Sixth International Symposium on Marine Propulsors
smp'19, Rome, Italy, May 2019

Numerical Investigations of Propeller-Rudder-Hull Interaction in the Presence of Tip Vortex Cavitation

Naz Yilmaz[1], Batuhan Aktas[2], Savas Sezen[1], Mehmet Atlar[1], Patrick A. Fitzsimmons[1], Mario Felli[3]

[1]University of Strathclyde, Glasgow, UK

[2]Fugro Structural Monitoring, Glasgow, UK

[3]CNR-INM, Rome, Italy

ABSTRACT

This paper presents the investigation of propeller-rudder-hull interaction by using Computational Fluid Dynamics (CFD), and Experimental Fluid Dynamics (EFD) approaches including the propeller's performance in cavitating conditions, particularly the effect of the tip vortex cavitation on the interaction phenomenon.

The investigation was focused on the recently generated benchmark test data for the research catamaran, *The Princess Royal*, and its scaled model which was tested for cavitation and noise investigations in the large circulating water channel of CNR-INM, Italy within the scope of the European collaborative project SONIC (SONIC, 2012). The hull-propeller-rudder interaction of a 1/3.4 scaled model of this vessel was simulated by using the commercial CFD software, STAR-CCM+, which implemented the Schnerr–Sauer cavitation model the cavitation effect. Large Eddy Simulations (LES) was used for a better resolution of high velocity and pressure gradients to model the tip vortex cavitation.

A new adaptive meshing technique using a Mesh Adoption Refinement Approach for Cavitation Simulations (MARCS) was applied for more effective modelling of the tip vortex cavitation during the propeller-rudder-hull interaction (Yilmaz et al., 2019). The results of the CFD simulations were compared with the EFD data, particularly for the cavitation dynamics.

Keywords

Tip Vortex Cavitation (TVC), Propeller-Rudder-Hull Interaction, CFD, LES.

1 INTRODUCTION

Computational modelling of tip vortex cavitation of marine propellers is still challenging for marine CFD community. This is particularly true in modelling the tip vortex cavitation (TVC) from all propeller blades simultaneously and tracing them through a rudder in the propeller slipstream for relatively large rudder-propeller clearances. The investigation of the effect of tip vortex cavitation (TVC) on the interaction between the propeller and the rudder in the presence of a non-uniform hull wake flow has great importance to the practising and academic naval architects.

Although the numerical modelling of sheet cavitation has been tackled successfully by many investigators, extending this model to include the TVC interacting with the rudder and in the presence of the unsteady wake are still under development (e.g. Windt & Bosschers, (2015), Lloyd et al. (2017) Viitanen & Siikonen, (2017), Yilmaz et al., (2019) etc.)

To understand the complex propeller-rudder-hull interaction, the problem is divided into its components; hull-propeller, hull-rudder and propeller-rudder, following the approach proposed by Kracht (1995). As a result, the mutual interaction effect due to the propeller – rudder combination will be investigated in detail as the main interest of this paper including the propeller cavitation, and TVC mainly.

The propeller-rudder interaction phenomenon may be investigated in two parts: (1) The influence by the rudder on the propeller's pressure distribution, where the propeller is operating in the flow in front of the rudder. (2) The effect of the propeller flow field on the rudder which sometimes manifests itself in cavitation erosion of the rudder structure (Carlton, 2007).

In the past 30 years, propeller-rudder interaction has been investigated experimentally by many researchers. (e.g. Molland and Turnock, (1991), Goodrich and Molland, (1979), Kracht (1995) and Felli et al. (2009), Felli and Falchi, (2011). There have been recent investigations by using CFD methods for better understanding of the cavitation phenomenon including the effect of rudder regarding the sheet cavitation developed on the blades and TVC (e.g. Boorsma and Whitworth, (2011), Paik, et al., (2013) and Mascio et al., (2015).

Although there have been pockets of useful EFD and numerical studies involving low-fidelity and CFD based numerical modelling, there is a clear gap in modelling the propeller-rudder-hull interaction phenomenon in propeller cavitating conditions. This paper was motivated by the desire to fill this gap utilizing the state-of-the-art commercial CFD code in comparison with EFD results.

In this paper, the recently developed new mesh refinement technique, which utilises a Mesh Adaptive Refinement approach for Cavitation Simulations (MARCS) (Yilmaz et al. 2019), was applied to the propeller-rudder-hull arrangement of the Newcastle University research vessel, *The Princess Royal*. The CFD predictions for the scaled model of this vessel were compared with the experimental data, which was generated by the CNR-INM within the scope of the SONIC Project (Felli et al., 2014) and the comparative results between the EFD and CFD were discussed.

Following the above introduction, the paper continues with the review of the EFD approach which was used for the experimental investigations of the propeller-rudder-hull interaction in the presence of the tip vortex cavitation by CNR-INM at §2. The CFD investigation of the complex interaction phenomenon is presented at §3 with the detailed description of the numerical model, computational domain preparation and mesh generation. An overall summary of the new meshing approach for simulating tip vortex cavitation is also given in §3 while the associated results and discussions are included in §4. The paper ends with the conclusions and future works presented in §5.

2 EXPERIMENTAL FLUID DYNAMICS (EFD) APPROACH

In order to investigate the propeller-rudder interaction by including the effect of the hull, the natural candidate for the EFD data would be *The Princess Royal's* data because of the accessibility of the authors to the most of the data available for this vessel through their involvement with the FP7-SONIC Project.

The model tests conducted in the Large Circulation Water Channel of CNR-INM in Rome, Italy could be a good candidate to simulate the propeller-rudder-hull interaction due to the availability of a range of good visual observations of the cavitation patterns, especially with the tip vortex cavitation extensions in the propeller's slipstream. The EFD data from this facility also included the effect of the free surface.

A review of these model tests including the general descriptions of the CNR-INM test facility, the model, test matrix and arrangements is presented. Further details of these tests and the data can be found in SONIC Project Report, Felli et al. (2014).

2.1 Model Tests at CNR-INM

The Large Circulating Water Channel of CNR-INM, Italy is a vertical plane, free surface, variable-pressure recirculating channel, having a capacity of 4 million of litres test water. The test section of the facility has 10 m length, 3.6 m width and 2.25 m maximum water depth. The facility is driven by two 4-bladed axial flow impellers operating in two separate and parallel trunks and developing power of 435kW at 1500 rpm. The maximum water speed in the test section is 5.2 m/s. The facility can be depressurised down to 30 mbar, by fitting a removable cover to the test section. An overall arrangement of the facility is shown in Figure 1.

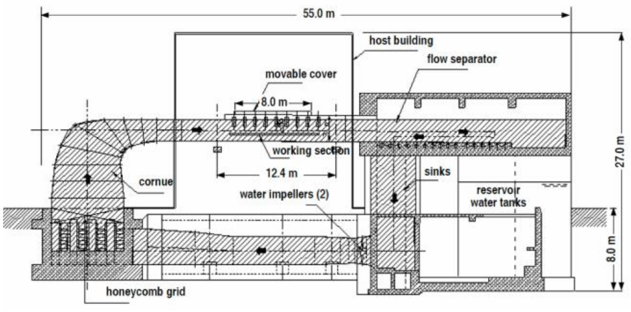

Figure 1 The Large Circulating Channel of CNR-INM

2.2.1 The Princess Royal and its scaled model

For the model tests, one demi-hull of *The Princess Royal* vessel and its propeller were modelled with a scale factor λ = 3.4, by taking advantage of the symmetry feature of the catamaran. Figure 2 shows the demi-hull model, which was made from fibreglass and the aft end details of the model. More detailed information about *The Princess Royal* and its rudder and propeller geometries can be found in Atlar et al., (2013).

Figure 2 *The Princess Royal* Research Vessel and the Propeller

2.1.2 Test Matrix

The original test matrix was prepared using the four most representative and reliable runs of the full-scale trials selected by the SONIC project partners which corresponded to the engine speeds of 600, 900, 1200 and 2000 rpm, with a reduction gear ratio of 1:1.75 as shown in Table 1. In this Table U is the cruising speed, V_M is the model testing velocity defined as $U/(\lambda)^{0.5}$, n is rotational speed of the propeller, J is the advance ratio defined as V_M/nD, where D is the propeller diameter, P_0 is the static pressure in the test section at the propeller immersion.

The cavitation tests were performed with an oxygen content of 0.25 mg/l, at a water temperature of about 14 °C.

Table 1 Test Matrix

Cond	U [kn]	V_M [m/s]	n [rps]	J [-]	P_0 [mbar]
1	4.775	1.33	10.53	0.57	70
2	7.1	1.98	15.80	0.57	80
3	9.35	2.61	20.96	0.56	70
4	10.53	2.94	26.31	0.51	75

2.1.3 Test Setup

As described in details in Felli et al. (2014), the cavitation observations were conducted using a high-speed camera located as shown in Figure 3. The model was set up according to the ITTC procedure 7.5-02-03-03.3 on "Cavitation Induced Pressure Fluctuation Model Scale Experiments" (ITTC, 2014).

The model was equipped with a dedicated dynamometer for the measurement of thrust and torque although no records of the thrust and torque were taken during these tests. Also, detailed flow measurements were undertaken to qualify the characteristics of the flow in the propeller region.

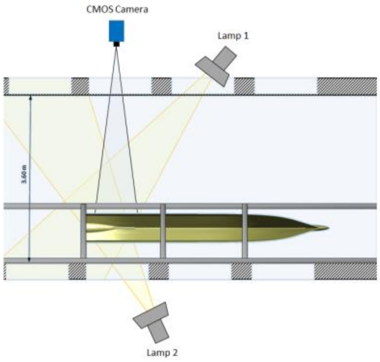

Figure 3 Stroboscopic Lights and Camera Arrangement for Cavitation Observations

The model tests also included measurement of fluctuating hull pressure at seven locations as shown in Figure 4.

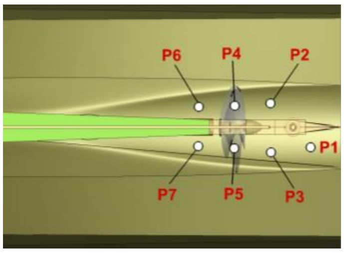

Figure 4 Nomenclature of Pressure Sensors and Relative Position, Felli et al. (2014)

3 COMPUTATIONAL FLUID DYNAMICS APPROACH

Amongst the four test conditions given in Table 1 “Condition 3”, which presented one of the strongest tip vortex cavitation dynamics scenarios, was selected to simulate for the CFD predictions.

For the computations, *The Princess Royal* model and the test section of the circulating water channel were represented as precise as possible with regards to the dimensions, location of the hull and cavitation conditions. The latter is the function of the inflow speed, propeller shaft speed and tunnel free surface height, tunnel reference pressure, saturated vapour pressure and fluid temperature etc. for the proper validation study between EFD and CFD investigations.

3.1 Numerical Model

The CFD simulation was conducted by using the commercial CFD software, STAR-CCM+ for marine applications (STAR-CCM+, 2018).

In order to simulate the free surface in the testing facility, two different type fluids (i.e. water and air) and two different flow phases (i.e. liquid and vapour) had to be modelled. While the free surface was defined between the water and air, a multiphase interaction was described between the liquid and vapour phases of the water for modelling cavitation.

For turbulence modelling in this study, Large Eddy Simulation (LES) was preferred for cavitation simulations. This is due to the fact that, in contrast to the Reynolds Averaged Navier-Stokes (RANS) model, scale-resolving simulations are able to solve the large scales of turbulence and model small-scale motions. For scale-resolving simulations, the two well-known approaches, Detached Eddy Simulation (DES) and Large Eddy Simulation (LES), are both available in STAR-CCM+. LES is used more commonly for simulating complex flows such as cavitation, especially for the tip vortex type of cavitation, as also used in this paper.

The cavitation was modelled by using the Schnerr–Sauer cavitation model which is based on the Rayleigh-Plesset equation. The bubble growth rate in the Schnerr-Sauer model (Schnerr & Sauer, 2001) is estimated by using Equation 1.

$$\left(\frac{dR}{dt}\right)^2 = \frac{2}{3}\left(\frac{p_{sat} - p_\infty}{\rho_l}\right) \quad (1)$$

The cavitation number, which is based on the rotational speed of the propeller, is defined in Equation 2.

$$\sigma_n = \frac{p - p_{sat}}{0.5\rho_l (nD)^2} \quad (2)$$

where p is the tunnel pressure, p_{sat} is the saturation pressure of water, ρ_l is the density of the fluid, n is the shaft speed and D is the diameter of the propeller.

The time step value is selected as $5x10^{-5}$s, which means 954.198 time steps per revolution (i.e. for a rotational speed of 20.96 rps; time per revolution is 0.0477s or angular blade displacement of 0.377 degree per time step).

3.2 Computational Domain

The computational flow domain is shown in Figure 5 which presents the position of the hull geometry and the test facility coordinate system with its origin located on the tunnel free surface at a 2.25m water depth in a static condition.

The hull geometry was located nearer to the right side of the tunnel walls to meet the symmetry condition with the breadth of the flow domain being 3.6m in total. The inlet and outlet patches of the flow domain were set at a distance of 2.5m from the bow of the ship and 2.0m from the aft of the ship, respectively (Figure 5).

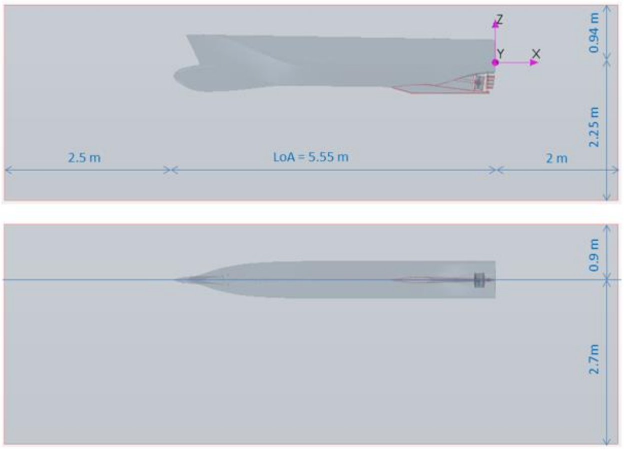

Figure 5 Computational Flow Domain Dimensions

The computational flow domain for the simulations included a rotating region, which was represented by the sliding and overset mesh, and a stationary region to represent the background as shown in Figure 6. While the background region covered the hull, keel and rudder geometries, the rotating region included the propeller geometry only. The overset region was prepared to cover the rudder geometry to be able to transfer the data (tip vortices) from the propeller blades through the rudder.

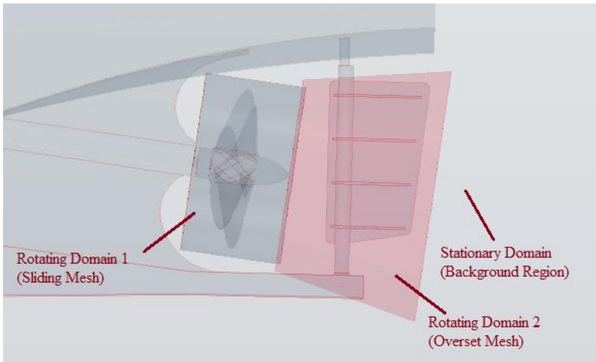

Figure 6 The Flow Domain including Rotating (Sliding and Overset Mesh) and Stationary (Background) Regions

3.3 Mesh Generation

3.3.1 Mesh Adoption Refinement Approach for Cavitation Simulations (MARCS)

A new mesh refinement approach, MARCS (Yilmaz et al., 2019), was used in this paper to simulate the TVC. MARCS (Mesh Adaption Refinement for Cavitation Simulations), was validated using the INSEAN E779A standard test propeller, the Postdam Propeller Test Case (PPTC) and *The Princess Royal* propeller as reported in, e.g. Yilmaz (2019) and Yilmaz et al. (2019).

In MARCS, the mesh was refined only in the region where the tip vortex cavitation may occur in the propeller slipstream. Before the application of this procedure, the simulation was run, and sheet cavitation was simulated using the coarse mesh arrangement without any refinement. At the end of this simulation, using the existing solution, the Q-Criterion limit was determined by creating a threshold region in the STAR-CCM+ software as shown in Figure 7. This limit was defined by visualising an iso-surface of the Q-Criterion, which is a vortex identification method, calculated using Equation 3.

$$Q = \frac{1}{2}(\|\Omega\|^2 - \|S\|^2) \tag{3}$$

where ||Ω|| and ||S|| represent the spin rate tensor and strain rate tensor, respectively.

MARCS was used for the simulation of the TVC trajectories from the propeller blades through the rudder. The details and the total number of cells generated for the sheet and tip vortex cavitation using MARCS are shown in Table 2 while Figure 8 illustrates the mesh generated by using MARCS for the tip vortex cavitation simulations.

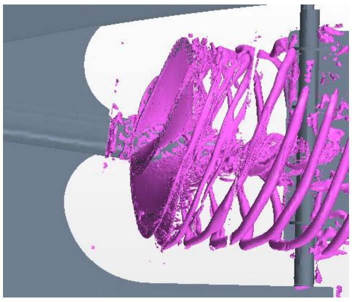

Figure 7 Isosurface of Q-Criterion = 10000 s^{-2}

Table 2 Mesh Details for *The Princess Royal* Propeller-Rudder-Hull Arrangement

Mesh Details	Sheet	Tip Vortex (MARCS)	Unit
Surface Size (Blade)	0.5/1.5	0.5/1.5	[mm]
Surface Size (Ref.)	1.0	0.25	[mm]
Number of Cells	21,119,336	49,664,725	[-]

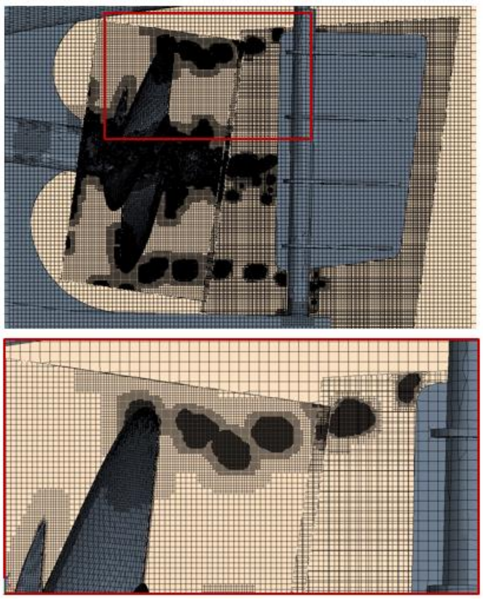

Figure 8 Generated Mesh using MARCS

4 RESULTS AND DISCUSSIONS

The CFD simulation results of “Condition 3” are presented and compared with the experimental results regarding the cavitation dynamics, hull pressure fluctuations and propeller hydrodynamic performance coefficients.

Table 3 displays the CFD predicted propeller thrust and torque coefficient. Since the model tests did not measure the thrust and torque data, no comparison of these coefficients can be made with the EFD.

Table 3 Hydrodynamic Propeller Performance for Propeller-Rudder-Hull Interaction

Cond	Method	K_T	$10K_Q$
3	CFD – Model Scale	0.183	0.267
	EFD – Model Scale	N/A	N/A

Figure 9 shows a comparison of experiments in cavitation tunnel (top) and CFD results (bottom).

As far as the TVC is concerned, despite the TVC was extended up to the rudder, thanks to the application of MARCS, the traces of the TVC could not be fully developed throughout the rudder. This lack of precision in the modelling can be explained with the nature of the dynamics of the TVC which can be observed from the experimental images where the tip vortices were also losing their strengths as they approached the rudder, which can be observed from the top two rows of Figure 9. This causes the disappearance of the tip vortices and hence requires a smaller surface mesh size for the refinement region in the CFD calculations to capture the TVC while they are losing their strength as they approach the leading edge of the rudder.

As shown in Figure 9, it was concluded that the TVC could not be extended clearly up to the rudder due to the interface problem between the sliding mesh and overset mesh regions at some propeller blade positions, although a combined system was used to try to eliminate this problem. The extra cavitating bubbles, which were produced at the interface surface between the sliding mesh and the overset regions due to the mesh refinement with the small mesh surfaces on the interface, were also observed in Figure 9. Unfortunately, these extra cavitating bubbles do not reflect the reality.

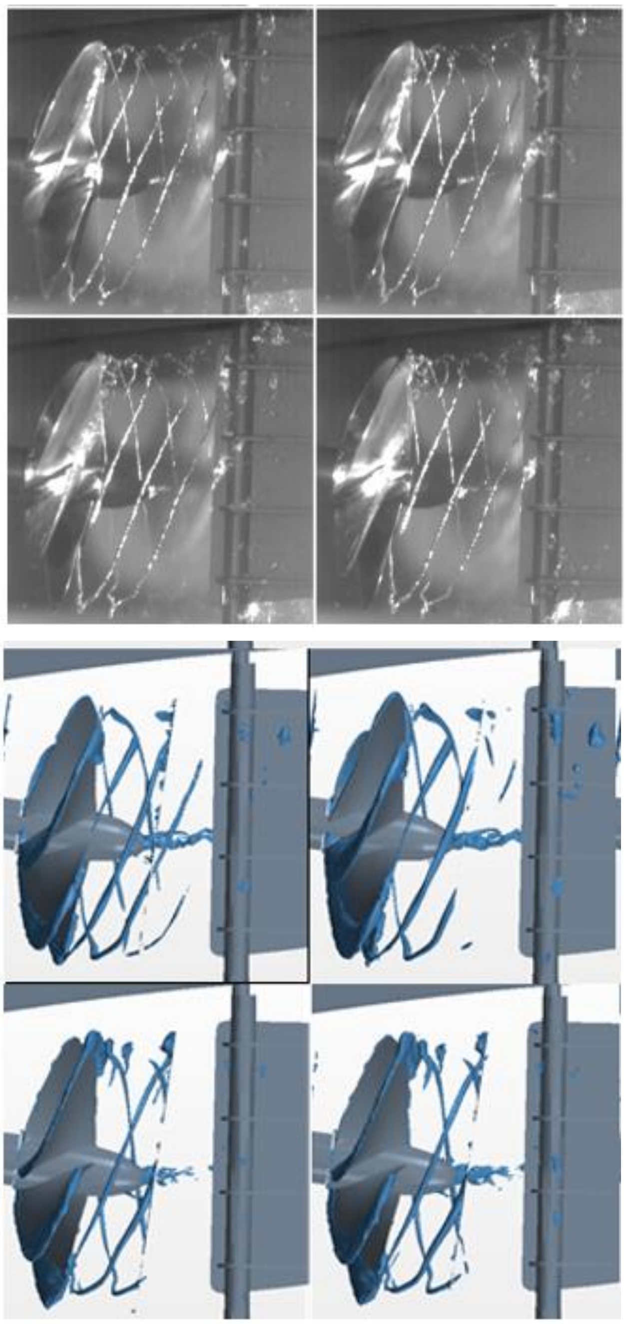

Figure 9 Comparisons between EFD (Top) and CFD (Bottom) Results in terms of Cavitation Patterns including TVC

In order to provide a more focused comparison of the TVC trajectories, Figure 10 presents a comparison of the EFD and CFD images overlaid at one blade position of the propeller. In this Figure, the experimental image is placed in the background, and the CFD image is placed on the top. As shown in these comparative images, in spite of the underpredicted tip vortex cavitation extent by the CFD, the correct trajectories of the tip vortex cavitation in the propeller's downstream can be clearly seen by proving the ability of the MARCS procedure including the effect of the non-uniform hull wake flow.

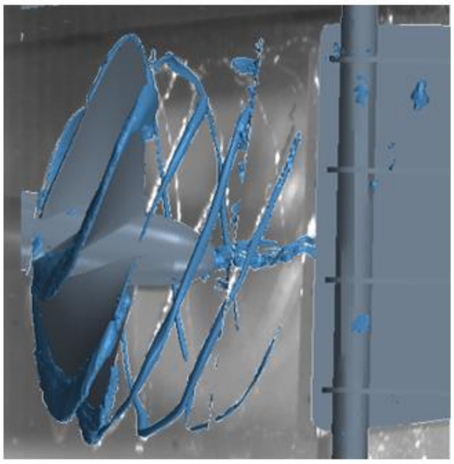

Figure 10 Comparisons between EFD and CFD Results in terms of Cavitation Patterns including TVC (Backward; EFD, Forward; CFD)

Figure 11 presents a comparison of the fluctuating hull pressures between measurements and computations at three pressure probe locations as shown in Figure 4. While the red line demonstrates the averaged value of pressure data on the model hull surface recorded during the experiments, the black line presents the corresponding pressure data based on the CFD simulations.

As shown in Figure 11, although the CFD simulations generally show a reasonable correlation for the magnitudes with the experimental data for all probe locations, in some cases phase shift between the EFD and CFD results are noticeable. For these comparisons, only the sheet cavitation was modelled due to the unstable cavitation dynamics of the tip vortex cavitation with the application of the MARCS.

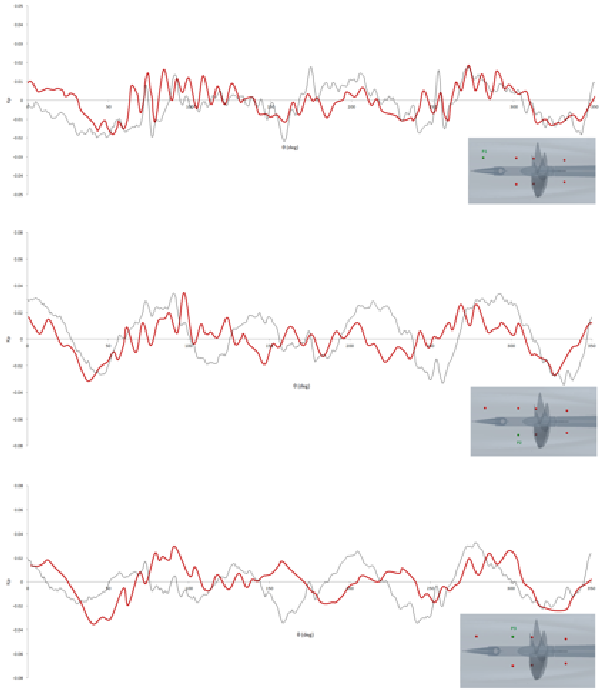

Figure 11 Hull Pressure Fluctuations Comparisons between EFD (Red) and CFD (Black) Results (From Top to Bottom; P1, P2 and P3)

5 CONCLUSIONS AND FUTURE WORKS

The main objective of the paper is to demonstrate the capability of the state-of-the-art commercial CFD code for an effective investigation of the propeller-rudder-hull interaction in the presence of the cavitation, especially the TVC.

To achieve the above purpose, firstly, the cavitation tunnel tests conducted in the Depressurized Large Circulating Water Channel of CNR-INM with the scaled model of *The Princess Royal* research vessel were simulated using the commercial code, STAR CCM+ and results were compared with the experiments for one of the test conditions (Condition 3) which displayed the strongest tip vortex cavitation.

In general, although the improvements have been achieved for extending the TVC in the propeller slipstream, the interaction between the TVC and rudder could not be simulated at the desired accuracy due to the complexity of the full interaction phenomenon amongst the propeller-rudder-hull in the presence of the hull geometry and its wake.

The new adaptive mesh refinement technique (MARCS) developed previously in Yilmaz et al., (2019) and Yilmaz (2019) could give unstable results at model scale with the inclined shaft and in non-uniform flow conditions. This would result in the appearance and disappearance of the tip vortex cavitation in solution time when the generated mesh matched with vortex trajectory and was no longer suitable, respectively, due to the rotation of the refined mesh region. The MARCS approach therefore still needs to be further developed applying re-meshing methods at each time-step to be able to keep the vortices matched with the refined mesh region in the propeller slipstream when the blade position changes in solution time.

6 ACKNOWLEDGEMENTS

The Principal Author of this paper is sponsored by the Turkish Ministry of Education during her doctoral studies which provided basis for this paper. Other author, Mr Savas Sezen's doctoral study is being partially sponsored by the Stone Marine Propulsion Ltd, UK. The model test data used in this paper was generated as part of the FP7-SONIC Project activities which was sponsored by the EC with Grant agreement no: 314394. The access provided to High-Performance Computing for the West of Scotland (Archie-West) thorough EPSRC grant no. EP/K000586/1 is gratefully acknowledged.

REFERENCES

Atlar M., Aktas B., Sampson R., Seo K.C., Viola I.M., Fitzsimmons P., Fetherstonhaug C., (2013) "A Multi-Purpose Marine Science & Technology Research Vessel For Full Scale Observations And Measurements", 3rd International Conference on Advanced Model Measurement Technologies for the Marine Industry (AMT'13), Gdansk, September.

Boorsma, A., Whitworth, S. (2011) "Understanding the Details of Cavitation", Second International Symposium on Marine Propulsors smp'11, Hamburg, Germany, June 2011.

Carlton, J. (2007). Marine Propellers and Propulsion, Elsevier Ltd.

Felli M, Camussi R, Guj G (2009) Experimental analysis of the flow field around a propeller-rudder configuration. Exp Fluids 46:147–164.

Felli, M., Falchi, M., (2011) 'Propeller tip and hub vortex dynamics in the interaction with a rudder'. Experiments in Fluids, Vol. 51, Issue 5, pp 1385-1402.

Goodrich, G. J., Molland A. F., (1979) "Wind Tunnel Investigation of Semi-Balanced Ship Skeg Rudder", Trans. RINA, Vol. 121.

ITTC Report, (2014) "Cavitation Induced Pressure Fluctuation Model Scale Experiments", in ITTC Recommended Procedures, 7.5-02-03-03.3.

Kracht, A. M., (1995) "Cavitation on Rudders" An International Conference on Propeller Cavitation (PROPCAV'95) to celebrate 100 years of Propeller Cavitation Research 16-28 May 1995, Newcastle Upon Tyne, UK.

Lloyd, T., Vaz, G., Rijpkema, D., Reverberi, A., (2017) "Computational fluid dynamics prediction of marine propeller cavitation including solution verification", Fifth International Symposium on Marine Propulsors smp'17, Espoo, June 2017

Mascio, A. D., Dubbioso, G., Muscari, R., (2015) "CFD Analysis of Propeller-Rudder Interaction", The 25th International Ocean and Polar Engineering Conference, 21-26 June, Kona, Hawaii, USA.

Molland, A. F. and Turnock, S. R., (1991) "Wind tunnel investigation of the influence of propeller loading on ship rudder performance", Southampton, UK. University of Southampton 151pp. (Ship Science Reports, 46)

Paik, K. J., Park, H. G., Seo, J. (2013) "RANS simulation of cavitation and hull pressure fluctuation for marine propeller operating behind-hull condition", Int. J. Nav. Archit. Ocean Eng. (2013) 5:502~512 pISSN: 2092-6782, eISSN: 2092-6790.

SONIC (2012) 'Suppression Of underwater Noise Induced by Cavitation', in European Union Framework Programme 7. FP7-SST-2012-RTD-1-SST.2012.1.1-1. - Assessment and mitigation of noise impacts of the maritime transport on the marine environment (coordinated topic within the framework of the 'Ocean of Tomorrow'), FP7, Grant agreement no: 314394.

Felli, M., Falchi, M., Grizzi, S., Mauro, L. and Orrico, M. (2014) 'Suppression Of underwater Noise Induced by Cavitation (SONIC)', Deliverable: D1.3-CNR INSEAN, FP7-SONIC Project Report, Grant agreement no: 314394, December.

Schneer, G. H., Sauer, J., (2001) "Physical and Numerical Modeling of Unsteady Cavitation Dynamics", ICMF-2001, 4th International Conference on Multiphase Flow, New Orleans, USA, May 27-June 1, 2001.

STAR-CCM+ User Guide, (2018)

Viitanen, V, M, Siikonen, T., (2017) "Numerical Simulation of Cavitating Marine Propeller Flows", 9th National Conference on Computational Mechanics, MekIT'17.

Windt, J., Bosschers, J., (2015). Influence of local and adaptive mesh refinement on the tip vortex characteristics of a wing and propeller. In: VI International Conference on Computational Methods in Marine Engineering, (MARINE 2015).

Yilmaz, N., Atlar, M., Khorasanchi, M., (2019) "An Improved Mesh Adaption and Refinement Approach to Cavitation Simulation (MARCS) of Propellers", Journal of Ocean Engineering, January 2019.

Yilmaz, N. (2019) "Investigation of cavitation influence on propeller-rudder-hull interaction", PhD Thesis, Dept of Naval Architecture, Ocean and Marine Engineering, The University of Strathclyde, Glasgow, February.