Suppression of Tip Vortex Cavitation noise using PressurePores™ technology: A numerical and experimental investigation

B. Aktas¹, N. Yilmaz², M. Atlar², N.Sasaki³, P. Fitzsimmons², D. Taylor³

¹Fugro Structural Monitoring, Glasgow, UK
²University of Strathclyde, Glasgow, UK
³OSCAR Propulsion, West Sussex, UK

ABSTRACT
This study aims to demonstrate the merits of PressurePores™ technology as a retrofit on marine propellers to mitigate tip vortex cavitation noise for a quieter propeller. Shipping noise originates from various sources on board a vessel, amongst which the propeller cavitation is considered to dominate the overall radiated noise spectrum above the inception threshold. Thus, by strategically introducing PressurePores™ technology to the propeller blades, that may be producing limited thrust for the operating vessel due to the presence of cavitation, a reduction in the overall cavitation volume can be achieved which would consequently result in a reduction of the radiated noise levels while retaining efficiency as much as possible or with the least compromise. The strategic implementation of the holes was mainly aimed to reduce the tip vortex cavitation which is a major contributor to the underwater noise emissions of a ship.

In this paper, the details and results of a unique combination of the numerical and experimental investigation are presented to shed light and further develop this URN mitigation concept. An overall finding from conducted research indicated that significant reduction from cavitation noise could be achieved (up to 17 dB) at design speed with the most favourable strategic arrangement of the pressure pores. Such reduction was particularly important in the frequency regions that are utmost important for marine fauna while the propeller is losing only 2% of its efficiency.

Keywords
PressurePores™; Pressure Relief Holes; Underwater Radiated Noise (URN); Cavitation Noise Mitigation; Experimental Hydrodynamics; Computational Fluid Dynamics (CFD).

1 INTRODUCTION
The technological developments of the last half-century have revolutionised the world that we live in at the moment. One of the main driving factors for such swift advancement is the globalisation of the world. Commercial shipping has made globalisation possible by providing the most efficient means of transportation. With the ever-increasing world population, the volume of commercial shipping has been experiencing an increasing trend over the last five decades. Unfortunately, this has also resulted in the elevation of emissions produced by the maritime industry IMO, (2013).

One of the most adverse by-products of the commercial shipping has been underwater radiated noise (URN) emission Ross, (1976). The extraordinary expansion of the world fleet has resulted in increased levels of the ambient noise in the world’s seas, especially in the low-frequency domain Frisk, (2012). Unfortunately, this domain is also utilised by marine mammals for various fundamental living activities. Thus, exposing them to such an abrupt change in ambient noise levels may disorient them or disrupt their communication signals, leading to behavioural changes of these mammals or local extinction Richardson et al., (2013), White & Pace, (2010).

Within the framework described above, the recently conducted PressurePores™ Technology development project (Patent Application Number PCT/GB2016/051129) aimed to explore the merits of implementing pressure relieving holes (PressurePores) on marine propellers to mitigate the cavitation induced noise for a more silent propeller. This paper presents a review and results of the experimental and computational investigation conducted within this project.

To achieve the aim of the project, first, a literature review was conducted in the related field. This revealed that in the late 90s, Sharma et al. from the Indian Institute of Technology in Bombay conducted research involving cavitation noise on marine propellers Sharma et al., (1990).

In that research study, Sharma et al. tried to delay the onset of the tip vortex cavitation and to reduce the produced noise without influencing the propeller performance adversely. Based on this rationale, they modified propellers by drilling 300 holes of 0.3mm diameter in each blade. These holes were drilled at the tip and the leading edge areas of the blades. Sharma et al’s tests indicated that the dominant cavitation type at inception was the tip vortex cavitation under any testing conditions. The modifications did not demonstrate any measurable influence on the performance characteristics of any of the propellers tested.
But, as it was expected, it had a great influence on the Tip Vortex Cavitation (TVC).

Regarding the acoustic benefit, there was a great improvement by the complete attenuation of the low-frequency spectral peaks, e.g. as shown in Figure 1. The tests with the original (unmodified) propellers showed a consistent rise of spectrum levels throughout the frequency range, as the advance coefficients were reduced, but this was not the case for the modified propellers. The advance coefficients had a weak effect on the noise levels. This was attributed to the consequences of the modification where the tips were unloaded, and the suction peak in the leading edge was reduced while the TVC strength was reduced due to the increase in the angle of incidence.

![Figure 1. Influence of blade modification on cavitation noise for J=0.38. Sharma et al., (1990).](image)

Figure 1 presents a comparison of the noise characteristics for the original and the modified propellers A and B at the advance coefficient of J=0.38. In such a low J value, the improvement was more significant. Particularly for low frequencies, between 1 and 2 kHz, a reduction of about 15 dB was observed in the noise levels of both propellers. In conclusion, “the modifications carried out had no measurable influence on the performance characteristics of the basic propellers”. However, they achieved a delay in the onset of the cavitation and significant noise reductions. One interesting point to note in Sharma et al.’s work was that they tested all propellers in uniform flow conditions. This inherently disregards the presence of the ship hull in front of the propeller which is one of the most significant contributors to the cavitation and hence induced radiated noise.

To shed further light on this concept, and explore the effect of hull wake, an independent pilot experimental study was conducted in the Emerson Cavitation Tunnel of Newcastle University as part of an MSc study Xydis, (2015) by following Sharma et al., (1990). This pilot study was conducted using rather heuristic hole arrangements and limited test cases without any numerical optimisation of these arrangements. While the study demonstrated some encouraging signs of the radiated noise reduction, the level of the reduction in cavitation extent to support this mitigation needed more sophisticated and detailed observations. Inspired from this MSc study, and based on the model propellers tested in the study, a comprehensive Computational Fluid Dynamics (CFD) based investigation was conducted by Aktas et al., (2018) to demonstrate the effectiveness of this mitigation method, which is later patented as the PressurePores™ Technology by the sponsoring company. Based on the outcomes of this investigation, the best performing cases with the strategically selected PressurePores™ were chosen to be tested at a towing tank for the efficiency measurements while at a cavitation tunnel for the cavitation and noise measurements to confirm the results of the CFD investigations. The results of the towing tank and cavitation tunnel test confirmed the findings of the high fidelity numerical simulation for the propeller efficiency and cavitation observations as well as confirming the significant reduction in the emitted cavitation noise levels (up to 17 dB). The reductions from the noise spectra are also found to be prominent in the frequency regions that can be important for some marine mammals while the propeller was loosing about 2% of its efficiency.

The details of the numerical and experimental investigation summarised above is presented in this paper by the following layout; after this introductory section, Section 2 presents the description of the propeller model used as well as the experimental set-up and test conditions for the CFD simulations and cavitation tunnel tests which were conducted in the University of Genova Cavitation Tunnel. Section 3 describes the details and results of the cavitation tunnel test observations while Section 4 presents the computational fluid dynamics modelling of the cavitating propeller. In Section 5 the details and results of the propeller performance tests, which were conducted in the CTO towing tank of Gdansk, and finally Section 6 presents the main conclusions obtained from the investigations.

### 2 Propeller Model, Experimental Set-up and Test Conditions

The experimental approach adopted in this study necessitated the use of several experimental artefacts. These included a propeller model that had two modified versions incorporating the PressurePores™ technology, a cavitation tunnel and a towing tank described in the following. The adopted experimental test matrix is also provided.

#### 2.1 Propeller Model

The propeller model used for both tests represented the port side propeller of the Newcastle University’s research catamaran, The Princess Royal with a scale ratio of 3.41, giving a 220mm model propeller diameter. The reason was selecting this propeller as the test case two folds: firstly this vessel has become almost benchmark vessel worldwide for the URN and cavitation investigations; secondly, the Authors had extensive information and access to this vessel which can be used for validating the investigation in full-scale as part of the PressurePore™ technology project.

The propeller model was manufactured with high accuracy by considering the cavitation testing as shown by the
deviation contour plot given in Figure 2. The principal dimensions of the full-scale propeller are given in Table 1.

Table 1: Propeller main characteristics and particulars

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Full Scale Diameter [m]</td>
<td>0.75</td>
</tr>
<tr>
<td>Pitch Ratio at 0.7R</td>
<td>0.8475</td>
</tr>
<tr>
<td>Expanded Blade Area Ratio</td>
<td>1.057</td>
</tr>
<tr>
<td>Number of blades</td>
<td>5</td>
</tr>
<tr>
<td>Rake angle</td>
<td>0°</td>
</tr>
<tr>
<td>Skew angle</td>
<td>0°</td>
</tr>
</tbody>
</table>

The application of the PressurePore™ technology to this benchmark test propeller utilised the knowledge and experience gained through the CFD investigations conducted with this propeller, as reported in Section 4, as well as another test case propeller, which belonged to a 95000 tonnes merchant tanker, and tested in the Emerson Cavitation Tunnel by Xydis, (2015). In these investigations, the pore configurations applied were simulated initially by using the earlier version of the TVC model developed, and later by using the state of the art adaptive mesh refinement technique of Yilmaz et al., (2017). Based on these investigations, two sets of PressurePores™ configurations were selected to be tested at the CTO towing tank for accurate prediction of the propeller open water performance parameters and in the University of Genoa cavitation tunnel for the cavitation observation and underwater noise measurements. The selected PressurePore™ configurations are shown in Figure 3 and 4, as applied on the model propeller, and represented by the following legend: “Modified Propeller” and “Modified Propeller-2”, respectively. The diameter of the pores of 1mm, and 33 pores were used for the Modified propeller while 17 pores for the Modified-2 propeller model.

2.2 Experimental testing facilities

As stated earlier, two testing facilities were used for the experimental investigations. These were the medium-size cavitation Tunnel of the University of Genoa (UNIGE) and the large towing tank of the Centrum Techniki Okrętowej S.A. (CTO) Model Basin in Gdansk.

The cavitation tunnel of the UNIGE is a Kempf & Remmers closed water circuit tunnel, schematically represented in Figure 5. The tunnel has a square testing section of 0.57m×0.57m, having a total testing section length of 2m. The nozzle contraction ratio is 4.6:1, allowing to achieve the maximum tunnel flow speed in the test section of 8.5 m/s. The tunnel is equipped with a Kempf & Remmers H39 dynamometer, which measures the propeller thrust, the torque and the rate of revolution.
The CTO towing tank is approx. 270 m x 12 m x 6 m in length, breadth and depth, respectively and is fitted out with a towing carriage of a maximum speed of 12 m/s. The performance of the propeller model before and after the application of the PressurePores™ technology was measured by using the standard open water dynamometer as shown in Figure 6.

![Figure 6: Towing tank open water test set-up](image)

### 2.3 Test set-up and test matrix

The test set-up used for the cavitation tunnel tests is shown in Figure 7. In order to simulate the tests in realistic operational conditions, the cavitation tunnel tests were carried out behind a simulated (nominal) wake field which was produced based on the wake survey conducted with The Princess Royal model at the Ata Nutku Towing tank of Istanbul Technical University, Korkut & Takinaci, (2013). For this purpose, a wire mesh wake screen was built in the cavitation tunnel test section and resulting wake field was verified by using a 2D Laser Doppler Velocimetry (LDV) device. The cavitation tunnel setup is schematically presented in Figure 7.

![Figure 7: Cavitation tunnel setup, longitudinal view](image)

The comparative velocity distributions of the simulated wake in the UNIGE tunnel measured by the LDV, and that of the nominal wake measured in the ITU towing tank can be seen in Figure 8. As shown in the top section of Figure 8, a part of the wake data measured in the UNIGE tunnel is missing due to the limitation of the optical access for the LDV which was caused by the obstruction of the propeller shaft. The LDV measurements could be carried out by the probe that could approach to the measurement zone only from the starboard side of the test section.

![Figure 8: Nominal wake field: Simulated in cavitation tunnel (top); Measured at towing tank (bottom)](image)

Based upon the typical in-service operational conditions of The Princess Royal, which correspond to 10.5kn and 15.1 kn vessel speeds, the cavitation tunnel test matrix is constructed as shown in Table 2.
In Table 2, STW represents the vessel speed through the water. K_r and K_0 are the standard thrust and torque coefficient, respectively, while the cavitation number is defined based on the propeller shaft speed using Equation (1):

$$\sigma_n = \frac{P_a + \rho g h_n - P_v}{0.5 \rho (nD)^2} \tag{1}$$

where \( P_a \) is the atmospheric pressure, \( g \) is the gravitational acceleration, \( \rho \) is the density of water, \( h_n \) is the shaft immersion of the propeller, \( P_v \) is the vapour pressure, \( n \) is the propeller shaft speed in rps, and finally \( D \) is the diameter of the propeller. Table 3 shows the non-dimensionalisation parameters used for some of the performance and operational characteristics of the propeller as well as others.

Table 3. Non-dimensional performance and operational parameters for propellers

<table>
<thead>
<tr>
<th>Performance Characteristics</th>
<th>Symbol</th>
<th>Formula</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thrust coefficient</td>
<td>( K_T )</td>
<td>( \frac{T}{\rho n^2 D^4} )</td>
</tr>
<tr>
<td>Torque coefficient</td>
<td>( K_J )</td>
<td>( \frac{Q}{\rho n^2 D^5} )</td>
</tr>
<tr>
<td>Advance coefficient</td>
<td>( J )</td>
<td>( \frac{V_a}{nD} )</td>
</tr>
<tr>
<td>Efficiency</td>
<td>( \eta )</td>
<td>( \frac{J \times K_T}{2\pi \times K_0} )</td>
</tr>
</tbody>
</table>

where \( T \) is the thrust, \( V_a \) is the advance velocity, \( Q \) is the torque and \( \eta \) is the propeller efficiency.

The model scale test conditions were specified according to the thrust coefficient identity. As shown in Table 2, while Condition V2 corresponded to the actual service speed (about 15 knots) of the research vessel, Condition V1 corresponded at 10.5kn speed condition.

The cavitation tunnel tests were completed in three stages: the first stage involved the tests with the original propeller model with no pore (Intact propeller). The second stage, the propeller model with 33-1mm pores on each blade (Modified propeller); and the third and final stage, with 17-1mm pores on each blade (Modified-2) which was achieved by closing a half of the pores on the Modified propeller with an adhesive material and smoothing them with care.

During the tests, the water quality was assessed based on the dissolved oxygen content of the tunnel which was monitored by using the ABB dissolved oxygen sensor, model 8012/170, coupled with an ABB analyser model AX400.

3 PILOT CAVITATION TUNNEL TESTS

In order to explore the merits of the pressure relief holes, as stated in the Introduction of this paper, a pilot experimental study was conducted in The Emerson Cavitation Tunnel (ECT) (Xydis, 2015). The model scale propeller used for this study was an existing propeller model with a diameter of 0.35m, which was based on the full-scale propeller of a 95,000 tones tanker with four blades and expanded blade area ratio (BAR) of 0.524. There were two further replicas of this model propeller, which were used for coating research previously, and they were all made from aluminum, as shown in Figure 9. In the pilot study, while the blue colored anodized model was utilized for the base line (or reference) performance measurements without any hole drilled, the other two propeller models were introduced with the pressure relieving holes on their blades and tested to compare their performances with those of the reference propeller. In order see the effect of the holes on the two distinct types of cavitation, which were observed with this model as the tip vortex and sheet cavitation, the 2nd model were introduced with number of small holes drilled at the blade tip region above 90% of the propeller radius, while the 3rd model propeller were introduced with the holes drilled at the mid as well as at tip regions above 60% of the blade radius, respectively. The three models used are shown in Figure 9 and called “Base”, “Tip modified” and “Sheet modified” related to their use and functionality of their effect on the tip vortex and sheet vortex, respectively.

Figure 9. From left to right: “Base” model propeller; “Tip” (region) modified model propeller; and mid and tip (region) modified, “Sheet” model propeller

A summary of the comparative propeller performance measurements in terms of the thrust and torque curves of the three model propellers is shown in Figure 10 based on the pilot study (Xydis, 2015). While the introduction of a large number of holes spread over the mid and tip region of the propeller blades had a significant reduction on the thrust, which would result in deteriorating propeller efficiency, as well as in torque for the “Sheet” propeller, a more conservative number of pressure relief holes concentrating around the tip region for the “Tip” modified propeller did not produce such a significant impact on the thrust and torque compared to the “base” propeller as shown in Figure . Most importantly, the experimental
results are in perfect agreement with the established CFD models (omitted to shorten the length of the paper) which shows up to 13.8% cavitation volume reduction and 0.5% loss from the efficiency.

Figure 10. Comparative non-dimensional thrust and torque coefficients over advance coefficient in open water and vacuum condition

The noise measurements corresponding to typical operating conditions of this vessel revealed up to a 10 dB reduction in the sound pressure levels (SPL) for the mid-frequency region (300 Hz to 2 kHz) as well as in the high-frequency region (10 to 20 kHz) for the tip modified propeller and for advance coefficients of J=0.55 and J=0.5 as shown in Figure and 12, respectively.

Figure 11. Comparative sound pressure levels of three model propellers tested for J=0.55 and vacuum condition

Figure 12. Comparative sound pressure levels of three model propellers tested for J=0.50 and vacuum condition

4 COMPUTATIONAL FLUID DYNAMICS APPROACH

Before moving on to the experimental investigations of the effect of the drilling holes on the propeller performance, cavitation dynamics and noise, the Base and Modified propeller (tip drilled) were also simulated using CFD approaches. Thus, cavitation volume reduction could be estimated at the modification stage of the propeller using CFD methods thanks to the new meshing refinement approach, MARCS, for cavitation simulations as explained in detail in the following.

4.1 Mesh Adaption Refinement Approach for Cavitation Simulations (MARCS)

An advanced mesh refinement method for capturing tip vortex cavitation in a propeller’s slipstream was proposed and preliminary results were presented for limited range of tip vortex extensions for two benchmark propeller models (PPTC and INSEAN E779A) (Yilmaz et al., 2017). The proposed method was further developed by using INSEAN E779A propeller, and further extension of the TVC in the propeller slipstream could be achieved (Yilmaz et al., 2019). Following this achievement, the same meshing refinement approach was applied on The Princess Royal propeller model with a greater extension of the tip vortex development in the propeller’s slipstream.

The new adaptive mesh refinement approach (MARCS) proposed by the Yilmaz was used to enhance the capture of tip vortex cavitation in a propeller slipstream. In MARCS, the adaptive mesh refinement was created only in the region where the tip vortex cavitation may occur. At the beginning of this application, the upper limit of absolute pressure in the solution was determined by creating a threshold region in Star CCM+ (Figure 13, Left). In such cavitation simulations, the volume fraction of vapour shows the area where the absolute pressure is below the saturation pressure of water, thus identifying the cavitating volume. A threshold region was created by increasing the saturation pressure from the default saturation pressure, 3169 [Pa] used by Star CCM+ to a higher value, 17,000 [Pa] thus generating, the pink region shown in Figure 13, Left. This artifice provided an indication of the volumetric trajectory on which to generate a fine mesh for accurately capturing the pressure-drop correctly and tracking the cavity bubbles in the propeller slipstream.

Figure 13. Tip Vortex Cavitation Refinement (MARCS)

The MARCS application provided an improvement for prediction of cavitation volume reduction and hence noise reduction for the base and modified propeller (tip drilled) before cavitation tunnel tests as shown in Figure 14 and Figure 15.

4.2 Numerical Model

The commercial CFD software, STAR-CCM+ finite volume stress solver, was used to solve governing equations (such as continuity and momentum) (STAR-CCM+ User Guide, 2018).

In CFD procedures fluid flows are simulated using various methodologies depending on the nature of the flow.
problem and the availability of computational resources. Numerical methods can be broadly categorized into Reynolds Averaged Navier–Stokes (RANS), Detached Eddy Simulation (DES), Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS). While RANS solver widely is used for open water simulations to predict propeller performance coefficients, scale-resolving simulations such as DES and LES models are commonly used in the calculations of turbulent cavitating flows.

With this selection of the suitable turbulence model for the tip vortex cavitation simulations of the base and tip drilled propellers, different time step values were tried based on the time steps recommended by ITTC and others in the open literature. Hence the time step was calculated such that the propeller rotates between 0.5 and 2 degrees per time step, ITTC (2014). Finally, a time step value of $\Delta t = 5 \times 10^{-5}$ s, which means 600.60 time steps per revolution (i.e. for a rotational speed of 33.3 rps; time per revolution is 0.03s or angular blade displacement of 0.59 degree per time step), was used for the cavitation simulations.

For cavitation modelling, while a Volume of Fluid (VOF) model was used in describing the multiphase flow, the Schnerr-Sauer model which is based on Rayleigh-Plesset Equation, was used as a cavitation model in STAR-CCM+ (STAR-CCM+ User Guide, 2018).

The numerical domain utilized throughout this study consisted of a static domain representing the cavitation tunnel and a rotational domain around the propeller employing a sliding mesh approach. The domain boundaries are defined as velocity inlet and pressure outlet. The tunnel wall as well as the propeller is defined as wall type of boundary conditions. The rotating domain passes through the gap between shaft and hub.

The numerical mesh is an unstructured trimmed grid, and basic prismatic cells are applied near the blade surface for resolving the boundary layer (trying to keep $y^+$ around 1). As stated earlier, MARCS approach was also used for local mesh refinement to be able to simulate TVC and evaluate the benefit of using the drilled holes in the presence of the TVC extension. Additionally, for the tip drilled propeller, different mesh arrangement was generated in direct relation with the drilled holes diameter. Smaller mesh size for the pressure pore surfaces was used to capture the cylindrical hole shape properly as shown in Figure 14.

### 4.3 Results - Cavitation Pattern

Figure 14 and Figure 15 presented the cavitation patterns using MARCS approach for the Base and Modified tip drilled propeller to predict propeller cavitation dynamics including tip vortex cavitation. Therefore, the cavitation volume reduction could be estimated using CFD approaches before experimental cavitation tests thanks to the MARCS technique.

![Figure 14. Cavitation Pattern Comparisons between Base and Tip Drilled Propellers (Left; Base propeller, Right; Tip Drilled propeller)](image)

![Figure 15. Comparison between EFD and CFD for Base and Tip Drilled Propellers (Top; EFD, Bottom; CFD)](image)
It was concluded that the application of the Pressure Pores in the tip region of the propellers provides cavitation volume reduction. This can be observed through the TVC reduction particularly in Figure 15. Although the cavitating noise predictions could not be conducted using CFD approaches in the scope of this study, it was expected that cavitating noise could be reduced due to the cavitation volume reduction (up to 20%) before cavitation tunnel tests. The CFD model based on MARCS was used extensively in the selection process of the most favourable Pressure Pores arrangements as described within more details in Aktas et al (2018).

5 PROTOTYPE TESTING

5.1 Cavitation Observations

To be able to make qualitative comparisons between the cavitation experienced by the intact and modified propeller cases, cavitation observations were carried out at the UNIGE Cavitation tunnel. For this purpose, a mobile stroboscopic system was utilised to visualize and record the cavitation phenomenon on and off the propeller blades. The cavitation recordings were made with three Allied Vision Tech Marlin F145B2 Firewire Cameras, with a resolution of 1392 x 1040 pixels and a frame rate up to 10 fps.

<table>
<thead>
<tr>
<th>Condition</th>
<th>K_τ</th>
<th>σ_N</th>
<th>Intact propeller observations</th>
<th>Modified propeller observations</th>
<th>Modified Propeller-2 observations</th>
</tr>
</thead>
<tbody>
<tr>
<td>V1</td>
<td>0.211</td>
<td>1.91</td>
<td>TVC everywhere, starting from blade L.E.; S.S. sheet cavitation at 0°, from 0.8R to the tip, for 15% of the chord at 0.8R, 100% at 0.97R; S.S. sheet cavitation at 180°, from 0.85R to the tip, for 10% of the chord at 0.85R.</td>
<td>Pores cavitation everywhere; TVC at 0° and 180°, only cloudy vortex at other positions; S.S. sheet cavitation at 0°-45° from 0.8R for 10% of the chord, merging with holes cavitation at outer radii; S.S. sheet cavitation at 180°, from 0.85R for 5% of the chord, merging with holes cavitation at outer radii.</td>
<td>Pores cavitation everywhere; TVC everywhere, at 90° and 270° the cavitating core is at inception; S.S. sheet cavitation at 0°, from 0.8R, for 15% of the chord, at 180°, from 0.85R for 10% of the chord.</td>
</tr>
<tr>
<td>V2</td>
<td>0.188</td>
<td>1.07</td>
<td>TVC everywhere, starting from blade L.E.; double vortex at 0°-60°; S.S. sheet cavitation at 0°, from 0.8R to the tip, for 50% of the chord at 0.8R, 100% at 0.85R; S.S. sheet cavitation at 90° and 270°, from 0.9R for 10% of the chord; S.S. sheet cavitation at 180°, from 0.83R to the tip, for 50% of the chord at 0.83R, 100% of the chord at 0.92R.</td>
<td>Pores cavitation everywhere; TVC everywhere, with double vortex at 0°-60°; S.S. sheet cavitation at 0°-45° from 0.8R for 30% of the chord, merging with holes cavitation at outer radii; S.S. sheet cavitation at 180°, from 0.83R for 20% of the chord, merging with holes cavitation at outer radii.</td>
<td>Pores cavitation everywhere; TVC everywhere, the cavitating core is now well developed but still presents unstable behaviour; double vortex at 0°; S.S. sheet cavitation at 0° from 0.8R for 40% of the chord, at 180° from 0.8R for 30% of the chord.</td>
</tr>
</tbody>
</table>

Table 4 Cavitation observations for V1 and V2 for Intact and Modified propeller
Remarks on the cavitation observation for the Intact propeller, Modified Propeller and Modified Propeller-2 cases for Condition V1 and V2 are provided in Table 4. Some sample images are also shown in Figure 16 and Figure 17 for the three propellers alongside Case V1 and V2 in the respective order. From the images and the remarks, it is evident that with the introduction of PressurePores™, tip vortex cavitation experienced by the intact propeller was disrupted. For the Modified propeller case, in both conditions, the tip vortex cavitation was almost disappeared. For the Modified Propeller-2 case it was observed that the application of the PressurePores™ changed the nature of the steady, solid tip vortex cavitation to a cloudy appearing line with less strength and reduced core diameter.

5.2 Radiated Noise Measurements
In this section the details of the test set-up for the noise measurements, the analyses and presentations of these measurements and results are presented.

Figure 7 showed a scheme of the setup adopted during these tests including the positions of the three hydrophones utilised. In particular, two hydrophones were mounted on fins at the downstream of the propeller: one on the port side at the same vertical position with the propeller shaft (H2); the other (H3) on the starboard at a lower vertical position. The third
hydrophone (H1) was mounted in an external plexiglass tank filled with water and mounted on the bottom window of the testing section. The measurements from H3 was used for the noise results presented throughout this manuscript.

Moreover, noise tests were also repeated at least three times. For the post-processing of the noise measured the ITTC, (2017) guidelines for the model scale noise measurements were followed.

The average Power Spectral Density, G(f) in Pa²/Hz, was computed from each sound pressure signal p(t) using Welch’s method of averaging modified spectrograms. The Sound Pressure Power Spectral Density Level Lp is then given by Equation (2):

$$L_p(f) = 10 \log_{10} \left( \frac{G(f)}{P_{ref}} \right) (\text{dB re } 1 \mu\text{Pa}^2/\text{Hz}) \quad (2)$$

where $p_{ref} = 1 \mu\text{Pa}$.

The background noise was measured reproducing the same condition for corresponding test conditions regarding the shaft revolution, flow speed and vacuum by replacing the propeller with a dummy hub. Only one series of the background noise measurements were carried out since the tunnel operational conditions do not vary significantly passing from the intact to the modified propeller cases.

Comparing the total noise measured with the background noise, the net sound pressure levels of the propeller were analysed as follows:

1. Signal to noise ratio higher than 10 dB:
   
   No correction made

2. Signal to noise ratio higher than 3 dB but lower than 10 dB:

   $$L_{PN} = 10 \log_{10} \left[ 10^{(L_{Ptot}/10)} - 10^{(L_{PNB}/10)} \right] \quad (3)$$

3. Signal to noise ratio lower than 3 dB:

   Results disregarded \( (4) \)

Also, the net sound pressure levels may be scaled to a reference distance of 1-meter exploiting measured transfer functions, or simply according to Equation (5):

$$L_{PN@1m} = L_{PN} + 20 \log_{10}(r) \quad (5)$$

where \( r \) is the distance between propeller (acoustical centre) and sensor.

The latter formulation has been used in present work. The acoustical centre of the propeller was defined with respect to the centre of the propeller disk.

Based upon the above-described post-processing, Figure 18 to 21 present the measured noise levels for the Intact propeller, Modified propeller and Modified Propeller-2 in the narrow and Third-octave band for condition V1 and V2. In both cases, significant reductions regarding the radiated noise levels can be observed over a frequency range from 200Hz to 1kHz. For the service speed condition V2, the reductions are consistent almost throughout the entire frequency range tested. For condition V1, the application of the PressurePores\textsuperscript{TM} Technology observed to cause some elevation of the URN in the high-frequency region.
and Modified Propeller-2 net noise levels at 1m (narrowband), condition V2, hydrophone H3

Figure 21. Comparison between Intact, Modified Propeller and Modified Propeller-2 net noise levels at 1m (one third octave band), condition V2, hydrophone H3

Figure 22 presents the net difference between the noise levels of the Intact propeller and both modified propellers for Condition V2 as measured by hydrophone H3 to demonstrate effectiveness of the PressurePores™ technology. As shown in this figure a maximum of 17dB reduction is possible by using this technology at the critical low frequency region of the URN spectrum.

Figure 22. Noise reduction with application of pressure relief holes in Third Octave band for condition V2, measured at hydrophone H3

5.3 Propeller Performance Tests

This section presents the details and results of the propeller open water tests that were conducted at the CTO towing tank. The purpose of these tests was to determine performance characteristics of the propeller regarding thrust, torque and efficiency before and after the introduction of the PressurePores™ technology.

During these tests, the rate of the propeller shaft revolutions was set over a range to assure the Reynolds Numbers above the critical threshold of 500,000. Also, to confirm the typical convergence of the measurements, for single advance ratio of J = 0.6, the tests were repeated for three additional values of the Reynolds number. The test data analysed for the thrust, torque and efficiency were presented by the 4th-degree polynomials for the three propeller test cases as shown in Figure 23. The operating condition of the Princess Royal propeller are very close to Advance Coefficient J=0.5. As shown in Figure 23, the open water tests indicated that there is a 2% loss of thrust and 4% gain in torque which consequently results in a propeller efficiency loss of 5.7% for the Modified Propeller compared to the Intact propeller. For Modified Propeller-2 case, with a half of the pores applied in the Modified propeller test case, the loss in thrust was about 0.1% while there was a 2.2% gain in torque which resulted in the efficiency loss of 2.3%.

Figure 23. Open water characteristics of the PR propeller before and after the application of PressurePores™ (Modified propeller and Modified Propeller-2)

6 CONCLUSIONS

A unique combination of the experimental and numerical investigation was conducted to develop and explore the benefits of the PressurePores™ concept to mitigate the URN levels of a marine propeller. Following a pilot cavitation tunnel testing, that showed promising results, extensive CFD simulations were conducted to further develop this technology and establish strategic application methodologies.

To accurately simulate the effects of PressurePores™ on TVC of a propeller, a novel adaptive meshing technique was used in a commercial CFD code to capture the cavitating propeller flow properties. With the understanding achieved through the CFD simulations, the PressurePores™ technology was applied on a prototype propeller and then validated by using model tests conducted in the University of Genova cavitation tunnel and CTO towing tank for the cavitation, noise and efficiency performances.

The test results conducted with the model propeller of a research vessel and for two different combinations of the PressurePores™ technology revealed that significant
reductions in the measured propeller noise levels can be achieved.

The comparative test results for the Modified Propeller test case indicated the noise reduction compared to the unmodified propeller can be as high as 17dB and particularly in the frequency region that are utmost important for some marine mammals. For the same configuration, the towing tank test data showed about a 2% loss in the propeller efficiency.

The test results for the Modified Propeller showed more superior underwater noise reduction in the high-frequency region but with a higher propeller efficiency loss, about 5.7%.

Most importantly, established CFD models which shows up to 13.8% cavitation volume reduction for the conventional commercial vessel propeller, for which cavitation tunnel tests has been conducted as the pilot study, has shown significant noise emission reductions with only 0.5% loss from the efficiency. This empowers the novelty of the PressurePores™ concept as a retrofit to the propulsion of marine vessels.

REFERENCES


