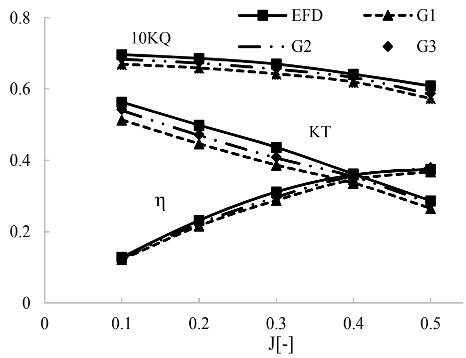


Figure 2. The open-water characteristics of the RDT with different grids

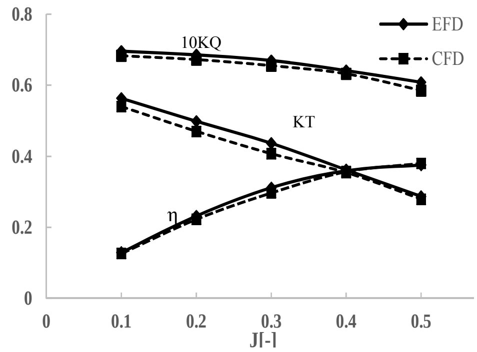


Figure 3. The open-water characteristics of the RDT with the SST k-ω turbulence model by employing grid G2

Cavitation Behavior of the RDT in Uniform Flow

As the numerical simulation of the cavitating flow demands higher grid resolution, the fine grid G3 has been selected to better simulate the cavitation behavior of the RDT in uniform flow and non-uniform flow. In order to simulate the real shaft-less situation as much as possible, the hub is ignored in the cavitation simulation. The cavitation behavior of the RDT at a specific cavitation regime with a cavitation number of σ_n=3.0 in uniform flow is explored by using the tow-phase RANS method with SST k-ω turbulence model and the Schnerr-sauer cavitation model. The operation condition for the current study is at an advance coefficient of J=0.3, which is the design point of the RDT. Transient computations are performed with a time step size of 1.7e-4s corresponding to 1° rotation of the RDT. The propeller rotation is simulated by a rigid body motion with a sliding mesh technique.

Figure 4 shows the iso-surface plots of instantaneous vapor volume fraction of α =0.2 observed in the current study with finer grid and larger number of the prism layer and those observed in our previous work [5]. In both cases, a sheet cavitation has been captured on the suction side of the blades, and none of the tip-vortex cavitation occurs at the blade tip due to the high integration between the rim and the blades. However, a larger extent of the sheet cavitation is observed at the leading edge but covers less area of around 35% of the blades' area. Moreover, kind of cloudy structures are captured to be shed from the sheet cavitation in the current study. When comparing the hydrodynamic

*Corresponding Author, Wei He: hwcuduca@163.com

performance of the RDT under the cavitation condition and the non-cavitation condition in uniform flow, it is found that the cavitating behavior causes the thrust fall down about 9.12%. Whereas a loss of 12.665% of the thrust was predicted by using the same numerical method but with the coarser grid and less number of prism layers[5]. It is supposed that the grid resolution at the gap between the rim part and the blades also has an impact on the simulation of the cavitation behavior of the RDT. However, the influence of the hub on the cavitation performance of the RDT demands further investigation.

(a) current study (no hub) (b) previous work by Tao Yang (2016)

Figure 4. Cavitation pattern of the rim-driven thruster in uniform flow at J=0.3, σn=3.0

Cavitation Behavior of the RDT in Non-uniform Flow

To investigate the cavitation behavior of the RDT in non-uniform, a 'Trible Triple Peak' wake field obtained from Boswell'experiment[6] has been applied in the current study, as shown in figure 5. And the inlet boundary changes to 1/3 D to avoid numerical dissipation while other numerical methods remain the same as those in the uniform flow condition. As we can see from figure 6, a sheet cavitation starts from the leading edge and extends to the trailing edge with kind of cloudy structures tending to shed form the whole sheet. The occupied area and the cavitation pattern on each blade are qualitatively similar as each other but with quantitative differences. It can be observed that the averaged cavitation area covers almost 50% of the blades which may result in larger loss of the thrust and torque by comparison with the hydrodynamic performance predicted when the cavitating RDT operates in uniform flow.

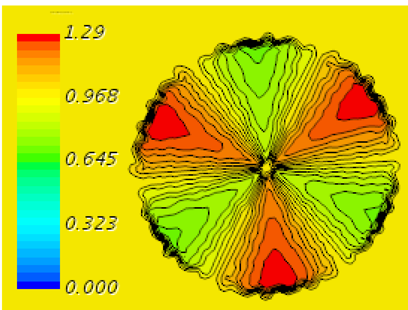


Figure 5. The velocity magnitude of the 'Triple Peak' wake field

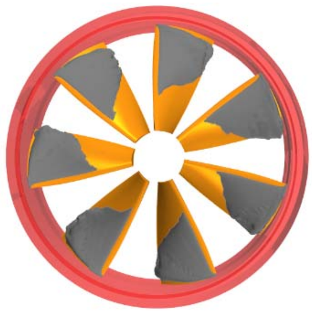


Figure 6. Cavitation pattern in non-uniform flow

Pressure Fluctuations in Non-uniform Flow

Even though in the non-cavitation condition, noise could be generated due to the water displacement by blades and the pressure difference between the suction and pressure sides of the blades of the RDT. When it comes to the cavitation condition, the dynamic behavior of large volume of cavitation and the sudden collapses of complex and various cavitation pattern generate strong pressure fluctuations, which would intensify the vibration and noise and has a close correlation with the erosion risk. In the present study, the pressure fluctuations generated by the RDT in the 'Triple Peak' wake field are examined by monitoring the pressure of four points that are located closely behind the RDT. The specific location of the four points are P1(0.015m,0m,0.1m), P2(0.015m,0m,-0.1m), P3(0.015m,-0.1m,0m), P4(0.015m,0.1m,0m).

For convenience of the analysis, a dimensionless physical quantity K_P is used to describe the pressure signal:

*Corresponding Author, Wei He: hwcuduca@163.com

$$K_P = \frac{P_k}{\rho n^2 D^2} \tag{1}$$

where : ρ is the density of the water, n is the rotation speed, D is the propeller diameter, P_k is the static pressure. The frequencies of the pressure fluctuations are obtained by transforming the pressure signals in the time domain into the frequency domain by Fast Fourier Transformation (FFT), as shown in figure 7. It can be found that strong pressure fluctuations are generated at blade passing frequency and its multiples, which can be related to the dynamic behavior of large volume of the observed sheet cavitation on the suction side of the RDT. The typical pressure fluctuating frequencies vary from the first order frequency of 116.67HZ, to the second order fluctuating frequency around 233HZ with an amplitude decrease of 60%.

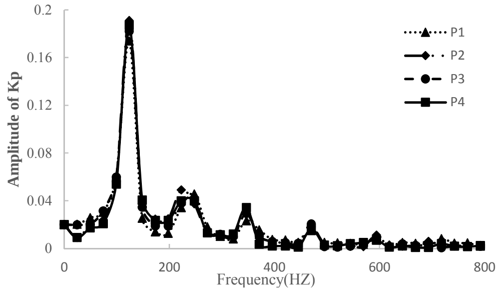


Figure 7. The pressure fluctuations observed at four specific locations in frequency domain

Conclusion

The following conclusions are drawn from the present study:

- The open-water performance of the RDT with the SST k-ω turbulence model by employing the medium grid G2 fairly agrees with the experimental data to within 6.7%. It is suggested that the sufficient resolution in the tiny gap between the duct and the rim has more important influence than the total grid number on the prediction of the hydrodynamic performance of the RDT.
- A sheet cavitation occurs on the suction side of the RDT in the uniform flow with a cavitation number of σ_n=3.0, and kind of cloudy structures are observed to be shed from the sheet cavity. None of the tip-vortex cavitation appears due to the high integration between the rim and the blades.
- The sheet cavitation that captured for the RDT in a ‘Triple Peak’ weak field occupies almost 50% of the blades, resulting in larger loss of the thrust and torque than the cavitating RDT operates in the uniform flow.
- A preliminary study of the pressure fluctuations is performed by examining four points closely behind the RDT. Strong pressure fluctuations that can be related to the dynamic behavior of large sheet cavitation are found to be generated at blade passing frequency and its multiples. Further investigations are recommended for the cavitation behavior of the RDT in complicated practical wake field.

ACKNOWLEDGEMENT

This work is financially supported by the National Natural Science Foundation of China (Project No.51609189, No.51309184), and the work is also supported by the Fundamental Research Funds for the Central University (WUT: 2016IVA010 and 2017IVB008).

References

[1] Tuohy, P., M., Smith, A. C., Husband, M. (2010). *Induction rim-drive for a marine propulsor*. International Conference on Power Electronics, Machines and Drives. (pp.1-6). IET.

[2] Song, B., W., Wang, Y, J., Tian, W, L. (2015). *Open-water performance comparison between hub-type and hubless rim driven thrusters based on CFD method*. Ocean Engineering. 103:55-63.

[3] Huyer, S, A., Dropkin, A. (2010). *Integrated motor/propulsor duct optimization for increased vehicle and propulsor performa nce*. Journal of Fluids Engineering, 133(4), 797-807.

[4] Dubas, A, J., Bressloff, N, W., Sharkh, S, M. (2015). *Numerical modelling of rotor–stator interaction in rim driven thrusters*. Ocean Engineering. 106:281-288.

*Corresponding Author, Wei He: hwcuduca@163.com

[5] Yang T., Li, Z., Chen, K., He, W. (2017). *Prediction of Hydrodynamic Performance for Rim-Driven Thruster*. International Society of Offshore and Polar Engineers. 17-409.
[6] Ye, J., Xiong, Y., Gao , X., Liu, Z. (2013). *Prediction method of low-order blade frequency noise of non-cavitation propeller in time domain*. Journal of Harbin Engineering University. 34(1), 1-6.
[7] Zhang, G, M., Li, Z, R., He, W. (2016). *Numerical simulation of open-water characteristics and cavitation performance for a highly skewed propeller*," 26th Int Offshore and Polar Eng Conf, Greece, ISOPE.

*Corresponding Author, Wei He: hwcuduca@163.com