Journal of Ocean, Mechanical and Aerospace
-Science and Engineering-, Vol.25

November 30, 2015

Hydrodynamic Effects of the Length and Angle of the Ducted Propeller

Sohrab Majdfar[a], Hassan Ghassemi,[b,*], and Hamid Forouzan[c]

[a,c)] *Department of Marine Engineering, Imam Khomeini University, Nowshahr,, Iran.*
[b)] *Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Iran.*

*Corresponding author: gasemi@aut.ac.ir

Paper History

Received: 14-November-2015
Received in revised form: 27-November-2015
Accepted: 30-November-2015

ABSTRACT

Ducted propellers used in many vessels, especially fishing vessels, trawlers and submarine which provide the higher efficiency. In this article, the effects of the duct length and duct angle are investigated on the hydrodynamic performance. First, did the modeling of duct 19A. The Kaplan propeller performance with nozzle 19A by turbulence model of SST-K-ω analyzed and validated with experimental results that indicate acceptable accuracy. Finally, by changing the nozzle at a rate of 10% and 20% of the original length of the nozzle and also change the angle of the nozzle, analyzed the effects of the changes made. The Kaplan propeller with 19A nozzle is selected for case study. A Reynolds Average Navier-Stokes (RANS) turbulence model of the SST-K-ω employed for the present calculations. Numerical results are included pressure distribution, hydrodynamic characteristics and velocity behind the propeller at various geometry and physical conditions. Comparisons of the results are shown with acceptable agreement by the experimental data. It is concluded that the position of the propeller and increasing the duct angle inside the duct may be limited.

KEY WORDS: *Ducted propeller, RANS, Hydrodynamic Characteristics.*

NOMENCLATURE

API	American Petroleum Institute
ΔT	Temperature Difference in and out
F_T	Thermal Expansion
L_A	Anchor Length
ΔL	Expansion
F_P	Pressure Force
F_F	Friction Force
ε_{sd}	Design Compressive Strain
ε_c	Critical Strain

1.0 INTRODUCTION

Ducted propellers consist of a combination of two principal components. The first is an annular wing which can be either symmetric with respect to the rotation axis or asymmetric to accommodate for the wake flow field variations. The second component, i.e. the propeller, differs from a typical open propeller because it has to be designed taking into account the mutual interaction between the duct and the rotor. In general, there are two main types of ducts: the accelerating (also called the Kort Nozzle) and the decelerating duct (also referred to as pumpjet that Stipa [1] and Kort [2] experimentally proved the increase of the efficiency which can be obtained by ducting the propeller with an accelerating nozzle.

The ability to accurately predict the thrust and torque of a ducted propeller in open-water conditions is very important for a calculation method used in the design stage. The RANSE methods have been progressively introduced for the calculation of ducted propeller systems, meeting considerable success in predicting open-water characteristics for the well-known Ka-series (Sanchez-Caja et al [3]), (Abdel-Maksoud & Jeinke [4]) and (Krasilnikov et al [5]). However, due to their relative complexity and time requirements, they are not yet routinely used in the design process, which is often still based on the use of inviscid flow methods. Krasilnikov studied mesh generation techniques for the Analysis of ducted propellers using a commercial RANSE solver and its application to scale effect.

Various numerical methods based on inviscid (potential) flow theory have been proposed for the analysis of ducted propellers. For example: Kerwin et al [6] used combination of a panel method, also known as boundary element method (BEM), to model the duct with a vortex lattice method for the propeller, and Lee and Kinnas [7] used a panel method for the complete ducted propeller system operating in unsteady flow conditions including blade sheet cavitations. Both methods applied a transpiration velocity model for the gap flow between propeller blade tip and duct inner surface, and analyzed duct with a sharp trailing edge. The sources indicate that the use of non-viscous flow model to ducted propeller, with its many benefits, there may be some serious limitations in the areas of flow where viscosity effects cannot be ignored to meet and should be modeled for the correct prediction of thrust and torque of ducted propeller. One of such region concerns the gap flow, which has a strong influence on the propeller and duct circulation distribution, and therefore, on the distribution of loading between propeller and duct, as studied in detail by Baltazar & Falcão [8]. In addition, there may be a considerable interaction between the vortices shed from the propeller blade tips and the boundary layer developing on the duct inner side, as found in the works of Krasilnikov et al and Rijpkema & Vaz [9]. This effect has not been studied before with potential flow methods and its importance is therefore unknown. Ducted propeller was also done in the field of design. Bobo et al [10] design of ducted propeller and model tests of a fishing research vessel for M.Cies Shipyards. Hughes [11] and Moon et al [12] presented a specific method to model the flow between the inner plate of nozzle and propeller tip. Falco and Campos [13] studied on the calculation of ducted propeller performance in axisymmetric flows. Hoekstra [14] presented a RANS-based analysis tool for ducted propeller systems in open water conditions. Zondervan, Hoekstra and Holtrop [15] researched on the flow analysis, design and model testing of ducted propellers. Gu & Kinnas [16] modeled the flow around a ducted propeller by a vortex lattice and finite volume methods. Haimov et al [17] research on ducted propellers as better propulsion of ship by calculations and practices. Baltazar et al [18] studied on open water thrust and torque predictions of a ducted propeller system with a panel method. An experimental and numerical study on wake vortex noise of a low speed propeller fan carried out by Sasaki et al [19]. A series work based on both potential method and RANSE solver for the whole geometry have been done for a multi-component linear jet optimization by Abdel- Maksoud et al [19] and Steden M et al [20]. In this study, we are trying to show that there are some numerical analysis software to predict and investigate of propellers, and with validation of one of them, the effects of changes on duct want to study on ducted propeller.

2.0 GEOMETRIC MODELING

The most common propeller for ducted propellers is Kaplan type. The Ka 4-70 propeller comes from the famous Wageningen propeller series. It is a traditional ducted propeller that has a large chord at the tip. For all results in this paper the Kaplan propeller with a P/D ratio of 0.8 is used. Geometric modeling of Kaplan propeller is done by Propcad and Solidworks software's that Kaplan geometric data and Nozzle is shown in Table 1.

Table 1: Kaplan geometric parameters and nozzle characteristics

Parameter	Value
Prop. Dia.	Dp=300mm
Number of blades	Z=4
Pitch ratio	P/D=0.8
Expanded Area Ratio	EAR=0.70
Nozzle length	L=0.5DP
Nozzle type	19A

The 19A and 37 nozzles are the most common type of nozzles due to the favorable hydrodynamic properties. In this article the 19A nozzle applied which is an accelerator nozzle. The nozzle length is equal to half of propeller diameter and the distance between the propeller tip and the inner surface of the nozzle is equal to one percent of propeller diameter (3 mm).

We found the nozzle data and made it in the Solidworks that shown in Figure 1. Then assembled duct and propeller are shown in Figure 2.

3.0 MESH GENERATION AND BOUNDARY CONDITIONS

After Modeling of ducted propeller and domains, divided it into 4 pieces and applied one piece with smaller cells because of higher accuracy in calculations.

The computational domain consists of an internal rotating cylinder containing the propeller and an external stationary cylinder with radius 1.5D. The inlet uniform boundary condition is located at 3D upstream of the propeller plane and the constant pressure condition is imposed 6.5D downstream that shown in Figure 3. At the inlet of the cylinder the velocity is prescribed and at the outlet the pressure. For a thrust producing operating condition of the propeller, the fluid through the duct is accelerated. Then ICEM meshing tools applied.

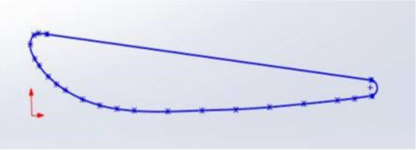


Figure 1: Section of duct in Solidworks

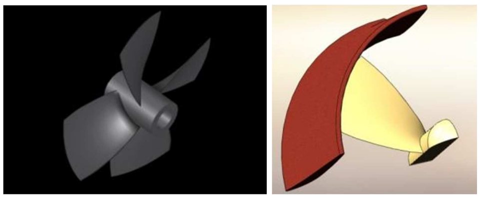

A B

Figure 2: A) 3D model of Kaplan propeller B) ducted propeller Assembled in Solidworks

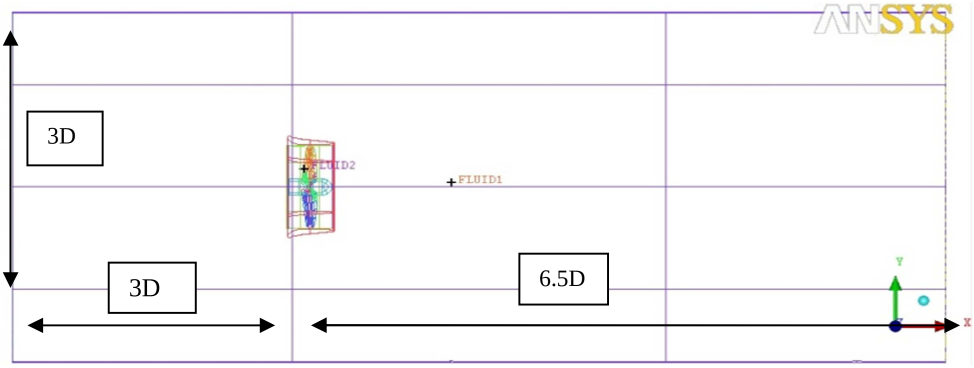


Figure 3: Computational domains dimensions

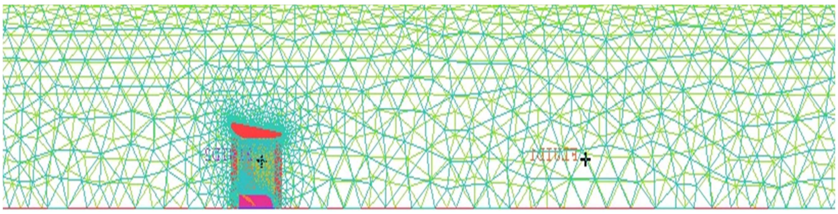
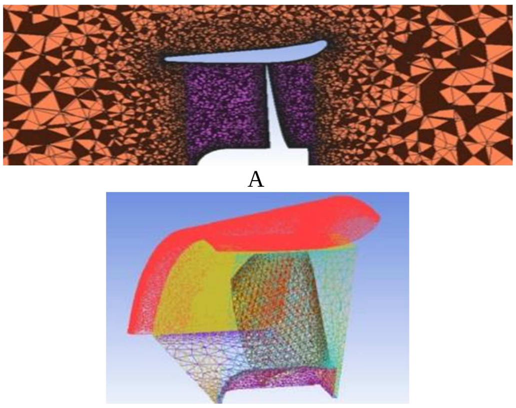


Figure 4: Division of calculation domain with mesh

In this analysis, the rotational velocity of the propeller is imposed by a moving reference frame (MRF) applied to the inner region of the domain because of low time in computation and acceptable accuracy in simulation. All domains divided into two sections: 1-main domain that is stationary domain with larger mesh 2-rotating domain with small mesh around propeller that shown in Figure 4.

The generated mesh size grows outwards with ratio 1.2 then defined boundary conditions that include inlet, outlet, rotating domain, open water, propeller and duct that Figure 5 shown meshes near of propeller and duct. First mesh with 1 million mesh used for model then smaller mesh used with 1.4, 1.5 and 1.7 millions cells and the results at advanced ratio of 0.4 compared. Comparison of results showed that minimum number of cells for this model is 1.4 millions and Figure 6 showed independence of results from meshes.



A

B

Figure 5: A, B) Mesh cells on the propeller and duct

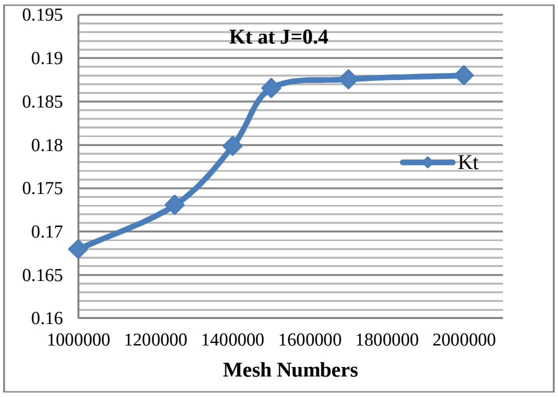


Figure 6: Independence of results from meshes

4.0 SOLVER SETTINGS

The CFD code applied is ANSYS CFX v.14. The Reynolds Averaged Navier-Stokes equations are solved numerically by a finite volume technique. High Resolution method used to discrete equations and first order method used for investigate of turbulence. SST model selected for turbulence because applied in most of the articles due to higher accuracy. 3000 iterations selected in determining the number of iteration to achieve convergence and the remaining amount is considered 0.0001.

5.0 VALIDATION

After completion of solver module, numerical results compared with experimental results to validate software. The experimental results of model tests normally present values of K_T, K_Q and

Efficiency plotted as a function of advance coefficient J for a fixed pitch ducted propulsor as shown in Marine Propellers and Propulsion book [21]. The software outputs were thrust coefficient, torque coefficient and efficiency that obtained by thrust and torque of propeller and duct. The equations 1-4 have shown method to calculate hydrodynamics performance of propeller in different advance coefficients.

$$advanced\ coefficient: \qquad J = \frac{V_A}{nD} \qquad (1)$$

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

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

$$Efficiency: \qquad \eta = \frac{K_T}{K_Q} \times \frac{J}{2\pi} \qquad (4)$$

where V_A is advanced velocity, n is angular velocity, D is Propeller Diameter, T is total thrust, ρ is water density and Q is total torque. Comparison of the numerical and experimental data is shown in Fig. 7. The relative error is about less than 10%. Also the pressure contours on front and back of propeller at advanced ratio 0.3 is shown in Fig. 8. The blade tip is located where the pressure lines converge. On the suction side of the blade tip a low pressure area can be observed.

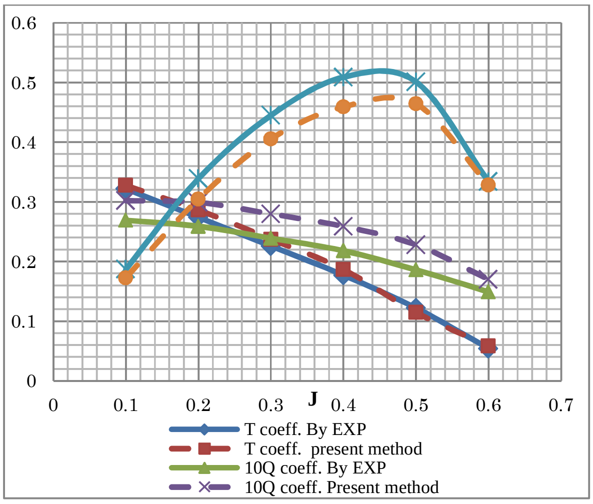
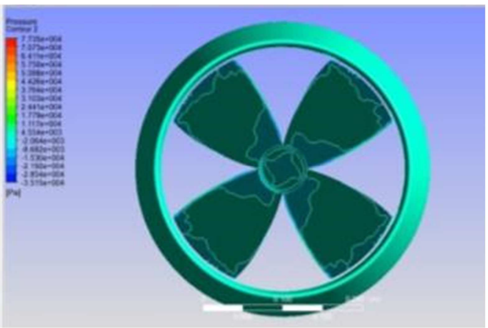
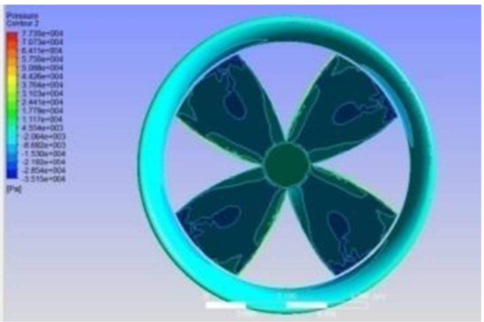


Figure 7: Comparison of the numerical and experimental hydrodynamics characteristics of ducted propeller



A



B

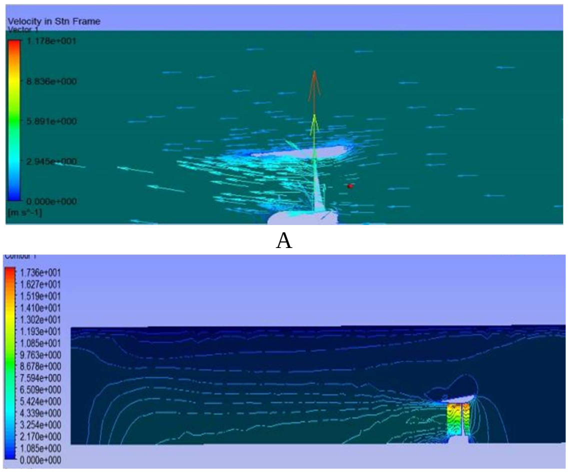
Figure 8: A: Pressure contours on back and B: face of ducted propeller at J=0.3

6.0 RESULTS

6.1 Increase the Nozzle Length

After validation of the software, 3D models of duct with 10 and 20 percent of first duct length created in Solidworks. Then save it with IGES format and import to ICEM for meshing with same settings of the original model. Then specify boundaries settings of model in Ansys CFX-pre and run solver to export the results. It should be noted that all nozzle types are 19A.

To examine the effect of increasing the length of the nozzle, numerical results obtained for the first model. Fig. 10 illustrates comparison of ducted propeller with increase of 10%duct length and standard length. In all advanced ratios, increasing of length showed higher efficiency about 2% and it shows positive effect on hydrodynamics characteristics of ducted propeller. Fig. 9A show the velocity vectors in case of increased length 10% at J=0.3 and Fig. 9B showed velocity contours. Fig. 11 illustrates pressure distribution on blade at various radiuses. The sudden pressure jump at the blade tip (x=0) is clearly visible. Some oscillation showed at leading edge but at chord length showed uniform distribution of pressure on blade. Fig. 12 showed pressure distribution on duct with increasing of length 10%.



A

B

Figure 9: A: Velocity vectors B: Velocity contours in 10% increase length at J=0.3

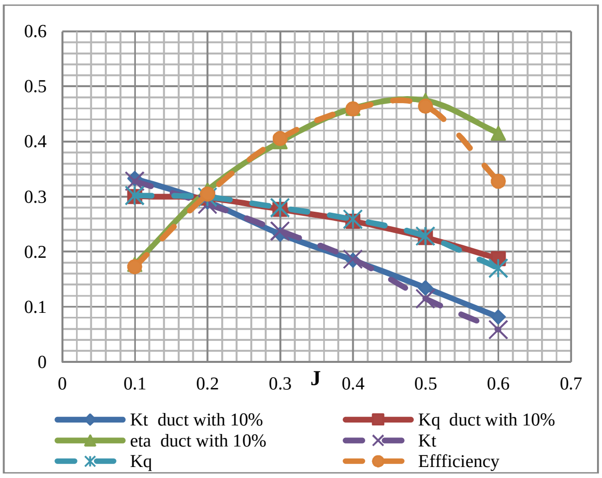


Figure 10: Comparison of the hydrodynamics characteristics of ducted propeller

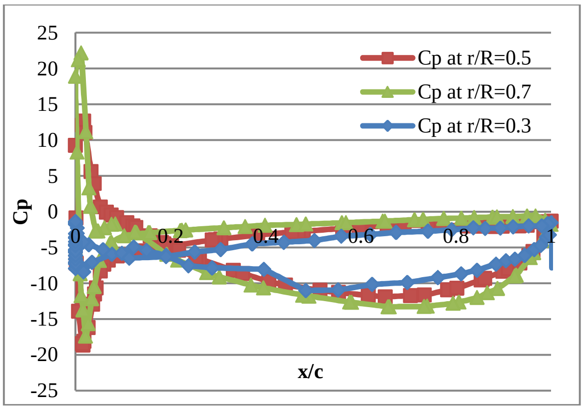


Figure 11: Pressure distributions on propeller at different radiuses in propeller with 10% increasing of duct length

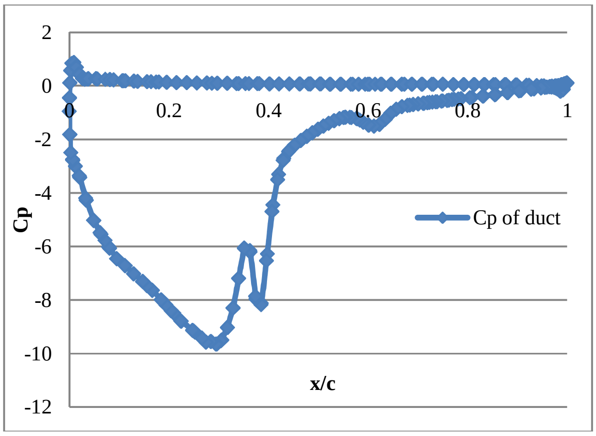
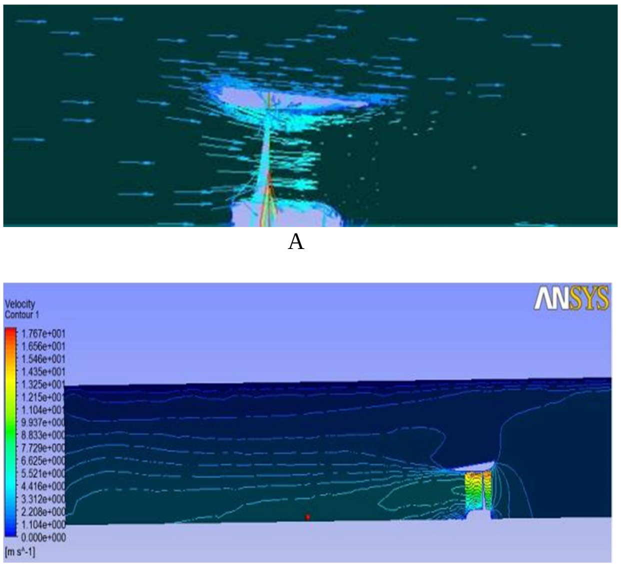


Figure 12: Pressure distributions on duct with 10% increasing of duct length

The numerical results obtained from increasing duct length 20% compared with first model have shown in Fig. 14. As can be seen, increase 20% length of duct had negative effect on the performance of propeller and nozzle, which reduces the coefficient values of thrust, torque and efficiency in comparison with increase of 10% of length. In this case study, increase of length caused to increase about 1.6% of efficiency. Fig. 13A and 13B show velocity vectors and contours in case of 20% increase of length at advanced ratio 0.3 that show decrease in values.



B

Figure 13: A: Velocity vectors and B: Velocity contours in 20% increase length at J=0.3

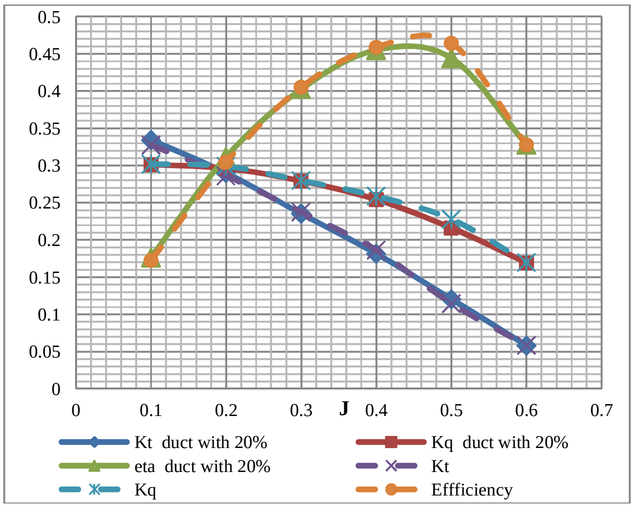
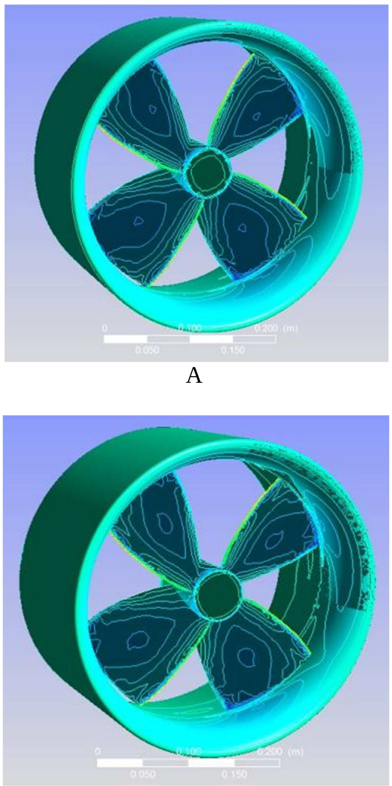


Figure 14: Comparison of the numerical and experimental hydrodynamics characteristics of ducted propeller



A

B

Figure 15: A: Pressure contours in ducted propeller with 10% increase of length B: pressure contours in ducted propeller with 20% increase of length

Fig. 15 shows the difference between pressure contours distribution on front side of propeller and nozzle in cases of increasing in duct length. Lower pressure is observed in model with 20% increase of length. Also Fig. 16 shows uniform pressure distribution on blade and some oscillation at leading edge but the amount of pressure has small decrease in compared with previous case study. Fig. 17 shows pressure distribution on duct with increase 10% of length that has less oscillation.

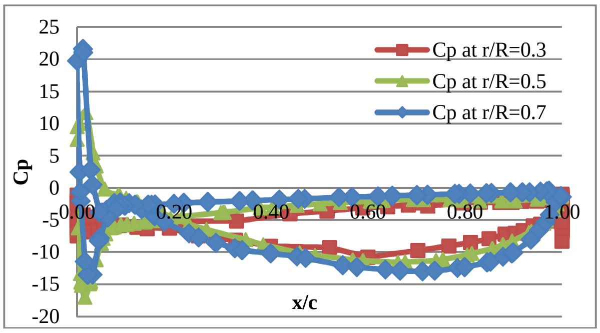


Figure 16: Pressure distributions on propeller at different radiuses in propeller with 20% increasing of duct length

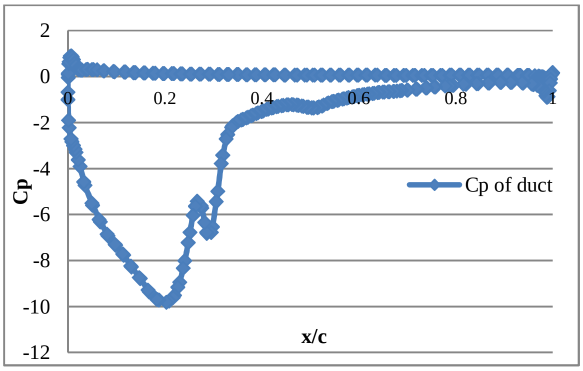


Figure 17: Pressure distributions on duct with 20% increasing of duct length

7.2 Model with Duct Angle 10 Degree

Now, for investigate the effect of increase in duct angle, change the model of duct. After modeling of new duct specified boundaries settings and run solver module to export results for different advanced ratios. The results compared with first model in Fig. 18. Increase in duct angle causes to increase thrust and torque coefficients that more effect of this change is showed at lower advanced ratios. In heavy conditions for this model, the thrust is more than first model but effect on torque is more than thrust and causes to decrease total efficiency of ducted propeller.

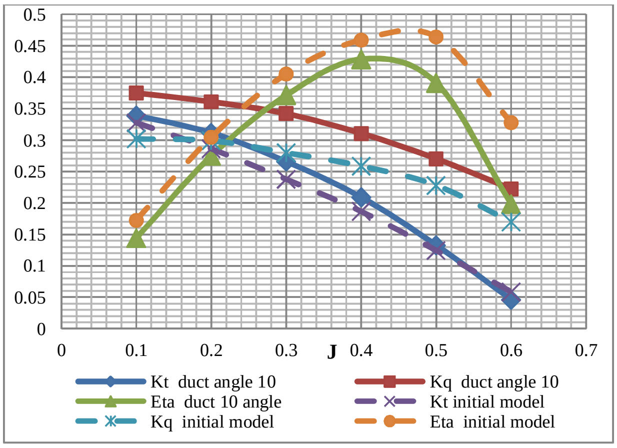


Figure 18: Comparison of hydrodynamics characteristics of the ducted propeller in cases normal duct and duct with 10 degree duct angle

7.0 CONCLUSION

In this paper, the Kaplan propeller with nozzles 19A analyzed by numerical method and the following results are concluded:

- ✓ Pressure coefficient distribution on the duct and blade are presented in contours and diagram. Negative low pressure coefficients are shown in back side and high pressure is given in face side of the blade. On the duct is also shown low pressure at suction side (mean inside the duct).

- ✓ To evaluate the effect of increasing the duct length on the performance of the propeller in open water, the nozzle section length increased to 10 and 20 percent. The results show that increasing 10% of the nozzle length has positive effect on the performance of the propeller but increase the nozzle length further 10% will have a negative effect on propeller performance.
- ✓ Effect of the duct angle up to 10 degree and the results are compared with the first model results. It is shown that with increasing duct angle the thrust and torque of propeller are also increased but the efficiency is diminished.

REFERENCE

1. Stipa, L. (1931). *Ala a turbina*. L'Aerotecnica, 923–953.
2. Kort, L. Der neue d. (1934).¨*usenschrauben-antrieb*. Werft, Reederei und Hafen, 15.
3. Sanchez-Caja, A., Rautaheimo, P. & Siikonen, T.(2000). *Simulation of incompressible viscous flow around a ducted propeller using a RANS equation solver*. Proceedings of the Twenty-Third Symposium on Naval Hydrodynamics,.
4. Abdel-Maksoud, M. & Heinke, H.-J. (2003). *Scale effects on ducted propellers*. Proceedings of the Twenty-Fourth Symposium on Naval Hydrodynamics, Fukuoka, Japan.
5. Krasilnikov, V. I., Sun, J. Y., Zhang, Z. & Hong, F. (2007). *Mesh Generation Technique for the Analysis of Ducted Propellers Using a Commercial RANSE Solver and its Application to Scale Effect Study*. Proceedings of the 10th Numerical Towing Tank Symposium (NuTTS'07).
6. Kerwin, J. E., Kinnas, S. A., Lee, J.-T. & Shih, W.-Z. (1987). *A surface panel method for the hydrodynamic analysis of ducted propellers*. Transactions of Society of Naval Architects and Marine Engineers 95.
7. Lee, H. & Kinnas, S. A. (2006). *Prediction of cavitating performance of ducted propellers*. Proceedings of the Sixth International Symposium on Cavitations.
8. Baltazar, J. &Falcão de Campos, J. A. C. (2009). *On the modeling of the flow in ducted propellers with a panel method*. Proceedings of the First International Symposium on Marine Propulsors, Trondheim, Norway.
9. Rijpkema, D. &Vaz, G. (2011). *Viscous Flow Computations on Propulsors: Verification, Validation and Scale Effects*. Proceedings of the Developments in Marine CFD.
10. Bobo M. J., De La Rosa J.-C., Masip J., Quereda R. & Pangusión, L. (2005). *Design of ducted propeller and model tests of a fishing research vessel for M.Cies Shipyards*. OTI 2233-2, CEHIPAR (in Spanish).
11. Hughes, M. J. (1997), *Implementation of a special procedure for modeling the tip clearance flow in a panel method for ducted propellers*. Proceedings of the Propellers/Shafting '97 Symposium.
12. Moon, I.-S., Kim, K.-S., and Lee, C.-S. (2002). *Blade tip gap flow model for performance analysis of waterjet propulsors*. In International Association for Boundary Element Methods, Austin, Texas, USA.
13. Falcao de Campos, J. A. C. (1983). *On the calculation of ducted propeller performance in axisymmetric flows.* PhD Thesis, Delft University, Wageningen, the Netherlands.
14. Hoekstra, M. (2006). *A RANS-based analysis tool for ducted propeller systems in open water conditions*. International Shipbuilding Progress 53, 205-227.
15. Zondervan, G-J., Hoekstra, M. & Holtrop, J. (2006). *'Flow Analysis, Design and Testing of Ducted Propellers*. Proceedings of Propeller/Shafting Symposium, Virginia Beach, United States.
16. Gu H and Kinnas S A. (2003). *Modeling of contra-rotating and ducted propellers via coupling of a vortex-lattice with a finite volume method*. In: Proceedings of Propeller/Shafting Symposium, SNAME, Virginia Beach, USA.
17. Haimov, H., Bobo, M.J., Vicario, J. & Del Corral, J. (2010). *Ducted propellers. A Solution for Better Propulsion of Ships. Calculations and Practice*. Proceedings of First International Symposium on Fishing Vessel Energy Efficiency, Vigo, Spain.
18. J. Baltazar, J.A.C. Falcão de Campos and J. Bosschers (2011). *Open-Water Thrust and Torque Predictions of a Ducted Propeller System with a Panel Method*, Second International Symposium on Marine Propulsors smp'11, Hamburg, Germany, June.
19. Sasaki S., Torise I., Hayashi H. (2012). *An experimental and numerical study on wake vortex noise of a low speed propeller fan*, Open Journal of Fluid Dynamics, 2, 290-296.
20. Abdel-Maksoud M, Steden M and Hundemer J. (2010). *Design of a Multi-Component Propulsor*. In: Proceedings of 28th Symposium on Naval Hydrodynamics, Pasadena, USA, September.
21. Steden M, Hundemer J, Abdel-Maksoud, M, (2009). *Optimization of a linearjet*, First International Symposium on Marine Propulsors, Trondheim, Norway.
22. Carlton, J. S. *"Marine Propellers and Propulsion"*, 3rd edition, 2012