NUMERICAL CAVITATION NOISE PREDICTION OF A BENCHMARK RESEARCH VESSEL PROPELLER

Savas Sezen, University of Strathclyde, UK
Mehmet Atlar, University of Strathclyde, UK
Patrick Fitzsimmons, University of Strathclyde, UK
Norihuki Sasaki, University of Strathclyde, UK
Giorgio Tani, University of Genova, Italy
Naz Yilmaz, Technical University of Bursa, Turkey
Batuhan Aktas, Fugro Structural Monitoring, UK

The study of hydrodynamic noise is an active research field due to the rapid increase of noise levels in the oceans as well as its impact on marine fauna. In this regard, non-mandatory regulations are becoming widespread to reduce noise levels induced by shipping activities. The development of reliable tools for the prediction of underwater radiated noise (URN) is required for both accurate design of the ship propulsors and predicting the noise levels precisely. This paper presents the preliminary results of a numerical study for noise prediction of a benchmark propeller in open water/uniform flow conditions. The experimental benchmark test data for the research vessel, “The Princess Royal”, in uniform flow conditions were used for validation purposes. The numerical analyses were implemented by using a viscous solver based on the finite volume method while the experimental data were obtained from model tests conducted at the Genova University Cavitation Tunnel. The main aim of the study is to predict propeller hydroacoustic performance under cavitating conditions. The hydrodynamic flow field was solved using a RANS (Reynolds-averaged Navier-Stokes) solver. The Schrerr-Sauer cavitation model based on a reduced Rayleigh-Plesset equation together with a VOF approach was used to model sheet cavitation on the propeller blades. The computed hydrodynamic characteristics and sheet cavity patterns are shown to be in good agreement with the Genoa experimental data thus providing a firm basis for cavitating noise predictions. The hydroacoustic performance of the model propeller is predicted by using a hybrid method. In the noise simulations, RANS equations were equipped with a porous FW-H (Fjowcs Williams-Hawkings) formulation. The different propeller operational conditions were simulated using this hybrid method. The numerical results were also validated with the experimental data for the propeller hydroacoustic performance. Such validations have shown good agreement with the benchmark test cases, especially in the low-frequency region.

Keywords: CFD; Cavitation; FW-H; The Princess Royal research vessel; Propeller Underwater Radiated Noise (URN); RANS.

1. Introduction

Cavitation is a complex fluid mechanics phenomenon for pumps, nozzles, injectors, turbines, propellers and a variety other fluid machinery components. It can lead to undesirable effects for marine propellers such as efficiency loss, hull/shaft vibrations, underwater radiated noise and blade erosion [1]. While there are different types of cavitation such as sheet, cloud, tip and hub vortex, cavitation on propeller blades, these can be also classified in two parts: cavitation attached to blade surfaces; and cavitation off the blade surfaces (e.g. tip vortex cavitation) [2]. The effects of such cavitation types on the propeller hydrodynamic and hydroacoustic performances are different. For instance, while tip vortex cavitation is a major contributor to broadband noise, especially in the presence of the vortex dynamics, sheet cavitation collapses can be seen in the low frequency noise region.
Underwater Radiated Noise (URN) from ships has received increased attention over the last decade, in the design of improved naval systems, fishing vessels and increased comfort in cruise vessels, but primarily due to concern for the possible impact on marine wildlife. The issue was put on the political agenda of the Marine Strategy Framework Directive, MSFD, D11 2017/848 for a relevant qualitative descriptor for a good environmental status in the marine environment by the European Commission in 2016. Shipping noise originates from various sources on board a vessel. At low ship speeds, onboard machinery noise is dominant until propeller cavitation inception occurs, thus dominating the overall radiated noise spectrum. While it is not possible to avoid cavitation at service speed conditions for efficient commercial ships, various full-scale URN measurements have shown room for improvement. Among the same type of vessels, full-scale measurements have reported up to 20 dB difference in the measured noise levels. This may suggest that current practices in ship design can be improved scrutinised in terms of the URN characteristics and hence may lead to minimising the impact on ever-increasing ambient noise levels in the world’s oceans. In order to reduce the ambient noise levels in the oceans, IMO (International Maritime Organization) has published a non-mandatory guideline for commercial ships [3]. The scope of this guideline is to give general advice related to mitigation of underwater radiated noise to designers, ship operators and owners.

In order to provide a better understanding of underwater radiated noise measurements, a Joint Research Programme (JRP)10, named “Noise Measurements”, was established with the Hydro Testing Alliance Network of Excellence (HTA-NoE), completing its mission successfully in 2011 (AMT’11, 2011). The members of this group further decided to continue of the URN investigations under the activities of a new working group entitled “Noise Community of Practice (Noise Cop) of the Hydro Testing Forum (HTF). A round robin (RR) test programme was organized for prediction of open water cavitation and hydroacoustic performance of the Newcastle University’s (UNEW) Deep-V type catamaran research vessel, “The Princess Royal”. In the scope of the RR test programme, different facilities (i.e. University of Genoa UNIGE, University of Strathclyde, NMRI, SSPA, KRISO, CNR-INM and MARIN) developed techniques to model cavitation and noise prediction and presented their results by means of hydrodynamic and hydroacoustic performance for the Princess Royal Propeller.

URN can be predicted in some of the following ways, including experimental methods in cavitation tunnels and depressurised towing tanks, semi-empirical formulations and computational fluid dynamics (CFD) methods. With the development of computer technologies and power, CFD is becoming a more popular tool for this research field although it is still considered to be in its infancy and requires more validation, e.g. [4], [5]. Numerical predictions of propeller noise are not complicated by using hybrid methods such as RANS, Large Eddy Simulation (LES) and Detached Eddy Simulation (DES) coupled with the FW-II equation. Furthermore, such numerical acoustic simulations are not dependent on the hydrophone location which may be somehow limited in typical experimental configurations. Additionally, experimental pressure sensors may be affected adversely by the facility reverberations during the measurements. The resulting combined structural and acoustic signal can be tricky to separate clearly [6, 7].

In recent years, studies for the prediction of propeller radiated noise have been increasing rapidly with the development of the numerical methods and experimental facilities. In the study of Hallander et al., (2012), the authors examined the underwater radiated noise induced by a propeller located behind a hull wake[8]. RANS with FW-H equations were used as a hybrid method for near and far-fields. Various numerical tools were applied to predict the propeller radiated noise level and their differences were investigated in details. Lafeber et al., (2015) predicted cavitating propeller noise for three different cases (i.e. a cruise liner, a container vessel and a catamaran) using both numerical and experimental methods [9]. The experiments were conducted in MARIN’s Depressurized Wave Basin while tip vortex and sheet cavitation noise models were implemented for numerical computations.
The resulting numerical and experimental results were compared with the full-scale sea trial data. Aktas et al., (2016a) conducted systematic cavitation tunnel tests of the Princess Royal model propeller both in uniform and inclined flow conditions [10]. These experiments were conducted in the Emerson Cavitation Tunnel. The main purpose of the study was to examine the effects of inclined flow angle on propeller hydrodynamic and hydroacoustic performances. The results showed that propeller efficiency increased with the increase in the shaft inclination angle whereas propeller noise was not affected. Kowalczyk and Felicjancik (2016) studied hydroacoustic performance of a marine propeller in non-uniform flow conditions by conducting experiments and numerical calculations [11]. In this study, the flow field around the propeller blades was solved by RANS, while pressure fluctuations were calculated as a hydrodynamic pressure without using the FW-H acoustic analogy. The resulting pressure fluctuations were considered as acoustic pressures and noise levels were calculated. Cavity patterns and noise levels were compared with the experimental results. The results revealed that numerical and experimental results are in a quite good agreement with the experiments.

Sakamoto and Kamiirisai, (2018) investigated near-field cavitating propeller noise using a viscous CFD solver and Brown’s semi-empirical formulation [12]. In this study, four different propellers for existing ships were of the interest and cavitation was modelled by simulating the full-scale ship wake. Time series pressure fluctuations were captured to predict the propeller tonal noise levels, while cavitation area was calculated using a viscous solver and used as input for Brown’s method in order to predict the propeller broadband noise. The results of the study showed that Brown’s formulation gives promising results to detect the upper bound limit of the broadband noise in comparison to experimental results.

In a recent study Sezen and Kinaci, 2019, predicted model scale propeller hydroacoustic performance using RANS with the FW-H approach [13]. In this investigation the pressures, which were gathered directly from a hydrodynamic solver, and the acoustic analogy were compared in the near field under non-cavitating and cavitating conditions. The results showed that the RANS based predicted pressures should not be extrapolated to the far-field by using the International Towing Tank Conference (ITTC) distance normalization. The FW-H solution is needed to predict the near field noise under cavitating conditions. In another study of Cianferra et al. 2019, a hybrid methodology was presented to predict the propeller radiated noise in open water conditions for a benchmark propeller, SVA-VP1304 [14]. The hydrodynamic field was solved by LES with a grid about 6*10^6 cells, and hydroacoustic performance was predicted with the porous FW-H formulation. The effects of different noise terms (i.e. thickness, loading and quadrupole) were examined at different hydrophone locations. In a very recent study Lidtke et al. 2019, the hydroacoustic performance of the INSEAN E779A model propeller was presented for non-uniform flow conditions [15]. RANS equations were coupled with the porous FW-H approach. The effects of significant simulation parameters such as time step and grid resolution were investigated. The results showed that the best porous data surface should be selected carefully in order to avoid signal corruption due to the vortex penetrating with the surfaces.

In the study of Tani et al. (2019), a general view of the ongoing round robin test programme with the Princess Royal vessel propeller was presented [16]. Included in this study, were details of the different facilities, experimental setups and data analysis methods. In addition, comparative results of the cavitation patterns and noise characteristics of the test propeller are presented and discussed based on the measurement results from the different facilities. In the scope of their study, the results of uniform flow conditions were given, and further studies were recommended.

Despite the recent experimental and numerical studies in the field of URN, reviewed above, the numerical prediction methods for the propeller URN still require more validation studies to predict the noise levels accurately, especially in the presence of the cavitation. The main aim of this study, therefore, is to present the preliminary numerical results for the benchmark Princess Royal model propeller by using the porous FW-H formulation in combination with the RANS method in the low-
frequency region as well as validation of the numerical hydroacoustic results with the experiments under cavitating conditions.

In Section 2 the paper first presents details of the experimental set-up and test conditions of the benchmark propeller. The numerical models used in the CFD solver is given in Section 3. Verification and validation studies for the numerical solver are presented in Section 4, and this is followed by the presentation and discussion of the numerical results in Section 5. Finally, in Section 6, concluding remarks are given, including suggestions for future work.

2. Description of Experimental Set-up and Test Conditions

The cavitating flow experiments and noise measurements were performed at the Cavitation Tunnel of the University of Genoa. This tunnel is a Kempf & Remmers closed water circuit tunnel, schematically presented in Figure 1. The tunnel has a 0.57 x 0.57 m square test section, and the total length of the test section is 2 m. The maximum tunnel flow speed is 8.5 m/s in the test section, and the nozzle contradiction ratio is 4.6:1 [17] and [18].

![Figure 1: UNIGE Cavitation Tunnel [17].](image)

A Kempf & Remmers H39 dynamometer is used to measure the propeller thrust, torque and propeller rotational rate in the tunnel. Cavitation on the propeller blades is observed by using a mobile stroboscopic system. Additionally, three Allied Vision Tech Marlin F145B2 Firewire Cameras, with a resolution of 1392 x 1040 pixels and a frame rate up to 10 fps, are used for the cavitation observation. During the measurements, the quality of the water in the tunnel was checked regularly following the ITTC guidelines.

The URN measurements of the model propellers were conducted using three hydrophones (i.e. Reson TC4014 and two Brul Kjaer type 8103) at different locations. Brul & Kjaer Type 2635 charge amplifiers were connected to hydrophones. The location of the hydrophones is shown in Figure 2.
Figure 2 shows hydrophone (H1) is placed in an acoustic chamber consisting of a small tank made of PMMA (Plexiglas), positioned on the observation window below the propeller and filled with water. The remaining hydrophones (H2 and H3) are placed inside the test section both downstream and outside the propeller slipstream. In the noise measurements, 2¹ samples records are acquired with a 200 kHz sampling frequency. The propeller operational conditions are identified according to thrust coefficient and cavitation number identity. In the measurements, by taking the limits of cavitating tunnel, propeller rotational rate is selected as high as maximum to set to maximum Reynolds number. To obtain the required thrust coefficient, flow speed is adjusted in the cavitation tunnel in accordance with the given propeller rotational rate.

The cavitation number is generally derived with respect to propeller rotational rate by using Equation 1:

$$\sigma_N = \frac{P_0 - P_V}{\frac{1}{2} \rho (nD)^2}$$

where $P_0$ is the static pressure, $P_0 = P_{atm} + \rho gh$, (Pa); $\rho$ is the density of water, (kg/m³); $g$ is the acceleration of gravity, (m/s²); $h$ is the propeller shaft immersion, (m); $P_V$ is the vapour pressure, (Pa); $n$ is the propeller rotational rate, (rps) and $D$ is the propeller diameter, (m). Propeller thrust ($K_T$), torque ($K_Q$) coefficients and efficiency ($\eta_0$) values are calculated for different advance coefficients. The advance ratio is defined in Equation 2:

$$J = \frac{V_\Lambda}{nD}$$

Thrust, torque coefficients and efficiency value are calculated as follows, respectively,

$$K_T = \frac{T}{\rho n^2 D^4}$$

$$K_Q = \frac{Q}{\rho n^2 D^5}$$

$$\eta_0 = \frac{J K_T}{2\pi K_Q}$$

where $T$ is the propeller thrust, (N); $Q$ is the propeller torque, (Nm); and $V_\Lambda$ is the propeller advance speed (this is ground speed or incoming flow at infinity upstream) (m/s). Reynolds number can be calculated for the marine propeller as follows:

$$Re = \frac{nD^2}{v}$$
where \( v \) is the kinematic viscosity, \((\text{m}^2/\text{s})\).

Within the scope of the round-robin test campaign, the model propeller of The Princess Royal was selected in order to predict the propeller underwater radiated noise, since information was available for the full-scale ship together with sea trials. Figure 3 and Table 1 show the geometry and its main particulars.

![Figure 3: Model scale of Princess Royal propeller.](image)

**Table 1: Model propeller main characteristics.**

<table>
<thead>
<tr>
<th>Characteristic</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Propeller Diameter (m)</td>
<td>0.22</td>
</tr>
<tr>
<td>Scale ratio (( \lambda ))</td>
<td>3.41</td>
</tr>
<tr>
<td>Number of Blades, ( Z )</td>
<td>5</td>
</tr>
<tr>
<td>P/D at 0.7R</td>
<td>0.8475</td>
</tr>
<tr>
<td>Expanded Blade Area Ratio</td>
<td>1.057</td>
</tr>
<tr>
<td>Hub to Propeller Diameter Ratio ( (D_h/D) )</td>
<td>0.18</td>
</tr>
<tr>
<td>Rake Angle (( ^\circ ))</td>
<td>0</td>
</tr>
<tr>
<td>Skew Angle (( ^\circ ))</td>
<td>19</td>
</tr>
<tr>
<td>Propeller Type</td>
<td>Fixed Pitch Propeller</td>
</tr>
<tr>
<td>Blade Loading Distribution</td>
<td>NACA ( a=0.8 )</td>
</tr>
<tr>
<td>Thickness Distribution</td>
<td>NACA 66 modified</td>
</tr>
</tbody>
</table>

During the round-robin test campaign, the propeller URN were predicted for both uniform and inclined flow conditions. However, in the scope of this study, experimental and numerical results are presented only for uniform flow conditions. The test matrix for uniform flow is listed in Table 2.

**Table 2: Operational conditions used in the experiments and CFD analysis.**

<table>
<thead>
<tr>
<th>Loading Condition</th>
<th>( J )</th>
<th>( \sigma_N )</th>
<th>( n(\text{rps}) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td>0.4</td>
<td>2.22</td>
<td></td>
</tr>
<tr>
<td>C2</td>
<td>0.4</td>
<td>1.30</td>
<td></td>
</tr>
<tr>
<td>C3</td>
<td>0.4</td>
<td>0.72</td>
<td></td>
</tr>
<tr>
<td>C4</td>
<td>0.5</td>
<td>3.48</td>
<td></td>
</tr>
<tr>
<td>C5</td>
<td>0.5</td>
<td>2.03</td>
<td>35</td>
</tr>
<tr>
<td>C6</td>
<td>0.5</td>
<td>1.13</td>
<td></td>
</tr>
</tbody>
</table>
3. CFD Methodology

In this section, details are given of the numerical models, computational domain & grid structure used to solve flow around non-cavitating and cavitating propeller.

3.1 Computational Domain & Mesh Generation

Figure 4 provides a general overview of the computational domain used in the hydrodynamic and hydroacoustic simulations. A cylindrical computational domain comprises the three regions, for noise calculations, namely static, rotating and porous surface regions. The upstream and downstream extents of the domain were set 4.5D and 10.5D, respectively, with the domain radius of 4D. This means that computation domain dimensions were extended with respect to the experimental set-up. Propeller blades and hub were defined as no-slip wall to satisfy the kinematic boundary condition.

![Figure 4. Computational domain and boundary conditions.](image)

The Porous Data Surface (PDS) was situated outside of the interface between the static and rotating region. The optimum location and size of the PDS, which encompasses all the non-linear effects, should be selected carefully to minimise any spurious effects and avoid signal corruption [15]. A view of the PDS and dimensions are given in Figure 5.
An unstructured grid structure was used in Figure 6 to discretise the computational domain in compliance with the Finite Volume Method (FVM). Additional volumetric control regions were applied around the propeller blades and in the near field to capture the flow field precisely. In the hydroacoustic simulations, the grid structure around the propeller blades and in the near field region was changed in comparison to the hydrodynamic simulations since acoustic pressures are more sensitive to grid refinement than hydrodynamic inputs which are obtained from the hydrodynamic solver [6].

3.2 Hydrodynamic Model

The governing flow equations were solved using the Star-CCM+ commercial CFD code that has been widely used to predict propeller hydrodynamic and hydroacoustic performance [29]. In this code, a segregated flow model was adopted to solve the steady and unsteady RANS equations with the SST k-ω turbulence model. Besides, momentum equations were discretised using the first-order scheme, except convection terms which were treated using a second-order upwind scheme as well as time discretisation.

Propeller rotational motion was simulated by Rigid Body Motion (RBM) and Moving Reference Frame (MRF) techniques. In the first step, MRF was adopted based on the steady solution of the governing equations. Although more realistic results can be obtained from the unsteady solution, due to the cost-savings, the propeller open water simulations can be generally conducted based on the
steady solution. However, the RBM technique is preferred for unsteady cases such as cavitation, noise and so on, since it is considered to be the most accurate way to obtain the change of flow variables with time. At each time step, the governing equations are solved with a given grid position. In this study, the MRF technique was performed to compute non-cavitating propeller hydrodynamic performance, whereas the RBM was applied for the propeller URN simulations under cavitating conditions.

Cavitation on the propeller blades was modelled with the well-known Schnerr-Sauer cavitation model by using Volume of Fluid (VOF) approach to elucidate the phase changing of liquid into vapour. This model is based on the reduced Rayleigh-Plesset equation and neglects the impact of bubble growth acceleration, viscous effects, and surface tension. However, the cavitation model is based on the homogenous distribution of bubble seeds throughout the liquid. In this model, the bubble growth rate is predicted using Equation 7;

\[
\left(\frac{dR}{dt}\right) = \frac{2}{3} \frac{p_{\text{sat}} - p_w}{\rho}
\]

(7)

where, \( p_w \) is the pressure of the liquid, (Pa); \( p_{\text{sat}} \) is the saturation pressure, (Pa).

3.3. Hydroacoustic Model

Hydroacoustic simulations were carried out using the Ffowcs Williams-Hawkins (FWH) acoustic analogy based on Lighthill's equation [19]. It is the re-arrangement form of the continuity and momentum equations into nonhomogeneous wave equations [20]. The generalised FW-H equation is given in terms of generalised function in Equation 8;

\[
D^2 p(x, t) = \frac{\partial}{\partial t} \{[\rho_0 v_n + \rho (u_n - v_n) \delta(f)] \} - \frac{\partial}{\partial x_i} \{[\Delta P_{ij} \delta_{ij} + \rho u_i (u_n - v_n)] \delta(f)\}
\]

\[
+ \frac{\partial}{\partial x_i} \frac{\partial}{\partial x_j} \{T_{ij} H(f)\}
\]

(8)

Here, \( p \) is the acoustic pressure disturbance, Pa; \( D^2 \) is the D'Alembert operator, \( T_{ij} \) is Lighthill stress tensor, \( N \); \( u_i \) and \( v_i \) is the fluid and surface velocity components, m/s respectively. \( \bar{\rho} = \rho - \rho_0 \) is the density perturbation, kg/m³ with respect to the undisturbed medium and \( n \) indicates the projection along the outward normal to the surface. The first term on the right-hand side of Equation 8 represents the thickness (monopole) contribution, whereas the second term on the right-hand side is loading (dipole) contribution. The latter part on the right-hand side of the equation is the quadrupole (nonlinear) noise term. The first two terms are evaluated by solving the surface integrals, and the latter one is evaluated using volume integrals. D’Alembert operator and \( T_{ij} \) Lighthill Stress tensor can be written as follows;

\[
D^2 = \frac{1}{c_0^2} \frac{\partial^2}{\partial t^2} - \nabla^2
\]

(9)

\[
T_{ij} = \rho u_i u_j + P_{ij} - c_0^2 \bar{\rho} \delta_{ij}
\]

(10)

c_0 \text{ is the sound speed, m/s; } P_{ij} \text{ represents the compressive stress tensor, } N \text{ and } \Delta P_{ij} = P_{ij} - \rho_0 \delta_{ij}. \text{ (} \delta_{ij} \text{ is the Kronecker operator). According to Williams and Hawkins’ approach, the noise surface to define the sound source should be impermeable, and so they had generally considered an impermeable surface to be used. By doing this, the surface terms are comprised of linear components, namely thickness and loading. This form of the FWH is known as Farassat 1A which is proposed as a solution of the FW-H equation using Green Functions by [21]. This form of the FW-H equation is widely used especially in the aeroacoustic community. For maritime problems, the Farassat 1A solution loses its importance, and it becomes less desirable because of the unsteady cavitation dynamics and effects of non-linear terms. Therefore, the “porous FW-H” formulation which was derived by Di Franciscantonio [22] and performed by Ffowcs Williams and Hawkins in their original study has}
become the most widely used formulation. In this formulation, the propeller is surrounded by an additional surface which is called the porous data surface (PDS).

It should be noted that the linear terms of the FW-H equation lose their physical meaning and thus, the non-linear terms, which are located within the PDS region are accounted for solving thickness and loading noise terms [23][24]. Since PDS encompasses all off the noise sources, non-linear effects are generally ignored outside of the PDS region. The possible numerical solution of the FW-H (Equation 8) is obtained by defining the two acoustic variables (Equation 11 and 12) under the porous FW-H hypothesis and incompressible flow assumption.

\[ U_i = u_i \]

\[ L_i = P_{ij} \hat{n}_j + \rho u_i (u_n - v_n) \]

where \( u \) and \( v \) are the fluid and porous surface velocities. In the assumptions of neglecting of density fluctuations \( \rho = \rho_0 \) and that there is a stationary data surface \( (v_n = 0) \), the porous FW-H equation becomes,

\[ 4\pi p(x, t) = \int_S \frac{\rho_0 \hat{U}_n}{r} \, dS + \int_s \left[ \frac{L_r}{c_0 r} \right]_r \, dS \int_s \left[ \frac{L_r}{r^2} \right]_r \, dS \]

(13)

Here, \( r \) is the radiation direction, and dot defines a source time derivative with respect to retarded time. Subscripts \( r \) and \( n \) define the dot product of a quantity with a unit vector either radiation or normal directions, respectively.

4 Verification and Validation of Hydrodynamic Model

The verification study was conducted for non-cavitating open water condition using the Grid Convergence Index (GCI) method based on Richardson, (1911) for extrapolation using three different grids (fine, medium and coarse) [25]. The uncertainty study was only applied for grid-spacing with respect to the defined methodology proposed by [26]. It was conducted for both thrust coefficient \( (K_T) \) and torque coefficient \( (10K_Q) \) at \( J=0.4 \) using the refinement factor \( r=\sqrt{2} \) since this should be greater than 1.3. For the sake of completeness, this procedure is summarized;

The difference between the three different grids solutions should be determined by Equation 14.

\[ \varepsilon_{21} = \varphi_2 - \varphi_1, \quad \varepsilon_{32} = \varphi_3 - \varphi_2, \]

(14)

Here, \( \varphi_1 \), \( \varphi_2 \) and \( \varphi_3 \) are fine, medium and coarse mesh grid solutions, respectively. The ratio of solution scalars is used to calculate the convergence condition by Equation 15

\[ R = \frac{\varepsilon_{21}}{\varepsilon_{32}} \]

(15)

The solution type is identified in accordance with the value of \( R \). Four different types of solution are available [27]. If the convergence condition \( (R) \) is found between 0<\( R <1 \), the procedure is implemented according to methodology [26]. Apparent order \( (p) \) of the method is calculated by Equation 16.

\[ p = \frac{\ln \left( \frac{\varepsilon_{32}}{\varepsilon_{21}} + q \right)}{\ln(r_{21})} \]

(16)

\[ q = \ln \left( \frac{r_{21}}{r_{32} - s} \right) \]

(17)

Here, \( r_{21} \) and \( r_{32} \) is the refinement factors which are calculated by \( r_{21} = 3 \sqrt{\frac{N_1}{N_2}} \). (\( N \) is the cell number).
The extrapolated value is:

$$s = sgn \left( \frac{\varepsilon_{32}}{\varepsilon_{21}} \right)$$  \hspace{1cm} (18)

The approximate relative error and extrapolated relative error are:

$$e_a^{21} = \frac{\varepsilon_2 - \varepsilon_1}{\varepsilon_1} \quad e_{\text{ext}}^{21} = \frac{|\varepsilon_{\text{ext}}^{12} - \varepsilon_1|}{\varepsilon_{\text{ext}}^{12}}$$  \hspace{1cm} (19)

Ultimately, the GCI index is calculated by:

$$GCI_{\text{fine}}^{21} = \frac{1.25 e_a^{21}}{r_{21}^p - 1}$$  \hspace{1cm} (20)

Table 3 gives element counts for the fine, medium and coarse grids, together with solution results and uncertainty values.

<p>| Numerical Uncertainty for $K_T$ and $10K_Q$ at $J=0.4$ |
|----------------------------------------|--------|--------|--------|--------|</p>
<table>
<thead>
<tr>
<th>Number of Cells</th>
<th>$\varphi_1$</th>
<th>$\varphi_2$</th>
<th>$\varphi_3$</th>
<th>$R$</th>
<th>$%GCI_{\text{FINE}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$K_T$</td>
<td>2.9M</td>
<td>1.1M</td>
<td>0.49M</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>$10K_Q$</td>
<td>0.2231</td>
<td>0.2227</td>
<td>0.2221</td>
<td>0.66</td>
<td>0.379</td>
</tr>
</tbody>
</table>

Based on the result of the verification study, the fine mesh was selected for the non-cavitating open water propeller simulations. Numerical and experimental results for different advance coefficients are given in Figure 7. In the hydrodynamic simulations, propeller rotational rate was constant at $n=20$ rps in compliance with the experiment. The same grid structure was not adopted for acoustic simulations since the FW-H acoustic analogy is more sensitive to grid refinement than inputs derived by the hydrodynamic solver. To avoid the numerical dissipation and spurious noise effects, the mesh around the propeller including receiver locations must resolve the acoustic waves. At least 20 cells per wavelength should be used to solve the shortest wavelengths in the simulations (29). For the FW-H analogy, pressures are extrapolated to the far-field using the near-field solution. Therefore, solution of the near field requires a finer grid structure in the vicinity of the propeller blades, while the open water hydrodynamic characteristics can be calculated with a smaller element count with respect to cavitation and acoustic simulations. In the noise simulations, a grid of about 6.8M cells was used to solve the near field accurately. Note that the uncertainty analysis was not performed for the acoustic simulations.

![Figure 7. Comparison of thrust and torque coefficients.](image-url)
5. Results and Discussions

Results and discussions for the cavitation observations, noise predictions and as well as the propeller thrust coefficients are presented for the six different loading conditions, (C1 to C6) as shown in Table 2.

5.1 Cavitation Observation

Figure 8 and Figure 9 show the predicted and experimentally observed cavitation patterns on the propeller blades for J=0.5 and J=0.4, respectively. In the round robin (RR) test campaign, sheet cavitation was observed slightly less extended towards inner radii than other facilities namely, SSPA, MARIN and ECT, especially for the condition C2 and C3. This difference is considered due to the development of the boundary layer. During the experiments in UNIGE cavitation tunnel, a slightly lower propeller shaft rate (i.e. low Reynolds number) was used and the portion of laminar boundary layer could be present at inner radii. On the other hand, numerical calculation is conducted in fully turbulent flow conditions, hence the slight overestimation of sheet cavitation in CFD computations may be partially because of this.

As can be seen in these figures, the numerical results show a similar sheet cavitation pattern whilst tip vortex cavitation could not be observed as a grid refinement technique, such as MARCS [30] was not implemented at the propeller blade tips. Tip vortex cavitation in the slipstream can be observed accurately using different adaptive mesh refinement techniques and scaling resolving simulations such as LES and DES, e.g. in [29, 30]. However, the RANS approach is still considered to be a reliable model to observe the sheet cavitation on the propeller blades, e.g. [28]. In the scope of this study, the main aim was to predict the propeller underwater radiated noise in the low-frequency region due to the limitation of the RANS approach for broadband noise prediction. However, the effects of tip vortex cavitation on propeller radiated noise might be stronger for the hydrophones located in the downstream region [6]. Ultimately, observation of tip vortex cavitation, which is the main contributor to broadband noise, especially in the presence of such off-blade dynamic phenomena or bubble collapse, was not considered this study.
Figure 8. Comparison of cavitation observation for three different conditions at $J=0.4$. 

C1, $J=0.4$, $\sigma_N=2.22$ (Left: Experiment, Right: RANS)

C2, $J=0.4$, $\sigma_N=1.3$ (Left: Experiment, Right: RANS)

C3, $J=0.4$, $\sigma_N=0.72$ (Left: Experiment, Right: RANS)
C4, $J=0.5$, $\sigma_N=3.48$ (Left; Experiment, Right; RANS)

C5, $J=0.5$, $\sigma_N=2.03$ (Left; Experiment, Right; RANS)

C6, $J=0.5$, $\sigma_N=1.13$ (Left; Experiment, Right; RANS)

*Figure 9. Comparison of cavitation observation for three different conditions at $J=0.5$*

Additionally, numerical hydrodynamic thrust coefficients ($K_T$) for different loading conditions were compared with the experiments. Table 4 shows the difference in thrust coefficients between the experimental measurements and numerical calculations.

*Table 4: Comparison of thrust coefficient ($K_T$) for different loading conditions.*

<table>
<thead>
<tr>
<th></th>
<th>$K_T$ (EFD)</th>
<th>$K_T$ (CFD)</th>
<th>Difference (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td>0.244</td>
<td>0.233</td>
<td>-4.5</td>
</tr>
<tr>
<td>C2</td>
<td>0.244</td>
<td>0.239</td>
<td>-2.0</td>
</tr>
<tr>
<td>C3</td>
<td>0.226</td>
<td>0.233</td>
<td>+3.0</td>
</tr>
<tr>
<td>C4</td>
<td>0.190</td>
<td>0.180</td>
<td>-5.3</td>
</tr>
<tr>
<td>C5</td>
<td>0.189</td>
<td>0.183</td>
<td>-3.2</td>
</tr>
<tr>
<td>C6</td>
<td>0.191</td>
<td>0.193</td>
<td>+1.0</td>
</tr>
</tbody>
</table>
5.2 Propeller URN Results

In this section, the results of the cavitation tunnel measurements and numerical prediction for the URN are presented and discussed. In the cavitation tunnel measurements, the corrections, which are explained ANSI/ASA S12.64-2009/Part1, were implemented to determine the net cavitation noise.

Background noise measurements were conducted with a dummy hub model, operating in the same condition as for the propeller. If the difference between the background noise and the propeller noise is smaller than 3 dB, the result is discarded. On the other hand, if the difference is between 3 dB and 10 dB, the correction is applied with respect to Equation 22. When the difference is more than 10 dB, no correction is included in the calculations [17] and [18].

$$SPL_N = 10 \log \left[ 10^{\frac{SPL_T}{10}} - 10^{\frac{SPL_B}{10}} \right]$$  \hspace{1cm} (22)

where, subscripts N, T and B show net, total and background noise, respectively. As the computed or measured noise levels are dependent on the distance between the noise source and the hydrophone, the distance normalisation is usually implemented as given in Equation 23,

$$SPL = SPL_p + 20\log\left(\frac{d}{d_{ref}}\right)$$  \hspace{1cm} (23)

where $SPL_p$ is the measured noise level at the specified distance, dB; $d$ is the distance between the noise source and hydrophone, m; $d_{ref}$ is the reference distance which is equal to 1m. The reference distance between the propeller and the hydrophone was kept at H1 (see Figure 2) for the numerical simulations. Having obtained the results at the HP1 location, numerical results were extrapolated to 1m using the ITTC distance normalisation equation. It should be noted that, the experimental results were presented as the averaged of spectra of three hydrophones. With the help of the transfer functions, the experimental results were extrapolated to 1m distance.

In the numerical URN, in the low-frequency region, predictions were conducted using a hybrid method in which the RANS was equipped with the porous FW-H equation. In the hydrodynamic simulations, the pressure fluctuation induced by the propeller rotation, velocity and velocity derivatives are taken as inputs to the hydroacoustic FW-H solver. Thus, FW-H carries the pressures from near field to far-field. Figure 10 and Figure 11 compare the sound pressure levels measured in the cavitation tunnel with the numerical predictions in the low-frequency region. It should be borne in mind that the RANS with FW-H analogy can be used as a suitable tool for prediction of the low-frequency region, especially to capture the blade harmonics while the RANS approach suffers from accurately capturing the unsteady behaviour of cavitation. Therefore, the broadband noise spectrum, which is strongly affected by the turbulence, can be predicted more precisely using more advanced turbulence modelling such as LES and DES with the FW-H approach to reduce the disadvantage of the RANS approach. Nonetheless, the precise modelling including the wake effect should be further investigated.

Figures 10 and 11 compare the numerical and experimental results up to 1 kHz for three different loading conditions. Rather similar trends were observed even if there is a considerable difference for the higher frequency bands. Moreover, 1st Blade Passage Frequency (BPF) values (i.e. 35*5=175 Hz) were well captured for all conditions in the numerical simulations, however the amplitude of this noise component seems significantly overpredicted, except for condition C6. It must be noted that the noise at blade rate measured in the experiments may be influenced by the complex acoustical coupling between the propeller and the tunnel test section, which is not taken into account in the current numerical simulation. In particular, in the experiments, there is an anomalous peak around 800 Hz which might be induced by the background noise.
Figure 10. Comparison of experimental and numerical results at J=0.4 for three different loading conditions.

Figure 11. Comparison of experimental and numerical results at J=0.5 for three different loading conditions.

Figure 10 represents the noise levels characterised by the higher loading conditions; a difference in mid-range levels was observed for C2 condition, although blade harmonics were captured in the numerical calculations. It might be since strong tip vortex cavitation was observed during the experiments. The numerical results for C1 and C3 condition show rather good agreement with the experimental results. Similar comments can also be made for other loading conditions in Figure 11. The noise levels measured in the experiments and those predicted by the numerical simulations also indicate a similar trends for C4 and C6 conditions whereas the difference in this trend can be observed in C5 with respect to other test conditions.

Finally, considering the results from the numerical simulations, lower noise levels were observed, especially for the C2 condition. However, the characteristic of the spectrum is in good agreement with the experiment. It is worth mentioning that, in the numerical simulations, unexpected pressure oscillations can also be observed due to several issues such as grid structure on the PSD, time step, the size of the PSD and so on. Additionally, even though the contribution of tip vortex cavitation is considerable for the continuous part of the noise spectrum (broadband), tip vortex cavitation may have also crucial role for the low-frequency range. Therefore, the difference between numerical results and experiment might be originated from the poor modelling of tip vortex cavitation as well as turbulence. Note that the porous FW-H approach is required to avoid the noise decay, especially in the far field, as well as including non-linearities into the calculations. On the other hand, the main disadvantage of the porous FW-H is that these surfaces can suffer from the hydrodynamic perturbations (i.e. vortices, shear layers) crossing through the surface and corrupting the signal. Acoustic pressure corruption arises as a result of vortices convected through the permeable surfaces. Thus, it is considered that selection of an optimal permeable surface is more of an art than a science.
6. Concluding Remarks

This study presents the initial results of URN predictions for The Princess Royal model propeller in the uniform flow conditions, and are compared with experimental results from cavitation tunnel tests. In the numerical predictions, first, the hydrodynamic performance of the benchmark model propeller was calculated, and cavitation on the propeller blades was simulated. A verification study was conducted using the GCI technique both for the thrust and torque coefficients for the non-cavitating case. The predicted hydrodynamic characteristics and cavitation pattern on the propeller blades were validated against the experimental results. Following this, the propeller URN was predicted using a hybrid methodology in the low-frequency bands for six different loading conditions. The results of the noise simulations give a rather acceptable prediction with respect to the experimental data in terms of the BPF values and shape of the spectrum. Further work is planned, as follows;

- The simulations for broadband noise prediction will be performed using scale resolving methods such as LES and DES for broadband noise prediction.
- An adaptive mesh refinement technique (MARCS) [30] will be implemented on the propeller blade tips to investigate the effects of tip vortex cavitation on the underwater propeller radiated noise.
- Different porous data surfaces will be tested and the optimum surface will be proposed in order to avoid signal corruption.
- These improvements will be used to investigate model, and full-scale propeller underwater radiated noise under non-uniform flow conditions and in the presence of the hull and rudder.

7. Acknowledgements

The first Author is sponsored by the Stone Marine Propulsion Ltd of UK and the University of Strathclyde during his PhD study. Also, the granted access to the High-Performance Computing for the West of Scotland (Archie-West) is gratefully acknowledged.

References

3. IMO, (2014), M. R. “Guidelines for the reduction of underwater noise from commercial shipping to address adverse impacts on marine life.


