Computational Analysis of Propeller Sheet Cavitation and Propeller-Ship Interaction

J. Bosschers, G. Vaz, A.R. Starke, E. van Wijngaarden
MARIN (Maritime Research Institute Netherlands)

SUMMARY

This paper describes a computational procedure which is being developed for the practical analysis of propeller sheet cavitation and propeller-ship interaction including hull pressure fluctuations. The computational procedure consists of three different, coupled methods: (i) a RANS method for the analysis of the viscous flow around the ship hull and the determination of the velocity field in the propeller plane, (ii) a boundary element method for incompressible potential flow developed for the analysis of propeller performance including sheet cavity extent and volume, and (iii) a boundary element method which solves the acoustic wave equation to compute the propeller radiated pressure fluctuations on the ship hull. The background to the three methods and the coupling procedures are briefly described followed by a discussion on validation work. The validation work concentrates on the ship wake field, the propeller sheet cavity extent and the hull pressure fluctuations for model scale as well as full scale. The differences found between model scale and full scale are used to discuss implications for model-scale testing.

1. INTRODUCTION

With the increase in fuel prices, the interest in more efficient ship propulsors is rapidly growing. In the last couple of decades, however, an opposite trend could be witnessed. High-comfort standards in terms of low noise and vibration have been obtained at the expense of propeller efficiency. The conflicting requirements of high efficiency and comfort force designers into balancing propeller designs with respect to cavitation performance and efficiency, and require the designer to pay more attention to the details of the propulsor and the propulsor-hull interaction. These requirements call for improved design tools to predict the flow around the ship and its propeller. In addition to model testing in a towing basin, the use of advanced computational tools will aid in striking the right balance between efficiency and comfort. The computational tools used in the day-to-day design process of the ship and its propeller must show fast turn-around times, robustness and consistently accurate results. A limiting factor in this process is the computation of the dynamic action of propeller cavitation and its effect on hull excitation (i.e. comfort) levels. Even though results for unsteady cavitating propellers obtained by solving Reynolds-averaged Navier-Stokes (RANS) equations are becoming available, the turn-around time of such computations are too long for practical applications. Therefore, a cavitation model based on potential flow has been adopted for the development of a practical computational procedure for the analysis of hull excitation forces caused by a cavitating propeller operating in the wake of a ship. The computational procedure is based on three different numerical methods:

(i) A RANS solver for the analysis of the steady flow around the ship to predict the wake field in the propeller plane;
(ii) A boundary element method for incompressible flow that predicts the unsteady flow around the propeller including sheet cavitation;
(iii) A boundary element method that solves the acoustic wave equation for the analysis of the hull diffraction, necessary to predict the propeller-induced hull pressure fluctuations.

The background of the numerical methods is briefly described in Section 2. Results obtained with the unsteady cavitation model are compared in detail with experimental data in Section 3. The coupling between the different numerical methods is described and tested in Section 4. The test case that is analysed is a container vessel for which both model and full-scale cavitation observations and hull pressure fluctuations are available. The paper ends with a discussion on scaling aspects including implications for model-scale testing and conclusions.

2. NUMERICAL METHODS

2.1 SHIP FLOW ANALYSIS

A viscous-flow analysis is necessary to predict the ship wake field in the propeller plane. Use is made of the PARNASSOS code [1, 2], a RANS solver developed and used by MARIN and IST (Instituto Superior Técnico, Lisbon, Portugal) dedicated to the prediction of steady viscous flow around ship hulls. It solves the discretised RANS equations for a steady, 3D incompressible flow. The code uses structured multiblock body-fitted grids. Of the various turbulence models available in the code, the one-equation model by Menter [3] has been used for the present computations. Unlike most other methods, PARNASSOS solves the momentum and continuity equations in their original form without resorting to e.g., a pressure-correction equation or an artificial-compressibility form. This is
achieved by retaining the full coupling of all equations during the solution process. After discretisation and linearisation, the momentum and continuity equations result in a matrix equation, which is solved for all variables simultaneously using GMRES with an incomplete LU-factorisation as a preconditioner. The fully coupled solution procedure has been found to be robust and quite insensitive to the mesh aspect ratio. This permits to easily carry out even full-scale calculations without using wall functions.

2.2 PROPELLER FLOW ANALYSIS

For the analysis of the flow past a propeller, use is made of a boundary element method (BEM) that solves the incompressible potential flow equations for lifting surfaces. The method, designated PROCAL, is being developed within MARIN’s Cooperative Research Ships (CRS) for the unsteady analysis of cavitating propellers operating in a prescribed ship wake. It has been validated for open water characteristics, shaft forces, and sheet cavitation inception and extent. The code is a low order BEM that solves for the velocity disturbance potential. For the cavitating flow, the non-linear kinematic and dynamic boundary conditions are iteratively solved assuming that the cavity surface coincides with the body surface, i.e. a partially non-linear model similar to [4]. 2-D computations have shown that, from a practical point of view, this model gives identical cavity extents and volumes as fully non-linear models while the CPU-time is significantly smaller [5]. The time derivatives of the cavity thickness in the kinematic boundary condition are taken into account implicitly, while the time derivatives of the velocity potential in the non-linear dynamic boundary condition are taken explicitly from the previous iteration level using a relaxation procedure to prevent oscillations. The cavity reattachment is based on the model proposed in [6]. The implemented cavitation model requires a small panel spacing at the leading edge of the propeller blade in order to obtain good agreement with cavity extents observed in experiments [7]. Initial validation studies and details on the mathematical and numerical model can be found in [7, 8].

2.3 HULL EXCITATION ANALYSIS

Once the analysis of the propeller flow has been performed, the effect of the cavitating propeller on vibratory hull excitation can be determined. This requires the computation of the diffraction effect of the wetted hull surface. For that purpose a boundary element analysis for acoustic scattering is performed using the code EXCALIBUR [9]. EXCALIBUR is a frequency domain BEM that solves the Kirchhoff-Helmholtz integral equation for the disturbance potential of the acoustic velocity using a method proposed by Burton and Miller [10]. The distribution of the diffracted velocity potential on the hull surface can be obtained by specifying the strength, location and type of the acoustic sources in the free field. The zero-pressure condition at the free surface is fulfilled by applying a standard double-body approach. The spectral components of the hull surface pressures are derived from the velocity potentials through the transient term in Bernoulli’s law. Straightforward integration of the pressure over the hull then yields the hull excitation force at each selected frequency.

3. VALIDATION OF THE CAVITATION MODEL

3.1 INTRODUCTION

In this section the current status of PROCAL’s sheet-cavitation model is illustrated by comparing the cavitation simulations for two propellers with experimental data. First, the computed cavity extent for a propeller of conventional planform is validated using photos and values for the cavity area obtained by INSEAN, Italy [11]. Hull pressure fluctuations are mainly caused by cavity volume accelerations and hence it is the cavity volume (or cavity thickness) that is important as well. The predicted cavity thickness distribution is validated with experimental data for a high skew propeller as obtained by NMRI, Japan [12], formerly known as SRI.

3.2 CAVITY EXTENT

A detailed validation of the cavity extent has been carried out for the INSEAN E779A propeller, which has been tested in a screen generated wake in the Italian Navy Cavitation Tunnel CEIMM [11]. This propeller has a conventional planform with small skew angle and constant pitch distribution. The layout of the propeller with the panel distribution is presented in Figure 1. The measured wake has been made effective by a force-field method [11] and the resulting wake is shown in Figure 2. The computations have been made at a thrust coefficient $K_T=0.175$ and cavitation number $\sigma_p=2.835$, similar to the values in the experiment. A panel distribution of 120 panels along the chord and 20 panels in the radial direction has been used, together with 144 time-steps per propeller revolution.

Figure 1: Panelling distribution for propeller E779A.
A comparison between the results obtained by PROCAL and the experimental data is shown in Figures 3 and 4. The experimental values for the cavity area have been determined using image analysis [11]. The agreement is reasonable for both cavity area and cavity planform. In Figure 3 it is seen that PROCAL overestimates the cavity extent during the growth phase, but the maximum area and extent during collapse are well predicted. The collapse phase of the cavity is faster than the growth phase, a fact well-known from experiments. The phase difference in the cavity growth between experiments and computations might be related to a lack of accuracy of the potential time-derivative in the dynamic boundary condition, which was accepted to increase robustness. This requires further study. It can also be observed from Figure 4 that PROCAL does not capture the cloud cavitation existing at $\theta=15^\circ$, which is inherent to the potential flow model.

3.3 CAVITY THICKNESS

The experimental determination of cavity thickness is very complicated and as a result experimental data is scarce. One of the few data sets available in literature has been obtained in the cavitation tunnel of NMRI for a model of the training ship SEIUN-MARU [12]. The cavity thickness was measured using the reflection of a pulse laser beam on the cavity surface. The pulse-laser beam was coupled to a CCD camera and an image processor and synchronised with the revolution rate of
the propeller. The system was calibrated to estimate the coordinates of the reflected beam from which the cavity thickness could be estimated. The maximum error in the thickness is given as 0.5 mm. However, near the leading edge the cavity was reported to be too transparent to reflect the laser beam. The beam is then deflected by the cavity and reflected by the propeller surface resulting in a negative cavity thickness. Such data points were removed in the analysis procedure. Both a conventional planform propeller and a highly skewed propeller (HSP) were analysed for one rotation rate.

Figure 5: Predicted cavity layout of the high skew propeller of the SEIUN-MARU at different blade positions.

Results are presented for the HSP operating at thrust coefficient \( K_T = 0.201 \) and cavitation number \( \sigma_n = 2.99 \). The panel distribution on each blade consists of 120 panels in chordwise direction (60 on each side) and 20 panels in radial direction. 120 time steps per revolution were used. Comparison with cavity sketches presented in [12] gives a similar trend as found for the E779A propeller: the computed cavity grows somewhat faster than observed in the experiment while the maximum extent is well predicted. The overall computed cavity layout is presented in Figure 5. The cavity thickness distribution, made non-dimensional with the chord of the blade section, is presented in Figure 6 for two blade positions close to where the maximum cavity extent occurs. On average, a fair prediction of the cavity thickness is obtained. Near the leading edge, the differences are probably caused by poor reflection of the laser beam on the cavity surface discussed before. The differences near the end of the cavity might be due to unsteady cloud shedding in the experiment which cannot be modelled by the potential flow model. Note that the computed supercavity extends beyond the trailing edge on the propeller wake surface. This part of the cavity is not shown in Figure 6.

4. PROPELLER-SHIP INTERACTION

4.1 INTRODUCTION

In the current approach, three steps are distinguished in the analysis of the propeller-ship combination. The first step is the computation of the strong interaction between the propeller and the flow around the ship. In the intended coupling procedure, the propeller analysis BEM is to provide the propeller force distribution for the viscous-flow computation, and the viscous-flow computation is to provide the inflow velocity field for the BEM. The velocity field that couples these two methods is designated the effective ship wake. In the second step this effective wake field is used for the analysis of the variation of the sheet cavity extent. The third step is the computation of the fluctuating pressure exerted on the ship hull. This pressure field is generated by the cavitating propeller and is influenced by scattering of the ship hull and the free surface. This scattering effect is computed using a BEM suitable for acoustic analysis. These three steps are discussed separately in the following sections on the basis of a test case.

The test case used is a large container carrier of 6800 TEU fitted with a single fixed-pitch propeller. The main particulars of ship and propeller are presented in Table 1. The model scale ratio \( \lambda \) equals 33.7. For the computations presented here the 85% MCR condition is selected. The ship and its propeller have been extensively analysed in the joint industry project CoCa (Correlation of Cavitation) at which time the ship’s name was P&O NEDLLOYD SHACKLETON. The CoCa project was a joint project initiated in 1999 by Wärtsilä Propulsion Netherlands BV (WPNL) and MARIN, with contributions from several ship yards and owners. The
project consisted of conducting full-scale cavitation observations and hull pressure fluctuations for five ships before performing model tests in MARIN’s Depressurized Towing Tank (DTT) under identical conditions. In addition, computations for model-scale and full-scale conditions were made by WPNL and MARIN. After the CoCa project further research has been carried out on P&O NEDLLOYD SHACKLETON sponsored by the Dutch government under their Nederland Maritiem Land (NML) grants scheme. The results of both studies have been published in [13]. That reference also contains the details of the full-scale trials and model-scale measurements in the DTT. The computational approach presented here is based on more advanced methods than used in these studies: the propeller flow is analysed using a BEM instead of a lifting surface method and a hull diffraction computation is now included.

Table 1: Main particulars of P&O NEDLLOYD SHACKLETON.

<table>
<thead>
<tr>
<th>Ship particulars</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Length (L_pp)</td>
<td>286.00 m.</td>
</tr>
<tr>
<td>Breadth</td>
<td>42.80 m.</td>
</tr>
<tr>
<td>Draft</td>
<td>12.15 m.</td>
</tr>
<tr>
<td>MCR</td>
<td>65,880 kW.</td>
</tr>
<tr>
<td>RPM at MCR</td>
<td>100.0</td>
</tr>
<tr>
<td>Propeller particulars</td>
<td></td>
</tr>
<tr>
<td>Diameter</td>
<td>8.750 m.</td>
</tr>
<tr>
<td>Number of blades</td>
<td>6</td>
</tr>
<tr>
<td>Expanded area ratio</td>
<td>1.033</td>
</tr>
</tbody>
</table>

4.2 EFFECTIVE WAKE FIELD

If the flow around the ship hull is analysed without propeller, the wake field in the propeller plane is designated nominal wake. If the flow is analysed including propeller, the wake field is designated total wake. Due to the interaction between the propeller and the ship flow, neither the nominal wake field nor the total wake field is representative of the inflow velocity field required by the propeller analysis method. The wake field that is considered representative is called effective wake field. This effective wake field can be defined as the nominal wake field corrected for the (propeller-ship) interaction velocities, or as the total wake field minus the propeller-induced velocities. Both approaches are discussed in the following paragraphs. Contrary to the nominal and total wake field, the effective wake field can not be measured.

An approach for correcting the nominal wake field is the force field method (FFM). It assumes a prescribed propeller thrust and radial loading distribution. Then disturbance velocities with respect to the nominal wake field are computed by solving the 3-D Euler equations including the force field. The disturbance velocities consist of the required interaction velocities and the induced velocities which are obtained using vortex cylinder theory. Addition of the interaction velocities to the nominal wake field yields an effective wake field. The method was developed at MARIN in the 1980s within the CRS organization, and has been used extensively since. Historically, the method is applied to a measured nominal wake field, but it can evidently be applied to a predicted wake field as well.

Nowadays fast and accurate viscous-flow solvers are available and the predicted wake fields can be used early in the design of a ship, and well ahead of model testing [14]. To show the capabilities of viscous-flow solvers, Figure 7 compares the measured nominal wake for the test case with the computed wake. The result obtained is representative of what is found for single-screw ship stern flows in general; a slight under-prediction of the strength of the longitudinal vortices that cross the top half of a propeller disk. Related to that is an under-prediction of the axial velocity in that area. The differences can be attributed to the choice of the turbulence model (Menter’s one-equation model); somewhat better results might be obtained by using another turbulence model, but only at the cost of robustness.

Figure 7: Comparison between the measured and computed nominal wake field at model scale.

In addition to the prediction of nominal wake fields, viscous-flow solvers can be used to compute total wake fields, for instance by adding the propeller as an actuator disk. Subtracting the propeller-induced velocities from the total wake field yields the effective wake field. This approach captures more physical aspects of the propeller-hull interaction than the FFM discussed above. An example is flow separation that may occur or disappear due to the propeller action. Furthermore, Reynolds scale effects on the wake field are automatically incorporated. In the coupled approach, the thrust distribution used in the actuator disk model is calculated from the output of the propeller analysis BEM. Since the effective wake field is input for the BEM, iteration between the viscous-flow solver and the BEM is required. The fundamental issue in this coupled approach is the determination of the velocity field induced by the actuator-disk model that is subtracted from the total viscous wake field. In the present approach, this velocity field is computed by the
BEM that has also been used for the determination of the thrust distribution. The plane for the computation of the wake field was selected slightly upstream of the propeller to prevent the plane from intersecting with the singularities of the BEM. However, open water computations showed some differences between the induced velocities from the actuator disk and the BEM. These differences, which are being further investigated, vary with the upstream location of the plane. The thrust distribution in the actuator disk model is varying in radial and circumferential direction. The corresponding induced velocity field computed by the unsteady BEM has been time-averaged. The iteration process uses a non-cavitating analysis of the propeller and converges quickly.

In the following paragraphs, the model-scale effective wake fields obtained with the FFM will be discussed first, followed by a comparison between the results of the coupled method and the FFM. The section concludes with a comparison between the model-scale and full-scale wake field.

Figure 8 presents the effective wake fields obtained from the measured and the computed nominal wake fields (see Figure 7) using the FFM. The velocity deficit in the upper part of the wake has decreased in both cases. For the lower part of the wake, the velocity deficit decreases near the hub while in the outer part the changes are very small. Note that in the FFM only a change in axial velocity components can be computed; the transverse velocities are taken identical to the velocities in the nominal wake field.

An example of an effective wake field computed with the coupled procedure is presented in Figure 9 together with a computed nominal wake made effective with the FFM. Along the outer radii of the disc, the effective wake fields are quite similar, but at the inner radii in the top half of the propeller disc the coupled approach gives axial velocities that are much smaller than obtained with the FFM. This is further illustrated by comparing the mean axial velocity in the wake field, listed in Table 2.

From this table it can be seen that the mean velocity for the measured and the predicted model-scale nominal wake fields is equal, even though the distribution of the axial velocities over the propeller disk is different, as shown in Figure 7. This different distribution causes a 5 per cent difference in the effective wake fields obtained using the FFM: the measured wake field which contains larger velocity gradients in the top half of the disk is more affected by the FFM than the computed wake field. The mean velocity obtained with the coupled approach, however, is approximately 15 per cent smaller than the mean value obtained from the FFM. This is mainly caused by the low effective axial velocities that occur with the coupled approach around the 12 o'clock position in the top half of the disk. The mean velocity for this wake field is even smaller than the mean value of the nominal wake field. This unexpected behaviour is probably due to an incorrect estimation of the induced velocities for the actuator disk model in the viscous-flow computations.

Figure 8: Comparison between model-scale effective wake fields using the FFM for the measured and computed nominal wake fields.

Figure 9: Comparison between model-scale effective wake fields using the coupled approach and using the FFM with the computed nominal wake field. The dotted line indicates the edge of the propeller disk.

Figure 10: Comparison between model-scale and full-scale effective wake fields using the FFM for the computed nominal wake fields.
The effective wake field for the ship is compared with the model-scale wake field in Figure 10. The presented effective wake fields are obtained from computed nominal wake fields made effective using the FFM. As expected the axial velocity deficit has significantly decreased, resulting in an increase of the mean axial velocity (see Table 2).

It must be kept in mind that the effective wake field cannot be validated directly with experimental data and is only needed because different computational procedures have been used for the flow around the ship and the flow around the propeller. The effective wake fields can be judged by analysing the results of the propeller computations. In the following sections the computed sheet cavity extent and hull pressure fluctuations for the different effective wake fields will be compared with experimental data. Another approach, not pursued here, is to compare measured and computed wake fractions at e.g. thrust-identity obtained from propulsion data and propeller open water characteristics.

Table 2: Review of mean axial velocities for the nominal and effective wake fields. The velocity is made non-dimensional with the ship speed.

<table>
<thead>
<tr>
<th>Wake field</th>
<th>Mean axial velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>Nominal wake fields</td>
<td></td>
</tr>
<tr>
<td>measured</td>
<td>0.71</td>
</tr>
<tr>
<td>computed model</td>
<td>0.71</td>
</tr>
<tr>
<td>computed ship</td>
<td>0.81</td>
</tr>
<tr>
<td>Effective wake fields</td>
<td></td>
</tr>
<tr>
<td>measured (FFM)</td>
<td>0.79</td>
</tr>
<tr>
<td>computed model (FFM)</td>
<td>0.74</td>
</tr>
<tr>
<td>computed model (coupled)</td>
<td>0.69</td>
</tr>
<tr>
<td>computed ship (FFM)</td>
<td>0.82</td>
</tr>
</tbody>
</table>

### 4.3 CAVITY EXTENT

This section shows the influence of the different effective wake fields on the cavity extent while the influence on the hull pressure fluctuations is discussed in Section 4.4. The computations have been carried out at thrust identity by changing the ship speed. The static pressure at the propeller shaft, necessary for the computation of the cavitation number, has been increased to account for trim, sinkage and stern wave height. The value has been determined by increasing the draft of the ship until the wetted part of the transom shows a similar width as observed in the experiment. The propeller computations were made using 120 panels along the chord, 25 panels in radial direction and 90 time steps per revolution. A view of the propeller and the panel distribution is presented in Figure 11.

The cavity extent at different blade positions in the model-scale experiment is presented in the left column of Figure 12. The indicated blade position for the experimental observations is for blade no. 1, so blade no. 2 is located 60° before the indicated blade position. The experimental observations show sheet cavitation inception just after θ = 324°. At blade position θ = 336° the cavity is located between 0.7R and 0.8R and is growing towards the tip at increasing blade positions. When the blade passes the top dead centre a cavitating tip vortex develops and at following blade positions the sheet cavity collapses towards the tip. The re-entrant jet formed at the aft end of the cavity causes a highly irregular vortical cavitating flow structure which may interact with the cavitating tip vortex. At blade position θ = 60° an isolated cavitating tip vortex is present.

The predicted cavity extent for the effective wake field based on the nominal measured wake field is presented in the right column of Figure 12. In general, good agreement is obtained for the sheet cavity extent. Comparison of the cavity extent for blade no. 2 at blade position θ = 36° again shows that during cavity growth the computed extent is larger than observed in measurements. The details of the cavity closure with the re-entrant jet and the cavitating tip vortex cannot be modelled. The results suggest, however, that the overall cavity layout is not too much influenced by these phenomena.

The results for the effective wake field based on the nominal model-scale PARNASSOS computations are presented in Figure 13 on the left. The slightly larger peak of the computed ship wake in comparison with the measured wake field leads to an increase of the cavity extent. Comparison with experimental observations shows that the character of the sheet cavity is still well predicted, but the computed chordwise extent is too large. To illustrate the sensitivity of the cavitation model to the ship wake, the cavity extent for the coupled wake field, presented in Figure 9, is shown in the right column of Figure 13. As expected, the cavity extent has further increased compared to the effective wake fields using FFM, and it can be concluded that further research for this procedure is necessary.

The full-scale cavitation observations are presented in Figure 14. In general, the cavity behaviour is as described for the model-scale observations with the exception that much more cavity clouds are being shed from the cavity and that a continuously cavitating tip vortex is present at full scale. The cloudy structure of the cavitating tip vortex, especially apparent in the
Figure 12: Cavity extent at different blade positions (given for blade no. 1). Left: Experiment in the DTT. Right: Computations using the measured nominal wake field made effective using FFM.

Figure 13: Computed cavity extent at different blade positions for model scale. Left: Using the computed nominal wake field made effective using FFM. Right: Using the effective wake field from the coupled approach.

Figure 14: Full-scale observations of cavitation on the back of the propeller through a port-side window, blade positions are indicative.

Figure 15: Computed cavity extent at different blade positions for the ship. Left: Using the computed nominal wake field for the ship made effective using FFM. Right: Using the measured wake field at model scale made effective using FFM, computation at $J_{wp}$-identity.
observation at $\theta=60^\circ$ is caused by the interaction between vorticity shed by the action of the re-entrant jet and the cavitating tip vortex through which the cavity is locally destroyed. This behaviour has been observed using high speed video and a more detailed discussion of the phenomena is given in [15].

Both radial and chordwise extents of the sheet cavity in Figure 14 are difficult to determine. For that reason the results of the full-scale computations are presented in Figure 15 from an identical viewpoint as the model-scale computations. A direct comparison between full-scale and model-scale cavity extent is then also possible.

The results of the computations using the full-scale PARNASSOS nominal wake field made effective using the FFM are presented in the left column of Figure 15. The chordwise extent of the cavity is significantly reduced in comparison with the computations made for the model-scale PARNASSOS wake field and somewhat reduced compared with the model-scale experimental wake field.

The comparison between the full-scale observations and the computation using the full-scale wake field shows similar behaviour of the sheet cavity. However, the full-scale observations can only be judged from a qualitative point of view. The radial extent of the cavity at blade positions $\theta=24^\circ$ and $\theta=36^\circ$ seems somewhat larger in observations made during the trials. The largest difference is the absence of the cavitating vortex in the computations. Instead, a sheet type of cavitation occurs in the tip region which completely collapses while the cavitating vortex only changes in size and does not collapse.

Model-scale experiments are usually carried out at identical average blade loading or identical advance ratio as on full scale. The difference between the model-scale and full-scale wake field implies that the local blade angle of incidence in the wake peak for the ship is smaller than at model scale which may lead to a reduction in loading, a reduction in cavity extent and thus to a reduction of the hull pressure fluctuations. To account for this scale effect on the ship wake in model-scale experiments, it has been proposed in [16] to apply equal local advance ratio of the propeller blade in the upper wake peak near the tip radius. This location coincides with the centre of the cavity source generating the hull pressure fluctuations. Identity for the local advance ratio in the wake peak is achieved by increasing the ship speed while maintaining RPM. The condition is here designated as $J_{op}$-identity. Based on a number of cases and ship wake computations, a statistical formulation for the increase of the ship speed has been derived. For the present ship this resulted in a 9% reduction of the average thrust coefficient. The computations at this thrust coefficient for a model-scale measured wake field made effective using the FFM are presented in the right column of Figure 15. The cavity extent is very similar to the computations for the computed ship wake field presented in the left column of Figure 15.

4.4 HULL EXCITATION

Having discussed propeller-hull interaction effects on the effective wake field and cavitation extent, this section presents some results of the numerical method for the computation of the fluctuating pressure field exerted on the wetted ship’s hull. It is assumed that the pressure field on the propeller blades as well as the formation of cavities is unaffected by the scattering effect of the hull and the free surface. The fluctuating pressure field can be computed when the propeller cavitation analysis is finished.

The excitation pressures on the hull have been computed using the concept of a solid boundary factor, defined as the ratio between hull pressure and (fictitious) free-field pressure at the same location. The cavitating propeller is modelled by a single acoustic monopole, placed at an appropriate location in the propeller disc. Then, EXCALIBUR is used to perform a scattering analysis at given frequency, which gives the hull diffracted surface pressures. The latter can be divided by their free-field counterparts to yield the solid boundary factors. The free field pressures computed by the propeller flow analysis at the location of the hull are then multiplied by these solid boundary factors to obtain the hull surface pressure distribution as a function of frequency. The advantage of this approach is that the effect of variables like compressibility can easily be investigated, as will be done in one of the following paragraphs. Furthermore, empirical formulas for the solid boundary factor can be used if the geometry of the ship hull is not available.

Another approach, which is currently being implemented, is to interpret the result of the propeller computation as a set of acoustic sources of varying strength rotating in the free-field. A Fourier decomposition of the strength of each source along its circular path then leads to a frequency domain description of the source field. Next, the diffracted hull pressures are computed by EXCALIBUR using this source field as input. This approach is more correct from a theoretical point of view, especially if the contribution of the non-cavitating propeller to the hull pressure fluctuations is considered.

The solid boundary factors for the ship with a flat free surface have been investigated for both model and full-scale conditions. Model-scale experiments in the DTT are performed at propeller rotation rates at identical Froude number as for the ship. This implies that the model-scale ratio between the acoustic wave length and the length of the ship is a factor of $\sqrt{\lambda}$ larger than for the ship, with $\lambda$ the geometric scaling factor ($>1$). Using identical ship geometry and size, the model-scale solid boundary factor can then be obtained by reducing the ship blade passage frequency by a factor of $\sqrt{\lambda}$. The result for the first blade passage frequency is presented in Figure 16. Directly above the propeller a small change in solid boundary factor is observed (typical difference of 0.2) while at a larger distance the changes are significant. The overall pattern is very similar. For higher blade passage frequencies the difference between model and
ship increases in magnitude and pattern. As already
mentioned in Section 4.3, the draft of the ship has been
increased to account for trim, sinkage and stern wave
height.

![Figure 16: Amplitude of the solid boundary factors for
the first blade passage frequency. Left: Model scale.
Right: Full scale.]

The distribution of the computed pressure fluctuations
across the model-scale ship hull at blade passage
frequency is presented in Figure 17. The distribution is
very similar to that generated by a single acoustic
monopole. Included in the figure are the locations and
values of the model-scale pressure transducers. All
pressure pulse amplitudes presented in this and following
figures have been made non-dimensional using the
maximum pressure amplitude measured during the ship
trials and are given for first blade passage frequency only.
The model-scale pressure pulses measured above the
propeller disk (indicated by the arrow in Figure 17) are
compared with the computed amplitudes in Figure 18.
The computations using the measured wake field made
effective using FFM, which gave the best prediction of
the cavity extent, gives a good prediction for the
transducers above the upcoming blade. However, the
computed maximum value and the values above the
downgoing blade are smaller than measured. The
computations using the computed model-scale wake field
(made effective using FFM) showed a somewhat larger
cavity extent and the hull pressure amplitudes have
increased accordingly.

The distribution of the hull pressure amplitudes for the
ship is presented in Figure 19. The figure includes the
location of the pressure transducers and their measured
amplitudes. The distribution of the computed pressure
amplitudes is very similar to the model-scale distribution
presented in Figure 17. The measured amplitudes above
the propeller disk are compared to the computed
amplitudes in Figure 20. The graph also includes the
model-scale measured and computed amplitudes using
\( J_{wp} \)-identity. This condition gives a more realistic local
advance ratio for the propeller blade in the wake peak as
discussed in Section 4.3. It is observed that the maximum
predicted pressure amplitude using the computed ship
wake field is 50% larger than measured during the sea
trials. This difference between predictions (using
PARNASSOS plus FFM wake fields) and measurements is
larger than for the model-scale situation where the
difference is only about 10%. The model-scale
experiment at \( J_{wp} \)-identity gives a good prediction,
although the maximum value is still 20% too large. This
prediction is a significant improvement on the
experimental results obtained at similar advance ratio
presented in Figure 18. The computations at \( J_{wp} \)-identity
also give an acceptable prediction of the full-scale
amplitudes and are close to values obtained in the model-
scale experiment.

4.5 DISCUSSION OF SCALING ASPECTS

The previous sections have treated several differences
between model-scale and full-scale computations. These
differences are reviewed here and the implications for
model-scale testing are discussed.

The most prominent scaling aspect is the difference
between the model-scale and full-scale ship wake due to
differences in Reynolds number. The model-scale wake
has a larger velocity deficit in the top part leading to a
higher loading of the propeller blade in the wake peak in
comparison to full scale. As illustrated, the common
procedure for model-scale cavitation testing (at identical
advance ratio or thrust coefficient as the ship) would
result in a large overprediction for the cavity extent and
hull pressure fluctuations for this particular ship. A
significantly improved prediction for the full-scale
pressure amplitudes is obtained by using \( J_{wp} \)-identity [13,
16, 17], explained in Section 4.3. Depending on ship type,
the method is regularly applied in DTT model tests, but
details need to be further investigated. The computation
using a computed full-scale wake field showed fair
agreement to the computation at \( J_{wp} \)-identity using a
model-scale wake field. The computational procedure
can therefore be used to investigate the influence of wake
scaling on cavity extent and hull pressure fluctuations in
more detail and to investigate the consequences of
changing the advance ratio of the propeller in model
experiments. However, it is necessary to first conduct
more extensive validation studies of the computational
procedure for different ships and propellers.

Since the sheet cavity model is based on potential flow
theory, no scale effects other than due to wake scaling
can be computed. Comparison of measured and
computed sheet cavity extent shows that the potential
flow model captures the overall cavity layout and that the
hull pressure amplitudes at the first blade passage
frequency are reasonably well predicted. Limitations of
the potential flow model are most pronounced in the
cavity closure region where the re-entrant jet is present
and in the formation of a cavitating leading edge or tip
vortex. Their influence on sheet cavitation and low
frequency hull pressure fluctuations is however to be
further investigated, for instance by using an unsteady two-phase RANS solver [18, 19]. The cavitating tip vortex is also an important contributor to the higher harmonic hull pressure fluctuations [20] which have not been discussed here.

For the hull excitation forces, differences in the solid boundary factors have been observed between model-scale and full scale. Even though these differences do not influence the hull excitation as much as the difference in the wake field, it can easily be taken into account in model-scale experiments using the concept of cavitation source strength in combination with the acoustic diffraction solver [9]. In addition, differences in the solid boundary factor may arise due to differences in the stern wave height for model and ship. 2-D computations of transom flows [21] have shown an influence of viscous effects and suggest that the wetted area is smaller for the ship than for the model. As these computations are not yet available for 3-D ships, the presented results for the solid boundary factor were made using identical stern wave height for model and ship.
5. CONCLUSIONS

A computational procedure for the practical analysis of hull excitation forces due to cavitating propellers operating in a ship wake has been described and applied to a container vessel for which both model-scale and full-scale measurement data are available. The procedure is based on the coupling of computational methods for the analysis of the ship flow, the analysis of the propeller flow and the prediction of the hull excitation forces.

The results show that for the container vessel under consideration, the computational procedure can give reasonable to good results for the nominal wake field, the cavity extent and the hull pressure fluctuations. The hull pressure fluctuations for model scale were more accurately predicted than for full scale. More validation studies are required to investigate if these conclusions can be generalised.

Several issues have been identified which need to be studied in more detail and which may improve the accuracy. One issue is the computation of the effective wake field using the coupled approach between the methods for ship and propeller flow analysis. This is expected to give more accurate results than correcting the nominal wake field with a force field method. Other issues are related with the accuracy of the sheet cavitation model, the absence of a model for the cavitating tip vortex and the direct computation of the hull pressure fluctuations by the acoustic diffraction solver.

The computational methods described can also be used to improve model-scale testing procedures. Knowledge of the model and ship wake field and propeller blade loading allows for similarity testing at local advance ratio (or local blade loading) instead of advance ratio (or average propeller loading). This procedure was shown to give an improved prediction for full-scale hull pressures using model-scale wake fields, experimentally as well as computationally.

6. ACKNOWLEDGEMENTS

The boundary element method for propeller analysis PROCAL is being developed within the Cooperative Research Ships (CRS) organization. The authors also like to acknowledge the contribution of Martijn Zijlstra of MARIN to the software development of the boundary element methods. The comparison between the measured and computed cavity thickness for the SEIUN-MARU propeller has been made by Bram van Eijkeren (MSc student of Twente University) during his training period at MARIN.

7. REFERENCES


8. AUTHORS’ BIOGRAPHIES

Johan Bosschers is project manager at MARIN and is involved in various aspects of two-phase flow and propulsor hydrodynamics.

Guilherme Vaz holds the position of CFD-researcher at MARIN’s R&D department. He is currently involved in the development of both potential and viscous-flow codes for multi-phase flow around hydrodynamic structures.

Bram Starke holds the position of CFD researcher. In the past years he has contributed to the development and the utilisation of MARIN’s viscous-flow solver PARNASSOS, with an emphasis on propeller-hull interaction.

Erik van Wijngaarden is senior project manager at MARIN’s R&D department and is involved in research activities concerning propeller hull excitation.