Designing and Analyzing the Duct used on Pump Jet Propulsion System for an AUV

Norouz Mohammad Nouri1, Mehrdad Kalantar Neyestanaki2, Saber Mohammadi3

1Associate Professor, Iran University of Science & Technology; mnouri@iust.ac.ir
2Master of Science Student, Iran University of Science & Technology; mkalantar@mecheng.iust.ac.ir
3P.H.D Candidate, Iran University of Science & Technology; smohamadi@iust.ac.ir

Abstract
Pump jet is one of the most important propulsion systems for AUVs in high velocities. The critical characteristic of the pump jet is its high efficiency. Pump jets are among the rare propulsion systems which are used non-autonomously in shallow waters. A simple type of pump jet consists of a rotor, a stator and a cylindrical duct covering the rotor and the stator. However, in some cases the propeller with counter rotating direction has been used instead of the stator in some recent applications (like the CRP system). According to the advantages of this propulsion system, it has been widely used in recent years in various naval and military applications. Therefore, there is an essential need to numerical and experimental analysis on these types of propulsion systems. Numerical and experimental analysis conducted on the propulsion systems with ducts, especially the ones with contra rotating propellers, have been done for the purpose of achieving to a configuration with higher efficiency and the lowest possibility of cavitation occurrence. The main object of this article is to design a proper duct for pump jet propulsion system through 2D axisymmetric analysis by means of computational fluid dynamics simulations.

Keywords: Pump Jet Propulsion System, Computational Fluid Dynamics, 2D analysis, duct

Introduction
A simple Pump jet consists of a rotor, a stator and a cylindrical duct covering them. Pump jets are one of the rare propulsion systems being employed non-autonomously in shallow waters. Other features of the pump jet could be mentioned as low risk of cavitation occurrence, low acoustical signals and not being affected by the surrounding perturbations. The reason for lower noise produced in this system compared to conventional open propeller could be mentioned as its prevention of tip vortex and cavitation occurrence. Furthermore, pump jets have smaller diameters compared to the open propulsion systems, since the duct allows the rotor blade to put its edges under pressure without losing thrust. The covering duct controls the water flow rate and reduces the produced sound waves. The shape of this duct could be in accelerating or decelerating form. In the form of accelerating duct, efficiency is increased through accelerating the water flow but, on the other hand, increases the possibility of cavitation. The decelerating duct, however, reduces the efficiency but, minimizes the cavitation occurrence possibility. The distance between the upper edge of the vane and the duct is supposed to be small compared to the vane length, so that the fluid could not move back toward the face of the propeller from the pressurized zone at the back of it. Typically this distance is selected equal to 3% of the vane length. The idea of using pump jet for cavitation prevention returns to 1945 when G. F. Wislicenus represented an initial design for an axial flow pump jet as the propulsion system for axisymmetric underwater vehicles. Others investigated the possibility of propelling underwater vehicles by means of pump jets and, eventually, a wide range research program in this area was started by the Naval Ordnance Test Center of the United States of America (NOTS) in Pasada. This program described using axial or mixed flow pump jets installed in front of the vehicle body in order to prevent inlet diffusion. Investigating the possibility of using pump jet propulsion system went on and some important reports in this area was published by Brumfield et al. in NOTS and some others in John Hopkins university. Finally, in the early 1950, the NOTS program led to conducting various experimental works on axial flow pump jets. In the middle of 1950, another program focused mainly on applicable methods of designing pump jets [1].

In 1984, Markatos did some calculations about wake and turbulent boundary layer of the circular bodies and represented some predictions [3]. In 1988, Furuya et al. attempted to design a pump jet using a two dimensional method employed for designing pumps. However, as the authors claimed, their method will not be effective in high velocities [2]. Brown and Vosper reported in 1996 that England has equipped its submarines to pump jets. In addition, it was mentioned that American Sea-Wolf submarines have been equipped to pump jets as well [4]. In 1999, Suryanarayana designed a pump jet as a axisymmetric underwater vehicle propulsion system in NSTL Institute and published it in an internal report [6]. A CFD simulation on the flow passing over a axisymmetric body with a pump jet was also conducted by Ivanell in 2001[5]. Abdel-Maksoud and others in Hamburg University (2010) tried to design multi-component ducted propellers with a view on cavitation reduction, which were consisted of a duct, a rotor and a stator and were much similar to pump jet system [11].

In 2011, Neelima Devi and some others studied on designing, finite element analyzing and manufacturing the stator blades of pump jets and published a paper on their activities [10] [9]. In recent decade, fluid dynamics has been used for designing various propellers for marine propulsion systems. This method has also been used in designs of modeling viscous flow around a propeller. There are some other numerical methods such as Lifting Line, Lifting Surface Model and Boundary Element Method which are associated whose simplified assumptions neglect a part of fluid behavior. Therefore, the data resulted from computational fluid dynamics are more validated than of other methods.
In this paper, the parameters effective on the duct performance are initially investigated. Then the primary design is made and finally, computational fluid dynamics simulations are carried out through two and three dimensional analysis.

**Case Study**

The duct or the surrounding duct of the ship propeller could be assumed as a ring hydrofoil which produces a lift force proportionate to the pressure gradient around the foil. This force could be positive or negative and its magnitude depends on the duct geometry. Adding a duct leads to a change in the velocity inside the propeller surface. Velocity variations depend on the propeller geometry and the operational conditions. Operating the propeller inside the duct and the induced velocities generated by it could cause an angle of attack at the inlet of the duct which could be positive or negative relative to the form of the duct and the loading on the propeller [12]. If the inlet angle to the duct is negative, a lifting force is applied to the inner side of the duct, similar to an airfoil, which has a positive axial component. Thereby, as observed in the Fig.1, two kinds of duct exist for propellers which could be used widely. These are called Kurt nozzle (Accelerating duct) and Decelerating duct. Operation characteristics of these two ducts are different to each other.

![Figure 1: (Right) Kurt nozzle (Accelerating duct); (Left) decelerating duct](image)

Decelerating duct is a diffuser duct that reduces the fluid velocity in the propeller location and increases the circumferential pressure in the propeller surface according to the enlargement of cross section. This device has two general benefits; first one is that the installed propeller inside this kind of duct would be resistive to cavitation and the second benefit is the lower noise produced due to the propeller performance because of the reduction in the relative velocity of the fluid passing over the propeller blades, tip vortex prevention and the postponement of cavitation occurrence as well. These features are considered as the main advantages for those kinds of marine vehicles which have to move in shallow waters and the penetration depth of their propulsion system would be negligible and are disposed to cavitation occurrence.

**Design Procedure**

It is necessary to initially investigate the effective parameters on the duct performance and estimate their variation influences on the system efficiency. In the designing procedure of the ducted propeller propulsion system, the purpose is to achieve a design in which either the requirements related to the propeller would be considered or the advantages of adding the duct including efficiency improvement and cavitation prevention could be obtained. These purposes are affected by the operational conditions of flow entering the propeller and the propeller operating pressure as well. This pressure is highly dependent to the depth of the propulsion system performance. In designing the duct, the main object has to be chosen between increasing the efficiency and making the system more resistant to cavitation occurrence. Choosing the main object will affect the duct type.

Changing the geometrical parameters of the duct depends on their effects on three parameters: the axial force produced by the duct, the pressure at the propeller location, the minimum effect of the duct profile on the flow and minimizing the swirling flow at the back of the duct. In fact, there should be no separation on the duct surface in appropriate geometry.

During the design process, one should reach to a design with the considered purposes by selecting different geometrical parameters and their approach of affecting on the propulsion system performance. Furthermore, the velocity variations inside the duct could lead to changes in the hydro dynamical coefficients of the propeller and consequently, getting the propeller out of performance in the maximum open water efficiency condition. Therefore, it will be necessary to repeat the process of selecting the propeller after calculating the velocity inside the duct.

The steps suggested for accessing the considered design are as following:

- Determining the duct length
  - The duct length is stated by terms of the diameter. In [14] the ratio of the duct length to its diameter in single propellers has been accounted as 0.5. Therefore, according to the suggested values for the duct length of the single propeller, one could add a gap between two propellers to obtain an initial design for the length of the pump jet.

- The distance between the blade tip and the internal surface of the duct
  - The distance between the blade tip and the internal surface of the duct has been considered equal to 1.5% in references like [12] and [14].
Geometry of the duct cross section

The next step is selecting the geometry of the duct cross section. This choice includes various geometrical parameters, such as camber ratio, thickness ratio, leading edge radius, position of the maximum thickness and the dihedral angle.

In this part, the initial design is inspired by the geometries suggested by previous works. Using standard geometries of NACA seems more appropriate due to their ability in applying changes on the camber length and moving from converging geometry to the diverging one in designing stage. Therefore, based on the kind of application, one could reach from a decelerating duct to an accelerating one by changing the camber and the dihedral angle. According to the previous applications, the initial value for the dihedral angle is considered between 7 to 11 degrees and the initial value of camber ratio is considered between -1 to 1%.

In order to obtain the proper dihedral angle and camber ratio, their effects on the axial force and the pressure field inside the duct has to be studied. One should observe their variation effects on the propulsion system with simplified assumptions and without any simulation analysis. In computing the axial force produced by the duct, some simplifying assumptions are required. These assumptions include limiting the duct drag coefficient in the flow, using lift coefficient from the thin airfoil theory and assuming the inlet flow to the duct parallel to its cross section chord line. However, the angle of onset flow to the duct has a negative value due to the end shape of the underwater body and also the propeller suction. This negative entrance angle produce a lift force in the direction of the duct which has a radial positive component parallel to the axis (Fig. 2).

![Figure 2: The axial force produced by the duct parallel to the axis](image-url)

\[
T = (C_L \sin \alpha - C_D \cos \alpha) \times 0.5 \rho V^2 \pi D_{\text{mean}} L
\]  
(1)

The above relationship calculates the axial force in the direction of motion in which \( C_L \) is the lift coefficient of the airfoil cross section which is estimated from the thin airfoil theory equal to \( 2\pi(\alpha + 2\frac{f}{c}) \). The amount of \( \alpha \), which is equal to the angle between the relative angle of the onset flow to the duct and the chord line, is considered as zero. In addition, the drag coefficient, \( C_D \), is neglected. In addition \( V \) is the advance velocity, \( D_{\text{mean}} \) is the duct average diameter and \( L \) is the cord length of the duct cross section. The values in the axial force produced are all in terms of Newton.

<table>
<thead>
<tr>
<th>%f/c</th>
<th>( \alpha )</th>
<th>7</th>
<th>8</th>
</tr>
</thead>
<tbody>
<tr>
<td>-1</td>
<td>1085.6</td>
<td>1239.8</td>
<td></td>
</tr>
<tr>
<td>0</td>
<td>0.0</td>
<td>0.0</td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>-1085.6</td>
<td>-1239.8</td>
<td></td>
</tr>
</tbody>
</table>

Table 1: The axial force produced in terms of N

As observed above, increasing the negative camber could lead to a rise in the axial force and increasing the positive one produces drag relative to the duct type.

Numerical Simulation

Two Dimensional Analysis

To simulate the flow, one should model the Navier-Stokes equations in two directions. According to the stability of the flow analysis and its independency to the time, lack of effect of gravity, lack of existence of external forces and compressibility of the structural equations of the flow are all simplified as the following:

\[
\begin{align*}
\partial_t u_j + \partial_j v^k u_k &= -\frac{1}{\rho} \frac{\partial \rho}{\partial x_j} + \nu \nabla^2 u_j \\
\partial_j u_j &= 0
\end{align*}
\]  
(2)
In this research, the body geometry and the duct are modeled asymmetrically alongside body and without the propeller. This way the system simulation has been simpler and less time consuming, but the propeller effects on the system have not been investigated and, instead, FAN boundary condition with the pressure jump equal to the desired thrust will be used. In this study the k-\omega model has been employed to model the turbulence flow, because this method calculates the wall effects on the flow with higher precision. Therefore, through this method, an initial analysis could be done on the duct geometry and the effects of the geometrical parameters changes could be investigated.

**Boundary Conditions**

Boundary conditions in this problem include the inlet velocity, outlet pressure, symmetry and wall. The inlet velocity condition starts from the upper edge of the body of AUV and include the inlet and upper region of the environment. This condition includes the specific velocity parallel to x axis. The reason of using the body profile of the submarine to this analysis is to investigate the boundary layer influences caused by the upstream body on the propulsion system performance. The pressure outlet condition is located at the end of the computational area at the back of the duct. The wall condition includes the body surface of the vehicle and the duct in which the no slip condition is considered. At the bottom and back of the body, the axisymmetric condition has been accounted for the two dimensional axisymmetric study.

![Figure 3: Calculation domain](image)

**The Fan Boundary Conditions**

The boundary conditions for fan have been employed for modeling the effects of propeller suction on the larger flow field. By means of this boundary condition, implementing the operational curve of the fan is allowed in which the head (pressure) and flow rate (velocity) are modeled. This boundary condition gives no details about the flow between the vanes, but the flow rate could be predicted. In this boundary condition, the fan is modeled as a thin plate and the pressure jump is varied as a function of the inlet velocity. This equation could be constant, linear or curved as a defined function by the user [15]. Modeling the fan could be three dimensional as a thin surface between cells and two dimensional as a line. By means of momentum and thrust theory considered for the propeller, the amount of pressure jump in this boundary condition could be modeled.

**Mesh Generation**

While mesh generation, one should pay the following notes:

- The point's concentration is determined based on the required exactness. In points of the flow with high velocity gradient, more concentration on nodes is required rather than points with low gradient. It is physically better to have higher accuracy in all regions, but according to the computational limitations and the computer capacity, this is not possible.
- The direction of nodes positioning should be in the flow path, so that the numerical solution is earlier converged.
- The flow behavior near the wall is considered as a region with high gradient.

![Figure 4: The mesh generated for the numerical simulation](image)

**Results Independency to the Mesh**

In all numerical simulations, one should investigate the independency of the results in regard with the number of meshes. To achieve this goal, the value of horizontal force is obtained by changing the number of meshes and its
independency to the mesh will be investigated. The results from simulations made through this method have been displayed in Fig. (5).

![Figure 5: Independency of the results to number of meshes](image)

Three Dimensional Simulation
The mentioned coding program is based on the finite volume method. The equation for an incompressible single-phase fluid has been used in the simulation, in combination with the implicit solver to find the stationary field of all hydrodynamic unknown quantities. The velocity-pressure coupling and overall solution procedure is based on a SIMPLE C for correction tetrahedral mesh around propeller [15].

Modeling

Through solving the flow and thermal fields in the turbulence regime, an appropriate turbulence model would be required. In this paper, we have employed the RNG k-e model. The RNG-based k-e turbulence model is originated from the instantaneous Navier-Stokes equations by means of a mathematical technique named as "renormalization group" (RNG) method. The analytical derivations lead to a model with constants which differ from the ones in the standard k-e model and the additional terms and functions in the transport equations for k and ε as well.

RNG k-e model is more suitable than the standard k-e model because of the following reasons:

- The RNG model has an additional term in its ε equation which could considerably improve the accuracy of the rapidly strained flows.
- The influence of swirl on turbulence is included in the RNG model which could lead to an enhancement in the accuracy of the swirling flows[15]

Convergence criterion of the calculations consists of two sections, one of which reaching the residual of all equations to 10e-5 and also approaching the resultant forces of the numerical simulation to a fixed value with a smile variation range.

Boundary Conditions

Boundary conditions include the velocity inlet which starts from the upper edge of the body of AUV and include the inlet and upper region of the environment. The pressure outlet boundary condition is considered for the outlet. At the back of the computational domain, length is prolonged the amount that the outlet flow reaches the stable state. The flow field has three computational zones that consist of two fields each one rotating in the rotational velocity (rps) of the adjacent propeller and the computational field for the rest of the environment which is stationary. The connection face between these zones is achieved by means of two planes with distinct mesh and interface boundary condition. When meshing the interface, it is attempted to have the meshes for these planes coincided on each other to have fewer errors while transferring information from the initial mesh to the secondary one.

In this method, all rotary walls with rotational velocities of the rotary coordinate system could be modeled. Such cases include the pumps and turbines vanes in which the relative rotational velocity is equal to zero. Therefore, the propeller surfaces have the boundary condition of rotation relative to the propeller axis. Stationary walls such as shaft or the duct could be modeled through the stationary coordinate system considering the fact that it has a rotational surface in regard with the rotating reference frame In this case, the absolute rotational velocity for these walls is put zero [15].

Grid

Hexagonal elements have been used for the overall domain due to their ability to follow the streamlines close to body, possibility of making them long parallel to the body and thin orthogonal to the body and, eventually, they need to lower node points numbers rather than tetrahedral element; however, for the complex geometry around the propeller, tetrahedral elements have been employed due to their easy generation of elements around it. For generating the hexahedral grid, the domain should be divided into some zones; two of the zones are near the blades, and other zones
are including the remaining parts. In the former, tetrahedral and in the latter hexagonal meshing has been generated. The grid near the blade surface should be fine for the effects of the boundary layer. The $\gamma^+$ in the final case study is ranged between 30 and 300.

The thrust of the propeller was checked for independency to the number of grids. It was found that the thrust varies by about 0.5% when the number of grids is more than 3.8 million (fig 4).

![Figure 6: Independency of the results to number of meshes](image)

**Validation**

At first, a standard propeller which tabulated experimental hydrodynamic coefficients is specified, the coefficients will be extracted from the tables and then the error of simulation results is compared to these values. In the considered propeller, the diameter is 250 mm, the rotation velocity is 2400 rpm and the flow velocity is 8 m/s [17]. According to the above equations, the hydrodynamic coefficients are extracted. Firstly the numerical data related to the propeller by expanded area ratio of 1.2 are compared to those of experimentations and the results are validated. The thrust force and the resultant torque are given here table 1.

<table>
<thead>
<tr>
<th>Thrust(N)</th>
<th>Torque(N.m)</th>
<th>rpm</th>
<th>input velocity(m/sec)</th>
<th>Diameter(m)</th>
<th>KT</th>
<th>KQ</th>
<th>J</th>
<th>Eta</th>
<th>Eta Exp</th>
<th>Error</th>
</tr>
</thead>
<tbody>
<tr>
<td>830</td>
<td>48.9</td>
<td>2400</td>
<td>10</td>
<td>0.25</td>
<td>0.133</td>
<td>0.031</td>
<td>1</td>
<td>0.676</td>
<td>0.690</td>
<td>0.021</td>
</tr>
<tr>
<td>1063.9</td>
<td>59.8</td>
<td>2500</td>
<td>10</td>
<td>0.25</td>
<td>0.157</td>
<td>0.035</td>
<td>0.96</td>
<td>0.680</td>
<td>0.696</td>
<td>0.022</td>
</tr>
<tr>
<td>2435.3</td>
<td>122.2</td>
<td>3000</td>
<td>10</td>
<td>0.25</td>
<td>0.250</td>
<td>0.050</td>
<td>0.80</td>
<td>0.635</td>
<td>0.642</td>
<td>0.011</td>
</tr>
</tbody>
</table>

**Result**

Two and three dimensional simulations have been done on a pump jet propulsion system occupied with an imposed submerged axisymmetric body. The results are mentioned below.

**Two Dimensional Analyses**

These analysis contains an annular duct formed by revolving a NACA 4-digit airfoil, which has a constant camber ratio, around body axis mounted in the eastern part of an axisymmetric submerged body. The selected dihedral angles of the duct cross section ($\alpha$) are between 7 to 11 degree and the FAN boundary condition is defined in the mid part of the duct and the free stream velocity is considered equal to 20 m/sec. During the CAD model preparation of the propulsion system, the middle diameter of the duct is considered to be constant which is proportional to the location of the FAN boundary condition. Therefore, in larger dihedral angles, the inlet diameter increases and the outlet one reduces. After simulating the various produced CAD models through CFD code, as expected, it could be seen that by increasing the dihedral angle the generated thrust force by the duct is reduced and pressure field rises and by having this angle reduced, the pressure inside the duct is fallen and the duct thrust is increased. In this stage, by compromising between resultant force produced by the duct and minimum pressure observed in the duct, an optimum dihedral angle could be found. The next stage of the design procedure is to find the optimum camber ratio for duct cross section. This could be done Like previous stage. After accomplishing the mentioned procedure, a proper geometry for the propulsion system
duct could be concluded. In this article, according to the object requirements, a duct with dihedral angle equal to 8 degree and cross section camber ratio equal to 0.5% is extracted.

<table>
<thead>
<tr>
<th>P min (pascal)</th>
<th>Duct thrust(N)</th>
<th>α</th>
</tr>
</thead>
<tbody>
<tr>
<td>-7.3e04</td>
<td>3311.67</td>
<td>8</td>
</tr>
<tr>
<td>-5e04</td>
<td>1621.87</td>
<td>9</td>
</tr>
</tbody>
</table>

Table 3. The results related to the dihedral angles variations

Figure 7: The velocity field around a duct mounted at the eastern part of a submerged axisymmetric body through 2D axisymmetric CFD simulation in 20 m/sec

Figure 8: The amount of pressure on a duct surface through 2D axisymmetric CFD simulation in 20 m/sec

**Three Dimensional Analyses**

Regarding the disability of the fan model in estimating the propeller operation accurately as well as computing the pressure field around the propellers, it is necessary to model the initial design obtained from two dimensional analyses stage, in the three dimensional simulation and optimize it to make it proper for 3D operation in coupling by real propeller. The dimensionless parameters employed for stating the pump jet propulsion system characteristics, could be expressed as the following:

The thrust constant:
\[
K_T = \frac{T_{fwd} + T_{aft} + T_{dact}}{\rho n_{fwd}^2 D_{fwd}^4} \quad (3)
\]

The torque constant:
\[
K_Q = \frac{Q_{fwd} + Q_{aft}}{\rho n_{fwd}^2 D_{fwd}^5} \quad (4)
\]

The propulsion constant:
In the above equations, $T_{fwd}$, $T_{aft}$ and $T_{duct}$ are the thrust for the first and second propeller and the duct, respectively, $Q_{fwd}$ and $Q_{aft}$ are torque of the first and the second propellers in N.m, respectively, $N$ is the rotational velocity of the first propeller in rps, $D_{fwd}$ is the front propeller diameter in m, $V_{fwd}$ is the fluid velocity entering the first propeller surface in m/sec and $\rho$ is the fluid density. $\eta_{fwd}$ and $\eta_{aft}$ are the propeller efficiency of the first and the second propellers, respectively. In the figure below, the performance curve of the pump jet propulsion system is displayed.

![Pump Jet with body characteristic](image)

**Fig 10:** The pump jet propulsion system operational curve

In figure 9, the produced thrust by the duct is also displayed. As observed here, by increasing the rpm i.e. decreasing $J$ and as a result increasing the propeller suction, the produced thrust by duct is increased.

![Duct Thrust via J variation](image)

**Figure 11:** The duct thrust via J variation

In figures 10 and 11, the velocity and pressure fields are shown around the body.

![Pressure field around the propulsion system](image)

**Figure 12:** The pressure field around the propulsion system in 20 m/sec
In this stage, some modifications on the duct geometry (around the extracted parameters values resulted from 2D analyses) have to be done to make the initial duct geometry to a proper one. As observed in this article, the produced axial force by means of the duct through 3D simulations in objected operating point, is well matched to the result obtained from 2D simulation in that point by an error below 20%.

**Conclusion and Discussion**

According to the numerical results obtained from the simulations carries out on the flow around the duct, both in two and three dimensions, it could be concluded that the two dimensional analysis is an appropriate method for obtaining the initial design for reducing expenses and time consumption of numerical simulations. In addition, in order to make the results reliable and to modify the duct geometry in coupling condition by real propeller, three dimensional model simulations and some modifications based on their results are required. Furthermore, as observed, based on the amount of the propeller suction, propulsion systems with duct could produce thrust or drag with an inverse relationship with the pressure field.

**Reference**

13. R. I. Lewis, " Turbo machinery Performance Analysis"

