VOL. 12, NO. 4, FEBRUARY 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

A CFD RANS CAVITATION PREDICTION FOR PROPELLERS

Iwan Mustaffa Kamal[1] and Tuan Muhammad Amier Tuan Mohd Yusof[2]
[1]Universiti Kuala Lumpur, Malaysian Institute of Marine Engineering Technology, Lumut, Malaysia
[2]IHC Engineering Malaysia Sdn. Bhd., Johor Bahru, Malaysia
E-Mail: iwanzamil@unikl.edu.my

ABSTRACT

Cavitation is a general fluid mechanics phenomenon which can occur whenever a liquid is used in a machine which induces pressure and velocity fluctuations in the fluid. The analysis of the cavitation of a screw propeller normally done in a cavitation tunnel is highly complex, difficult and time consuming. With the advances of the computer, the numerical prediction of cavitations can be done quite faster compared to the tedious manual experimentation in a cavitation tunnel. Nowadays new orientations in analysis of propeller in steady and unsteady flow are Computational Fluid Dynamics (CFD) method: using Reynolds Averaged Navier-Stokes (RANS) solver. The flow around propeller can be derived from the equations of motions using boundary conditions. Therefore the inception of the cavitation can be predicted. Therefore this paper main aim is to present the application of CFD in simulating the cavitation behaviour of ship propellers. The propeller chosen for this study is the Gawn KCA series. The code selected for the CFD in this project is FINE™/TURBO. The results of the simulated cavitation in CFD were validated with existing experimental data. The sheet cavitations at the suction side of the propeller blade modelled by the CFD were found to be in close agreement with the existing experimental results. It was found that the CFD somehow failed to model the tip vortex cavitation occurring at the tip of the propeller blades.

Keywords: CFD, propeller, RANS, cavitation.

INTRODUCTION

Cavitation has been a challenging battle for a naval architect and propeller designers alike, since the year 1897 when Charles Parson encountered problems with propeller cavitation in his newly designed steam powered vessel named 'Turbinia' [3]. Cavitation on ship propellers can cause propeller thrust breakdown, noise, vibration and serious erosion. It is a primary importance for a propeller designer to reduce the chances of cavitation occurrence by changing the propeller geometry such as the chord length and the blade area ratio. One of the tools necessary to study propeller cavitation is by using a cavitation tunnel. The study of propeller cavitation in a cavitation tunnel is highly complex, difficult and time consuming. With the advances of the computer, the numerical prediction of cavitation can be done quite faster compared to the tedious manual experimentation in a cavitation tunnel.

With the advances of the computer technology, more and more of the propeller cavitation analysis are done using Computational Fluid Dynamics (CFD). The analysis of a propeller in steady and unsteady flow using Reynolds Averaged Navier-Stokes (RANS) method can be conducted.

Examples of this application can be found in Salvatore *et al.* [12], where results from the Rome 2008 Workshop on cavitating propeller modelling were presented. Seven computational models by RANS, LES and BEM were benchmarked against a common test case addressing the INSEAN E779A propeller in uniform flow and in a wake field. Salvatore el al highlighted that for the cavitating flow in a steady flow discrepancies in cavity extent were observed. In the case of a propeller operating in a non-uniform flow, difficulties to correctly model the inflow to the propeller were observed.

Vaz *et al*. [16] presented the state-of-the-art in cavitation simulation for the cavitation prediction of the INSEAN E779A propeller in open water condition and in behind condition. This study was a part of the Cooperative Research Ships SHARCS project which was performed by ten different institutions using eight different flow codes, ANSYS®CFX®, ANSYS Fluent®, Excalibur, FINE™/Marine, OpenFOAM®, PROCAL, ReFRESCO and Star CCM+®. The cavitation patterns simulated by all these codes were compared with photographs of blade cavitation obtained from experiments conducted in cavitation tunnel. Vaz *et al.* reported that the predicted cavity extent is in good agreement with the photographs. However there were large differences in the cavity size between the codes. This was probably due to the different turbulence modelling used between the codes.

Abdel-Maksoud [1] conducted a case study using a commercial CFD code, CFX-TASC flow from AEA Technology. The propeller used in the case study was the Potsdam Propeller Test Case (PPTC) propeller of SVA Potsdam. The study was done for 15 cavitation numbers. The cavitation pattern simulated using the code showed a good agreement with the experimental study carried out in the cavitation tunnel of the Potsdam Model Basin. However, there were some difficulties in simulating the cavitation above σ=6.42, especially at the back of the propeller blade. The reason for this was the coarse grid used at the back of the propeller blade, and it was not possible to predict the correct pressure reduction in the propeller slip stream.

Other examples can be found in Sipilä and Martio [15] in their work at VTT Technical Research Centre of Finland. The work was carried out under the EU's 6th Framework Programme project VIRTUE. The cavitation simulation was carried out using a general purpose RANS

solver FINFLO developed by Helsinki University of Technology in the late 1980s [14]. The Fluid Dynamics Unit of Aalto University in Finland further developed this code by implementing a cavitation model in the code. Sipilä and Martio conducted two case studies, one using the INSEAN E779 propeller test case and the other using the PPTC propeller of SVA Potsdam. Sipilä and Martio highlighted that the sheet cavitation simulated in FINFLO occurred earlier compared to the results from cavitation tunnel experiments. Sipilä and Martio concluded that this could be attributed to laminar separation of the flow at the leading edge of the model propeller in the tunnel tests. Other work which is related to cavitation prediction using CFD can be found in Bosschers *et al.* [2], Guilmineau *et al.* [6], Ji *et al,* Kimura *et al.* [7], Lindau *et al*. [9] and Sanchez-Caja [13].

In this paper a different code was used to simulate the cavitation inception on the propeller blade. A commercial CFD RANS code FINE^{TM}/TURBO was used. The work presented in this paper is focused to address the accuracy of this RANS solver to predict the cavitation using a case study. In this case study a well-known propeller series, Gawn-KCA series, was used, as there is abundance of cavitation results from cavitation tunnel testing. A comparison was made with the CFD simulated results against results from the tunnel testing. The differences between these two results were discussed. The author was unable to make a case study using a much more recent propeller test case such as the INSEAN E779A propeller and the SVA Potsdam Test Case (PPTC) Propeller, due to unavailability of the cavitation data from INSEAN and SVA Potsdam at the time.

PROPELLER PARTICULARS

The propeller chosen for this case study was taken from three bladed Gawn-Burrill KCA systematic propeller series. The propeller chosen was a KCA420 propeller with an expanded area ratio of 0.65 and a pitch ratio of 2.0. This systematic series of flat-faced segmental section propellers were open-water tested by Gawn in 1953 [4]. Gawn and Burrill [5] further tested these systematic series in 1957 in a cavitating environment. The main reason in choosing this propeller for this case study is that these propellers are still widely used for high-speed vessel where at high speed the propeller is highly susceptible to cavitation. A detailed explanation on the Gawn KCA series can be found in Mitchell *et al.* [10].

Table-1. Principal characteristics of the 3 bladed Gawn-KCA series propeller – KCA420.

Parameter	Model scale
Propeller diameter (mm)	240
Number of blades (-)	3
Pitch ratio (-)	2.0
Skew angle at blade tip (deg)	0
Rake (deg)	0
Expanded area ratio (-)	0.65
Hub diameter (mm)	28
Hub length (mm)	37.5
Rotation	Right

The three-dimensional propeller geometry was developed using PropCAD^{TM}, a propeller modelling software and Autodesk Inventor, a CAD modelling software. The principal characteristics of the KCA propeller are presented in Table-1. The propeller geometry was converted into IGES file and imported to AutoBlade^{TM}. The AutoBlade^{TM} is the interface for the blade geometry input within the FINE^{TM}/TURBO software.

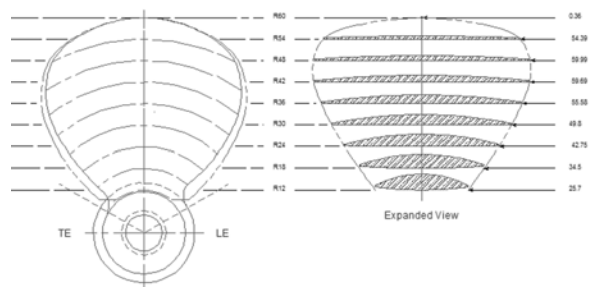

Figure-1. The developed (left – dotted line), projected (left – solid line) and expanded view (right) of the KCA420 propeller with the blade section at each radii.

NON-DIMENSIONAL FORM FOR PROPELLER

In the KCA propeller experiments conducted by Gawn and Burrill [5], the propellers were tested in over a range of advance coefficient J and cavitation number σ_V. In the KCA experiments, the free stream cavitation number was used, σ_V = 6.3, 2.0, 1.5, 1.0, 0.75 and 0.5.In the experiments the tunnel velocity V_T were held constant whilst the measurements of the thrust, torque and shaft revolution were made over a range of advance coefficient. The non-dimensional cavitation number and the advance coefficient are as defined in Equation (1) and Equation (2) respectively:

$$\sigma_V = \frac{p_O - p_V + \rho g h}{\frac{1}{2}\rho (V_T)^2} \quad (1)$$

$$J = \frac{V_T}{nD} \quad (2)$$

where (p_O – p_V) is pressure at the propeller centreline, p_O is the atmospheric pressure at sea level, at 101300 N/m^2, $p_V$is the vapour pressure of water, at 1700 N/m^2 at 15°C, ρ is the sea water density, g is the gravitational acceleration at 9.81 m/s, h is head of water at the propeller centreline, in metre, V_T is the tunnel velocity in m/s, n is the propeller shaft revolution in rev/secs and D is the diameter of the propeller, in metre.

The cavitation number σ_V = 1.0 was chosen for this study. Three advance coefficient were chosen J = 1.10, 1.40 and 1.60 as shown in Table-2. All these test conditions were replicated in the CFD cavitation simulation.

Table-2. Test condition chosen for the case study.

Case	1	2	3
Cavitation number σ_V	1.0	1.0	1.0
Advance coefficient J	1.10	1.40	1.60

COMPUTATIONAL WORK

A numerical analysis using a commercial finite volume based CFD software FINE^{TM}/Turbo from NUMECA [11]was carried out to determine the propeller cavitation severity. This CFD code is a RANS flow solver with the capability to solve steady and unsteady flow for all types of fluids, incompressible or fully compressible. Within this CFD suite, a geometry modelling module, AutoBlade^{TM} and a grid generator module, AutoGrid^{TM} are available for modelling and grid generation respectively.

The numerical analysis using this code consisted of a model replicating the experimental work done in Gawn and Burrill.

FLUID DOMAIN AND GRID GENERATION

The grid generation was done by using the most practical way which is using the AutoGrid^{TM} module, which is based on the surface imported from AutoBlade^{TM} module as described earlier. In this module, the structured grids were generated automatically once the geometry of the blade was imported to the module. A few settings were required before any grids were generated such as blade row type configuration, periodicity or number of blades, rotation speed, spanwise grid point number and wall cell width. The generated grid on the propeller blade is shown in Figure-2.

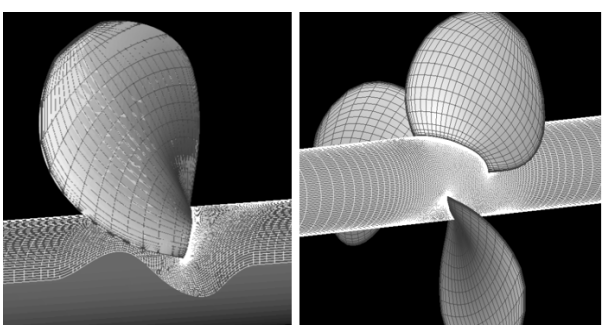

Figure-2. (Left) The mesh of the propeller blade at the pressure side and (Right) the mesh of the propeller blade at the suction side. Note for the resolution of the mesh at the propeller blade and the hub. Local mesh refinement was made at the hub area and the blade at the edge region.

No shroud control was used in this set-up, as shroud control is meant for turbo machinery casing definition i.e. compressor casing and turbines stator housing. Therefore no rotor-stator interaction is used in this setting. The inlet for the flow domain was located at a distance of three times the diameter of the propeller from mid of the chord of the propeller blade root section. The outlet was located at a distance of four times the diameter of the propeller from the same point at downstream. In radial direction domain, the far-field boundary was considered up to a distance of four times the diameter of the propeller from the axis of the propeller hub.

Local refinement of the grid cell was needed near to the blade wall. The grid sizes was generated in such a way than cell sizes near the blade wall were small and the cell sizes increased towards the outer boundary as shown in Figure-3. This is necessary as it was expected that a large pressure gradient will occur near to the wall. Therefore a denser group of grids were required near to the blade wall as to capture the flow properties with significant quality. Figure-4 shows the grid over the entire domain and propeller used for cavitation simulation using FINE^{TM}/Turbo.

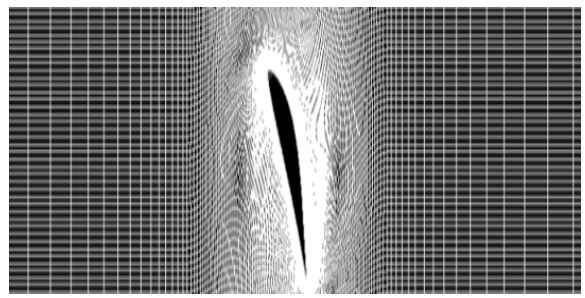

Figure-3. Local refinement of the grid showing finer resolutions was made at the region in the proximity to the blade. A coarser grid was used in the far-field region.

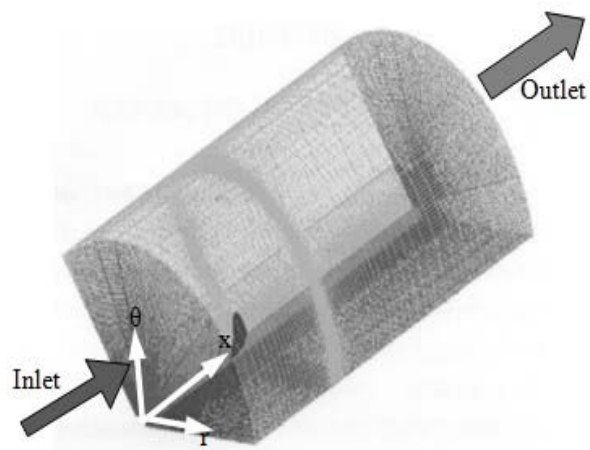

Figure-4. The fluid domain over the entire one propeller blade and the cylindrical coordinate system used in the CFD computation.

A grid independence study was conducted with the grids refined until the results are independence to the grid cell sizes. The total numbers of cells generated for the entire grid were3million. Modelling one third of the fluid domain containing one propeller blade is sufficient to solve the entire flow domain. The effect of the other blades was taken care by imposing periodic boundary condition on meridional planes at the two sides of the one third sections.

RANS SOLVER SETTINGS

The CFD code FINE^{TM}/TURBO was used to solve the three dimensional viscous incompressible flow. The grid unit of the propeller in the CFD computation was set in SI unit in meter. The coordinates system chosen for the computation was cylindrical coordinate system represented by x, r and θ as shown in Figure-4. The computations were set to begin with a non-cavitating model, and then continued in sequence of a cavitating model at vapour pressure of 0 N/m^2, 500 N/m^2, 1000 N/m^2 and 1700 N/m^2. This step of computation starting with a non-cavitating model and the vapour pressure increased step by step was necessary as to ensure a smooth convergence of each partial differential equation of the grids.

Parameter configurations settings in FINE^{TM}/TURBO consisted of configurations of the fluid model, flow model, rotating machinery input, boundary conditions, numerical model, initial solution, and output selection. In the fluid model settings in FINE^{TM}/TURBO, water with a density of 1000 kg/m^{3}was chosen as an incompressible fluid in the fluid model library in FINE^{TM}/TURBO.In the flow model configuration settings, the mathematical model used was the Turbulent Navier-Stokes model with Spallart-Allmaras (Allmaras and Spallart) as the turbulence model. A steady-state condition was chosen in the flow model for the cavitation computation.

At the inlet boundary and at the external boundary, the velocity component of the given inflow speeds calculated by the advance coefficient were imposed to the flow model. On the blades and the hub of the propeller, no-slip conditions and a wall-function were used. On the blades and on the hub, the average y+ was below 0.6. A periodic boundary was imposed to the flow model and a courant number of 1 was used for the flow model.

VALIDATION WITH EXPERIMENTAL TEST RESULTS

A detailed validation of the cavity extent was made to the three cases of the CFD simulations, against experimental results of the Gawn-KCA 420 propeller tested by Gawn and Burrill[5] in King's College Cavitation Tunnel, Newcastle in 1955.The cavitation patterns simulated by FINE^{TM}/TURBO were compared with photographs of blade cavitation obtained from experiments conducted in the King's College cavitation tunnel. The validations were focused only at the blade which is at the top dead centre position. All observations were made only at the suction side of the blade.

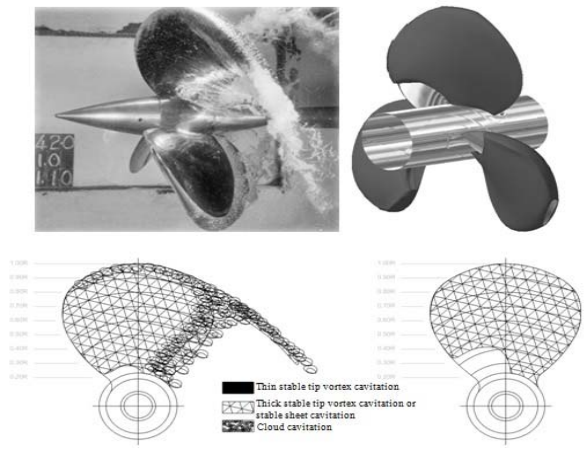

Figure-5. (Top)Case 1 – Cavitation number σ_V = 1.0 and advance coefficient J = 1.10, (Left) Experiments (Right) CFD. (Bottom) The sketches according to the ITTC procedure - (Left) Experiments (Right) CFD.

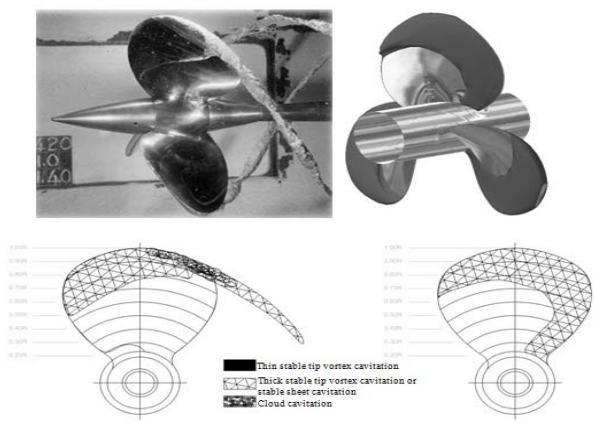

Figure-6. Case 2 – Cavitation number σ_V = 1.0 and advance coefficient J = 1.40, (Left) Experiments (Right) CFD.(Bottom) The sketches according to the ITTC procedure - (Left) Experiments (Right) CFD.

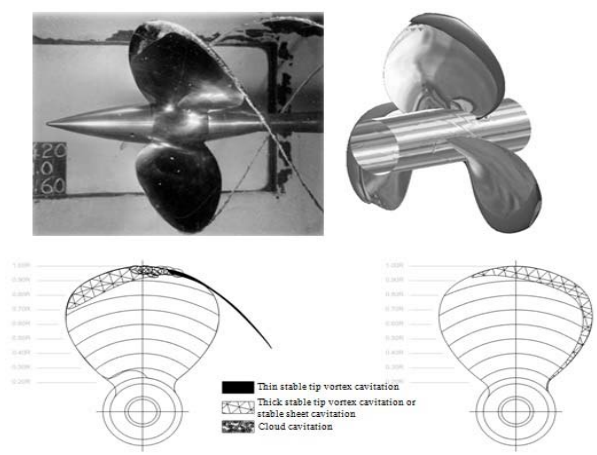

Figure-7. Case 3 – Cavitation number σ_V = 1.0 and advance coefficient J = 1.60, (Left) Experiments (Right) CFD. (Bottom) The sketches according to the ITTC procedure - (Left) Experiments (Right) CFD.

Figure-5 (Top) shows the computed cavity shape for case 1, Cavitation number σ_V = 1.0 and advance coefficient J = 1.10. These results were sketched following ITTC recommended procedure 7.5-02.03-03.2. The sketches are shown in Figure-5 (Bottom). The CFD simulation was able to predict the sheet cavitation which covered the suction side blade surface from 0.2R to 0.9R. The CFD simulation was also able to predict some root cavitation which is in agreement with the experiment photograph. However, the CFD simulation under predicted the sheet cavitation at somewhere around 0.2R to 0.3R at the leading edge side of the suction side blade surface. The CFD simulation was unable to simulate the cavity volume of the attached tip vortex cavitation, the detached tip vortex cavitation leaving the tip of the blade in the slipstream and some cloud cavitation occurred at the blade tip.

Figure-6(Top) shows the computed cavity shape for case 2, Cavitation number σ_V = 1.0 and advance coefficient J = 1.40.The sketchesof the results are shown in Figure-6 (Bottom). The CFD simulation was found to over predict the cavity extent especially at the trailing edge approximately around 0.7R to 0.2R and over predict the cavity extent at the propeller root. The simulation was unable to simulate the detached tip vortex cavitation and some cloud cavitation at the blade tipas shown in the experiment photograph.

Figure-7(Top) shows the computed cavity shape for case 3, Cavitation number σ_V = 1.0 and advance coefficient J = 1.60.The sketchesof the results are shown in Figure-7 (Bottom). The CFD simulation was able to simulate the tip vortex cavitation that extended from 0.9R to the edge of the tip, but slightly over predict the cavitation at the propeller root on the trailing edge side. The simulation was unable to predict some sheet cavitation at the leading edge of the propeller blade and some cloud cavitation at the blade tip. Again the simulation was unable to simulate the detached tip vortex cavitation which left the propeller blade approximately at 0.5R.

DISCUSSIONS

The failure of the CFD simulation to accurately predict the detached propeller tip vortices was due to an inappropriate grid resolution used in the region just after propeller tip. This is the challenging task of CFD simulations as the entire slipstream of the propeller cannot be filled with high resolution grids as this would increase the computation time excessively. Therefore, it is up to the naval architect or the propeller designer to choose the best grid resolution without undermining the accuracy of the simulation.

Inappropriate turbulence modelling is likely to be another cause of failure in capturing the tip vortex cavitation. Further research is needed to explore other turbulence model such as k-ε, k-ω and Shear Stress Transport (SST) model.

The predicted sheet cavitation on the suction side of the KCA propeller was more extensive in Case 2 than that observed in the experiment photographs. This discrepancy is likely to be caused by inappropriate turbulence modelling at the trailing edge.

CONCLUSIONS

The cavitating performance of a Gawn-KCA series propeller was numerically simulated using a RANS solver. The RANS solver used in this study was a commercial CFD code FINE^{TM}/TURBO. A turbulent Navier-Stokes numerical model was used along with Spallart-Allmaras turbulence modelling in the CFD computations.

The extents of the cavitation predicted by CFD for case 1 are in good agreement with the experiment photograph. The predicted sheet cavitation on the suction side of the KCA propeller was more extensive in Case 2 than that observed in the experiment photographs. The simulation was unable to predict some sheet cavitation at the leading edge of the propeller blade and some cloud cavitation at the blade tip, for all cases.

More complex types of cavitation, such as cloud cavitation are unlikely to be captured by the CFD simulation. The failure to capture these type of cavitation are likely cause by the grid resolutions and the turbulence modelling used in the computation.

The overall numerical results suggest the possibility of the cavitation model in the RANS solver to be used for practical applications in the propeller cavitation analysis as a complementary tool to the cavitation tunnel test. However further research is needed to overcome the issues in capturing the detached propeller tip vortices and cloud cavitation which is related to the turbulent characteristics and the grid resolutions.

NOMENCLATURE

D	diameter of the propeller
g	gravitational acceleration at 9.81 m/s
h	head of water at the propeller centreline
J	advance coefficient of propeller
n	propeller shaft revolution in rev/secs
p_O	atmospheric pressure at sea level, at 101300 N/m^2
p_V	vapour pressure of water, at 1700 N/m^2 at 15ºC
V_T	cavitation tunnel's speed
ρ	water density
σ_V	cavitation number

ACKNOWLEDGEMENTS

The author acknowledges the support from Universiti Kuala Lumpur through the funding of a short term research grant under the project "Cavitation and Evaluation using CFD for Standard Series Screw Propeller", Project ID: STR10024. The author would also like to acknowledge the support from NurulAsimaZainon and NorainiDaud of Universiti Kuala Lumpur, Malaysian Institute of Marine Engineering Technology for assisting in modeling the propeller.

REFERENCES

[1] Abdel-Maksoud, M. 2003. Numerical and Experimental Study of Cavitation Behaviour of a Propeller. Proceeding STG Sprechtag Cavitation. Hamburg, Germany.

[2] Bosschers, J., Vaz, G., Starke, A.R. and van Wijngaarden, E. 2008. Computational Analysis of Propeller Sheet Cavitation and Propeller-Ship Interaction. MARINE CFD2008, 26-27 March 2008, Southampton, UK.

[3] Carlton, J. 2007. Marine propellers and propulsion. Butterworth-Heinemann. Oxford, UK.

[4] Gawn, R.W.L.1953. Effect of Pitch and Blade Width on Propeller Performance. Trans INA, Volume 95, pp. 157-193.

[5] Gawn, R.W.L.and Burrill, L.C.1957. Effect of Cavitation on Performance of a Series of 16in. Model Propellers. Trans INA, Volume 99, pp. 690-728.

[6] Guilmineau, E., Deng, G.B., Leroyer, A., Queutey, P., Visonneau, M. and Wackers, J. 2015. Numerical Simulations of the Cavitating and Non-cavitating Flow around the Potsdam Propeller Test Case. Fourth International Symposium on Marine Propulsors smp'15. Austin, Texas, USA.

[7] Ji, B., Luo, X., Peng, X., Wu, Y. and Xu, H. 2012. Numerical analysis of cavitation evolution and excited pressure fluctuation around a propeller in non-uniform wake. International Journal of Multiphase Flow 43. Elsevier. pp. 13 -21.

[8] Kimura, K., Kawamura, T., Fujii, A., Taketani, T. and Huang, Z. 2009. Study on Unsteady Cavitating Flow Simulation around Marine Propeller using a RANS CFD code. Proceedings of the 7th International Symposium on Cavitation CAV2009, August 17-22, 2009. Ann Arbor, Michigan, USA

[9] Lindau, J.W., Moody, W.L., Kinzel, M.P., Dreyer, J.J., Kunz, R.F. and Paterson, E.G. 2009. Computation of Cavitating Flow through Marine Propulsors. First International Symposium on Marine Propulsors SMP'09. Trondheim, Norway.

[10] Mitchell, G.H.G., Sampson, R. and Atlar, M. 2013. A modern approach to the representation and use of the KCA systematic propeller series. Third International Symposium on Marine Propulsors smp'13. Launceston, Tasmania, Australia.

[11] NUMECA. 2011. FINE™/TURBO User Manual. Numeca. Belgium.

[12] Salvatore, F., Streckwall, H. and van Terwisga, T. 2009. Propeller Cavitation Modelling by CFD – Results from the VIRTUE 2008 Rome Workshop. First International Symposium on Marine Propulsors smp'09. Trondheim, Norway.

[13] Sanchez-Caja, A. 1998. P4119 RANS calculations at VTT, 22nd ITTC Propeller RANS/Panel method Workshop. France.

[14] Siikonen,T., Hoffren, J., Laine, S., 1990. A Multigrid LU Factorisation Scheme for the Thin-Layer Navier-Stokes Equations. Proceedings of the 17th ICES Congress, Vol. 90. Stockholm, Sweden.

[15] Sipilä, T. andMartio, J. 2008. FINFLO Analysis of Cavitating and Non-cavitating propeller in Uniform and Non-uniform flow. VIRTUE WP4 Workshop II on Propeller RANS Calculations. Rome, Italy. pp. 29 -30.

[16] Vaz, G., Hally, D., Huuva, T., Bulten, N., Muller, P., Becchi, P., Herrer, J.L.R., Whitworth, S., Mace, R. and Korsstrom, A. 2015. Cavitating flow calculations for the E779A propeller in open water and behind conditions: Code comparison and solution validation. Fourth International Symposium on Marine Propulsorssmp'15, Austin, Texas, USA.