An Improved Tip Vortex Cavitation Model for Propeller-Rudder Interaction

[1]Naz Yilmaz*; [1]Mehmet Atlar, [1]Patrick A. Fitzsimmons

[1]*University of Strathclyde, Glasgow, UK*

Abstract

The paper starts with the computational modelling of the tip vortex cavitation in uniform flow conditions with an isolated propeller in detail and provides experimental validation. It then moves onto further modelling to include the effect of non-uniform flow and the presence of a rudder placed in the propeller slipstream. The propeller-rudder arrangement of the Newcastle University research vessel, The Princess Royal, and associated experimental data were used for Experimental Fluid Dynamics (EFD) analysis to validate the modelling. The cavitation simulations were conducted using commercial CFD software, Star CCM+. A new meshing technique, which utilizes a Mesh Adaptive Refinement approach for Cavitation Simulations (MARCS), recently developed by the authors, has been applied successfully to simulate the tip vortex cavitation, particularly to trace its extension up to the rudder in the propeller slipstream. The comparison of the CFD and EFD methods for the isolated propeller in cavitation tunnel conditions showed very good agreement in terms of the thrust and torque coefficients of the propeller as well as the sheet and tip vortex cavitation patterns observed. The cavitation simulations have been extended for the same propeller by using the new mesh refinement approach to include the effect of the hull wake and the presence of the rudder. Although the latter simulations fall short of the EFD results and hence they are still under development, the paper presents the developments and results so far to achieve the ultimate aim of this study, i.e. computational modelling of cavitating tip vortices of a propeller interacting with a rudder.

Keywords: Tip Vortex Cavitation, EFD, CFD, Propeller-Rudder Interaction

Introduction

Sheet cavitation of marine propellers can be predicted accurately with existing methods such as lifting surface, BEM and even more accurately with computational fluid dynamics methods (CFD) due to better modelling of the physics of the flow, thanks to new developments in computational power and technology. However, computational modelling of the cavitating tip vortices of a propeller has its challenges, in particular for extending these vortices from the blades up to a rudder in the wake of a ship hull and interacting with the rudder. Even sheet cavitation predictions sometimes give unstable results due to the lack of accurate tip vortex cavitation modelling [1].

There are many numerical and computational studies to predict tip vortex cavitation in literature [2, 3, 4 and 5], using especially CFD methods in which RANS based models for tip vortex cavitation simulations of marine propellers are preferred [6]. Although the RANS model is recognized as a reliable method for sheet cavitation simulations, further studies are still required particularly for accurate predictions of tip vortex cavitation [6]. In contrast to the RANS model, scale-resolving simulations can model small-scale motions and resolve the large scales of turbulence. Within this context, there are two popular approaches for scale-resolving simulations which are known to be Detached Eddy Simulations (DES) and Large Eddy Simulations (LES) models. Recently, the turbulence models based on these two approaches have been preferred widely by the CFD community for simulating complex physical phenomenon such as cavitation and especially for the tip vortex type. These two approaches are also implemented in the commercial CFD software, Star CCM+ [7] which is used in this study as described below, where the LES model is preferred for simulating the tip vortex cavitation.

Detailed and different grid generation techniques have been investigated in literature with regards to bubble dynamics, cavitating bubble diameter and cavitation inception phenomenon. Although, each bubble cannot be modelled, followed and tracked in space using the Rayleigh-Plesset model, which is implemented in the Star CCM+ code, cavitation phenomena have been investigated by creatively pushing the limits of mesh generation capabilities of the software for capturing tip vortex cavitation. In terms of bubble dynamics, Rayleigh published the first analysis

*Corresponding Author, Naz Yilmaz: naz.gorener@strath.ac.uk

about cavitation and bubble dynamics results [8]. After that, Rayleigh-Plesset equation was presented by Plesset and Prosperetti including bubble growth and collapse phenomena by neglecting bubble-bubble interaction and assuming the bubble was spherical in shape [9]. Based on the Rayleigh-Plesset equation, Schneer and Sauer developed a new volume of fraction (VOF) method for time dependent growth and collapse of cavitating bubbles [10]. The Schneer-Sauer cavitation model, neglecting surface tension, viscous effects and bubble growth acceleration, is implemented in the Star CCM+ Code and hence also used for cavitation simulations in this study [7].

Recently, numerical modeling of the tip vortex cavitation phenomenon has been the focal point by some researchers using CFD methods and commercial codes by creating mesh refinement regions around a propeller's tip area for capturing cavity bubbles in the slipstream including the authors, e.g. [12, 13]. However, if the mesh is generated using a larger than required surface size in the tip vortices region, tip vortex cavitation cannot be captured in the propeller slipstream. On this basis there is a need for studies to investigate the cavitating bubble radius for determining surface size in the mesh refinement region based on some useful experimental data e.g. [11]. Within this framework, the authors have developed a new mesh refinement technique, which utilizes a Mesh Adaptive Refinement approach for Cavitation Simulations (MARCS), based on the relationship between the surface size of the generated mesh and the cavity bubble radius [12]. In this paper, the MARCS approach has been applied to the propeller-rudder arrangement of the Newcastle University research vessel, the Princess Royal by taking advantage of the experimental data which were generated by the authors in their previous work [17] and the results are discussed.

Numerical Approach

For cavitation simulation, the Volume of Fluid (VOF) method was used in defining two states of the fluid (water and vapour) in flow domain. DES and LES turbulence models were preferred for the reasons explained in the introduction.

The Schnerr-Sauer cavitation model used in this study is based on a reduced Rayleigh–Plesset (RP) equation within Star CCM+. It neglects the influence of surface tension, bubble growth acceleration, viscous effects and the bubble-bubble interaction but includes scaling of the bubble growth and collapse rates for both single-component and multi-component materials. Using this cavitation model, the cavitation bubble growth rate can be calculated as follows,

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

Where p_{sat} is the saturation pressure, p_∞ is the pressure of the liquid and ρ_l is the liquid density.

The cavitation number (rotational speed, σ_n) is also defined as,

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

The new adaptive mesh refinement approach (MARCS) proposed by the authors is 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+. In such cavitation simulations, the volume fraction of vapour shows the volume 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 1. This artifice provides 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.

*Corresponding Author, Naz Yilmaz: naz.gorener@strath.ac.uk

In applying this idea on a generated mesh, two field functions were created in order to prepare a table including coordinates and the surface size of the new refined mesh for generating the adaptive mesh. These were needed in order to extend the tip vortex cavitation, much as shown in Figure 1. One of the two field functions, called "Cell Width", specifies the one dimension of each cubic cell. Whereas the other field function termed "Refinement" is defined for creating a refinement table which includes coordinates of each cell in x, y and z directions and the surface size of the new mesh to be used while an adaptive mesh is being generated. The "Refinement" field function represents each mesh cell, where the absolute pressure below 17000 [Pa] and above 3169 [Pa], is sub-divided by three in the three dimensions. The upper limit of absolute pressure was defined by creating a threshold (17000 [Pa]) and checked visually as shown in Figure 1. The lowest limit of the pressure was determined by the saturation pressure (3169 [Pa]). Figure 1 shows, respectively, the threshold below 17000 [Pa] and mesh generated using the new adaptive mesh refinement approach (MARCS).

The most important part of the MARCS approach is to determine the surface size of the each cubic cell in the tip vortex region for capturing cavity bubbles in the slipstream. Kuiper [11] investigated and measured cavitation inception, including tip vortex cavitation, using a model scale propeller (Propeller V) at J-values of 0.3, 0.4 and 0.5. Additionally, he defined an experimental relationship between the cavitating core radius (a_c) and cavitation index (σ_n) by preparing equations and graphs in this range of J values. In his study, the minimum radius of each bubble (a_i) was consistently found to be about 0.25 mm on a 250 mm diameter propeller at the cavitation inception stage. According to these investigations, the core radius (a_c) always tends to go to the minimum core radius (a_i). On the basis of Kuiper's study, similar relations between bubble radius, mesh size and simulating tip vortex cavitation are determined within the new adaptive mesh approach. The mesh size was always required to be maintained below 0.25 mm (it is approximately 0.22 mm for this case) for capturing the tip vortex cavitation structure in the propeller slipstream. Using a mesh size larger than 0.25 mm, the tip vortex cavitation could not be simulated as extended as shown in Figure 2.

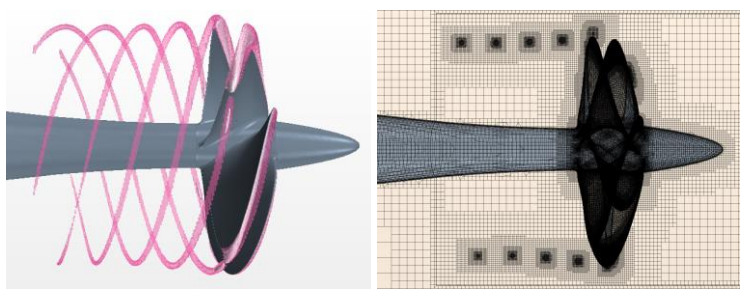

Figure 1. Adaptive Mesh Refinement (MARCS)

Results

Cavitation phenomena, particularly the cavitating tip vortex, have been investigated using EFD and CFD methods for the Princess Royal Propeller. This propeller has been recommended recently by the ITTC [14] as a benchmark propeller for test ranging from open water tests to noise measurement studies.

In 2017, this propeller was tested in the cavitation tunnel of the Shanghai Jiao Tong University (SJTU) as a part of a collaborative study between the University of Strathclyde (UoS) and SJTU. The model propeller was manufactured using a 3.41 scale factor, resulting in model scale diameter of 0.22 m for SJTU model propeller. The main particulars of the propeller and the CAD model of it were supplied by the UoS [15]. The cavitation test matrix was composed according to the test conditions that have been prepared and published before in literature as a part of a round-robin test campaign [16]. Accordingly, 12 different conditions were determined with 2 different J (0.4 & 0.5) and 3 different σ_V values (13.9, 8.1 and 4.5-5.5) using J and K_T similarities. CFD simulations of each condition were conducted simultaneously to allow validation of the computational model for an isolated propeller. Figure 2 shows the comparisons between EFD and CFD results for only one condition at J=0.4 and σ_V=8.1 (Condition 2) where the strongest tip vortex cavitation was observed and with sheet cavitation rolling up to the tip vortex. This condition was selected for comparison with CFD results due to the behavior of the strong tip vortex cavitation. The CFD

*Corresponding Author, Naz Yilmaz: naz.gorener@strath.ac.uk

predictions of the cavitation patterns were created using a volume of vapour (VOF) value of 0.1. The comparison results showed good agreement in terms of propeller performance coefficients -deviation is 2.5% and 5% for K_T and K_Q respectively- as well as the sheet and tip vortex cavitation patterns as shown in Figure 2.

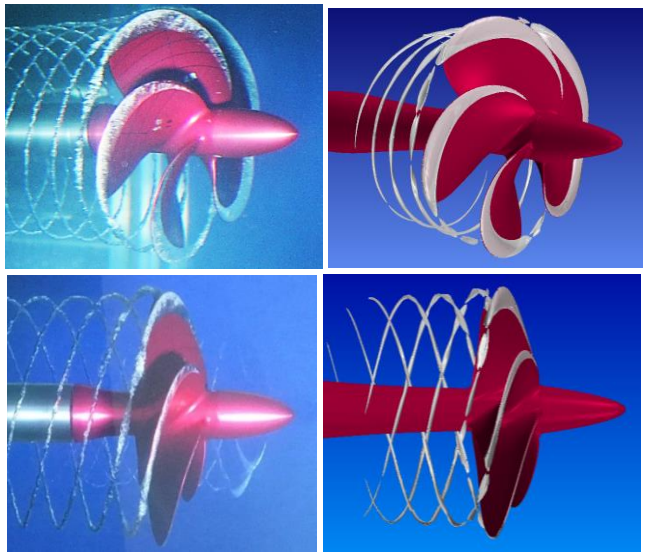

Figure 2. Cavitation comparisons between EFD and CFD Results (Left; EFD, Right;CFD)

Following this achievement in simulating the tip vortex cavitation of the Princess Royal Propeller in uniform flow, the study was extended to simulations in non-uniform flow in the presence of a rudder in order to investigate propeller-rudder interaction with cavitating tip vortices. Thus the Princess Royal Propeller with the rudder arrangement in non-uniform hull wake was simulated using the same mesh refinement approach (MARCS) to allow the tip vortices to interact with the rudder. While the sliding mesh method was used for the case of an isolated propeller, the "overset mesh" method was preferred in order to eliminate interface problems between the rotating and stationary domains for the propeller-rudder interaction simulations.

Figure 3 shows cavitation pattern comparisons between the EFD and CFD results including sheet and tip vortex cavitation. The EFD results were obtained from cavitation tests conducted in the Emerson Cavitation Tunnel, University of Newcastle, as presented in the literature [17]. It can be observed that although the same mesh refinement approach was used for the propeller-rudder interaction case, the CFD prediction of the tip vortex cavitation could not be extended as far as in the isolated propeller simulations (Figure 2 and 3) and hence currently presenting a discrepancy compare to the EFD results.

Bearing in mind the differences in physical conditions for the isolated propeller and the propeller-rudder arrangement cases, which include shaft inclination, non-uniform flow (wake screen) and the presence of a rudder for the latter case, the discrepancy between the two simulation cases can be investigated using Figure 4. In this figure, CFD simulations at 35000, 60000 and 70000 iterations are presented (from Left to Right) while the simulation was running and the generated mesh is rotating with the propeller. At 35000 iterations, a new mesh was generated using the MARCS method as described above. During the simulation, other images from 60000 and 70000 iterations were captured as seen in Figure 4. It can be observed that the generated mesh at 35000 iterations cannot match with the tip vortices that are produced at subsequent time steps in the simulation (i.e. 60000 and 70000 iterations) for the same shape at 35000 iterations. While the vortices were produced for each blade position and each time step and the propeller is rotating with inclined shaft, in non-uniform flow and presence of a rudder, the mesh generated at the beginning was no longer suitable during the subsequent solution time. Thus, tip vortex cavitation cannot be extended for propeller-rudder interaction simulation as much as in the isolated propeller simulations. Although the latter simulations are still under development during the preparation of this paper, so far the tip vortex cavitation simulations have been encouraging for evaluating the propeller performance in cavitating conditions interacting with the rudder in its slipstream.

*Corresponding Author, Naz Yilmaz: naz.gorener@strath.ac.uk

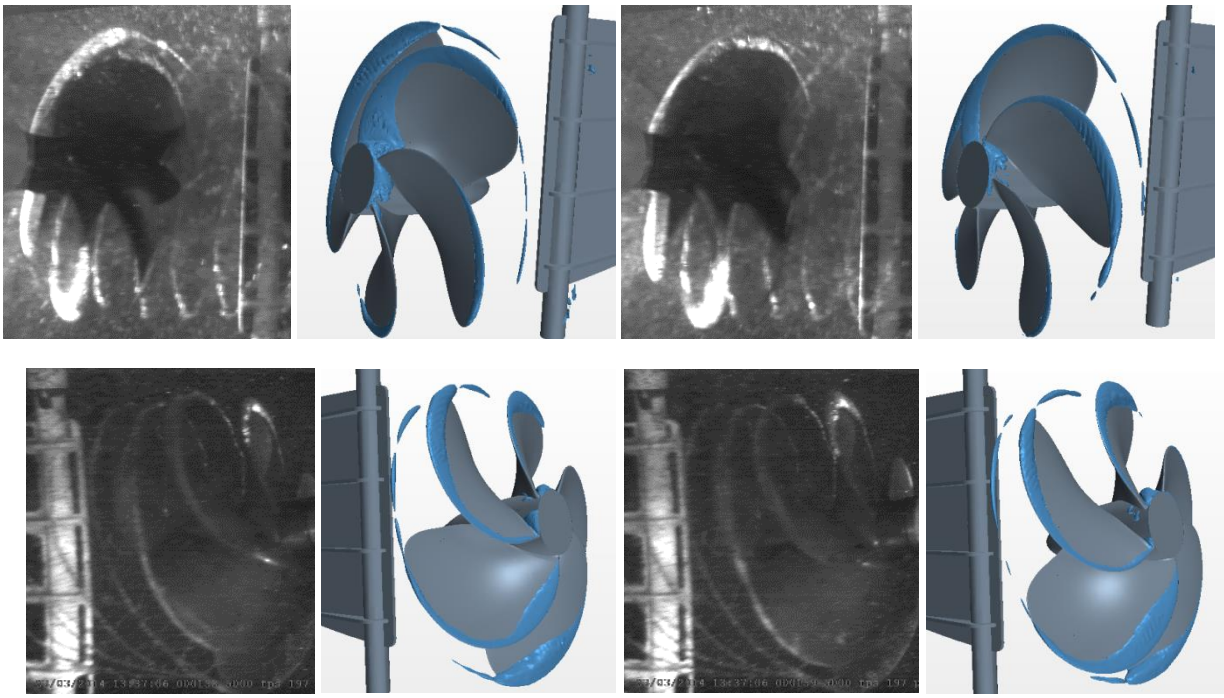

Figure 3. Cavitation comparisons between EFD and CFD Results (Left; EFD, Right;CFD)

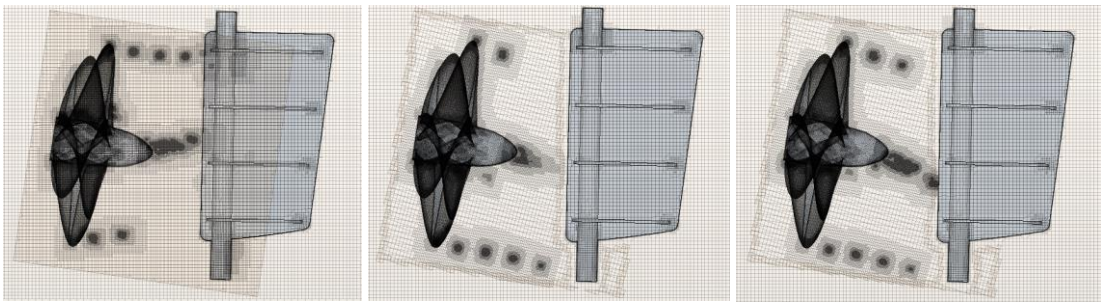

Figure 4. Adaptive Mesh Refienement (MARCS) during solution time (From Left to Right; Mesh from 35000, 60000 and 70000 iterations)

Conclusion

This paper has presented a new and efficient Mesh Adaption and Refinement approach for Cavitation Simulation (MARCS) of marine propellers, particularly for tip vortex cavitation. Successful simulations on an isolated propeller were achieved for the extension of the cavitating tip vortices well downstream of the propeller plane using the new approach such simulations. Further, the Princess Royal propeller was simulated using the same method in non-uniform flow and in the presence of a rudder. The simulation results so far for the latter case present a discrepancy compared to the EFD results and hence require further development in order to extend the tip vortex cavitation through the rudder. It is hoped, in the near future, to achieve further improved simulations of the propeller, rudder and hull flow interaction leading to more accurate prediction of the performance of propellers in cavitating conditions both at model and full-scale which is the ultimate aim of this study.

Acknowledgments

The principal author of this paper is sponsored by the Turkish Ministry of Education during this study. For CFD simulations, the access provided to High Performance Computing for the West of Scotland (Archie-West) through

*Corresponding Author, Naz Yilmaz: naz.gorener@strath.ac.uk

EPSRC grant no. EP/K000586/1 is gratefully acknowledged. The authors also thank to Prof. Chen-Jun Yang providing opportunity to the principle author for attending the cavitation tunnel tests in SJTU cavitation tunnel.

References

[1] Fine, N. E. 1992, Nonlinear Analysis of Cavitating Propellers in Nonuniform Flow, PhD thesis, Department of Ocean Engineering, MIT.

[2] Lee, H., 2002, Modeling of Unsteady Wake Alignment and Developed Tip Vortex Cavitation, PhD Dissertation, University of Texas at Austin.

[3] Hsiao, C. T., Pauley, L. L., 1998, Numerical Computation of Tip Vortex Flow Generated by a Marine Propeller, 1998 ASME Fluids Engineering Division Summer Meeting FEDSM'98 June 21 - 25, 1998 Washington D.C.

[4] Lee, H., Kinnas, S. A., 2004, Application of a Boundary Element Method in the Prediction of Unsteady Blade Sheet and Developed Tip Vortex Cavitation on Marine Propellers, Journal of Ship Research, Vol. 48, No. 1, March 2004, pp. 15–30.

[5] Hsiao, J. T., Chahine. G.L., 2008, Scaling of Tip Vortex Cavitation Inception for a Marine Open Propeller, 27th Symposium on Naval Hydrodynamics Seoul, Korea.

[6] Gaggero, S., Tani, G., Viviani, M. and Conti, F., 2014, A Study on The Numerical Prediction of Propellers Cavitating Tip Vortex, Ocean Engineering 92(2014)137–161.

[7] Star CCM+ User Guide, 2018

[8] Rayleigh, Lord, 1917, Phi/os. Mag. 34: 94-98

[9] Plesset, M. S., Prosperetti, A., 1977, Bubble Dynamics and Cavitation, Ann. Rev. Fluid Mech. 1977. 9: 145-85

[10] 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

[11] Kuiper G., 1981, Cavitation Inception on Ship Propeller Models, Wageningen.

[12] Yilmaz, N., Khorasanchi, M., Atlar, M., 2017, An Investigation into Computational Modelling of Cavitation in a Propeller's Slipstream, Fifth International Symposium on Marine Propulsion smp'17, Espoo, Finland, June 2017

[13] 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

[14] ITTC Report, 2017, Specialist Committee on Hydrodynamic Noise Final Report and Recommendations to the 28th ITTC.

[15] 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, Jan.

[16] Tani, G., Aktas, B., Viviani, M., Atlar, M., 2017, Two medium size cavitation tunnel hydro-acoustic benchmark experiment comparisons as part of a round robin test campaign, Ocean Engineering 138 (2017) 179–207.

[17] Aktas, B., Atlar, M., Turkmen, S., Shi, W., Sampson, R., Korkut, E., Fitzsimmons, P. Propeller cavitation noise investigations of a research vessel using medium size cavitation tunnel tests and full scale trials, Ocean Engineering Volume 120, 1 July 2016, Pages 122-135.

*Corresponding Author, Naz Yilmaz: naz.gorener@strath.ac.uk