Numerical investigation of different tip clearances effect on the performance of

Pumpjet Propulsor

*Qin Denghui, †Pan Guang, Huang Qiaogao, Lu Lin and Luo Yang

College of Marine, Northwestern Polytechnical University, Xi'an, China

*Presenting author: 18729869394@163.com †Corresponding author: panguang601@163.com

Abstract

Tip clearance loss is a limitation of the improvement of turbomachinery performance. Previous studies show the tip clearance loss is generated by the leakage flow through the tip clearance, and is roughly in close relation with the gap size. In this study, a pumpjet propulsor with different size of tip clearance ($\delta = 0.2 \text{mm}$, 0.5 mm, 1 mm, 2 mm, 3 mm) has been presented to investigate the influence of the tip clearance on a pumpjet propulsor. This analysis was carried out by solving Reynolds Averaged Navier-Stokes (RANS) method with the commercial Computational Fluid Dynamic (CFD) code CFX14.5, and the SST $k - \omega$ turbulence model is applied. In order to verify the accuracy of numerical simulation method, calculations were carried out with a worldwide employed propeller (the E779A propeller). Simulation results show that the open water efficiency decreases gradually in the same advance coefficient (J) with the increasing of tip clearance. However, the open water efficiency is basically unchanged after the tip clearance is bigger than 2mm. The effects of tip clearance on the tip-separation vortex and the tip-leakage vortex are discussed, the area affected by the tip-separation vortex and the tip-leakage vortex is becoming bigger and bigger as the tip clearance increases. And as the tip clearance increases, the core of tipseparation vortex and tip-leakage vortex are "contact", they transfer to each other. The position of the contact is moved to following edge in the axial direction, and the position is 1/3 of the blade tip away from leading edge in the case δ =3mm. The main effected area of different tip clearance, which is the low pressure area, is mainly focus at the area above 0.9 spanwise of the suction side of rotor blade.

Keywords: Pumpjet Propulsor; Tip clearance; Computational fluid dynamic (CFD). The tip vortex structure.

Introduction

Pumpjet propulsor is a new type of underwater propulsion system, which adopts single-rotor propulsion and decelerating duct. The application of decelerating duct improves the cavitation performance of the propulsion system at a lower velocity.

At present, the research on the characteristics of pump jet propulsion, domestic and international published literature mainly concentrates on the test and numerical calculation of hydrodynamic performance. Ch. Suryanarayana et al[1] make experiment on hydrodynamic performance of the underwater vehicle equipped with pumpjet propulsor. They verify the advantages of the rear stator pumpjet propulsor and indicate that the rear stator can absorb the rotational energy of the rotor and reduce the radial component in the wake, and so as to improve the efficiency of the propulsion. Stefan Ivanell [2] uses computational fluid dynamics method to calculate the hydrodynamic performance of the torpedo with pump jet, and the rationality of the method is verified by comparing with the experimental results. The numerical results show that the stator has contributed about 20% of the thrust. Song Baowei et al.[3] calculate the hydrodynamic performance of a type of pump jet propulsor based on CFD method; using high quality structured grid and using sliding mesh technology. The numerical results and the experimental results are in good agreement. Pan Guang et al [4] carry numerical calculation to the vehicle equipped with a certain type of water pump jet propulsion. The open water performance curve of pump jet propulser is given and it indicates that

the pumpjet propulsor has higher efficiency, and ideal balance performance. The pressure of rotor blades and stator blades in relative height is analyzed. The morphology and the principle of the rotor tip vortex are explained. In the flow of pump jet propulsor gap, Wang Tao et al [5] carry numerical simulation for complex viscous flow field of pumpjet propulsor. By analyzing the local flow field, the influence of the clearance flow on the flow field (including velocity and pressure fields) is revealed. In addition the most of the study about the flow of tip clearance are aimed at the duct propeller or axial flow pump. For example, T. Lee Y. et al [6] study the flow of tip clearance of the duct propeller by solving the three-dimensional RANS equation method. The calculated results are in good agreement with the experimental results. It is shown that the numerical method is feasible for the study of the tip clearance flow. Although the duct propeller and axial flow pump are different with pumpjet propulsor, the results of the duct propeller and axial flow pump research have a good reference to the research of tip clearance flow of pumpjet propulsor.

In this paper, according to the 0.2mm, 0.5mm, 1mm, 2mm, 3mm pumpjet propulsor model, the high quality structured grid is generated based on the block grid coupling technique. By means of numerical simulation, based on the sliding grid technique, the numerical simulation of threedimensional full channel steady turbulent flow is carried out. The open water performance of the pumpjet propulsor with different tip clearances, the influence of the rotor tip-separation vortex and tip-leakage vortex and the rotor blade surface pressure field is analyzed.

Numerical simulation method

Governing equations

For an incompressible and single phase fluid, the governing equations for Reynolds Averaged Navier-Stokes (RANS) can be written as the mass and momentum conservations in the following tensor form:

$$\frac{\partial \rho U_j}{\partial x_j} = 0 \tag{1}$$

$$\frac{\partial(\rho U_i U_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\mu \frac{\partial U_i}{\partial x_j} \right) + \frac{\partial \tau_{ij}}{\partial x_j} - \frac{\partial P}{\partial x_i} + S_M$$
(2)

where i = 1, 2, 3, j = 1, 2, 3, ρ is the fluid density, x_i and x_j are the Cartesian coordinate components. S_M , U_i and U_j are different values depending on different situations. For an inertial frame, S_M equals to zero, U_i and U_j represents the absolute velocity component. For a relative rotating frame, S_M is the sum of Coriolis ($2\omega \times U$) and centrifugal forces ($\omega \times (\omega \times r)$), U_i and U_j represent the relative velocity components. μ is the dynamic viscosity, t is the time, τ_{ij} denotes the Reynolds stresses, and P and U represent the pressure and the time averaged velocity, respectively.

Turbulence model

According to the existing study by Ji et al. [7], the $k-\omega$ shear stress transport (SST) 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. 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.

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. [8] and presented in Tables 1. This propeller has been widely tested for several years and a large number of reliable experimental data are available (see Salvatore et al. [9]). The computational domain and boundary conditions for E779A marine propeller is a 1/4 cylinder passage as shown in Figure 1.

P	ropeller diameter (D_t)	227.3 mm		
	P/D_t ratio	1.1		
	Skew angle	4°48 ["]		
	Rake	4°3 ["]		
	Blade area ratio	0.689		
	Hub diameter (D _H)	45.53 mm		
	Free Slip Wall			
Velocity Inlet	E779A Propeller		Pressure Outlet	

Table 1: Parameters of the E779A propeller

Figure 1: Computational domain and boundary conditions for E779A propeller.

The advance ratio J is defined, respectively, as $J = U_{\infty} / (nD_t)$, where U_{∞} denotes the free stream velocity, n is the blade rotating velocity, p_{out} is the outlet pressure. The thrust coefficient $K_T = Thrust / (\rho_t n^2 D_t^4)$ and torque coefficient $K_0 = Torque / (\rho_t n^2 D_t^5)$ are defined, respectively. The numerical simulations are carried out at three different typical values of advance ratio J = 0.71, 0.77 and 0.83. The numerical results of K_T and K_O at three different advance ratios are compared with the experimental data and summarized in Table 2.

Table 2: Comparison of thrust and torque coefficient with experimental data								
_	Numerical Results		Experimental Results		Errors (%)			
J	K_T	10 <i>K</i> _Q	K_T	10 <i>K</i> _Q	K _T	10 <i>K</i> _Q		
0.71	0.2485	0.4327	0.2474	0.4449	0.44	2.74		
0.77	0.2206	0.3913	0.2184	0.4031	1.01	2.93		
0.83	0.1913	0.3488	0.1888	0.3590	1.32	2.84		

From Table 2 we can see that the numerical prediction results are in good agreement with the experimental results, and the errors of K_T and K_Q are less than 3%. Consequently, it is indicated that the numerical simulation method with the *SST* $k - \omega$ turbulence model is applicable and reliable for pumpjet propulsor flows.

Steady numerical simulation for pumpjet propulsor

Pumpjet propulsor model

Pumpjet propulsor simulation model in this study is shown in Figure 2: the propeller has 11 rotor blades, 9 stator blades. The rotors are in front of the stators, and the rotors rotate clockwise (seen from the front of the model). The diameter of pumpjet propulsor is D = 0.26 m and the length of pumpjet propulsor is L = 0.17 m.



Figure 2: Pumpjet propulsor model

In order to simulate the flow better and get more precise result, two half ellipsoid type flow-guide caps have been added in front and rear of the propulsor model, respectively. In order to study the effects of different tip clearance effect on the performance of pumpjet propulsor, different diameter of the duct has been selected to get different tip clearance. Five models with 0.2mm, 0.5mm, 1mm, 2mm and 3mm tip clearances have been selected. In order to facilitate the discussion, the δ has been defined to represent the tip clearance.

Computational domain and mesh

The computational domain and boundary conditions are shown as Figure 3 and Figure 4. The computational domain is a length of 10*L*, diameter of 5*D* cylinder surrounding the model, whose axis coincides with the symmetry axis of Propulsor model. The inlet is located 3*L* from the front face of Propulsor model, and the outlet is situated 7*L* from the front face of Propulsor model. 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, and the other two domains are stationary domains. The rotor and stator domains are embedded in the external flow field domain. The interaction between the rotor domain and stator domain and the interaction between the rotor domain and external flow field domain are solved by using the sliding mesh method.



Figure 3 The computational domain and boundary conditions



Figure 4 The sliding interface

The computational grid quality directly affects the results of numerical simulations. The structured grid has the advantage of using less memory and is very favorable for the boundary layer calculation. Therefore, the three computational domains are filled with structured grids. Multi-block grid method are used to generate high-quality structured grid by ANSYS ICEM. The grids around PJP adopt H hybrid grids, The PJP surface and propulsor blades are surrounded by O-hexahedral girds. Figure 5 shows the rotor and stator blades surface grids. In addition, in order to accurately capture the phenomenon of tip vortex, the gap is encrypted and the boundary layer is 0.05mm, Figure 6 shows the encrypted mesh between the rotor blades and the duct. The number of entire computational domain grids is approximately 3×10^6 .



Figure 5 the rotor and stator blades surface grids Figure 6 the encrypted mesh between the rotor blades and the duct

Boundary condition

Software ANSYS CFX is applied in numerical simulation. For computational domain boundary conditions, 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 stator blades. The free-slip wall boundary is imposed on the cylinder surface. The averaged static pressure is 0 Pa at the outlet. The interface between the rotor domain and stator domain is set to frozen rotor. The finite volume method is used to discrete control equations and the turbulence model. The pressure and velocity coupling using the

SIMPLEC algorithm and the spatial derivatives are calculated using a second-order upwind algorithm.

Results and discussion

To facilitate the discussion of calculation results, the non-dimensional physical quantities are shown in Table 3.

Table 5. Non-dimensional physical quantities					
physical quantities	definition				
advance coefficient	$J = \frac{v}{n * D}$				
thrust coefficient of rotor	$K_{T_t} = \frac{T_t}{\rho n^2 D^4}$				
the torque coefficient of rotor	$K_{M_t} = \frac{M_t}{\rho n^2 D^5}$				
thrust coefficient of stator and duct	$K_{T_s} = \frac{T_s}{\rho n^2 D^4}$				
the torque coefficient of stator and duct	$K_{M_s} = \frac{M_s}{\rho n^2 D^5}$				
Total thrust coefficient	$K_T = K_{T_t} + K_{T_s}$				
Total torque coefficient	$K_M = K_{M_s}$				
The open water efficiency	$\eta = \frac{J}{2\pi} \frac{K_T}{K_M} \delta$				

Table 3. Non-dimensional physical quar	ntities
--	---------

In the table, v is the far field flow velocity; n is rotor speed (r/s); D is the diameter of the rotor; ρ is the fluid density; T_t is the thrust of rotor; T_s is the thrust of stator and duct; M_t is the torque of rotor and M_t and M_s is the torque of stator and duct.

Different tip clearances effect on the open water performance of PJP

In the case of $\delta = 3mm$, maintain the velocity of inlet equal to $25.72ms^{-1}$ and change *n* from 2400rps to 4200rps to obtain different advance ratios. Figure 7 shows the thrust and torque coefficient and open water efficiency curves.



Figure 7 Thrust and torque coefficient and open water efficiency curves

Maintain the velocity of inlet equal to $25.72ms^{-1}$ and calculate models with 0.2mm, 0.5mm, 1mm, 2mm and 3mm tip clearance. Figure 10 shows the open water efficiency curve of the five models.



Figure 8 the open water efficiency curve of different tip clearance models

It can be seen from Figure 8: The rotor provides the main thrust because the rotor has much more thrust than stator and duct system; As J increases in the calculation range, the thrust and torque of rotor and the stator and duct system are gradually reduced; As advance coefficient increases in the calculation range, a linear relationship between thrust coefficient of the stator and duct system and advance coefficient. As J increases, the thrust coefficient of the stator and duct system change from trust to resistance; The torque coefficient of rotor and the stator and duct system are close to each other in the calculation range, and the maximum relative error is only 5.86% when advance coefficient is 2.53. It indicates that the PJP used in this study have a ideal balance performance. As J increases in the calculation range, the open water efficiency increased first and then decreased. The PJP has a maximum open water efficiency about 71.5% when J is 1.9 In the case δ =0.2mm.

tip clearance increases, the open water efficiency decreases gradually in the same J. The open water efficiency is basically unchanged after the tip clearance is bigger than 2mm.

Different tip clearances effect on the tip vortex structure of PJP

You et al. [10] found that the tip vortex structure of ducted propeller is formed by three parts: the tip-separation vortex, the tip-leakage vortex and the induced vortex. The tip-leakage vortex is caused by the pressure different between the pressure and the suction side. The tip-separation vortex is formed due to flow separation underneath the blade tip. The induced vortex is generated by the tip-leakage vortex. Although the tip vortex structure of ducted propeller may be different with PJP, the research conclusion has a great impact on PJP. Because the strength and the influence area of induced vortex are small, the effect of different tip clearance on the tip-separation vortex and the tip-leakage vortex has been mainly analyzed. Figure 11(a) shows the vortex core of rotor blade in the case δ =3mm using the λ_2 vortex-identification (Jeong and Hussain, [11]). Figure 9(b) shows the flow streamlines near the rotor blade tip. The pressure contours of pressure side and rotor suction side of rotor in the case of δ =3mm are illustrated in Figure 10 (a) and (b).



(a) the vortex core of rotor blade (b) the flow streamlines near the rotor blade tip Figure 9 In the case $\delta=3$ mm



Figure 10 the pressure contours of rotor blade in the case δ =3mm

It can be seen from figure 9(a) that the rotor tip-separation vortex is caused by the flow separation at the leading edge of the rotor blade tip. The rotor tip-separation vortex spreads in the axial direction along the intersecting line of the rotor blade tip and pressure side of the rotor blade, leaves the trailing edge of rotor blade tip, and spreads to the stator passage finally. The tip-separation vortex moves toward to the intersecting line of the rotor blade tip and suction side in the circumferential direction with the spread of the vortex in the axial direction.

As shown in Figure 9(a), the rotor tip-leakage vortex is formed at the blade tip of the suction surface of the rotor blade. It can be seen from Figure 10 that there is obvious area of low pressure on the tip of suction side of the rotor blade near the leading edge. Simultaneously, obvious area of high pressure is formed on the tip of pressure side of the rotor blade near the leading edge. The fluid flow is sucked to the low pressure area of the suction side due to the pressure difference, which causes appearance of the rotor tip-leakage vortex. The tip-separation vortex left the suction side of the rotor blade and moved toward to mid-passage with the spread of the vortex in the axial direction. By Figure 9(b) can be seen that the rotor tip-separation vortex and tip-leakage vortex are not completely independent. A portion of the fluid separates from tip-separation vortex and integrates into tip-separation vortex. The low pressure center of vortex is also called vortex core. Figure 9 shows that the tip-separation vortex core and tip-leakage vortex core are "connected", they transfer to each other.

The vortex core of rotor blade in the case δ =0.2mm; δ =1mm and δ =3mm when J=1.9 are illustrated in Figure 11.



The flow streamlines near the rotor blade tip in the case $\delta = 0.2$ mm; $\delta = 1$ mm and $\delta = 3$ mm when J=1.9 are illustrated in Figure 12.



Figure 12 the flow streamlines near the rotor blade tip of different tip clearance

As shown in Figure 11 that the tip-separation vortex spreads toward the suction surface as the tip clearance increases, and the affected area is becoming more and more large. The tip-separation vortex almost covers the whole area of the tip of rotor blade in the case of δ =3mm. As for tip-

leakage vortex, it can be seen from figure 12 that as the tip clearance increases, the affected area of tip-leakage vortex is more and more large too. The affected area is only focus on the area near the leading edge of the rotor blade in the case of δ =0.2mm, but the tip-leakage vortex almost affects the entire rotor passage in the case of δ =3mm. Moreover, the distance between the tip-leakage vortex core and the suction side is larger with the increasing of the tip clearance, and the tip-leakage vortex core has moved to about 1/2 in the middle of the passage in the case δ =3mm. Last but not least, as the tip clearance increases, the core of tip-separation vortex and tip-leakage vortex are "connected ", they transfer to each other. The position of the "connected" is moved to trailing edge in the axial direction, and the position is 1/3 of the blade tip away from leading edge.

Different tip clearance effect on the pressure field of Rotor blade

Figure 13 and Figure 14 show the pressure contours on the pressure side and suction side of the rotor blade with different tip clearance:



By Figure 14 can be seen that the main effected area of different tip clearance is mainly focus at the area above 0.9 spanwise of the suction side of rotor blade, and the effect of the pressure side is not very obvious.

The blade tip loading at constant span of 0.98 is illustrated in Figure 15.



Figure 15 The blade tip loading at constant span of 0.98.

As shown in Figure 15, as the tip clearance increases, the low pressure area appears in the area above 0.9 spanwise of the suction side of rotor blade. The low pressure area gradually moved from the leading edge to the trailing edge in the axial direction, and the effected area gradually increases. The lowest Cp appeares in the axial position at about 30% streamwise, and the low pressure zone affects the area from streamwise 10% to streamwise 50% of rotor blade at constant span of 0.98.

Conclusions

In this study, a pumpjet propulsor with different size of tip clearance($\delta = 0.2 \text{ mm} \\ 0.5 \text{ mm} \\ 1 \text{ mm} \\ 2 \text{ mm} \\ 3 \text{ mm}$) has been presented to investigate the influence of the tip clearance to pumpjet propulsor. This analysis was carried out with RANS method, and the *SST* $k - \omega$ turbulence model is applied. In order to verify the accuracy of numerical simulation method, calculations were carried out with a worldwide employed propeller (the E779A propeller). It is indicated that the numerical simulation method with the *SST* $k - \omega$ turbulence model is applicable and reliable for PJP flows. The influences of the clearance on pumpjet propulsor are reflected in five aspects mainly.

1)As tip clearance increases, the open water efficiency decreases gradually in the same J. The open water efficiency is basically unchanged after the tip clearance is bigger than 2mm.

2) The tip-separation vortex spreads toward the suction surface as the tip clearance increases, and the affected area is becoming bigger and bigger. The tip-separation vortex almost covers the whole area of the tip of rotor blade in the case of δ =3mm.

3) The rotor tip-leakage vortex is formed at the blade tip of the suction surface of the rotor blade and left the suction side of the rotor blade and moved toward to mid-passage with the spread of the vortex in the axial direction. Moreover, the distance between the tip-leakage vortex core and the suction side is larger with the increasing of the tip clearance, and the tip-leakage vortex core has moved to about 1/2 in the middle of the passage in the case $\delta=3$ mm.

4) As the tip clearance increases, the core of tip-separation vortex and tip-leakage vortex are "contact", they transfer to each other. The position of the contact is moved to following edge in the axial direction, and the position is 1/3 of the blade tip away from leading edge in the case δ =3mm.

5) The main effected area of different tip clearance, which is the low pressure area, is mainly focus at the area above 0.9 spanwise of the suction side of rotor blade, the effect of the pressure side is not very obvious.

6)As the tip clearance increases, the low pressure area gradually moved from the leading edge to the following edge in the axial direction, and the effected region gradually increases. The lowest point appeared in the axial position at about 30% streamwise, and the low pressure zone affects the area from streamwise 10% to streamwise 50% of rotor blade.

References

[1] Suryanarayana Ch, Satyanarayana B, Ramji K, et al(2010). Experimental evaluation of pumpjet propulsor for an axisymmetric body in wind tunnel. *International Journal of Naval Architechture and Ocean Engineering*, 2: 24-33.

[2] Ivanell S. (2001) Hydrodynamic simulation of a torpedo with pump jet propulsion system. Stockholm: *Royal Institute of Technology*.

[3] Liu, Z., Song B., Huang Q., (2010) Simulation method of hydraulic performance of pump jet propulsion system based on CFD Technology28(5): 724-729.

[4] Pan, G., Hu, B., Wang, P., Yang, Z., and Wang, Y. (2013), "Numerical Simulation of Steady Hydrodynamic Performance of a Pump-jet Propulsor," *Journal of Shanghai Jiao Tong University*, Vol. 47, 932-937.

[5] Wang, T., Zhou, L., (2004), "Study on numerical simulation and mechanism of interaction with the mainstream flow pump jet clearance" Proceedings of the conference on hydrodynamics of ships, 212-223.

[6] Y. T. Lee, C. Hah and J. Loellbach. (2003) Flow Analyses in a Single-Stage Propulsion Pump.J. Turbomach, 118(2): 240-248.

[7] Ji, B., Luo, X., WU, Y., Liu, S., Xu, H., and Oshima, A. (2010), "Numerical Investigation of Unsteady Cavitating Turbulent Flow around a Full Scale Marine Propeller," Proceedings of the 9th International Conference on Hydrodynamics, October 11-15, Shanghai, China.

[8] Subhas, S., Saji, V.F., Ramakrishna, S., and Das, N.H. (2012), "CFD Analysis of a Propeller Flow and Cavitation," International Journal of Computer Applications, Vol. 55, pp 26-33.

[9] Salvatore, F., Testa, C., and Greco, L. (2003), "A Viscous/Inviscid Coupled Formulation for Unsteady Sheet Cavitation Modelling of Marine Propellers," Fifth International Symposium on Cavitation (CAV2003), November 1–4, Osaka, Japan.

Suryanarayana, Ch., Satyanarayana, B., and Ramji, K. (2010), "Performance Evaluation of an Underwater Body and Pumpjet by Model Testing in Cavitation Tunnel," International Journal of Naval Architecture and Ocean Engineering, Vol. 2, pp 57-67.

[10] You, D., Wang, M., Moin, P. and Mittal, R., 2007. Large-eddy simulation analysis of mechanisms for viscous losses in a turbomachinery tip-clearance flow. *Journal of Fluid Mechanics*, 586, .177-204.

[11] Jeong, J. and Hussain, F., 1995. On the identification of a vortex. Journal of fluid mechanics, 285, .69-94.