Viscous/Potential Flow Coupling Study for Podded Propulsors

Eren ÖZSU¹, Ali Can TAKİNACI², A.Yücel ODABAŞI³

¹TÜBİTAK Marmara Research Center Energy Institute (TÜBİTAK MRC), Kocaeli, Turkey
²,³Department of Naval Architecture and Marine Engineering, Istanbul Technical University (İTU), Istanbul, Turkey

ABSTRACT
Podded propulsors are being widely used in marine propulsion industry in the last decade. Pod designs are varying in terms of pod/strut shapes to improve flow characteristics around the podded propulsors. Flow related properties such as propulsion performance, pod/propeller interaction and vibration/acoustics are investigated by numerical methods.

The main objective of this paper is to present a design approach based on viscous/inviscid coupling for podded propulsor design. Surface panel method is used to calculate the flow around the well known pod/strut model namely Santyr pod A with a given length of 4.53 m. Santyr pod has a NACA66 strut connected to the pod gondola with a smooth intersection line. In the calculations, the surface is divided into separate patches defining the strut and the pod connected with this intersection line.

The pod/strut geometry has been analyzed both in potential and unsteady viscous RANS solver. The viscous flow characteristics around the pod were evaluated. The near field velocity components are derived from the viscous solution and applied to the potential solver as modified boundary condition. Viscous velocity components are set as the new panel source strengths and the potential flow computation is finalized with viscous near field data. The difference in propeller plane velocity contours was plotted and the effect of viscous flow is presented. The results for the potential and viscous calculations are discussed.

Keywords
Pod, forward fillet, boundary layer, velocity coupling

1 INTRODUCTION
Flow around podded propulsors using numerical methods has been analyzed by researchers in the last decade. This important research area focused on critical issues related with propulsion performance, hydrodynamic modeling, cavitation, maneuvering, operational aspects as well as experimental studies and tank tests. "(Atlar (ed.) 2004)" and "(Billard, (ed.) 2006)"

Form design is an important part of hydrodynamic performance since geometry of pod shape and strut highly affect the flow characteristics in the aft part of the ship.

Due to the structural requirements and physical nature of using podded propulsors as a ship rudder, strut sections are relatively large compared to conventional appendages. The diameter of electric propulsion motor is not directly a key parameter for overall pod geometry where strut is a foil shaped component connected to the pod gondola. "(Goubault 2004)" Pod and strut geometry related to hydrodynamic shape is the driving parameter for podded propulsor design. Limited number of publications exists on parametric investigation of pod/strut geometry. Study on systematic series and hydrodynamics was published by "(Islam et al 2006)" and "(Molloy et al 2004)".

The fore part of pod/strut intersection has an important effect on flow characteristics. The present study focuses on leading edge side of pod/strut intersection region.

2 POTENTIAL FLOW MODELLING OF POD/STRUT
Pod and strut is modeled using a surface panel method. The surface discretization is made with 964 quadrilateral panels. The length of the Pod is 4.53[m] and the model is divided into 5 patches. The pod model used in potential flow computation is shown in Figure 1.
In the calculations, free stream velocity is taken as \( V_{\text{inf}} = 4.53 \text{ m/sec} \). Only pod and strut is used in the analyses and the propeller was not modeled.

### 2.1 Off-Body Velocity Calculation on Propeller Planes

The off-body velocity computations were made on propeller planes defined at \( x = 1.075 \text{m} \) and \( x = 3.455 \text{m} \). Propeller plane grid is size is \((80 \times 80) \) panels starting from the local radius of the pod at corresponding \( x \) station to 1 meters height. This constant height value covers the maximum propeller radius sufficiently. More planes are defined in the near field of strut leading and trailing edges in order to calculate the induced velocities around the pod/strut intersection line. Figure 2 shows these off-body planes.

![Figure 2 Off-Body Planes for Velocity Calculations](image)

### 2.2 Wake Modeling

Wake sheet of two pod length is placed starting from the trailing edge of the strut along the longitudinal symmetry axis. A time stepping wake model is used where the shape of the wake sheet is updated in every time step and the local velocities are derived repeatedly. The deformed wake shape can be observed in unsteady analyses at different angles of attack. Figure 3 presents the pod and the wake model used in simulations.

![Figure 3 Wake Sheet Behind Strut Trailing Edge](image)

### 3 RANS MODELLING OF POD/STRUT

Pod/strut is modeled in a commercial viscous RANS solver. The main objective of using a viscous solver is to investigate the velocity profiles in the near field of the pod surface. The free stream velocity is set to \( V_{\text{inf}} = 4.53 \text{ m/sec} \). Using the same velocity input of 4.53 m/sec, both for potential and viscous solvers makes a baseline for comparison and velocity data exchange. \( k-\omega \) SST turbulence model is used for the unsteady solution.

#### 3.1 Grid Size

3.26 Million elements are used to model the flow domain with the unstructured tetrahedral mixed mesh scheme. The grid is shown in Figure 4.

![Figure 4 Grid for Fluid Domain Used in RANS Solver](image)

### 3.2 Unsteady Simulation

In the unsteady solution, time step is selected as 0.5 and the total simulation time is set to be 75 seconds. It is assumed that this 75 seconds time period is sufficient for the unsteady flow to be developed around the pod. Figure 5 shows the velocity contours at \( t = 75 \) seconds. Velocity contours around the pod can be seen shown on XY and YZ planes at desired position. The velocity at the boundaries decreases to zero according to the wall boundary condition. The width of the flow domain can be seen in Figure 5. Symmetry boundary condition is used in XY plane where simulations are carried out for half of the model.

![Figure 5 Velocity Contours at \( t = 75 \) sec.](image)
4 MODELLING OF HORSESHOE VORTEX
Pod/strut junction is a complicated region where the flow properties rapidly change. In the upstream side of the strut, stagnation occurs from free stream velocity to zero where the maximum velocity figures arise over the thick part of the strut. Flow tends to separate in front of the junction forming an horseshoe vortex wrapping in the stagnation region and moving downstream.

As a result of the experimental data based on three different wing nose sections, "(Mehta 1984)" states that the nature of the secondary flow or an horseshoe vortex is highly dependent on wing nose shape. Wing shape is the major factor and largely determines the pressure gradients in front of and around the wing and the rate of stretching of the horseshoe vortex. "(Olcmen & Simpson 2006)"

Formation of a horseshoe vortex around the relatively thick strut causes undesirable effects in the flow field. Acoustic and vibration may be the major considerations because of the existence of propeller action before and after strut. Uniform velocity distribution in the propeller planes is important for propulsion performance.
Streamlines and velocity vectors are used to simply understand the existence of the horseshoe vortex around the pod/strut intersection region. Streamlines starting from pod/strut intersection are plotted both for viscous and potential flow solver. Results are obtained for the same velocity input $V_{inf}= 4.53$ m/sec at zero angle of attack.

4.1 Horseshoe Vortex Computed in Panel Method
Streamlines calculated from surface panel code are shown in Figure 6. Figure 7 and Figure 8 shows the off-body streamlines regarding to the vortex formation by top and side views respectively. The distance where the streamlines are located in the x direction is 0.1 meters starting inside the fluid domain to the strut surface along the symmetry axis. The calculated velocity magnitude can be seen in contour map format for the stagnation region. Streamlines show the wrapping vortex near the symmetry plane and moving downstream. Velocity vectors on the streamlines provide information about local velocities around the pod.

4.2 Horseshoe Vortex Computed in RANS Solver
Streamlines calculated from unsteady viscous solver are shown in Figure 9. Figure 10 shows the difference in wrapping streamlines and freestream in front of the strut and along the pod.
BOUNDARY LAYER VELOCITY CALCULATIONS

Unsteady viscous analyses were made to obtain velocity profiles around the pod. The inflow was taken as $V_{in} = 4.53 \text{ m/sec}$ at zero angle of attack. Turbulence model is selected as $k-\omega$ SST and the $5\%$ turbulence intensity is set with a viscosity ratio of 10. The main objective is to analyse the near field velocity profile.

5.1 Boundary Layer Velocity Profiles

Point velocities are obtained within the boundary layer in viscous solver to provide data for velocity coupling between two models. The boundary condition in inviscid solver is then modified by the exported viscous near field velocities. The velocities in inviscid solver is then recalculated with modified panel source strengths and viscous effects around the pod/strut model is represented. Figure 11 presents the velocity distribution in front of the pod/strut junction where the local velocity decreases in the stagnation region.

6 GEOMETRY MODIFICATION FOR POD/STRUT JUNCTION

A foil shaped strut and an ellipsoid have curved surface patches intersecting on a line. After discretization of real surfaces by quadrilateral panels, the edges of the neighboring panels are matched along the intersection line. With all the section data are known it is possible to place an additional surface patch to construct the junction smoothly. Constant or variable radius patches can be used for this purpose. This geometry modification process can be done either using complex surface construction methods or more simple section selection based on designer’s experience or aid. Research publications exist on surface construction in the literature.

6.1 3D Surface Blending for Pod/Strut

The intersection line can be replaced by using surface blending in two options.

6.1.1 Constant Radius Blending

The intersection line is fully replaced by a constant radius surface patch. Figure 13 presents an example of placing a 0.16 meter radius surface patch. Strut has two fillets on the symmetry axis.

6.1.2 Forward Fillet with Variable Radius

The intersection line can partly be replaced by a varying fillet radius resulting a blended forward fillet. The maximum width of the fillet is set to the strut thickness. Trailing edge of the strut and pod left original. Table 1 shows the starting and ending points of the fillet on the symmetry plane along x axis.
Table 1 Forward Fillet Geometry

<table>
<thead>
<tr>
<th>Forward Fillet*</th>
<th>Radius in meters</th>
<th>Starts at x =</th>
<th>Ends at x =</th>
<th>Width</th>
</tr>
</thead>
<tbody>
<tr>
<td>FF02</td>
<td>0.2</td>
<td>1.460</td>
<td>1.665</td>
<td>0.518</td>
</tr>
<tr>
<td>FF03</td>
<td>0.3</td>
<td>1.355</td>
<td>1.665</td>
<td>0.518</td>
</tr>
<tr>
<td>FF04</td>
<td>0.4</td>
<td>1.249</td>
<td>1.665</td>
<td>0.518</td>
</tr>
</tbody>
</table>

* FF0X notation: Forward Fillet with R=0. meters radius

7 RESULTS

4 different pod/strut geometry were analyzed including Santyr Pod and three pod models fitted with forward fillets. In the analyses, the effect of propeller has not been taken into consideration and propeller plane velocities were focused.

The difference on surface pressure distribution was plotted in contour map format for the fore part and the blended region of the pod body. Streamlines and velocity vectors on each streamline were plotted starting from the intersection region. In order to avoid repeating graphics and figures, results of Santyr Pod and the pod that has the largest fillet, FF04 were used for comparison.

7.1 Streamlines

Streamlines are shown in Figure 14. Streamlines located in the near field of fillet move smoothly upwards to form a horseshoe vortex at the unblended part of the strut.

7.2 Propeller Plane Velocities

Propeller planes are defined at two locations in the upstream and downstream part of the strut at x = 1.075 and x = 3.455 m respectively. Black solid contour lines represent the pod with forward fillet radius of 0.4 meters.

7 CONCLUSION

The purpose of this paper is to present the effects of leading edge filleting to the propeller plane velocity distribution based on a viscous/inviscid velocity coupling.
Forward filleting positively affect propeller plane velocity distribution. Investigation of the location of the vortex core for different fillets, capturing more accurate boundary layer velocities with more dense meshes with a reduced $y^+$ values will be the topics for future studies.

Moreover, covering propeller effects and the induced velocities over propeller planes will be valuable contributions to build up a quick and reliable design approach.

REFERENCES
Atlar, M. (ed.) (2004). First International Conference on Technological Advances in Podded Propulsion, School of Marine Science and Technology, University of Newcastle, UK.


