CFD Simulation for Propeller Four-Quadrant Flows

Sing-Kwan Lee (M)


ABSTRACT

A Reynolds-Averaged Navier-Stokes (RANS) method has been employed in conjunction with an overlapping moving grid approach to provide accurate resolution of four-quadrant propeller flows under both the design and off-design conditions. It is well known that some off-design propeller flow phenomena are dominated by viscous effects and cannot be accurately predicted by the potential flow methods. In order to properly account for viscous effects, it is necessary to employ accurate and robust numerical methods which can provide detailed resolution of the propeller boundary layer, turbulent wake, leading edge separation, and unsteady ring vortices induced by propeller operations under off-design conditions. In this study, time-domain simulations are performed for the DTRC 4118 propeller under ahead, bollard-pull, crash-ahead, crash-astern, and backing conditions and compared with the available experimental data.

INTRODUCTION

The potential flow methods based on the assumptions of inviscid fluid and irrotational motion are widely used in propeller flow analysis. Using the methods, propeller performance at near design conditions can be predicted quite accurately. However, some off-design propeller flow phenomena are dominated by viscous effects and cannot be accurately predicted by the potential flow methods. Off-design conditions in fact can occur in all four quadrants as defined by the ship velocity $V_s$ and the propeller angular velocity $\omega$. The four modes of propeller operation are defined as ahead or forward ($+V_s, +\omega$), backing or astern ($-V_s, -\omega$), crash-ahead or reverse backing ($-V_s, +\omega$) and crashback or crash-astern ($+V_s, -\omega$).

During crash-astern and crash-ahead operations, the reversal of propeller rotation creates a relatively large angle of attack, causing the flow to separate at the leading edge of the blade. Figure 1 shows the velocity vectors from RANS simulation for the DTRC 4118 propeller under crash-astern operation (Chen & Lee, 2004). As shown, in the mid-span region of the blade passage, a large separation region can be clearly seen. Also, the water tunnel measurements performed by Jiang et al. (1997) demonstrated that the unsteady ring vortices could appear near blade tip region even when the propeller is operated in the steady crash-astern condition. In addition to these complex flow separation and unsteady ring vortices, another unusual feature in crash-astern/ahead operation is its inflow direction. As known, for ahead condition, the propeller inflow direction is always same as the free stream flow velocity. However, under crash-astern/ahead operation, depending on the value of advanced coefficient, J, the inflow towards propeller disk may be in the opposite direction of the free stream flow due to the strong recirculation flows – ring vortices. Figure 2 is Laser Doppler Velocimetry measurement results (Jessup et al., 2005) for a conventional propeller 4381 under crash-astern condition at $J = -0.5$. According to Jessup’s observation, at $J = -0.5$, these tip ring vortices moved inward and then outward to propeller center, however, no obvious frequency was observed. Unlike Jessup’s observation, in Jiang et al’s experiment for the same propeller at $J = -0.472$, it is reported that the ring vortex generated first at blade tip and broke up into two in downstream. This periodic ring vortex shedding repeated with a period of 2.5 seconds and caused the measured shaft force oscillating with 2.1 second period.

Fig. 1 Large separation flow around leading edge (trailing edge for ahead condition) of DTRC 4118 propeller under crash-astern operation.
As seen in Figure 2, the time-averaged flow pattern of this unsteady ring vortex flow shows its center at around 1.5 propeller radius slightly downstream to the propeller disk. Also, as mentioned earlier, propeller inflow direction is no longer along the free stream direction (from left to right) but opposite to free stream direction (from blade back side to face side).

It should be also noted that the opposite flow direction in crash-astern condition is dependent on propeller geometry and operating J. Figure 3 shows velocity vector plot based on RANS simulation for the other propeller (DTRC 4118 propeller) at J = -0.4 under crash-astern condition. Unlike the 4381 propeller, basically, the inflow toward propeller still maintains in the free stream flow direction even though a clear ring vortex appears near blade tip.

In order to properly account for viscous effects and complex flow pattern, it is necessary to employ accurate and robust numerical methods which can provide detailed resolution of the propeller boundary layer, turbulent wake, leading edge separation, and unsteady ring vortices induced by propeller operations under off-design conditions.

In this paper, a Reynolds-Averaged Navier-Stokes (RANS) method is employed in conjunction with an overlapping moving grid approach to provide accurate resolution of four-quadrant propeller flows under both the design and off-design conditions. In this study, time-domain simulations are performed for the DTRC 4118 propeller under ahead, bollard-pull, crash-ahead, crash-astern, and backing conditions and compared with the available experimental data.

**FORMULATION AND NUMERICAL METHOD**

Details of the numerical methods and turbulence model used in the present study are based on Potaza et al (2005) and Chen and Petal (1988). The generalized equations and solution procedures are described as follows.
Governing Equations and Turbulence Model

Consider the non-dimensional Reynolds-Averaged Navier-Stokes equations for incompressible flow in the most general form in curvilinear coordinates ($\xi, \tau$):

$$ U_j = 0 $$

$$ \frac{\partial U^i}{\partial t} + U^j U_{ij} + u^i u_{,j} = \frac{1}{Re} g^{ik} U_{,jk} = F^i $$

where $U^i$, $u^i u^j$ and $p$ represent the contravariant mean velocity, Reynolds stress, and fluctuating pressure, respectively. The quantities $g_{ij}$ and $g^{ij}$ represent metric and conjugate metric tensors in general curvilinear coordinates with the following relationship:

$$ g^{ij} = \frac{1}{J} (g_{ij} u^i - g_{ij} u^j) $$

where both $(i,j,k)$ and $(m,n,l)$ are in cyclic order, and $J$ is the associated metric Jacobian. In equation (2), $t$ is time and $F^i$ are the body forces. The Reynolds number $Re = U_L L/v$ is based on the reference length, $L$, and velocity, $U_L$, used to non-dimensionalize the equations. Equations (1) and (2) represent the continuum and mean momentum equations, respectively. The equations are written in tensor notation with the subscripts, $j$ and $jk$, represent the covariant derivatives.

If turbulence is assumed isotropic and the Boussinesq approximation is employed, Reynolds stresses can be written:

$$ -u^i u^j = 2\nu \epsilon^{ij} - \frac{2}{3} g^{ij} k $$

$$ \epsilon^{ij} = \frac{1}{2} (g^{ik} U_{ij} + g^{jk} U_{ji}) $$

where $\nu$ is the eddy viscosity and $k = g_{ij} u^i u^j / 2$ is the turbulent kinetic energy. The quantities $\epsilon^{ij}$ represents contravariant and physical components of the rate-of-strain tensor respectively. Substituting (4) into (2) yields momentum equations ready for eddy viscosity turbulence modeling:

$$ \frac{\partial U^i}{\partial t} + U^j U^i_{,j} + \epsilon^{ij} (p + \frac{2}{3} k)_{,j} - 2\nu \epsilon^{ij} = -\frac{1}{Re} g^{ik} U_{,jk} = F^i $$

the quantity $1/Re = 1/Re \cdot \nu / \epsilon$ represents the effective turbulent viscosity.

Equations (6) are closed using a modified version of the two-layer turbulence model of Chen and Patel (1988). The approach utilizes a two-equation $k-\epsilon$ model for most of the flow field, but a one-equation $k-l$ model in the viscous sub-layer and buffer zone. The prescribed length scale circumvents numerical problems often encountered with near wall dissipation calculations, and results in a more realistic sub-layer profile. Switching between $\epsilon$ and $l$ dissipation models is performed automatically using the turbulent Reynolds number based on the distance to the nearest wall. If $R_e = Re \sqrt{\kappa y} < 300$, the $k-l$ model is employed.

Conservation equations for turbulent kinetic energy and (in the fully turbulent region) its dissipation rate can be written:

$$ \frac{\partial \epsilon}{\partial t} + U^j e_{,j} - \frac{1}{Re} g^{ik} e_{,k} = G + \epsilon = 0 $$

$$ \frac{\partial e}{\partial t} + U^j e_{,j} - \frac{1}{Re} g^{ik} e_{,k} = \frac{\epsilon}{k} C_{ij} G $$

$$ + C_{ef} e^2 \frac{\epsilon}{k} = 0 $$

The effective viscosities in (7) and (8) are taken as $1/Re = 1/Re + \nu / \epsilon$, and $1/Re = 1/Re + \nu / \epsilon$, respectively, and the coefficients $(C_\mu, C_{ii}, C_{ij}, \sigma_\nu, \sigma_\sigma, \sigma_l)$ are fixed constants equal to $(0.09, 1.44, 1.92, 1.0, 1.0, 1.0, 1.3)$. Eddy viscosity is computed using the isotropic relation $\nu_l = C_{ij} k^2 / \epsilon$.

In the near-wall region, the rate of turbulent dissipation is specified in terms of $k$ rather than being computed from (8). From Chen and Patel (1988):

$$ \nu_l = \frac{k^{3/2}}{l_e} $$

where $l_e$ is a dissipation length scale equal to $C_{ij} [1 - \exp(-R_e / A)]$. With $k$ and $\epsilon$ known, the eddy viscosity is found from:

$$ \nu_l = C_{ij} \sqrt{k \rho} / \gamma_{ij} [1 - \exp(-R_e / A)] $$

the constant $C_{ij}$, $A_I$ and $A_e$ are chosen to yield a smooth distribution of eddy viscosity between the two regions, and take the values as follows:

$$(C_{ij} = \kappa C_{ij}^{-3/4}, \kappa = 0.418, A_I = 70, A_e = 2C_{ij})$$

To handle the practical calculation around complex geometries and evaluate the associated geometry related coefficients, equations (6) thru (8) are first written in Cartesian coordinate and then transformed into body-fitted coordinates. The required transformation can be defined as a mapping between physical coordinate (Cartesian coordinate) $(x', x', x', t)$ and the generally non-orthogonal body-fitted computational space $(\xi', \xi', \psi', \tau)$, i.e.,

$$ t = \tau, \quad x' = x' (\xi', \psi', \psi', \xi') $$

Using the relations given in Chen et al (1990), equation (1) and (6) thru (8) are transformed to:
\[
\frac{1}{J} \sum_{j=1}^{3} \sum_{i=1}^{3} \frac{\partial}{\partial \xi^i}[b_j U^i] = 0
\]  
(12)

\[
\frac{\partial U^j}{\partial \tau} + \sum_{m=1}^{3} C_{jm}^i \frac{\partial U^i}{\partial \xi^j} - \frac{1}{R_e} \nabla^2 U^i + S_{ij} = 0
\]  
(13)

\[
\frac{\partial \phi}{\partial \tau} + \sum_{m=1}^{3} C_{jm}^i \frac{\partial \phi}{\partial \xi^j} - \frac{1}{R_e} \nabla^2 \phi + S_{ij} = 0
\]  
(14)

\[
\frac{\partial \phi}{\partial \tau} + \sum_{m=1}^{3} C_{jm}^i \frac{\partial \phi}{\partial \xi^j} = \frac{1}{R_e} \nabla^2 \phi + S_{ij} = 0
\]  
(15)

where

\[
C_{jm}^i = \frac{1}{J} \sum_{k=1}^{3} b_j^i [U^k - \frac{\partial x^k}{\partial \tau} - \frac{1}{J} \sum_{m=3}^{3} b_m^i \frac{\partial \phi}{\partial \xi^m}]
\]  
(16)

\[
s_{ij} = -(G - \varepsilon)
\]  
(18)

\[
s_j = -R_e \frac{\varepsilon}{\alpha} (C_{ij} G - C_{ij} \varepsilon)
\]  
(19)

Expression for \( C_{ij} \) and \( C_{jm}^i \) are obtained by replacing \( \alpha \) in equation (16) with \( \alpha_i \) or \( \alpha_j \). The curvature parameters \( K_{ij} \), geometric coefficients \( a_{ij} \), and Laplacian operator \( \nabla^2 \), can be referred to Chen et al (1990).

**Finite-Analytic Discretization**

Numerical method used for advection-diffusion type momentum equations (2) for velocities, (7) for \( k \), and (8) for \( \varepsilon \) in the present RANS simulations is based on ‘Finite-Analytic’ discretization approach (Chen et al, 1990, Pontaza et al, 2005). Each equation is first rewritten in the form of a general convection/diffusion problem. Using \( \phi \) to represent one of the conserved quantities (\( U^i \), \( k \), \( \varepsilon \)), the genetic transport equation becomes:

\[
\sum_{j=1}^{3} (g^i - a_{ij}^i \frac{\partial \phi}{\partial \xi^j} = R_{\phi} \frac{\partial \phi}{\partial \tau} + S_{\phi}
\]  
(20)

where

\[
a_{ij}^i = R_{\phi} C_{ij}^i - f^i
\]  
(21)

and

\[
S_{\phi} = R_{\phi} \frac{\partial \phi}{\partial \tau} - 2 (g^{i2} \frac{\partial \phi}{\partial \xi^2} + g^{i3} \frac{\partial \phi}{\partial \xi^3} + g^{i1} \frac{\partial \phi}{\partial \xi^1} + \frac{\partial \phi}{\partial \xi^j})
\]  
(22)

The quantities \( f^i \) result from the transformed Laplacian and can be referred to Chen et al (1990).

In the finite-analytic approach, Equation (20) are linearized in each local element and solved analytically by the method of separation of variables. Evaluation of the analytic solution at the interior node provides a stencil for the center point in terms of its nearest neighbors. Using a backward Euler representation of the temporal terms, the final analytic solution is a 27 node variables formula and require the evaluation of three infinite series (Chen, 1982).

Instead of the computationally expansive 27 node formula, Pontaza et al (2005) construct a computationally less-demanding set of coefficients, by superimposing sets of two-dimensional coefficients in such a manner that the three-dimensional governing equation is satisfied. To illustrate, consider the three-dimensional form of transport equation (20) in Cartesian coordinate,

\[
u \frac{\partial \phi}{\partial x} + v \frac{\partial \phi}{\partial y} + \nu \frac{\partial \phi}{\partial z} = \frac{\partial \phi}{\partial x} + \frac{\partial \phi}{\partial y} + \frac{\partial \phi}{\partial z} = S_{\phi}
\]  
(23)

where \( S_{\phi} = R_{\phi} \frac{\partial \phi}{\partial \tau} + S_{\phi}

Equation (23) can be rewritten in any of the following three alternative forms:

\[
u \frac{\partial \phi}{\partial x} + v \frac{\partial \phi}{\partial y} - \nu \frac{\partial \phi}{\partial z} = g_{\phi} , \ g_{\phi} = S_{\phi} - u \frac{\partial \phi}{\partial x} + \frac{\partial \phi}{\partial z}
\]  

\[
u \frac{\partial \phi}{\partial x} + v \frac{\partial \phi}{\partial y} + \nu \frac{\partial \phi}{\partial z} = g_{\phi} , \ g_{\phi} = S_{\phi} - u \frac{\partial \phi}{\partial x} + \frac{\partial \phi}{\partial z}
\]  

\[
u \frac{\partial \phi}{\partial x} + v \frac{\partial \phi}{\partial y} - \nu \frac{\partial \phi}{\partial z} = g_{\phi} , \ g_{\phi} = S_{\phi} - u \frac{\partial \phi}{\partial x} + \frac{\partial \phi}{\partial z}
\]  

For each of the above as two-dimensional equations in constant x-, y-, z-, planes, a two-dimensional local analytic interpolant is derived (Pontaza et al, 2005). When the interplant is evaluated at an interior node, located at \( x = 0, y = 0, z = 0 \), the final finite analytic solutions are as follows:

\[
\phi(0) = \sum_{i=1}^{3} \alpha_{ij} \phi_i - \alpha_{ij} \phi_j
\]  

\[
\phi(0) = \sum_{i=1}^{3} \alpha_{ij} \phi_i - \alpha_{ij} \phi_j
\]  

\[
\phi(0) = \sum_{i=1}^{3} \alpha_{ij} \phi_i - \alpha_{ij} \phi_j
\]  

The local analytic interpolants \( \{\alpha_{ij}\}_{i=1}^{3} \) are functions of geometry of the element, the velocity field, and the Reynolds number. Details of the solution procedure by the method of separation of variables can be referred to Pontaza et al (2005).
In view of equation (23), which is equivalent to requiring \( g_x - g_y + 2S_{\chi} = 0 \), the three-dimensional discretization of (23) is

\[
\phi(0) = \frac{1}{\alpha_{ij}^x} \left( \sum_{m} \alpha_{jm}^x \phi_m^x + \sum_{n} \alpha_{im}^y \phi_m^y + \sum_{o} \alpha_{in}^z \phi_m^z - 2(R_x \frac{\partial \phi}{\partial x} + S_x) \right)
\]

(24)

from which a stencil relating 19 nodal variables is extracted. Additional details can be also found in Chen et al. (1995), where this approach first appeared.

If a backward Euler representation of the temporal terms \( R_x \frac{\partial \phi}{\partial x} \), for velocity momentum equation, Equation (24) can be written as follows

\[
[1/\alpha_{ij}^x + 1/\alpha_{ij}^y + 1/\alpha_{ij}^z] \phi(0) + \frac{\Delta t}{\alpha_{ij}^x} \phi(0) = \sum_{n} \alpha_{in}^x \phi_n^x + \sum_{o} \alpha_{in}^y \phi_n^y + \sum_{m} \alpha_{in}^z \phi_m^z
\]

(25)

It should noted that in the source term \( S_x \), \( \Delta \nabla p \) is used.

**Pressure/Velocity Coupling**

The velocity-pressure coupling is achieved by a discrete projection method. The velocity field is first decomposed by projecting out the divergence-free-producing part of the field, i.e., the pressure gradient, \( \vec{u} = \vec{u} + \nabla p \)

(26)

Then, by requesting a solenoidal velocity field, \( \nabla \cdot \vec{u} = 0 \), where \( \nabla \) is the discrete divergence operator, Poisson equation for pressure can be obtained:

\( \nabla \cdot \vec{u} = 0 \)

(27)

If velocity is specified at a boundary, then pressure need not be specified there and is computed consistently by extending the projection to that boundary. If velocity is specified on the entire boundary, only a pressure datum extending the projection to that boundary. If velocity is specified at a boundary, then pressure need not be specified there and is computed consistently by extending the projection to that boundary. If velocity is specified on the entire boundary, only a pressure datum extending the projection to that boundary.

Choosing to retain a co-located degree of freedom arrangement, where the velocity degrees of freedom and pressure degrees of freedom share the same nodal locations, care needs to be exercised in choosing the grid representation of the discrete divergence operator \( \nabla \). If \( \nabla = \nabla_p \), then the pressure Poisson equation computational stencil will allow spurious checkerboard-type pressure solutions (Ferziger and Peric, 1996). In solving the Poisson equation (24), here, \( \nabla \) is defined to have a grid representation with a staggered node arrangement to suppress the possibility of spurious pressure solution. This approach is common practice in finite-difference and control-volume discretizations, when choosing a co-located degree of freedom arrangement (Ferziger and Peric, 1996). For more detailed numerical formulations for equations (26) and (27), readers can be referred to Chen et al., 1995 and Pontaza et al., 2005.

**General Solution Procedure**

In the solution procedure, the discretized momentum equations are solved first, using the latest pressure field. Then, the Poisson equation for the pressure is solved based on the latest velocity. Upon obtaining the pressure field, the velocity field is made solenoidal by projecting it onto a div-free space via equation (24). Step-by-step details on the solution strategy are given as follows:

1. Compute the finite analytic coefficients, \( \{\alpha_{ij}^x, \alpha_{ij}^y, \alpha_{ij}^z\} \), at all interior global nodes using the latest velocity field.
2. Construct and solve the set of algebraic equations associated with the discretized momentum equation (25). Use the latest pressure field to evaluate the pressure gradient on the right-hand side of (25), i.e. treat the pressure field explicitly.
3. Project the resulting velocity field onto a div-free space as follows:
   - Construct and solve the set of algebraic equations associated with the Poisson equation for the pressure (27). Use the latest velocity field to evaluate the right-hand side of (27), i.e. treat the velocity field explicitly.
   - Make the velocity field solenoidal by adding back the pressure gradient, using equation (26).
4. Repeat the projection step (3), until the initial divergence of the velocity field drops by one or two orders of magnitude. This is typically achieved in at most two projection steps.
5. Repeat steps (1-4) to account for the fixed point linearization of equation (25), i.e. to account for the nonlinearity associated with the momentum equations.
   - At this point, under-relaxation of the pressure field may be necessary in order to allow the velocity field in step (2) to respond smoothly to the pressure gradient that causes it to be solenoidal. Typically, under-relaxation is only needed if the time-step size is large or the transient of the problem are significant.
   - If the time step is small, then the finite analytic coefficients in step (1) need not be re-evaluated. This essentially amounts to linearizing the momentum equation about the previous time step. In such an approach, iterations at this level are still needed if pressure is under-related. Instead of "nonlinear iterations", these iterations receive the name of "outer-iterations".
   - Perform time advancing when a tolerance criterion between successive nonlinear/outer iterations is met. The recommended convergence measure is the L1 norm of the momentum residuals (Ferziger and Peric, 1996) as the normalized difference between
successive nonlinear/outer iterations may be misleading if under-relaxation is used.

- If time accuracy is not desired, e.g. marching fast towards a steady-state, then time-advance the velocity and pressure fields immediately after step (4).

6. Repeat step (1-5) until the desired time level is reached or a steady-state is achieved.

On a give time step and nonlinear/outer iteration, the system of equations associated with the velocity components are solved iteratively using a line-by-line tri-diagonal alternating-direction-implicit (ADI) algorithm. Iterations are stopped when the initial residual drops by one or two orders of magnitude, typically achieved in 3-5 ADI iterations. In turn, the system of equations associated with the pressure is solved iteratively using Stone’s strongly implicit procedure (Stone, 1968) (SIP) – which gives superior convergence rates when compared to an ADI algorithm for the same equation. Iterations are stopped when the initial residual drops by one or two orders of magnitude, typically achieved in 5-10 SIP iterations.

RESULTS AND DISCUSSIONS
In this study, the aforementioned numerical method and turbulent model are used in four-quadrant propeller flow simulations. Time-domain simulations are performed for a conventional propeller DTRC 4118 (Figure 5) under open water ahead, bollard-pull, backing, crash-astern, and crash-ahead operating conditions.

For the open water propeller flow computations, it is possible to solve the propeller flow on a non-rotating coordinate system with fixed grids by including the centrifugal and Coriolis forces in the RANS equations. Alternatively, one may solve the unsteady RANS equations for rotating propeller directly on an earth-fixed coordinate system. In the present study, the later approach is adopted so that the method can be easily extended to complete ship and propeller flow simulations with the propeller operating in a non-uniform ship wake. The use of rotating grid also greatly simplify the far field boundary condition where uniform flow can be specified without considering the swirling velocity components.

For the cases demonstrated in this paper, the flow field computation is initialized with free-stream flows in the entire computational domain and the propeller is allowed to rotate until a steady state or periodic response of the thrust coefficient is obtained. The flow conditions for ahead, backing, crash-astern and crash-ahead correspond to an advance coefficient \( J = 0.833 \) and a Reynolds number \( Re = 1.46 \times 10^5 \), where \( nD \) (\( n \) is propeller rotation speed and \( D \) is propeller diameter) is used as the characteristic reference velocity. For bollard-pull condition, \( J \) is set as zero and \( Re \) is maintained the same value \( 1.46 \times 10^5 \).

Numerical Grids
Surfaces of the three-dimensional multiple block grids are shown in Figure 6. The composite grid consists of 5 blocks: 3 blocks representing each of the 3 blades, 1 block for the propeller shaft, and 1 block to represent the far-field (the cylindrical grid block in Figure 7). The 5 blocks are shown in different colors in the figures.

![Fig. 6 Grids embedded in propeller blades](image)
In the present study, each propeller blade is covered by a small $62 \times 31 \times 31$ volume grid around the propeller root and shaft junction. Cross-sections of these small grids around the blades are constructed by resembling O-type grids around airfoils. The propeller shaft is covered by a $41 \times 17 \times 122$ volume grid. The propeller blades and shaft grids are all embedded in a cylindrical grid of size $61 \times 35 \times 122$. The total number of grid points is slightly over half million.

For the block representing the free-stream, the inflow boundary, where the free-stream enters the computational domain, is located 1.2 propeller diameters from the center of the shaft. The outflow boundary is located 1.8 propeller diameters downstream. The lateral boundaries are 1.6 diameters on each side. The grid distribution is coarse towards the inflow and outflow, and graded in the vicinity of the propeller.

In the time domain simulation, the non-dimensionalization is such that one revolution of the propeller is achieved in one dimensionless time unit. The time step size is chosen such that one revolution is computed in 60 time steps, or equivalently $\Delta t = 0.01667$. The first grid point away from solid walls is placed at a distance $y^+ \approx 0.10$, so that near-wall effects are computed and not modeled by an artificial boundary condition. In order to maintain the free stream direction to be $+x$ direction in all the simulation cases, the propeller geometry for backing and crash-ahead is reversed. Accordingly, the rotation directions for backing and crash-ahead conditions are changed to be same as the rotation directions in ahead and crash-astern conditions. The following table summarizes the free stream and propeller rotation directions used in this study.

<table>
<thead>
<tr>
<th></th>
<th>Ahead</th>
<th>Crash-astern</th>
<th>Backing</th>
<th>Crash-ahead</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free stream direction</td>
<td>$+x$</td>
<td>$+x$</td>
<td>$+x$</td>
<td>$+x$</td>
</tr>
<tr>
<td>Propeller direction</td>
<td>$-x$</td>
<td>$+x$</td>
<td>$-x$</td>
<td>$+x$</td>
</tr>
<tr>
<td>Propeller geometry</td>
<td>original</td>
<td>original</td>
<td>reversed</td>
<td>reversed</td>
</tr>
</tbody>
</table>

**Ahead Condition**

In the ahead condition, the time history of thrust (Figure 8) shows that the solution has reached state steady after 5 revolutions (about $300 \Delta t$ time). The predicted thrust coefficient $K_T = 0.145$ is in good agreement with the corresponding experimental measurement of $K_T = 0.150$ (van Gent, 1977). The minus sign of the thrust value in Figure 8 means propeller thrust is in $-x$ direction.

Pressure distributions on the blade surface are plotted in Figure 9 for pressure side and Figure 10 for suction side. As seen, the pressure distributions show a typical pattern that a very high pressure gradient occurs at the leading and trialing edges.
To have more comprehensive views of the propeller induced flows, velocity vector plot in longitudinal plane is drawn in Figure 11. As shown, due to the action of propeller, fluid is sucked to the propeller disk and starts to accelerate in front of the propeller toward downstream direction. Cross flows at propeller downstream are given in Figure 12. As usual, a strong swirling flow can be seen. Finally, Figure 13 shows the particle traces released from the propeller blade at three different radii – around the blade tip, mid-span and root locations. Typical contraction of the flow can be seen in the plot. It is noted also that due to the higher linear rotation speed at the outer radius, particle trace shows smaller pitch at the outer radius than at the inner radius area.

**Bollard-Pull Condition**

In the bollard-pull condition, the inflow velocity is set to be zero and the propeller is forced to rotate with the same rpm as the ahead condition. Therefore, the Reynolds number $Re$ is maintained same as the ahead condition with a value $1.46 \times 10^5$ but the advance coefficient $J$ becomes zero. The thrust time history is plotted in Figure 14. Similar to the pervious ahead case, $K_T$ becomes stable after few cycles (about 5 revolutions). It is noted that although there is still small continuous oscillations of the calculated $K_T$ after five revolutions, the change of the $K_T$ value is very small. In this simulation, the averaged $K_T$ value is about 0.40 which is smaller than the experimental measurement $K_T = 0.52$ (van Gent, 1977).
To obtain some ideas of the causes of the $K_T$ oscillation and discrepancy between the predicted $K_T$ and experimental value, a velocity vector plot is drawn (Figure 15). From the plot, it is seen due to the high thrust (caused by relatively high angle of attack at bollard pull operation), fluid is sucked from the back side of the blade around the tip area and generate a strong recirculation flow at tip area similar to vortex ring in crash-astern or crash-ahead operations. Also, at downstream behind the propeller cap another opposite flow recirculation exists clearly. It is believed that these complex recirculations could make the flow more unstable and cause the small fluctuation of the propeller thrust. In simulating this kind of large-scale unsteady turbulence flow, the limits and errors of RANS simulation are well-known. In general, it is believed that LES (Large Eddy Simulation) could provide more accurate resolutions than RANS simulation.

The pressure distributions are plotted in Figure 16 for the pressure side and Figure 17 for the suction side. Compared to the ahead operation, higher pressure occurs for the bollard pull condition. Also, instead of the large difference of pressure patterns between pressure and suction sides in the ahead condition, more similar pressure patterns appears on both side in the bollard pull condition.
It is also worthwhile to compare the velocity vector plots around the propeller blade for the ahead and the bollard-pull conditions (see Figure 19 and 20). For the ahead condition, the inflow is aligned closely with the designed blade angle at the propeller leading edge. Under the bollard-pull condition, however, the inflow approaches the propeller leading edge with a large angle of attack producing a larger thrust force and torque on the propeller.

**Backing Condition**

Under the backing operation, the ambient flow approaches the propeller from the propeller cap and the propeller also reverses its rotating direction. This leads to a reversal of the propeller leading and trailing edges and camber geometry. Time history of the thrust is plotted in Figure 21. Again, similar to the ahead case, a convergent thrust $K_T = 0.085$ is obtained after about five propeller revolutions. However, compared to the ahead $K_T$, the thrust under backing is smaller (about 60% of the thrust under ahead operation). After careful check of the flow near the blade surface, no flow separation was found. The reduction of the thrust is mainly due to the effect of camber reversal.

Pressure distribution contours for this backing case are plotted in Figure 22 for pressure side and in Figure 23 for suction side. It is noted that even at the pressure side a large low pressure area can be observed. Basically, except near the area around the leading edge (the original trialing edge under the ahead operation), the remaining blade areas on both pressure and suction sides are under low pressure loads.
To have a general view of the section flow, the velocity vector around the propeller blade is plotted (Figure 24) at radius 0.65R. As seen, the inflow angle of attack is small and almost is parallel to the chord line of the section. This confirms the small thrust obtained before for this backing condition ($J = 0.833$).

Particle traces are plotted in Figure 25. Basically, since the backing operation is quite similar to the ahead condition, particle traces under the backing condition show typical helix curves with flow contraction as seen in the ahead operation. This implies that for a backing condition, traditional potential method could be workable similarly to an ahead case although the round trailing edge Kutta condition may need a special treatment.

In Figure 25, it is also noted that compared to the ahead condition (Figure 13) the pitch of the particle path-lines in the backing condition is smaller. This is mainly due to the low thrust of the propeller which in turn causes the low downstream direction velocity.

**Crash-ahead Condition**

Crash-ahead condition is also performed in this study with the same advance coefficient $J = 0.833$ as the ahead case. The time history of the thrust is plotted in Figure 26. Unlike the ahead and backing cases, since large separation (Figure 27) occurs around the leading edge (the trialing edge for the ahead condition), the computation needs to take longer time to converge (about 10 revolutions) to a steady value. The predicted $K_T$ is 0.675 which is much higher than the ahead case ($K_T = 0.145$). High $K_T$ values under crash-ahead operation are also found in the experiment measurement for a conventional propeller P4381 (see Figure 28) at high $J$ value.
The pressure distributions at the pressure and suction sides are plotted in Figure 29 and 30. Unlike the previous cases, in this crash-ahead condition, very low pressure (minus pressure) is found at the area from leading edge to certain downstream distance on the pressure side of the blades. This is because there are large separation flows around that area as shown previously in Figure 27. Through the pressure pattern plotted in Figure 29 and 30, it seems the pressure on blades is still under unsteady state even though the change of the thrust is quite small after 10 revolutions in our computations. It is still uncertain in this study that the cause of this problem is due to whether the numerical convergence issues or the physical flow unsteadiness of this complex crash-astern case.

To have a general view of the flow pattern, a velocity vector plot is given in Figure 31. Under the crash-ahead operation, the propeller sucks the flow in the opposite direction of the freestream flow and the freestream flow becomes slow down in front of the propeller. However, as seen in Figure 31, fluids away from the propeller tip keep their downstream direction and cause a strong shear flow around tip area. Hence, a well-known “vortex-ring” is formed.

For completeness, particle traces are also plotted (Figure 32). As seen, compared to the pervious ahead, backing and bollard pull cases, more irregular path-lines appear in the crash-ahead condition. Depending on the locations of the releasing particles, they may either flow all the way downstream or be trapped inside the ring vortex. In Figure 32, the particle released near the blade tip area is found to flow downstream leaving the ring vortex area but for those particles released from the blade mid-span and the root locations, they are trapped around the propeller.
Crash-astern Condition

For the crash-astern case, the time history of the thrust and instantaneous pressure on the pressure and suction side of the blades are plotted in Figure 33, 34 and 35, respectively. Unlike the crash-ahead case, the thrust in the crash-astern condition converges quite rapidly. About three revolutions, the $K_T$ reaches a constant value 0.65. Although it was observed in the experiments (Jiang et al, 1997, Jessup et al, 2004) that unsteady periodic vortex shedding occurs behind the P4381 propeller, the current RANS simulation for the DTRC 4118 propeller doesn’t show this phenomena as also evidenced by the stable thrust value in Figure 33. It is still uncertain now this is due to the limit of RANS simulations or the real physics of the flow. More detailed experimental measurements and Large Eddy Simulations (LES) for DTRC 4118 propeller may require clarifying this issue in future.

As mentioned earlier, in the simulation of the propeller under crash-astern condition, the directions of rotation and inflow are maintained same as the crash-ahead condition but only the propeller geometry is changed by switching the leading and trailing edges of the blades and reversing the blade section camber. Since the simulations are performed under the same $J = 0.833$ for both cases, the inflow conditions for both conditions should be quite close one another such that due to the high angle of attack, inflow to blade is directly impinging the blade inner area (the maximum pressure spot in Figure 36) and generate relatively high thrusts. To have general idea of the similarity of those flows, velocity vector plots near the blade surfaces under the crash-astern and crash-ahead conditions are plotted for blade pressure and suction sides (Figure 36-39). As seen, due to the similar velocity distributions in crash-astern and crash-ahead conditions, the pressure contours for both conditions are also quite similar. The small differences between them are attributed to the camber and thickness effects.

From the pressure contours plot on pressure side (Figure 34), it is noted that a maximum pressure spot appears in each of the blade inner area. These pressure spot imply the existence of stagnation points. The stagnation point can be clearly seen through the velocity vector plot shown in Figure 36.
To obtain a more comprehensive view of the flow, velocity vector plot in a longitudinal plan is drawn in Figure 40. As seen, a strong vortex ring appears. Compared to the crash-ahead case, this crash-astern vortex ring shows an appearance of stronger swirling recirculation and is located further downstream and more outer to the propeller disk.

From Figure 40, it is noted that although the flow from upstream (left side) passes the blade pressure side, due to the action of propeller, the flow near the suction side is reversed. In Figure 41, the flow in the blade passage...
is plotted. As seen, due to the opposite flows near pressure and suction sides, two recirculation flows are generated.

Fig. 41 Flow in blade passage

Finally, for completeness, particle traces are also plotted. Similar to the crash-ahead case, quite irregular path-lines appear. Again, whether the particle can flow downstream is highly depends on the location where the particle is released.

Fig. 42 Particle traces plot for crash-astern condition

CONCLUDING REMARKS

A Reynolds-Averaged Navier-Stokes (RANS) method has been employed in conjunction with an overlapping moving grid approach to simulate the four-quadrant propeller flows under both the design and off-design conditions. The capabilities of this Chimera approach allowed for the numerical solution of the RANS in an earth-fixed coordinate system, where the propeller grids are allowed to rotate with respect to fixed grids for the ship geometry and surrounding ambient flow.

In this study, time-domain simulations are performed for the DTRC 4118 propeller under ahead, bollard-pull, crash-ahead, crash-astern, and backing conditions and the results are compared with the available experimental data. The predicted thrust coefficient for the ahead condition is in good agreement with the measurement result. For bollard-pull, crash-ahead and crash-astern cases, basically, the main physics of the flows have been captured in the simulations, however, it is found that the simulation results still have some discrepancies compared to experiment observations performed previously. It is still uncertain these discrepancies are from the limitation of RANS model or the difference (propeller geometry and operating conditions) between the propeller P4381 (experiment propeller) and the propeller 4118. More detailed experimental measurement and Large Eddy Simulations (LES) for DTRC 4118 propeller may need to clarify this issue in future.

REFERENCES
