NUMERICAL COMPUTATION OF TIP VORTEX FLOW GENERATED BY A MARINE PROPELLER

Chao-Tsung Hsiao and Laura L. Pauley
Mechanical Engineering Department
The Pennsylvania State University
University Park, PA 16802

ABSTRACT
The uniform flow past a rotating marine propeller was studied using incompressible Reynolds-averaged Navier-Stokes computations with the Baldwin-Barth turbulence model. Extensive comparison with the experimental data was made to validate the numerical results. The general characteristics of the propeller flow were well predicted. The current numerical method, however, produced an overly diffusive and dissipative tip vortex core. Modification of the Baldwin-Barth model to better predict the Reynolds stress measurements also improved the prediction of the mean velocity field. A modified tip geometry was also tested to show that an appropriate cross section design can delay cavitation inception in the tip vortex without reducing the propeller performance.

1. INTRODUCTION
The tip vortex flowfield in the vicinity of the tip region is a very complicated three-dimensional viscous flow phenomenon. The flow in the tip region can significantly affect the performance of lifting surfaces in both aerodynamic and hydrodynamic applications. For example, tip vortex cavitation is of major concern for marine propellers since it is an important source of noise. In order to avoid or control tip vortex cavitation, the underlying flow physics needs to be fully understood.

Using advanced flow visualization and non-intrusive measurement techniques, experimental studies can now reveal the detailed features of the tip vortex flow around a marine propeller configuration. They, however, still suffer from the non-conclusive study of the pressure field which is always crucial to the prediction of cavitation inception. Navier-Stokes computations, which can provide a detailed pressure field, have been recently used to predict the tip vortex flow. However, most of the numerical efforts (e.g. Copenhaver, et al., 1994; Furukawa et al., 1995; Lee et al., 1996) are made on the tip-clearance flow found in turbomachinery-type geometries while relatively few studies have been done on the open propeller-type tip vortex flow. Unlike the tip-clearance flow, the tip vortex generated by the marine propeller is usually more concentrated and has a tighter structure, which will require a more refined grid within the tip vortex core. For the propeller flow, Stern and Kim (1990) and Kim (1993) presented numerical results for a simplified propeller with infinite-pitch rectangular blades and for a practical propeller model respectively. These works demonstrated how a Navier-Stokes computation can simulate the general features of the propeller flow, but did not provide the detailed features of the tip vortex flow. The numerical calculation of Oh and Kang (1995) is one of the few numerical studies which focus on the propeller-type tip vortex flow. Although Oh and Kang were able to adequately estimate the thrust and torque coefficients in comparison with experimental data, they failed to predict the strength of the tip vortex. It is known that both turbulence model and grid resolution within the tip vortex core have a profound effect on the prediction of the tip vortex. Regardless of the turbulence model used, it is unlikely that they would obtain accurate prediction due to insufficient grid resolution within the tip vortex core since only 0.2 million grid points were used in their numerical calculation.

In the present study, the three-dimensional Navier-Stokes flow solver INS3D-UP developed by Rogers et al. (1991) is used to calculate the flow around a marine propeller configuration. The one-equation turbulence model developed by Baldwin-Barth (1990) is applied to test its accuracy and efficiency on the prediction of the tip vortex flow. In order to validate the numerical predictions, the present numerical method is applied to calculate the David Taylor Propeller 5168 with uniform flow conditions. Chesnakas and Jessup (1998) carried out extensive experiments for the present model using an LDV system in the water tunnel and provided substantial measured data on the flow velocities in the tip vortex. Extensive comparison between calculation results and experimental data is made to demonstrate the capability of the current Navier-Stokes computations to handle propeller flows.

Copyright © 1998 by ASME
2. NUMERICAL IMPLEMENTATION

2.1 Geometry and Grid Generation

The current study considers a rotating propeller operating in a uniform flow. The propeller model considered is the David Taylor Propeller 5168 which is a five-blade propeller with 15.856 inch diameter. Since the hub and blade root area of the propeller blade are overly complicated, some geometry simplifications are made for the numerical study. The blade flange, root fillets and a root trailing edge cut-out, which produces a gap between the blade and the hub, are ignored. Instead, propeller blades are assumed to be mounted on an infinite constant-radius hub/shaft cylinder. The computational domain is established as one blade-to-blade passage in which there are two side-boundaries, one for suction side and one for pressure side, formed by following the inlet flow angle. The complete computational domain is constructed by locating the inlet boundary 1.8 propeller radii, R, upstream and locating the outlet boundary two propeller radii downstream of the propeller midplane. The outer boundary in the radial direction is located at two propeller radii while the inner domain is bounded by the hub/shaft surface which is assumed to be a constant-radius cylinder.

To create an appropriate three-dimensional grid inside the computational domain, a combination of algebraic and elliptic grid generation techniques as described in Hsiao and Pauley (1996) is applied. The advantage of this grid generation technique is that one can easily distribute the initial grid according to the flow field with an algebraic method and then smooth the grid by applying the elliptic smoothing routine. An H-H-type single-block structure grid is generated for the current computational domain. The grid generation procedure starts with generating the surface grid on the blade and hub/shaft surfaces as shown with every other grid line in Figure 1. A two-dimensional grid is created on each constant-radius plane based on the surface grid. Each two-dimensional grid is generated using only the algebraic method or a combination of algebraic and elliptic methods. The three-dimensional initial grid is established by stacking all two-dimensional grids. Finally, the elliptic smoothing routine is applied to smooth the initial grid. An H-H type grid with a total of 2.4 million points is created for the current computational domain. In this grid, 10³ of the 211 streamwise grid points and 81 of 111 radial grid points are used on the propeller blade surface while 101 grid points are used in the blade-to-blade direction. The first grid spacing is specified as $1 \times 10^{-5}$ and $1 \times 10^{-4}$ chord length on the blade surface and on the hub/shaft surface respectively. As suggested in previous numerical studies of tip vortex flow around a finite-span hydrofoil (Dacles-Mariani et al., 1995 and Hsiao & Pauley, 1996), a grid spacing of at least 15 points across the vortex core is used to obtain a reliable near-field tip vortex flow. The overall features of the computational grid and surface grid are shown with every other grid line in Figure 2.

2.2 Numerical Method

The present computations are conducted on a rotating frame which is fixed on the propeller blade. The steady-rotating reference frame source terms, i.e., the centrifugal and Coriolis force terms, therefore, are added to the Reynolds-Averaged Navier-Stokes equations derived in the inertial frame. The three-dimensional incompressible Navier-Stokes flow solver INS3D-UP, developed by Rogers et al. (1991), is applied to calculate the rotating propeller flow. The INS3D flow solver is based on Chorin’s artificial-compressibility approach (1967). In the artificial-compressibility method, a time derivative of pressure is added to the continuity equation to couple it with the momentum equations. As a consequence, a hyperbolic system of equations is formed and can be solved using a time-marching scheme. This method can be marched in pseudo time to reach a steady-state solution. In this code, the first-order Euler implicit difference formula is applied to the pseudo-time derivatives. The spatial differencing of the convective terms uses a fifth-order accurate flux-difference splitting based on Roe’s method (1981). A second-order central differencing is used for the viscous terms. The resulting system of algebraic equations is solved by a Gauss-Seidel line-relaxation fully implicit method in which several line-relaxation sweeps through the computational domain are performed before the solution is updated at the new pseudo-time step. In the present study, a steady-state solution is acquired when the maximum residual reduces four orders.
The INS3D-UP code is accompanied by the Baldwin-Barth
one-equation turbulence model (Baldwin & Barth 1990) which
is derived from a simplified form of the standard k-ε equations.
This model is not only simpler than the two-equation model,
but also eliminates the need to define the turbulent mixing
length which is required in Baldwin-Lomax algebraic model.

2.3 Boundary Conditions

All the boundary conditions are treated in an implicit
manner except the outlet boundary. The boundary conditions
on each of the boundaries are as follows: free stream
conditions are specified for all variables at the inlet and outer
radial boundaries; the no-slip condition is applied on the blade
and the hub/shaft surfaces; a periodic boundary condition is
specified on the side-boundaries. For the outlet boundary, a
mass and momentum weighted extrapolation method as
suggested by Chang et al. (1985) is adopted. A first order
extrapolation is applied to update all velocity components, \( \tilde{u}^n \),
and pressure, \( p^n \), at the outlet boundary from interior points.
Then the velocities are weighted to conserve the inlet mass
flux as

\[
\tilde{u}^n = \frac{\tilde{m}_\text{in}}{\tilde{m}_\text{out}} u^n
\]

where \( \tilde{m}_\text{in} \) and \( \tilde{m}_\text{out} \) are the mass fluxes obtained by
integrating the velocity across the inlet and outlet boundaries.
Corresponding to the mass-weighted velocities, a new pressure
must be provided to satisfy the momentum equations in order
to maintain stability. As a result, the pressure at the outlet
boundary corresponding to the mass-weighted velocities is
derived from \( \xi \)-momentum equation as

\[
p^n = p^n - \frac{1}{\xi_x} \left[ (uU)^\xi - (uU)^n \right] + \frac{v}{\xi_x} \left( \nabla \xi \cdot \nabla \xi \right) \left[ \frac{\partial u}{\partial \xi} \tilde{u}^\xi - \frac{\partial u}{\partial \xi} u^n \right]
\]

where \( u \) is the velocity component in the axial, \( x \), direction and \( U \)
is the contravariant velocity in the streamwise, \( \xi \), direction.
Equation 2 is derived by assuming that the variation of the
velocities normal to the \( \xi \) direction is negligible, i.e. the
streamlines near the outlet boundary are assumed to be nearly
straight. In numerical computations, if a mild discrepancy
from this assumption occurs, the resulting pressure will show a
discontinuity in the region where changes in the streamlines
are significant. Since the streamlines for the tip vortex flow
still show a high curvature near the outlet boundary, a
modification is made for the momentum-weighted pressure by

\[
p^n = \left( \frac{\tilde{I}_p}{I_p} \right) p^n
\]

where \( \tilde{I}_p \) and \( I_p \) are obtained by integrating the pressure \( p^n \)
and \( \tilde{p}^n \) across the outlet boundary respectively. The velocities
and pressure obtained by this method will maintain the proper
shape of the streamlines and at the same time conserve the
mass and momentum fluxes.

3. RESULTS

In the present study the numerical computations were
conducted at three different advance coefficients, \( J = V/nD =
0.98, 1.10, 1.27 \), where \( V \) is the axial velocity, \( n \) is the
propeller rotating rate, and \( D \) is the propeller diameter. These
three advance coefficients correspond to three different
Reynolds numbers, \( Re=3.40, 4.19, 3.88 \times 10^6 \), which are based
on the propeller blade chord length at 0.7R section and the
vector sum velocity of the inflow velocity and the rotational
component. The numerical results are first extensively
compared with the experimental data measured by Jessup
(1996) to validate the current numerical method. A modified
tip geometry is then applied to investigate its effect on the tip
tip vortex structure in which we demonstrate how the current
numerical method can be applied to be a powerful design tool.

3.1 Validation

Comparison of three velocity components (axial velocity, \( V_x \),
tangential velocity, \( V_r \), and radial velocity, \( V_z \)) for the basic
case, \( J = 1.10 \), between the present numerical solution and the
experimental data at downstream location \( x/R = 0.2386 \) is
shown in Figures 3a-c. The axial direction, \( x \), is measured
from the propeller midspan at the hub. In Figures 3a-c, the
experimental data did not include the flow information near
the hub/shaft surface. From the comparison, it is seen that the
current numerical result shows a very good agreement with the
experimental data in the tip vortex flow, wake and blade-to-
blade flow. For a better description of the tip vortex structure,
a new primary/secondary coordinate system is defined in Figure
4. In this coordinate system, the primary velocity, \( V_p \), is defined
as being in the axial-tangential \( x-t \) plane at the propeller pitch
angle, \( \phi \). The secondary velocities (tangential velocity, \( V_r \),
and radial velocity, \( V_z \)) are the velocity components on the
secondary-flow plane which is normal to the primary velocity.
Since the propeller pitch angle varies in the radial direction,
the primary and tangential velocities are actually calculated at
each radial location by

\[
V_s = V_x \sin \phi(r) + V_r \cos \phi(r)
V_c = -V_r \cos \phi(r) + V_z \sin \phi(r)
\]

In this coordinate, the tip vortex axis is virtually normal to the
secondary-flow plane and the tip vortex structure such as the
vortex core can be better defined. Figure 5 shows comparison of
the primary velocity between the numerical result and
experimental data at \( x/R = 0.2386 \).

The most interesting area of the flow field is the tip
vortex. To validate the current prediction of the tip vortex, the
close-up view of the tip vortex is compared and shown in
Figures 6 at \( x/R = 0.2386 \). It is seen that the current numerical
result under-predicts the tip vortex strength with about 10%
difference in the minimum \( V_s \). For better quantitative
comparison between the numerical and experimental results,
Figure 3a Comparison of axial velocity $V_x$ between numerical and experimental results at $x/R=0.2386$.

Figure 3b Comparison of tangential velocity $V_y$ between numerical and experimental results at $x/R=0.2386$.

Figure 3c Comparison of radial velocity $V_r$ between numerical and experimental results at $x/R=0.2386$.

Figure 4 The primary/secondary coordinate system.

The line plots of velocities, $V_x$, $V_y$, $V_r$, across the tip vortex center in the tangential direction are shown in Figures 7 and 8 at $x/R=0.1756$ ($r/R=0.934$) and 0.2386 ($r/R=0.918$). It is noted that the position of the vortex center is specified as $\theta=0$ in these plots. The vortex center is defined at the location where the minimum $V_r$ occurs within the tip vortex and the experimental data is obtained by averaging the velocities from all five tip vortices. In Figure 7 and 8, from left to right the first valley of $V_x$ and $V_r$ corresponds to the wake while the second valley is associated with the tip vortex. It is seen that the tip vortex is better predicted at the location closer to the propeller while the wake is better predicted at the farther location. This may indicate that the eddy viscosity calculated from the turbulence model is too large within the tip vortex.
and leads to an overly diffusive and dissipative tip vortex. Since the current grid has at least $16 \times 20$ grid points within the tip vortex core, which has shown to be adequate for resolving the tip vortex flow in the previous numerical studies, the discrepancy between the numerical and experimental results is likely caused by the one-equation turbulence model used in the current study. Further discussion on the turbulence modeling will be given in next section.

From the numerical result, the pressure distribution along the tip vortex core is of interest in the study of cavitation inception since the experimental study is unable to directly provide this information with a non-intrusive measurement. To show the pressure distribution along the tip vortex core, we define the vortex core as the location where the local minimum pressure coefficient occurs at the local cross plane normal to the axial direction. The pressure coefficient along the tip vortex core, as shown in Figure 9, indicates that the location of minimum pressure coefficient, $C_{p_{\text{min}}}$, is very close to the tip trailing edge. This also implies that the tip vortex cavitation inception will also occur near the tip trailing edge. It is seen that the negative minimum pressure coefficient increases as the advance coefficient is decreased, i.e. the propeller loading is increased. Although real flow effects such as random turbulent fluctuation, water quality, etc. are known to influence cavitation inception, it is expected that the $-C_{p_{\text{min}}}$ and cavitation inception number, $\sigma$, should be comparable. Table 1
show a comparison between \(-C_{\text{pmin}}\) from the computations and \(\sigma_t\) measured by Jessup for three different advance coefficients. Also listed is \(x/R\), the streamwise location of \(C_{\text{pmin}}\). The experimental uncertainty for \(\sigma_t\) is ±0.4.

### Table 1. Comparison between \(-C_{\text{pmin}}\) and \(\sigma_t\) for three different advance coefficients \(J\)

<table>
<thead>
<tr>
<th>(J)</th>
<th>(\text{Re})</th>
<th>(-C_{\text{pmin}})</th>
<th>(\sigma_t)</th>
<th>(x/R)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.98</td>
<td>(3.40\times10^6)</td>
<td>4.67</td>
<td>4.88</td>
<td>0.054</td>
</tr>
<tr>
<td>1.10</td>
<td>(4.19\times10^6)</td>
<td>2.46</td>
<td>2.04</td>
<td>0.058</td>
</tr>
<tr>
<td>1.27</td>
<td>(3.88\times10^6)</td>
<td>1.36</td>
<td>1.11</td>
<td>0.059</td>
</tr>
</tbody>
</table>

Since the performance of the marine propeller is usually determined by the thrust and torque, it is also of interest to compare the thrust coefficient, \(K_t\), and the torque coefficient, \(K_q\). The \(K_t\) and \(K_q\) are defined as

\[
K_t = \frac{T}{\rho n^2 D^4}, \quad K_q = \frac{Q}{\rho n^2 D^5}
\]

where \(T\) and \(Q\) are the thrust and torque acting on the propeller in the axial direction. The computation of thrust and torque included the surface friction. The experimental data of the thrust and torque coefficient for Prop 5168 were measured in an open water test. Comparison of \(K_t\) and \(K_q\) between numerical and experimental results are shown in Table 2 for three different advance coefficients. It is seen that current numerical result predicts \(K_t\) and \(K_q\) well for higher advance coefficient, but the discrepancy increases as the advance coefficient is decreased.

### Table 2. The comparison of \(K_t\) and \(K_q\) for three different \(J\)

<table>
<thead>
<tr>
<th>(J)</th>
<th>Comp.(K_t)</th>
<th>Exp.(K_t)</th>
<th>Comp.(K_q)</th>
<th>Exp.(K_q)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.98</td>
<td>0.404</td>
<td>0.371</td>
<td>0.0901</td>
<td>0.0888</td>
</tr>
<tr>
<td>1.10</td>
<td>0.333</td>
<td>0.313</td>
<td>0.0781</td>
<td>0.0783</td>
</tr>
<tr>
<td>1.27</td>
<td>0.238</td>
<td>0.229</td>
<td>0.0606</td>
<td>0.0618</td>
</tr>
</tbody>
</table>

### 3.2 Modification of Turbulence Model

From comparison of Figure 6, it is seen that the present computation over-predicted the level of eddy viscosity in the vortex core which lead to an overly diffusive and dissipative tip vortex. To demonstrate the effect of eddy viscosity on the solution a simple modification of the Baldwin-Barth one-equation model as described by Dacles-Mariani et al. (1995) is applied. In the standard Baldwin-Barth one-equation model the production term \(P\) is approximated by

\[
P = C_l \nu R_c S
\]

where \(C_l\) is a constant, \(\nu\) is the laminar viscosity, \(R_c\) the turbulent Reynolds number, and \(S\) is a scalar measure of the deformation tensor. Since \(S\) is approximated by the magnitude of vorticity \(\omega_l\) in the Baldwin-Barth model, it is expected that the eddy viscosity will be over-predicted if the flow conducts pure solid body rotation such as in the vortex core. To reduce the eddy viscosity in the region where the vorticity exceeds the strain rate, Dacles-Mariani et al. modified the production term as

\[
P = C_l \nu R_c (\omega_l + c \min(0, 1 - \omega_l))
\]

where \(\omega_l\) is the strain rate and \(c\) is an arbitrary constant. Since the factor \(c\) is chosen arbitrarily, this modification only represents an attempt to empirically adjust the production term for vortex-dominated flow. Figure 10 shows the primary velocity \(V_t\) around the tip vortex for \(c=2\). As compared to Figure 6, one can see the prediction of minimum of \(V_t\) is significantly improved. To show the influence of the modification on the eddy viscosity, one of the Reynolds stress components \(v_x v_t\) is shown in Figure 11 for experimental and numerical results. It was found that although the magnitude of the Reynolds stress is improved with the modification, the distribution of strain rate is not predicted well. This indicates that it is difficult to accurately model the Reynolds stress for a vortex-dominated flow using a simple eddy viscosity turbulence model. Since the turbulence around a tip vortex is thought to be highly anisotropic and the Coriolis force in the rotating propeller can also induce anisotropic turbulence, anisotropic turbulence models such as Reynolds-stress models may, therefore, be required to resolve accurately the tip vortex of the rotating propeller.

### 3.3 Effect of Modified Tip

Previous experimental studies (Fruman et al. 1991, Pauchet et al. 1993) have confirmed that the hydrofoil cross section has a profound influence on the tip vortex cavitation inception and desinence. Hsiao (1996) further numerically studied the cross section effect on the pressure distribution along tip vortex core and on the hydrofoil surface. It was found that the cross section which induces an early tip vortex rollup will entrain more low momentum boundary layer flow into the tip vortex and attenuate the tip vortex strength. They concluded that an appropriate hydrofoil cross section can be designed to delay cavitation inception both on the hydrofoil surface and in the tip vortex.

In the present study, a modified tip geometry designed by Jessup in ONR's progressive research of propeller tip vortex flow is numerically investigated. The modification was made
Figure 10  $V_r$ contour around the tip vortex for modified production term case at $x/R=0.2386$.

$J = 1.1 \times R = 0.2386$ Experiment

$J = 1.1$ Pressure Side

Original Tip

Modified Tip

Figure 11 Comparison of the Reynolds stress component at $x/R=0.2386$.

Figure 12a Comparison of $-C_p$ on the pressure side between the original and modified tip geometries.

Figure 12b Comparison of the surface pressure coefficient on the suction side between the original and modified tip geometries.

by increasing the thickness of the blade tip about 50% gradually from 0.8R section to the tip. The computation for the modified tip case was conducted at $J = 1.10$. The pressure...
distribution on the blade surface is shown in Figure 12a-b for the original and modified tip geometry cases. From the pressure distribution on the suction side, one can see that the modified tip has a larger low pressure region near the tip trailing edge, which indicates that the tip vortex rollup position moved further upstream for the modified tip case. Comparison of the pressure distribution along the tip vortex as shown in Figure 13 confirms that this earlier tip vortex rollup reduces the negative minimum pressure coefficient in the tip vortex core. Since the $K_1$ and $K_2$ are almost the same for both cases (less than 1% difference), it is concluded that an appropriate cross section design can delay the cavitation inception in the tip vortex without reducing the propeller performance.

4. CONCLUSIONS

The tip vortex flow generated by a marine propeller was numerically studied using Reynolds-Averaged Navier-Stokes computations with a Baldwin-Barth turbulence model. The general characteristics of the propeller flow including the blade-to-blade flow, wake, and tip vortex are well predicted by the present numerical method. The close-up view of the tip vortex, however, shows that the numerical result slightly under-predicts the tip vortex strength. Modifying the turbulence model to better match the experimental measurements of the Reynolds stresses also improved the prediction of the mean velocity field.

The numerical computation of the modified tip geometry shows that increasing the thickness of the blade cross section near the tip will induce earlier tip vortex rollup and result in a weaker tip vortex.

ACKNOWLEDGMENTS

This research has been supported by the Office of Naval Research under contract N00014-96-1-1149 monitored by Dr. Edwin P. Rood. Computational facilities were provided by the NAVOCEANO Supercomputer Center and the CEWES High Performance Computing Center. These contributions are grateful acknowledged.

REFERENCES


