

# **A Simple Method for Designing a Duct for a Multi-Component Ducted Propulsion System**

**N. M. Nouri \***

Department of Mechanical Engineering,  
Iran University of Science and Technology, Iran  
E-mail: mnorui@iust.ac.ir  
\*Corresponding author

**M. Kalantar Neyestanaki**

Department of Mechanical Engineering,  
Iran University of Science and Technology, Iran  
E-mail: mehrdad.kn.89@gmail.com

**S. Mohammadi**

Department of Mechanical Engineering,  
Iran University of Science and Technology, Iran  
E-mail: smohamadi@iust.ac.ir

**Received: 8 May 2017, Revised: 23 June 2017, Accepted: 11 July 2017**

**Abstract:** The present paper numerically discusses the design procedure of marine ducts used for multi-component ducted propulsion systems at the stern of an axisymmetric submerged body. The results are presented in the form of tables showing the effects of dihedral angle as well as camber ratio of the duct as the two most important geometrical parameters on hydrodynamic performance of the propulsion system. Furthermore, a correlation has been extracted between the results of two and three dimensional analysis of ducted propellers. The results show that the design procedure of the duct used for a ducted propulsion system could be performed using some two dimensional analyses. The simulations are performed using a Reynolds averaged Navier Stokes Equations (RANS) based Computational Fluid Dynamics (CFD) tool.

**Keywords:** Axisymmetric submerged body, CFD, Ducted propulsion system, Propeller

**Reference:** Nouri, N. M., Kalantar, M., Mohammadi, S., "A Simple Method for Designing a Duct for a Multi-Component Ducted Propulsion System", *Int J of Advanced Design and Manufacturing Technology*, Vol. 10/ No. 3, 2017, pp. 57-64.

**Biographical notes:** **N. M. Nouri** received his PhD in Mechanical Engineering from University of National Polytechnic Institute of Lorraine, in 1995. He is currently Professor at the Department of Mechanical Engineering, Iran University of Science and Technology, Tehran, Iran. His current research interest includes Hydrodynamics. **M. Kalantar Neyestanaki** received his MSc in Mechanical engineering from Iran University of Science and Technology, in 2013. His current research focuses on fluid mechanics. **S. Mohammadi** received his MSc in Mechanical engineering from Iran University of Science and Technology, in 2011. He is currently PhD student at the Department of Mechanical Engineering, Iran University of Science and Technology, Tehran, Iran. His current research interest includes Hydrodynamics of Marine Propulsion Systems.

---

## 1 INTRODUCTION

---

Ducted propellers are among the most complicated marine propulsion systems which are widely used in marine applications. The shape of a marine duct could be in accelerating or decelerating form. In the form of accelerating duct, the efficiency is increased in the low speed of advance by accelerating the water flow through the duct, but it would increase the possibility of cavitation occurrence. The decelerating duct would reduce the efficiency; however, it would minimize the cavitation occurrence possibility because of reducing the flow speed passing the propeller. Therefore, the main objective of using such systems is to increase the hydrodynamic efficiency in low speed of advance or decreasing the cavitation risk in high speeds. This propulsion system could be used as a multi-component propulsion unit for axisymmetric submerged bodies. A simple configuration could involve rotor-stator composition combined with a covering duct. The necessity of combining the rotor with a stator is to cancel the rotational energy of the outflow from the propulsion unit in order to increase the efficiency. In recent years, it is seen that a CRP set is used instead of rotor-stator configuration. This system is able to cancel the rolling motion of the body. Multi-component ducted propellers could be used in shallow water applications with purpose of decreasing the risk of cavitation occurrence, lowering the produced noise and making the system less affectable by the environment [1].

The idea of using multi-component ducted propulsion systems in order to prevent cavitation dates back to 1945 when G. F. Wislicenus [2] presented an initial design for an axial flow ducted system as propulsion unit of axisymmetric underwater vehicles. The others investigated the possibility of propelling underwater vehicles by means of multi-component ducted systems and, eventually, a wide range of research programs in this area was started by the Naval Ordnance Test Centre of the United States of America (NOTS) in Pasada. Yu et al. (2013) [3] studied the Ka-series propellers performance with the 19A duct by employing the panel method panMARE and the RANSE code. They tried to optimize ducted propellers under open water conditions by using CFD method (2015) [4]. Suryanarayana et al. (2015) [5] designed a pump jet for specific under water vehicle. They predicted the performance of the propulsion system using CFD method. Furthermore, they analyzed the cavitation performance of the propulsion unit by means of cavitation tunnel. Ghassemi et al. (2017) [6] performed a numerical analysis over ducted propellers propulsion system using ANSYS-CFX software. The main objective of this research has been finding a simple method for designing the duct surrounding a

multi-component propulsion system. In this study, the applied cross sections of the ducts are four-digit NACA airfoils which are very similar to the common cross sections applied in marine applications. Therefore, we could confidently conclude that the obtained results would be in agreement with common cases in marine applications. The results obtained from this research would help the marine designers to simply design a duct for a ducted propulsion system. In this research, the effective geometrical parameters of the duct have been initially investigated. Then, the primary design has been made and finally, computational fluid dynamics simulations have been carried out through two-dimensional analysis. At the end, some three dimensional simulations have been performed in order to extract a correlation between the results of two and three dimensional analysis of a similar system.

---

## 2 CASE STUDY

---

The duct which is surrounding a marine propeller could be assumed as a semi-cylindrical passage which is made by rotating an airfoil about axis of body. The mentioned airfoil makes the duct produce axial force proportionate to the pressure gradient distribution. The resultant force is a vector whose components depend on the duct geometry in a certain regime of flow. On the other hand, the propellers revolution affects the upstream flow and causes it to have a different angle of attack relative to the ducts cross section which could be positive or negative with regard to the geometry of the duct, physics of the entrance flow and the loading condition on blades of the propeller [7].

In order to start the design procedure of a marine duct, it is necessary to find the effective geometrical parameters on its hydrodynamic performance and to estimate some allowed ranges for them. In the design procedure of a ducted propeller propulsion system, the main purpose is to achieve a design in which either the requirements relative to the propeller performance or the advantages of adding the duct including the efficiency improvement or the cavitation prevention are considered. These features are determined by physics of the flow entering to the propeller disk as well as the propeller operating pressure. This pressure is highly dependent on operating depth of the propulsion system and the loading condition on propeller blades. Therefore, in order to design a duct, the main objective has to be choosing between increasing the efficiency and making the system more resistant against cavitation occurrence.

In this research, some geometrical parameters of the duct geometry are fixed and the design procedure is limited to estimate two most important parameters of the duct cross section, i.e. camber ratio and dihedral

angle (the angle between the chord line of the duct cross section and the rotation axis of the duct). In order to obtain the proper dihedral angle and appropriate camber ratio for the duct cross section, their influences on the axial force produced by the duct and the pressure field inside the duct has to be studied.

The constraints suggested for accessing the appropriate geometry of a duct are as following:

- Duct length (L) is expressed by means of the duct diameter,  $D_{\text{mean}}$ . In [8] the ratio of the duct length to its diameter for the case of single propeller has been accounted as 0.5. In this research, the mean diameters of the ducts are equal to 375 mm. So the lengths of the ducts are considered equal to 188 mm.
- The distance between the blades tips and the internal surface of the duct has been considered to be 1.5 percent of the duct mean diameter as mentioned in [7] and [8].
- The four-digit NACA cross sections are used for the ducts. Location of maximum thickness and maximum camber is 50 percent of the length of the chord from the leading edge of each duct.
- The diameter of the submerged body is equal to 750 mm.
- Radius of leading edge of the duct is 1 mm in all cases.
- The maximum thickness ratios of the ducts cross sections are equal to 5 percent in all cases.

Using standard airfoils like four-digit NACA for the duct cross section seems to be appropriate due to the following reasons:

- 1- Ease of using their corresponding mathematical formulations
- 2- Acceptable similarity between the common ducts cross sections and this type of airfoils.

### 3 GOVERNING EQUATIONS

The governing equations consist of momentum, mass and energy conservation equations. Since the flow is isothermal, the energy conservation equation could be neglected. Thereby, the governing equations based on Reynolds averaging technique could be stated as following [9]:

$$\begin{aligned} \frac{\partial \bar{p}}{\partial t} + \frac{\partial}{\partial x_i} (\bar{\rho} \bar{u}_i) &= 0 \\ \bar{\rho} \frac{\partial \bar{u}_i}{\partial t} + \bar{\rho} \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} &= -\frac{\partial \bar{p}}{\partial x_i} + \bar{\rho} f_i \\ + \mu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} + \frac{\mu}{3} \frac{\partial}{\partial x_i} \left( \frac{\partial \bar{u}_j}{\partial x_j} \right) &- \bar{\rho} \frac{\partial}{\partial x_j} \left( \overline{u_i' u_j'} \right) \end{aligned} \quad (1)$$

where  $p$  is the static pressure,  $\bar{u}$  and  $\bar{u}'$  are the averaged and fluctuating components of the velocity vector,  $f_i$  are external forces,  $\mu$  is the viscosity and  $x$  is

the coordinate.  $\overline{\rho u_i' v_j'}$  is called Reynolds stresses and could be computed by means of eddy viscosity hypothesis which relates this tensor to the turbulent viscosity and mean velocity gradients of the flow. In order to model the turbulent viscosity, we need to use the turbulence models such as  $k-\epsilon$  or  $k-\omega$ .

### 3.1 NUMERICAL SIMULATIONS

In this study, the Reynolds averaged Navier-Stokes Equations (RANS) have been solved by using Computational Fluid Dynamics (CFD) tool. The finite volume method has been applied for discretization of governing equations. The pressure-velocity coupling is based on SIMPLEC algorithm [10]. The method employed for pressure discretization is PRESTO and for convective terms the Quadratic Upwind Interpolation for Convective Kinematics (QUICK) has been employed [11]. The applied turbulence model is RNG- $k-\epsilon$  which is based on the standard  $k-\epsilon$  model involving some improvements in modeling the rotating flows and streamlines curvatures [12]. The standard semi-empirical wall function has been used as well in order to model the boundary layer effects. We perform the simulations in the form of steady state.

### 3.2 TWO DIMENSIONAL ANALYSIS

The geometries used in this research have been axisymmetric bodies equipped with the ducts which have been modelled symmetrically relative to the axis of the body without the rotor-stator blades. The simulations are performed as the steady axisymmetric two dimensional ones. In this way, the simulations would be simply performed and would be less time consuming relative to the three dimensional cases. The main difference between these types of analyses with three dimensional one is that the effects of the real rotor-stator is neglected on the system and, instead, the FAN boundary condition with the pressure jump equal to the desired thrust of rotor-stator would be employed. Therefore, some simple analysis could be performed on different geometries of ducts in order to evaluate the

effects of different geometrical parameters on the duct hydrodynamic performance.

### 3.3 BOUNDARY CONDITIONS

Boundary conditions considered here include the inlet velocity, outlet pressure, zero gradients and wall. The inlet velocity condition starts from the upper edge of the submerged body and includes the inlet edge of the environment in the left edge of figure 1. This condition includes the specific velocity parallel to the axis of the body. The  $P = P_\infty$  and  $\frac{\partial \phi}{\partial x} = 0$  are considered as the outlet boundary condition at the end of the computational area at the right end of Fig. 1. The wall condition is surface of the vehicle and the duct in which the no-slip condition has been considered. On upper edge of the computational domain, zero gradients boundary condition has been applied. At the bottom and back of the body, the axisymmetric condition has been accounted for the two-dimensional axisymmetric study. The computational domain is shown in Fig. 1.



**Fig. 1** The computational domain for two dimensional numerical simulations

FAN boundary condition has been employed for modelling the effects of rotor-stator suction on the upstream flow field. By means of this boundary condition, operational parameters of an impeller involving the head (pressure) and flow rate (velocity) would be modelled. This boundary condition gives no details about the flow between the vanes. In this boundary condition, the fan is modelled as a thin plate at the middle part of the duct. Its produced pressure jump could be either a constant value or a function of inlet velocity. In this research, a constant pressure jump would be used over the fan boundary condition. By means of the relationship between momentum and thrust of the rotor-stator, the amount of pressure jump for this boundary condition could be computed as following:

$$\Delta P = \frac{T_{desired}}{A} \quad (2)$$

Where,  $\Delta P$  is pressure jump of the fan,  $T_{desired}$  is the desired thrust of the rotor-stator and  $A$  is the cross section of duct at its mid plane.

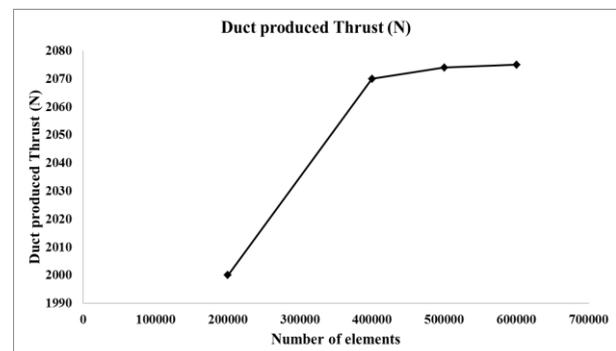
### 3.4 GRID GENERATION

While generating the grid, one should pay attention to the following points:

- 1- Concentration of the nodes is determined by the required accuracy of simulation. In regions of the flow with high velocity gradients, more concentrations of nodes are required rather than the regions with low gradients.
- 2- The elements should be aligned in the flow direction, so that the numerical solution could converge more rapidly.
- 3- The flow behavior near the wall is considered as a region with high gradients.
- 4- The grid quality near the walls must be high enough to model the boundary layer effects on the flow properly. This subject is related to the  $y^+$  parameter which is adjusted about 100 on the body and on the duct surfaces in order to use the semi-empirical wall functions.

### 3.5 INDEPENDENCY OF THE RESULTS FROM NUMBER OF ELEMENTS

In all numerical simulations, one has to investigate the independency of the results from the number of elements. In this research, dependency of the thrust produced by the duct on number of elements has been studied. The results are displayed in Fig. 2. It could be observed that independency of the results from number of elements has been obtained by using about 500,000 elements in the computational domain.



**Fig. 2** Independency of the results from the number of elements

### 3.6 THREE DIMENSIONAL SIMULATIONS

Some three dimensional analyses have been performed in the last part of this study. Therefore, the applied

procedure for performing three dimensional simulations has been explained here. In three dimensional cases one would model the whole axisymmetric body equipped with the entire propulsion unit. Thereby, three dimensional simulations must be performed so that the rotational effect of rotor could be considered. The simplest method for solving the problem with rotary boundary conditions containing the axially uniform inlet velocity to the rotary parts is the moving reference frame method. In this method, the rotational motion is imported to every node of the computational domain in the opposite direction and the problem would be solved in the steady-state form in a frozen state. The stationary walls could be modelled with respect to the stationary coordinate system as fixed walls and the rotary ones would be modelled as fixed walls with respect to the rotational coordinate system.

It is preferred to use hexahedral elements for the entire domain because the accuracy of the solution is in the maximum level using such elements in the grid. Additionally, using such elements, one would be able to follow the streamlines close to the body and to make a grid with lowest number of nodes. However, for complex geometries like propellers, it is common to use tetrahedral elements since they could be easily generated around such objects. In this way, a hybrid grid was generated in this problem. For generating such a hybrid grid, the domain was divided into several zones. One of them was around the propulsion system and the others were the remaining parts. In the former, tetrahedral and in the latter hexagonal grid was generated. The quality of the elements near the body, duct and rotor-stator blades was fine enough for proper modelling of the boundary layer effects using semi-empirical models. For this purpose, the  $y^+$  parameter on the body, duct and rotor-stator was adjusted about 100. Thrust of the rotor has been checked for independency from the number of elements. It was found that the thrust varied by about 0.5% when the number of elements was more than 3.8 million (Fig. 3). The generated mesh around the propulsion unit could be observed in Fig. 7.

#### 4 VALIDATION

To validate the three dimensional numerical approach in the case of open propeller, a 6-bladed B-wageningen standard propeller with pitch ratio of 1.2, expanded area ratio of 1 and diameter of 25 cm has been considered. After performing CFD simulations, its hydrodynamic coefficients have been extracted out and compared to the experimental results [13]. The propellers thrust coefficient, torque coefficient and

efficiency have been demonstrated in Fig. 4. In this figure:

- $K_T = T / \rho n^2 D^4$  is the thrust coefficient.
- $K_Q = Q / \rho n^2 D^5$  is the torque coefficient.
- $\text{Etta} = TV / 2\pi n Q$  is the hydrodynamic efficiency.
- $J = V / nD$  is the advance coefficient.

Where  $T$  is the thrust (N),  $Q$  is the torque (N.m),  $n$  is the rotational speed (rps),  $V$  is the advance coefficient (m/sec) and  $D$  is the propellers diameter (m). It could be observed that the simulations have been performed with maximum error of eight percent in predicting the thrust of the propeller. Therefore, the applied numerical approach seems to be accurate enough for performing further simulations in this research.

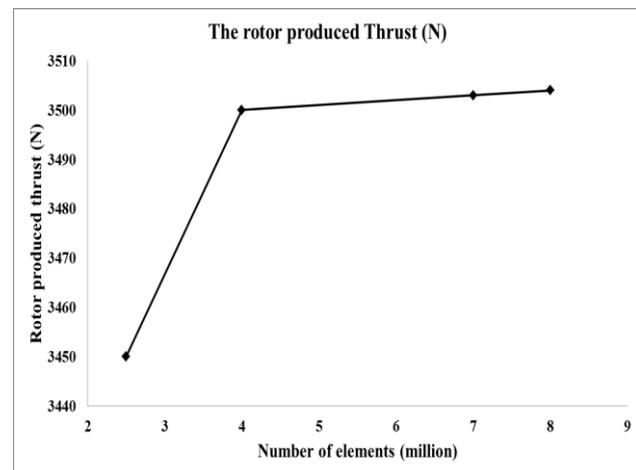


Fig. 3 Independency of the results from number of elements

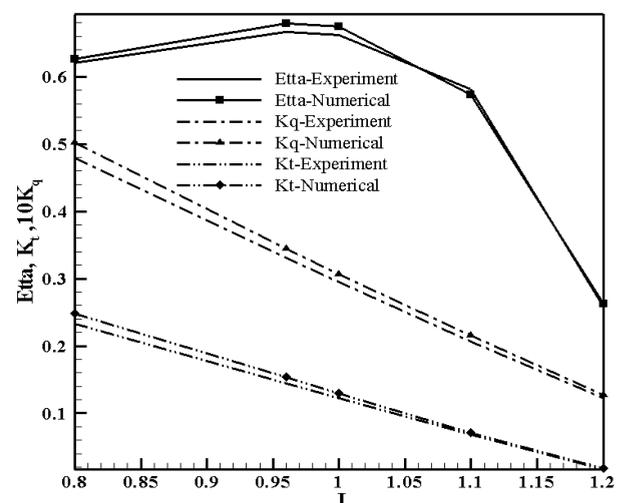


Fig. 4 Validation diagram of B-series propeller

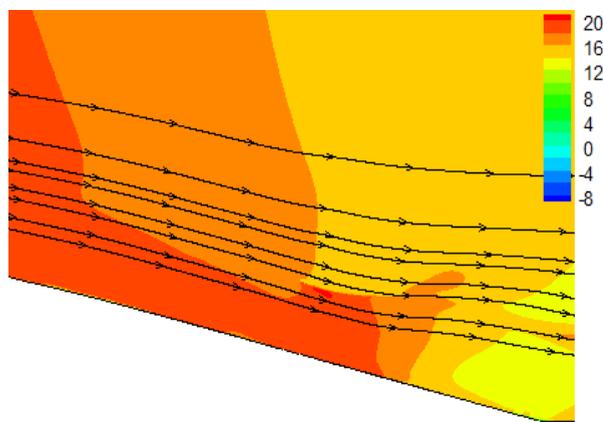
## 5 RESULTS

Some two and three dimensional simulations have been performed on the mentioned multi-component ducted propulsion system placed behind of the introduced axisymmetric submerged body. The velocity of the inlet flow has been assumed to be 10 m/sec in the simulations. The pressure jump on FAN boundary condition has been calculated so that the produced thrust is about 6000 N.

### 5.1 TWO DIMENSIONAL ANALYSIS

#### Estimating the Best Dihedral Angle for the Duct:

According to the fact that an appropriate duct should have the lowest level of confusion on physics of the flow surrounding the duct, dihedral angle of the duct could be estimated. For this purpose, physics of the flow should be investigated around the stern of the bare body. The initial dihedral angle of the duct could be estimated with respect to configurations of the streamlines of the flow at stern of the bare body. In this study, these values have been selected between 7 to 11 degrees with respect to Fig. 5.



**Fig. 5** Streamlines at stern of the bare body in addition to velocity contours

Therefore, the selected dihedral angles of the ducts cross sections have been placed between 7 to 11 degrees. FAN boundary condition has been defined in the mid part of the duct. The desired pressure jump corresponding to the desired thrust has been applied on this boundary condition. While preparing different models of the ducts with different dihedral angles, the middle diameters of the ducts corresponding to the locations of FAN boundary condition have been considered to be constant. Thereby, in larger dihedral angles the inlet diameter increases and the outlet one reduces. In this step, an optimum dihedral angle would

be found when comparing the resultant forces of the duct and the body with the minimum pressure over the duct (0). In Table 1, the minus sign means that the force is exerted in the opposite direction.

As observed in 0, increasing dihedral angle could lead to reduction of the generated thrust by the duct and to increase of the minimum pressure over the duct. On the other hand, having this angle reduced, the minimum pressure over the duct would be decreased and the thrust produced by the duct is increased. According to the fact that it would be desirable to have a greater value of minimum pressure over the duct in addition to a positive thrust produced by the duct, one would choose the desired dihedral angle between 7 and 8 degrees.

**Table 1** The results related to dihedral angle

Axial force of the body (N)	P <sub>min</sub> (Pascal)	Axial force of the duct (N)	Camber (%)	Dihedral angle (degree)
-10554	$-1 \times 10^5$	2077	0	7
-9688	$-7.3 \times 10^4$	1133	0	8
-8485	$-5 \times 10^4$	-439	0	9
-6865	$-8 \times 10^3$	-3076	0	11

#### Estimating the Best Camber Ratio for the Duct:

The next step has been to find an adequate camber ratio for the duct cross section (Table 2). where the submerged body would accompany a duct with dihedral angles of 7 and 8 degrees. The camber ratios have been supposed to be -1%, 0%, and 1%. Comparing the resultant forces of the duct and the body with the minimum pressure over the duct in this step, an optimum camber ratio could be found (Table 2). In Table 2, the minus sign means that the force is exerted in the opposite direction.

**Table 2** The results related to the camber ratio

Axial force of the body (N)	P <sub>min</sub> (Pascal)	Axial force of the duct (N)	Camber r (%)	Dihedral angle (degree)
-8216	$-6 \times 10^4$	-916	1	8
-8828	$-6.41 \times 10^4$	-36	0	8
-9688	$-7.3 \times 10^4$	1133	-1	8
-9195	$-9.95 \times 10^4$	101	1	7
-10074	$-1.13 \times 10^5$	1228	0	7
-10711	$-1.14 \times 10^5$	2218	-1	7

As observed in Table 2, through comparing the results obtained from the performed analyses with regard to the axial forces exerted on the duct and the body with magnitude of the minimum values of pressure over the duct, one could consider the dihedral angle of 8 degree and a camber ratio of 0 percent as the desired initial parameters for geometry of the duct in order to minimize the risk of cavitation combined with a small positive axial force. It is noticeable that the selected case could be changed with regard to other design criteria; for example, neglecting the risk of cavitation over the duct. The velocity and the streamlines around the final geometry is demonstrated in Fig. 6. As presented in Fig. 6, final geometry of the duct has a small influence on the streamlines around the body.

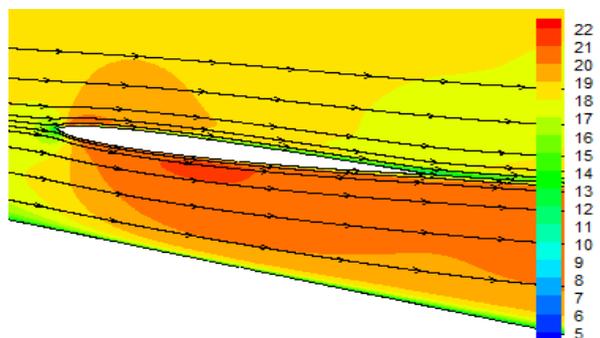


Fig. 6 Velocity and streamlines around final geometry of the duct

### 5.2 THREE DIMENSIONAL RESULTS

Regarding the disability of FAN model in estimating the rotor-stator performance, it would be necessary to combine the resulted geometry of the duct with the rotor-stator combination in order to perform some three dimensional simulations. The generated mesh around the complete system is presented in Fig. 7. The velocity field in a cross sectional plane after performing the simulations is displayed in Fig. 8 as well.

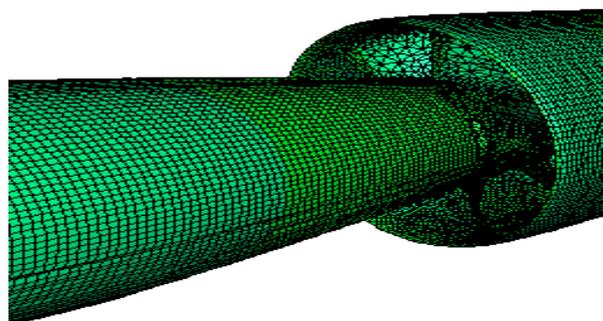


Fig. 7 The multi-component ducted propeller in three dimensional configuration

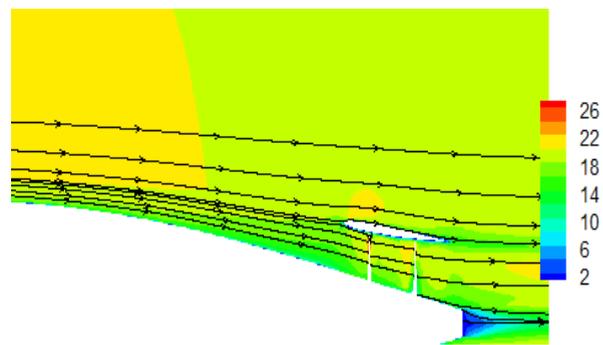


Fig. 8 Velocity field around the propulsion system in 20 m/sec

In this step, hydrodynamic parameters of the propulsion system were obtained through three dimensional simulations. Inlet velocity of the flow through the domain was as the same as the two dimensional analysis. Rotational speed of the rotor was adjusted so that the thrust produced by the rotor-stator combination was about 6000 N which was used previously in order to compute the pressure jump on FAN boundary condition in two dimensional analyses. The result showed that there was a significant difference between the axial force produced by the duct in the corresponding 2-D and 3-D states. In the other words, one could find out that there is a significant difference between operation of the 2-D FAN boundary condition and the 3-D rotor-stator combination which has made the duct to produce different magnitudes of the axial forces under equal loading conditions (equal magnitudes of the produced thrusts). In order to solve this problem, a rotational speed for the rotor was found so that to made the duct produce the axial force equal to its corresponding 2-D state. It was observed that the thrust produced by the duct through 3-D simulations was equal to the 2-D one only if the thrust produced by the propellers became 2.1 times greater than the corresponding thrust used in 2-D analysis. Therefore, equation (2) requires a correction factor in calculating the pressure jump which could be defined as followed:

$$K = \frac{T_{desired,3d}}{\Delta P_{Fan} A} = 2.1 \rightarrow \Delta P_{Fan} = \frac{T_{desired}}{KA} = 0.4764 \frac{T_{desired}}{A} \quad (3)$$

This equation could be a function of the rotor rate of revolution for the specified geometry, but it could be considered as a constant for rotational speeds which are rather close to each other (and also for rather similar duct geometries). 0 gives the results obtained by applying this correction factor in other three states. As observed here, applying the extracted correction factor, made the magnitude of the axial force produced by the duct in 2-D and 3-D sates equal to each other with a slight difference.

**Table 3** Comparison between 2-D and 3-D analysis using correction factor of K

Propeller thrust (N)	3-D duct thrust (N)	2-D duct thrust (N)	Difference between 2-D and 3-D results
6000	1269	1259	0.8%
7385	813	798	1.84%
8000	776	761	1.96%

Therefore, by means of this correction factor, the duct could be designed almost near to its final shape in two dimensional environment for a specific axisymmetric body. Regarding the fact that the computational costs of three dimensional analyses are about 10 times greater than two dimensional ones in these cases and because of the extracted correlation between 2-D and 3-D results, this method could be satisfactory in designing a duct. It has to be mentioned that this correction factor could be a function of the axisymmetric body shape and design constraints and would change for another problem.

## 6 CONCLUSIONS

A simple method was presented for designing an acceptable geometry of multi-component ducted propulsion systems using two dimensional CFD analyses. Furthermore, it was shown that for a fixed advance velocity and a special rotor-stator geometry with a specific desired thrust, there could be a correlation between the results of two and three-dimensional analyses. It was shown that, in order to find an acceptable geometry for the duct of a multi-component ducted propeller, it would be possible to find an adequate dihedral angle for the duct cross section using CFD analyses of the bare submerged axisymmetric body. After that, it would be possible to find an adequate magnitude of camber ratio of the duct cross section using some other two dimensional simulations.

## 7 ACKNOWLEDGEMENT

We do appreciate the personnel of the Applied Hydrodynamics Laboratory of Iran University of Science and Technology, who helped us in this project.

## 8 REFERENCES

- [1] Carlton, J., "Marine Propellers and Propulsion", Butterworth-Heinemann, 2011.
- [2] Henderson, R. E., McMahon, J. F., and Wislicenus, G. F., "A Method for the Design of Pumpjets", Pennsylvania State Univ State College Ordnance Research Lab, 1964.
- [3] Yu, L., Greve, M., Druckenbrod, and Abdel-Maksoud, M., "Numerical Analysis of Ducted Propeller Performance Under Open Water Test Condition", Journal Of Marine Science And Technology, Vol. 18, No. 3, 2013, pp. 381-394.
- [4] Druckenbrod, M., Greve, M. and Abdel-Maksoud, M., "Numerical and Experimental Analysis of a Ducted Propeller Designed by a Fully Automated Optimization Process Under Open Water Condition", China Ocean Engineering, Vol. 29, No. 5, 2015, pp. 733-744.
- [5] Suryanarayana, C., Satyanarayana, B., Ramji, K. and Rao, M. N., "Cavitation Studies on Axi-Symmetric Underwater Body with Pumpjet Propulsor in Cavitation Tunnel", International Journal of Naval Architecture and Ocean Engineering, Vol. 2, No. 4, 2010, pp. 185-194.
- [6] Motallebi-Nejad, M., Bakhtiari, M., Ghassemi, H., and Fadavie, M., "Numerical Analysis of Ducted Propeller and Pumpjet Propulsion System Using Periodic Computational Domain", Journal of Marine Science and Technology, 2017, pp. 1-15.
- [7] Bertram, V., Schneekluth, H., "Ship Design for Efficiency and Economy", Butterworth-Heinemann, 1988.
- [8] Oosterveld, M. W. C., "Wake Adapted Ducted Propellers", Doctoral dissertation, TU Delft, Delft University of Technology, 1970.
- [9] Tropea, C., Yarin, A. L., and Foss, J. F., Springer handbook of experimental fluid mechanics, Springer Science & Business Media, Vol. 1, 2007.
- [10] Van Doormaal, J. P., Raithby, G. D., "Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows", Numerical Heat Transfer, Vol. 7, No. 2, 2007, pp. 147-163.
- [11] Kaya, F., Karagoz, I., "Performance Analysis of Numerical Schemes in Highly Swirling Turbulent Flows in Cyclones", Current Science, Vol. 94, No. 10, 2008, pp. 1273-1278.
- [12] Orszag, S. A., Yakhot, V., Flannery, W. S., Boysan, F., Choudhury, D., Maruzewski, J., and Patel, B., "Renormalization Group Modeling and Turbulence Simulations", Near-wall turbulent flows, 1993, pp. 1031-1046.
- [13] Barnitsas, M. M., Ray, D., and Kinley, P., Kt, Kq and efficiency curves for the Wageningen B-series propellers, University of Michigan, 1981.