Propeller-Induced Hull Vibration – Analytical Methods

Sing-Kwan Lee, Min Liao, and Suqiu Wang
American Bureau of Shipping
ABS Plaza, 16855 Northchase Drive, Houston, Texas, USA

Proceedings of the 2nd International Ship Noise and Vibration Conference
London, UK, June 28, 2006

ABSTRACT
The recent increase in size and speed requirements for certain ship types, particularly container and passenger ships, accentuate the need for a more sophisticated understanding of, and response to, vibration of the hull structure due to propeller excitation. This paper overviews the methodologies and the state-of-the-art computational analysis tools that ABS developed in order to more accurately estimate propeller-induced hull vibration.

INTRODUCTION
In ship vibration, the propeller is frequently a trouble source which can cause an excessive ship stern vibration problem. The consequences of excessive vibration in the stern area can be severe. Deterioration of the structural members can be accelerated as a result of fatigue caused by long term cyclic vibration. Excessive vibration can damage or adversely impact the in-service performance of the ship’s mechanical and electrical equipment. Prolonged exposure to vibration can also contribute to crew and passenger discomfort, increasing the opportunities for human error.

Increased flexibility of the hull girder of larger, and particularly longer, ships with a fine, underwater form can significantly increase susceptibility to vibration. Moreover, as the weight and distribution of steel within ship structures are optimized as shipbuilders attempt to control production and material costs, the propensity for vibration-related troubles, particularly in the stern section of the vessel, increases. As the demand for higher service speeds for many of these vessels also increases, attendant increases in the propulsion power are required. This translates into higher loads on propellers, which in turn lead to greater propeller excitation and an increase in the risk of vibration and vibration-induced failures. Stern vibration problems arise from the unsteady cavities that attach to the surface of the propeller blades. These create an intense, fluctuating pressure impact on the ship’s hull. With modern propeller design, a small to moderate amount of sheet cavitation is often unavoidable in order to maintain the required propulsion efficiency. Reconciling the challenges posed by these conflicting technical and operational demands is essential if further improvements in the speed-power-size ratio are to be realized, particularly for ultra-large containerships.

To predict propeller-induced hull vibration is not simple. It is a synthetical analysis involving methodologies of many cross-field topics such as Computational Fluid Dynamics (CFD), Finite Element Method (FEM), and fluid cavitation dynamics. In propeller induced hull vibration assessment, the prediction of stern flow is central to the problem of unsteady propeller loads, cavitation, and propeller-induced hull pressure. The solution to these problems requires detailed knowledge of the turbulent stern flow (including thick and perhaps separated boundary layers), bilge vorticity, and propeller/hull interaction. Traditionally, in ship design the technology for these predictions was mainly based on regression and empirical formulae. At best, the use of ship flow codes was restricted to potential flow calculation augmented by boundary layer predictions to approximate viscous effects. Propeller calculations were performed using empirically generated effective wakes, and the propeller’s interaction with the hull was approximated with a thrust deduction coefficient.

The use of CFD in ship hydrodynamics has increased dramatically over the past decade. The increase is due to continuous advances in computational methods together with the increase in performance and affordability of computers. Also, due to the emergence of many unconventional propulsor designs such as tractor PODs, tip plate propellers, and propellers with wake equalizing ducts/spoilers, empirical methods based on the historical databases developed for conventional propeller designs become questionable in the innovative designs. More sophisticated analyses based on direct simulation using CFD and FEA methods are required to associate with the model tests for propeller-induced vibration studies.

Nowadays, with advances in CFD techniques, more comprehensive analyses can be performed for propeller/hull interaction flow problems. It has been demonstrated that CFD simulation, particularly using RANS-based methods, can provide more flow details in understanding the complex propeller/hull interaction process. This paper provides an overview of the methodologies and the state-of-the-art computational analysis tools that ABS has developed in order to more accurately estimate propeller-induced hull vibration.
INTEGRATED SIMULATION SYSTEM

Overview

Complexity of ship stern flow

As is well known, a propeller installed at ship stern area is operated in a turbulent flow environment. Unlike turbulent flows around airfoil with a thin boundary layer, turbulent flows around ship stern are thick boundary layer flows. At typical laboratory Reynolds numbers of around 5 million, the extent of the viscous flow (boundary layer) normal to hull at the stern is often of the order of 0.5 draft around midgirth. To model this thick boundary layer flow, any simplifications of the Navier-Stokes equations based on boundary layer theory are no longer valid. Instead, the three dimensional full Navier-Stokes equations have to be used. As is well-known, the correct simulation of fluid vortical turbulent motion is critical to an accurate prediction of propeller cavitation and induced hull pressure. In order to capture these physics correctly, Reynolds Averaged Navier-Stokes (RANS) equations with turbulence models is adopted in the current simulation system.

Iterative computations for propeller/ship interactive flow

In ship design practice, in order to enhance the efficiency, a propeller is usually designed to keep its diameter as large as possible. Therefore, it is common for the propeller tip to rotate closely to the hull surface, sweep the thick turbulent boundary layer attached to the hull and strongly interact with it. To correctly simulate this strong interactive flow, a propeller performance program (lifting surface code) needs to couple with a ship flow simulation program (RANS solver) in an interactive and iterative manner to predict the ship wake flow including the propeller effect. Based on the final convergent ship wake flow the blade pressure, cavitation, and propeller-induced hull pressure are in turn to be calculated.

Modules in Simulation System

Three modules are included in the integrated simulation system, namely they are the Propeller/Ship Flow module (PSF), the Hydro-Load Assessment module (HLA), and the Finite Element Analysis module (FEA). Figure 1 shows the details of the analysis programs and their functions involved in the modules.

In the simulation system, the PSF module is mainly for the propeller and ship flow simulations. Four components are included in the module. They are the propeller flow analysis program, MPUF3A, developed by Lee et al. (2003) for propeller performance and cavitation analyses; the Chimera RANS program (RANS model) developed by Chen at al.(2002) for turbulent ship flow simulation; the GBFLOW program (Euler model) developed by Choi et al (2002) for propeller effective wake calculation; and the propeller/ship flow interaction program SHIP-PROP (Chen and Lee, 2003a, 2003b, Lee and Chen, 2003) developed in ABS for propeller/ship flow coupling calculations.

In the HLA module, hydrodynamic loads of fluctuating pressure induced by propeller cavitation, unsteady bearing forces/moments and blade pressure due to non-uniform ship wake are assessed through the programs HULLFPP developed by Young and Kinnas (2002), PUF3HRM developed by MIT, and PropS2 developed in ABS (Lee, 2000). In the FEA module, a commercial FEM package NAStRAN is used for stress and vibration analyses.

Procedure of Comprehensive Analysis

The sequence of comprehensive analyses in the integrated simulation system is summarized as follows:

1. Bare hull wake field (nominal wake) simulation
2. Simulation of wake field under propeller-ship hull interaction (effective wake)
3. Propeller performance analysis (thrust and torque coefficients, \( K_T \) and \( K_Q \))
4. Propeller cavitation analysis (cavity patterns on propeller blades)
5. Hydrodynamic loading assessment (pressure on blades, propeller induced hull pressure and bearing forces/moments)
6. FEM analysis for vibration and stress on ship hull, shaft and propeller blades

Propeller ship flow analysis is initiated from nominal wake simulation, which is the first guess for the effective wake solution. In the current stage of the ABS integrated simulation system, effective wake simulation can be performed based on either the viscous flow model – MPUF3A/RANS coupled calculation or the inviscid-rotational flow model – MPUF3A/GBFLOW coupled calculation. If propeller-ship hull interaction is not strong such as the cases of tractor PODs, twin screw propellers or propeller with large aperture, effective wake simulation can be performed based on the inviscid flow model – MPUF3A/GBFLOW coupled calculation. For a strong propeller-ship hull interactive case such as a single screw propeller with a small propeller/hull clearance, the viscous flow model – MPUF3A/RANS coupled calculation needs to be applied. After propeller ship flow analysis, hydrodynamic loadings, such as fluctuating pressure on ship hull due to propeller cavitation, bearing forces and moment on shaft, and blade pressure on propeller, can be calculated by programs HULLFPP, PUF3HRM, and PropS2, respectively (Figure 1). Then, FEM vibration and stress analyses are performed for hull structure, shafting system and propeller blades based on the calculated loads. For structural safety check, ABS provides vibrations and stresses criteria (see ABS Rules (2003), ABS guidance note on ship vibration (2006) and related ABS technical reports (Lee and Seah, 2000; Lee, 2001)).
DIRECT SIMULATION STUDIES

In this paper, several simulation cases are selected to demonstrate the simulation capabilities of the integrated simulation system. The first group of the simulations focuses mainly on the hydrodynamic aspects of strong propeller-ship hull interaction for a model ship with its propeller operating under design (ahead) and off-design (backing and crash-astern) conditions. In the second group of simulations, the structural vibration problems induced by propeller excitation are addressed through a comprehensive analysis for a tanker.

**PSF module – Propeller/Ship Flow simulation**

- Viscous flow model: MPUF3A/RANS coupled calculation for strong propeller/ship interaction

[propeller flow calculation] - MPUF3A \(\Rightarrow\) SHIP-PROP \(\Rightarrow\) RANS - [ship flow calculation]

[data conversion & management]

- Inviscid flow model: MPUF3A/GBFLOW coupled calculation for weak propeller/ship interaction

measured nominal wake data \(\Rightarrow\) MPUF3A \(\Rightarrow\) GBFLOW - [effective wake calculation]

**HLA module – Hydro-Load Assessment**

- HullFPP [propeller induced hull pressure]
- PUF3HRM [bearing forces and moments]
- PropS2 [propeller blade loads]

**FEA module**

Ship hull vibration analysis
shifting system vibration
propeller blade stress

Figure 1: Procedure for comprehensive analyses using integrated simulation system

**Propeller/Hull Interaction Flows Simulation**

Propeller can operate in 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 crash-astern or crash-back \((+V_s, -\omega)\). 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. These off-design propeller flow phenomena are dominated by viscous effects and cannot be accurately predicted by the potential flow methods. Jiang et al (1991) extended the inviscid flow propeller design methods for the simulation of backing and crash-astern conditions. They adopted a simplified approach in propeller flow analysis program PSF with three-dimensional correction factors to account for the leading-edge separation under crash-astern conditions. More recently, Chen and Stern (1999) solve the RANS equations for propeller flow including forward, backing, crash-ahead, and crash-astern conditions. However, the above backing and crash-astern simulations were limited to the open water conditions with uniform inflow to the propeller without any propeller-ship hull interactive effect.

In this direct simulation case, calculations were performed for a series-60, \(C_B = 0.6\), ship hull with a MAU propeller (Toda et al., 1990) using the MPUF3A/RANS coupled model in the integrated simulation system. Figure 2 shows the chimera grids used in this calculation. In the present chimera domain...
decomposition approach, a 122x35x31 O-type numerical grid (block 1) was used around the ship hull to provide detailed resolution of the turbulent boundary layer and wake flows generated by the ship motion. Since the ship keel plane is a branch cut in the ship grid, a small 61x35x3 (block 2) was constructed around the keel plane so that the solution on the branch cut can be obtained directly from the RANS calculation. Both grid block 1 and 2 were embedded in a 121x81x31 rectangular grid (block 3) representing the ambient water. Furthermore, a 21x31x62 cylindrical grid (block 4) was generated behind the ship stern for propeller body-forces calculation. The cylindrical grid is completely embedded in the ship grid (block 1) to provide a more accurate resolution of the inflow to the propeller as well as the wake flow induced by the propeller thrust and torque.

Figure 2: Chimera numerical grid for CFD simulations

Ahead design condition – CFD validation
For the ahead operating condition – propeller design condition, calculations were performed first for the bare hull case with Froude number, \( \text{Fr} = 0.16 \), and Reynolds number, \( \text{Re} = 3.94 \times 10^6 \), based on Toda et al’s experiment condition (1990). Then the bare hull solution was used to initialize the RANS/MPUF3A coupling calculation. According to the experiment condition, advance coefficient \( J = 0.88 \) and cavitation number \( \sigma = 155.65 \) were used in MPUF3A for the propeller flow analysis. In the present MPUF3A/RANS coupling approach, the RANS method was employed to provide the inflow to the propeller (i.e., effective wake) and the MPUF3A was used to compute the propeller loading distributions based on the effective wake at the propeller plane.

It should be noted that the propeller loading distribution depends on the inflow to the propeller and the propeller inflow, in turn, depends on the propeller operating conditions. In order to obtain the correct inflow to the propeller or the effective wake, it is necessary to couple the MPUF3A and RANS codes in an interactive manner for accurate resolution of the propeller-ship interactions. Convergence histories of RANS/MPUF3A coupling for \( K_T \) and \( K_Q \) values are shown in Figure 3.

Figure 3: Convergence history of RANS/MPUF3A calculation

In Figure 3, the values of \( K_T \) and \( K_Q \) for the 0th coupling are based on open water condition and for the first coupling, \( K_T \) and \( K_Q \) were calculated based on the bare hull solution (nominal wake). Details of the \( K_T \) and \( K_Q \) values for the propeller responding to open water, nominal wake and effective wake inflow are summarized in Table 1. The final \( K_T \) and \( K_Q \) with propeller-hull interactive effect are 0.2420 and 0.0405. Compared to experimental data, \( K_T \) is over-predicted about 3.4% of and \( K_Q \) under-predicted about 1.5%. For the nominal inflow condition, the predicted thrust coefficient is \( K_T = 0.2073 \). It is about 11.4% lower than the measured value, \( K_T = 0.234 \). The corresponding torque coefficient \( K_Q = 0.0354 \) was also under-predicted by 14% compared to the measured \( K_Q = 0.0411 \). This indicates that the nominal wake is not accurate enough for propeller loading prediction. In order to improve the prediction, it is necessary to apply the interactive RANS/MPUF3A method to compute the effective wake resulting from the complex interactions between the ship hull and the propeller.

In addition to the evaluation of propeller general performance through thrust and torque coefficients, it is desirable to examine the detailed ship stern flows induced by the propeller to facilitate a better understanding of the propeller-ship interactions. Figure 4 shows the axial velocity contours and crossflows induced by the propeller. In order to quantify the propeller effects,
nominal wake solution is also plotted as a baseline for comparison in the figure. It is seen from Figure 4a & 4b that for nominal wake solution, the axial velocity and secondary flows are quite small at the propeller plane since the propeller is located in the thick boundary layer and wake region behind the ship stern. However, for effective wake solution shown in Figure 4c & 4d, strong axial flow accelerations and swirl at the propeller plane and behind the propeller can be seen. Moreover, the inflow to the propeller was also found to be significantly stronger in comparison with the nominal wake.

To further evaluate the performance of the RANS/MPUF3A calculation, the axial velocity contours at $x/L=0.98125$ (front of the propeller disk, where $x$ is the distance from ship bow to downstream location and $L$ is the model ship length) and at the propeller disk ($x/L=0.9875$) are plotted for the bare hull and the interactive solutions (Figure 5). As can be seen, although some discrepancies exist due to the absence of modeling the propeller shaft in the RANS calculation, the basic flow pattern and magnitude of axial velocity are still captured by the calculations as compared to the experimental measurement.

Off-design condition flows
Off-design conditions for propeller under backing and crash-astern operations were also performed in this direct simulation study. The simulation results are summarized in Figure 6. For the backing case, the ship is moving backward and the propeller also reverses its rotating direction. Therefore, the flow is moving towards the ship stern and the propeller wake impinges directly on the ship stern. As seen in Figures 6a and 6b, the flow is nearly uniform at the inflow plane location, which is behind the propeller for this backing case, but significant axial flow accelerations were observed at the propeller plane. It can also been seen from the figures that the propeller produced a strong suction near the root of the propeller blades. However, a distinct ring vortex with axial flow reversal was observed near the propeller tip.
For the crash-astern case, the ship is moving forward but the propeller is operating in the reverse (counterclockwise) direction. This not only leads to negative thrust and torque, but also results in a negative angle of attack with respect to the propeller inflow. Due to the negative thrust and torque, the propeller produces a net suction towards the ship stern and the propeller-induced swirling flow is rotating in a counterclockwise direction. The negative thrust leads to a local flow reversal in the axial direction, as seen in Figure 6a. It is also interesting to note that there is a large flow recirculation region above the propeller shaft. In addition, a smaller recirculation region was also observed in the blade tip region just behind the ship keel line.

**Figure 5: Axial velocity contours near propeller disk**
Hull Vibration Analysis
After demonstrating the general performance of the integrated simulation system for propeller/hull interaction flow analysis, the next group of simulations will be focused on the propeller-induced structural vibration problem for a tanker.

Propeller cavitation and its induced hull pressure
As known earlier, once the effective wake inflow solution (propeller/hull interaction flow) is obtained, propeller cavitation analysis can be performed. In the integrated simulation system, cavitation analysis is performed by the program MPUF3A, which can compute the time history of the blade cavity volume based on the effective wake input. In fact, the rapid change of cavity volume is the main source of intensive fluctuating pressure on ship hull. In general, both full and ballast load conditions are needed to be checked to see whether or not vibration problem occurs.
For this specific tanker selected, the draft at ballast condition is reduced about 11 meters compared to full load condition thereby causing a large reduction of suppression pressure to blade cavitation. As expected, large blade cavitation occurs in ballast load condition compared to full load situation. Figure 7 shows the time history of the cavity volumes for the both load conditions. As seen, cavitation obviously increases in the ballast load case. This also can be seen in the comparison plots of the cavity pattern between full load and ballast condition given in Figure 8.

Figure 7: Cavity volumes for both load conditions

Propeller-induced hull pressure has been calculated based on the previous cavity volume results by using the HULLFPP program. Hull pressure distributions for 1\textsuperscript{st} and 2\textsuperscript{nd} blade rate mode under full load and ballast conditions are plotted in Figures 9 and 10. As seen in the figures, due to the draught change in the ballast condition, the wet area subjected to propeller-induced pressure becomes much smaller. This finally influences the total vertical force under the ballast condition to be smaller compared to the full load condition even though the hull pressure in the ballast condition is much higher than the full load condition (see Table 2). Also, it is interesting to note propeller-induced hull pressure of the lowest blade rate mode (1\textsuperscript{st} blade rate) shows a more concentrated pattern compared to the higher blade rate hull pressure. This feature appears more clearly in the full load condition (see Figure 9(a) and Figure 10(a)).
Table 2 Comparison of 1st & 2nd blade rate pressure and vertical force for full and ballast conditions

In the integrated simulation system, HULLFPP not only can provide the pressure amplitude for different blade rate frequencies but can also compute the phase difference associated with the pressure amplitude for different locations on the hull. As will be shown later, this phase difference can have an important influence on the final structure vibration responses.

Hull vibration
As seen in Table 2, compared to ballast condition, the total forces at full load condition have higher values and are expected to cause more structure vibrations. For the sake of brevity, later discussions will be mainly focused on the full load case.

To have a general picture of the propeller-induced surface forces for this tanker, a summarized table for the first three blade rate forces are provided as follows:

Table 3 Propeller surface forces for full load condition

Here, $F_x$, $F_y$, $F_z$ are the total forces obtained by integrating the hull pressure including the phase difference effect. The directions of x, y, and z are in longitudinal, vertical and starboard directions, respectively (Figure 11). The phase definition is 0° at 12 o’clock position and increases in a counterclockwise direction with 90° to port side, 180° to centerline (downward) and 270° to starboard side.

As seen in Table 3, all the surface forces decrease at higher blade rate frequencies and vertical force $F_y$ is the dominated force for excitations. Compared to vertical force $F_y$, horizontal forces $F_x$ of the 1st, 2nd, and 3rd blade rate frequencies are small which are 20%, 8.5% and 6% of the 1st blade rate vertical force $F_y$.

Vibration analyses are performed based on the propeller-induced pressure obtained earlier (see Figure 10) with phase difference effect included. Results of vibration velocity of the whole ship in x, y and z directions are plotted in Figure 12 for the 1st blade rate frequency excitation.
direction. As seen, there is a large area in transom under the vertical vibration due to the propeller excitation but almost negligible vibration in x and z directions at that area. It is also interesting to note at the bridge wings of the desk house structure the vibration velocity (~1.12 mm/s) in x direction shows the same order of magnitude as the vertical vibration velocity (~1.8 mm/s) at the transom area. This is mainly because the local x-direction stiffness of the bridge wing structure is weak and it is easy to excite the x direction vibration even though we know the x direction force \( F_x \) is only about the 40.8% of the vertical force \( F_y \) (see Table 3). Basically, as seen in Figure 12, maximum vibration velocities in x and z directions occur at the bridge wing ends and the top of funnel as the structures have the weak local stiffness in these directions. However, the lateral force \( F_z \) (21.8% of \( F_y \)) is small compared to \( F_y \) and \( F_x \). Consequently, the z direction vibration shows a much smaller value (~0.6 mm/s at the top of the funnel) than the x and y direction vibrations.

To have a general picture of the vibration reduction at the higher blade rate excitations, the vertical vibration velocities due to the 1st, 2nd, and 3rd blade rate propeller excitations are also plotted (Figure 13). Here, all the color contours are drawn based on the same level range – the one used for the 1st blade rate. As seen, stern vibrations reduce rapidly at the high excitation frequencies. Roughly, the y direction vibration velocities (maximum values) at transom locations are about 1.75 mm/s at 1st blade rate frequency, 0.7 mm/s at 2nd blade rate frequency and 0.03 mm/s at 3rd blade rate frequency.

To investigate the phase difference effect on the structure responses for the same tanker, vibration analyses are performed by applying the same propeller induced pressure amplitude but without phase difference. Figure 14 shows the comparisons of the x and y vibration velocities for with and without phase difference effects under the 1st blade rate frequency excitation. Here, for convenience of comparison, contour level ranges are roughly set to be same for the plots (0 ~ 0.0019 for x velocity plot and 0 ~ 0.0021 for y velocity plot). As seen in the figure, if propeller-induced pressure is applied without phase difference, the structure vibrates more. The detailed vibration velocities for some critical locations such as transom and bridge wings are also summarized in Table 4. As expected, phase difference has a significant effect on the structure responses. Due to the phase difference, the total net forces, including \( F_x \), \( F_y \), \( F_z \), on the hull are reduced and the structure tends to have less vibration.
Table 4 Comparison of the vibration velocity at critical locations – with and without phase effects

As known previously, the net forces given in Table 3 are calculated by taking the phase difference into account. These net forces may be regarded as the equivalent loads which can generate the same structure responses as the previous calculations if they are applied at the center of the forces. However, this presents an interesting question to ask. If the vertical force is applied as the only excitation how much is the discrepancy of the vibration result compared to the previous one? For this tanker, vibration analyses are performed again by using the three blade rate vertical forces. The application point of the forces is located at the upper point of the propeller disk on the hull. The vertical velocity contours due to the 1st, 2nd, and 3rd excitations are plotted in Figure 15.

As seen in the figure, basically, this vertical force tends to over-predict the vertical vibrations. A detailed comparison between the vertical vibrations caused by the vertical forces and the vertical vibrations caused by the distributed hull pressure are given in Table 5 for the maximum values picked up in the transom area. In general, the over-prediction of the vertical velocity is about 1.5–2.6 times depending on the blade rate frequency.

Table 4  Comparison of the vibration velocity at critical locations – with and without phase effects

<table>
<thead>
<tr>
<th>Location</th>
<th>x vibration (mm/s)</th>
<th>y vibration (mm/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>phase effect</td>
<td>no phase effect</td>
</tr>
<tr>
<td>Bridge wing – port side</td>
<td>1.01</td>
<td>0.75</td>
</tr>
<tr>
<td>Bridge wing – starboard side</td>
<td>0.96</td>
<td>1.93</td>
</tr>
<tr>
<td>Transom – port side</td>
<td>1.75</td>
<td>1.48</td>
</tr>
<tr>
<td>Transom – starboard side</td>
<td>0.79</td>
<td>2.11</td>
</tr>
</tbody>
</table>

Figure 14: the comparisons of the x and y vibration velocities for with and without phase difference effects
Figure 15: Vertical vibration due to the 1st, 2nd, and 3rd vertical forces

Table 5 Vertical velocity comparison at transom area

<table>
<thead>
<tr>
<th>Excitation</th>
<th>y-direction velocity due to distributed hull pressure mm/s</th>
<th>y-direction velocity due to vertical force only mm/s</th>
</tr>
</thead>
<tbody>
<tr>
<td>1st blade rate</td>
<td>1.75</td>
<td>2.52</td>
</tr>
<tr>
<td>2nd blade rate</td>
<td>0.70</td>
<td>1.82</td>
</tr>
</tbody>
</table>

Figure 16: x-direction velocity due to the 1st blade rate vertical force.

CLOSEUP

An integrated simulation system for a more accurate estimate of propeller-induced hull vibration is presented in this paper. In this simulation system, state-of-the-art CFD and FEM tools are assembled together for propeller/ship interaction flow simulation and propeller-induced hull vibration analysis.

In propeller/hull interaction flow simulation for a series 60 hull with MAU propeller, it is found that propeller/hull interaction plays an important role on the propeller performance and effective wake prediction. By comparing the calculated thrust/torque coefficients ($K_T$ and $K_Q$) and wake field velocity with the available experimental measurement data, it shows that the CFD simulation can provide reasonable accuracy for the propeller/ship interaction flow resolution.

In propeller-induced hull vibration analysis, a tanker is selected to perform numerical study for different loading applications, namely, they are distributed hull pressure with phase difference considered, distributed hull pressure without phase difference, and a net vertical force. The findings of the vibration analyses results are summarized as follows:

- Pressure phase difference has an important influence in structure vibration.
- For the load without phase difference considered, all the vibration velocity in stern area tends to become more violent.
- For a net vertical force excitation, vertical vibration tends to be over-predicted.
- Also, this vertical force excitation can not capture the x-direction vibration on the structure especially at the critical locations such as the ends of the bridge wing.

In structure vibration analysis, in order not to over-predict/under-predict the structure responses, it is recommended that propeller-induced pressure with the phase difference should be used.
ACKNOWLEDGEMENTS

The authors wish to thank the ABS management team, especially Dr. Roger Basu, Dr. Han Yu, Mr. Ernst Kotte, and Dr. Bill Shi, for permission to perform this study. Dr. Sing-Kwan Lee would also like to acknowledge Mr. Yoshi Ozaki’s support and encouragement in publishing this paper. Thanks are also due to Mr. James Speed for helping to proofread this paper.

REFERENCES


