Available online at www.elixirpublishers.com (Elixir International Journal)

Mechanical Engineering

Elixir Mech. Engg. 90 (2016) 37695-37699

ISSN: 2229-712X

Propeller Cavitation Analysis using CFD

I.Manoj Krishna[1], R.Alekhya and B.Srikanth[1]

[1]Sri Aditya Engineering College, Surampalem, Kakinada

[2]Sri Sai Aditya Institute of Science and Technology, Surampalem, Kakinada

ARTICLE INFO

Article history:
Received: 21 April 2015;
Received in revised form: 8 January 2016;
Accepted: 13 January 2016;

Keywords
Propeller,
CFD,
Cavitation,
Multi phase flows,
Open water characteristics.

ABSTRACT

Cavitating flows are highly complicated because it is a rapid phase change phenomenon, which often occurs in the high-speed or rotating fluid machineries. It is well known that the cavitating flows rise up the vibration, the noise and the erosion. Therefore, the research on the cavitating flows is of great interest. Numerical method is highly important approach for studying the cavitating flow.The propeller is the predominant propulsion device used in ships. The performance of propeller is conventionally represented in terms of non-dimensional coefficients, i.e., thrust coefficient (KT), torque coefficient (KQ) and efficiency and their variation with advance coefficients (J). It is difficult to determine the characteristics of a full-size propeller in open water by varying the speed of the advance and the revolution rate over a range and measuring the thrust and torque of the propeller. Therefore, recourse is made to experiments with models of the propeller and the ship in which the thrust and torque of the model propeller can be conveniently measured over a range of speed of advance and revolution rate. Experiments are very expensive and time consuming, so the present paper deals with a complete computational solution for the flow using STAR-CCM+ software. When the operating pressure was lowered below the vapor pressure of surrounding liquid it simulates cavitating condition. In the present work, STAR-CCM+ software is also used to solve advanced phenomena like cavitation of propeller. The simulation results of cavitation and open water characteristics of propeller are compared with experimental predictions, as obtained from literature [1].

Introduction

A marine propeller is a propulsion system which turns the power delivered by the engine into thrust to drive the vessel through water. Propeller cavitation is a general problem encountered by the ship owner, whereby it causes vibrations, noises, degradation of propeller performance, deceases engine efficiencies, effects the life cycle of the ship and also results in high cost of maintenance. The basic physics of cavitation occurs when the pressure of liquid is lower or equal to the vapour pressure, which depends on the temperature, thus forming cavities or bubbles. The compression of pressure surrounding the cavities would break the cavities into smaller parts and this increases the temperature. Collapse of bubbles in contact with parts of the propeller blades will create high localized forces that subsequently erode the surface of the blades. Simulation on cavitating flow using CFD can be carried out to determine the performance of the propeller. A model is generated in CATIA and fluid-flow physics are applied to predict the fluid dynamics and other physical phenomena related to the propeller. Ref. [2] stated that, CFD can provide potential flow analysis such as flow velocities and pressure at every point in the problem domain as well as the inclusion of viscous effects.

During recent year's great advancement of computer performance, Computational Fluid-Dynamics (CFD) methods for solving the Reynolds Averaged Navier-Stokes (RANS) equation have been increasingly applied to various marine propeller geometries. While these studies have shown great advancement in the technology, some issues still need to be addressed for more practicable procedures. These include mesh generation strategies and turbulence model selection. With the availability of superior hardware, it becomes possible to model the complex fluid flow problems like propeller flow and cavitation.

Sanchez-Caja [3] has calculated open water flow patterns and performance coefficients for DTRC 4119 propeller using FINFLO code. The flow patterns were generally predicted with the k-ε turbulent model. He has suggested a better prediction of the tip vortex flow, which requires a more sophisticated turbulence model. Bernad [4] presented a numerical investigation of cavitating flows using the mixture model implemented in the Fluent 6.2 commercial code. Senocak et al. [5] presented a numerical simulation of turbulent flows with sheet cavitation. Sridhar et al. [6] predicted the frictional resistance offered to a ship in motion using Fluent 6.0 and these results are validated by experimental results.

Salvatore et al. [7] performed computational analysis by using the INSEAN-PFC propeller flow code developed by CNR-INSEAN. Experiments are carried to know the open water performance, evaluation of velocity field in the propeller wake and prediction of cavitation in uniform flow conditions. Bertetta et al. [8] presented an experimental and numerical analysis of unconventional CLT propeller.Two different numerical approaches, a potential panel method and RANSE solver, are employed. Zhi-feng and Shi-liang [9] studied the cavitation performance of propellers using viscous multiphase flow theories and with a hybrid grid based on Navier-Stokes and bubble dynamics equations. Pereira et al. [10] presented an experimental and theoretical investigation on a cavitating propeller in uniform inflow. Flow field investigations by advanced imaging techniques are used to extract quantitative

Tele:
E-mail addresses: i.manojkrishna@gmail.com

information on the cavity extension. Pereira and Sequeira [11] developed turbulent vorticity-confinement strategy for RANS-based industrial propeller-flow simulations. The methodology aims at an improved prediction of tip vortices, which are an origin of cavitation.

The numerical or experimental analysis and comparison of results highlight the peculiarities of propellers, the possibility to increase efficiency and reduce cavitation risk, in order to exploit the design approaches already well proven for conventional propellers also in the case of unconventional geometries. The simulated flow pattern agrees with the experimental data in most cases. However, the detailed shape of the wake behind the propeller blades is not captured. The present methodologies give in local disagreement with the experimental data, especially around blade wake and tip vortex. However, in order to clear the reason of these disagreements, more study using other turbulence models or other mesh patterns is necessary.

So in the present paper, the CFD STAR-CCM+ software is used to solve advanced phenomena like cavitation of propeller. The investigation is based on standard K-Є turbulence model in combination with a volume of fluid implementation to capture the interface between liquid and vapour. The open water characteristics of a propeller are estimated in terms of the advance coefficient J, the thrust coefficient KT, the torque coefficient KQ and the open water efficiency η0 in both non cavitating and cavitating condition of propeller. The simulation results of cavitation and open water characteristics of propeller are compared with experimental predictions, as obtained from literature [1].

Description of Model Propeller Vp1304

The propeller is a controllable pitch propeller. This affects the propeller blade design near the hub and results in a 0.3 mm gap between hub and propeller blade near the leading and trailing edge of the propeller.. The trailing edge for the upper propeller radii is sharp. The propeller was designed to generate a tip vortex. Two coordinate systems are specified. The ship coordinate system (SCS) is used for the open water tests and corresponds to the usual coordinate system. For the velocity measurements (LDV) a second coordinates system was introduced, the propeller coordinate system (PCS). The intention was to measure the axial velocities inline with the positive x-axis and thereby obtain positive axial velocities.

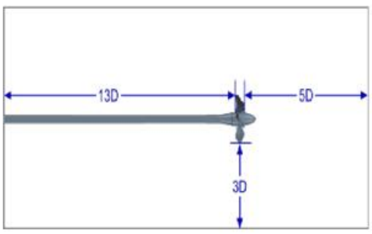


Figure 1.PPTC Model Propeller VP1304

Figure 2 Domain for full propeller simulations

Grid Generation

The flow domain is required to be discritized to convert the partial differential equations into series of algebraic equations. This process is called grid generation. A solid model of the PPTC propeller was taken. The complexity of the blade and complete domain is shown in Fig.2.

STAR-CCM+ was used for Unstructured grid with tetrahedral cells. The inlet was considered at a distance of 5D (where D is diameter of the propeller) from mid of the chord of the root section. Outlet is considered at a distance of 3D from same point at downstream. In radial direction domain was considered up to a distance of 13D from the axis of the hub. This peripheral plane is called far-field boundary. The mesh was generated in such a way that cell sizes near the blade wall were small and increased towards outer boundary. Fig. 3 shows the grid over the entire domain and propeller used for flow and cavitaton simulations using STAR-CCM+.

After convergence total number of cells generated for entire grid was 6.3 million. It is clearly shows that denser mesh is near propeller surface to capture the flow properties with significant quality.

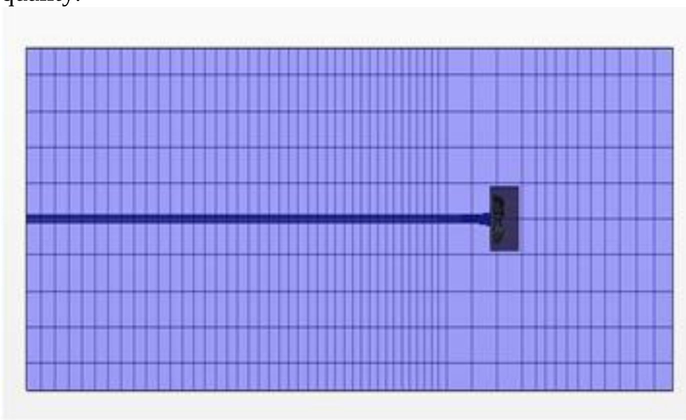
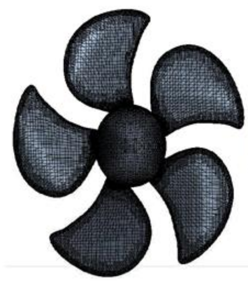


Figure 3. Grid over the entire domain



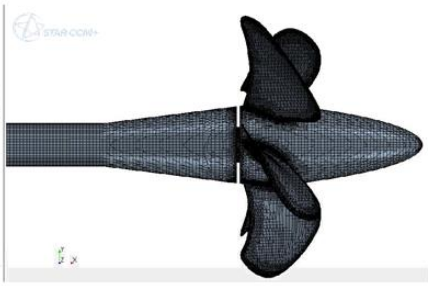


Figure 4

Figure 5. Grid over the propeller

The numerical mesh is an unstructured grid which comprises tetrahedral basic cells and prismatic cells for resolving the boundary layer around the surface. In this analysis, Standard K-Є turbulence model was used and $y+ < 1$ is highly recommended to use this model. Therefore, the first layer meshes around the surface were generated to satisfy this recommendation

Boundary Conditions

The continuum was chosen as fluid and the properties of water were assigned to it. A moving reference frame is assigned to fluid with a rotational velocity (9.98rps, 14.98rps, 15.026rps). The wall forming the propeller blade and hub were assigned a relative rotational velocity of zero with respect to adjacent cell zone. A velocities 0.39m/sec, 2.00m/sec, 3m/sec, was prescribed at inlet. At outlet outflow boundary condition was set. The far boundary (far field) was taken as Inviscid wall and assigned an absolute rotational velocity of zero. Fig. 6 shows the boundary conditions imposed on the propeller domain.

Numerical Model

STAR-CCM+ software was used for all simulations presented in this paper. STAR-CCM+, which is the finite volume solver for general purpose, implements low-Reynolds number type turbulence models, cavitation models, a moving mesh method etc.

In this paper, we used the Standard K-Є model, which can simulate transitional flows, for analyses of open water tests.

Additionally, we used the full-cavitation model for the simulations of cavitating flows.

Flow Solution and Solver Settings

The CFD code STAR-CCM+ was used to solve the three dimensional viscous incompressible flow. The Software simultaneously computes the flow equations using multiple processors. The software can automatically-partition the grid into sub-domains, to distribute the computational job between available numbers of processors. Table2 and Table 3 shows the details of Open water tests and cavitating details of the flow respectively.

Results and Discussion

Analysis of Open Water Tests

The open water tests conditions and the propeller model geometry were given by SMP'11 workshop. Table 1 shows the main characteristics and propeller shape of PPTC which is the propeller for the calculation. The analysis conditions are listed in Table 4.

Figure 6 shows the computational domain which comprises the inner rotational part containing the propeller, and the outer stationary part whose size is the same as the size of the towing tank. The inner rotational part and outer stationary part connect discontinuously. The inlet/outlet boundary is in the stationary part, and the constant velocity and zero pressure conditions are applied to it. In this analysis, the simulation is operated as the steady-state analysis. The inner rotational part is not actually rotated. Instead, the rotation is converted to the force which is applied to the part.

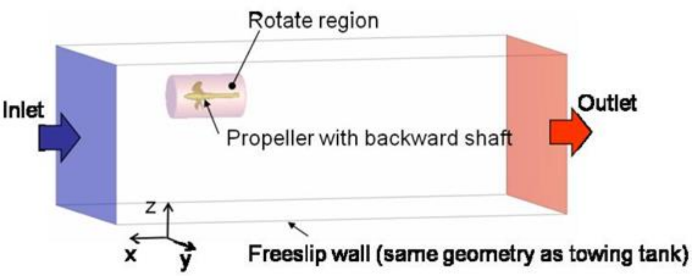


Figure 6. Computational domains of open water tests

The performance of propeller is conventionally represented in terms of non-dimensional coefficients, i.e., thrust coefficient (KT), torque coefficient (KQ) and efficiency and their variation with advance coefficients (J). A complete computational solution for the flow was obtained using STAR-CCM+ software. The software also estimated thrust and torque from the computational solutions for different rotational speeds (rps) of the propeller. These were expressed in terms of KT & KQ. The estimated thrust and torques are shown in Table 4. Comparison of estimated non-dimensional coefficients and efficiency (η) against experimental predictions, as obtained from literature [1], are shown in Table 5.It shows the comparison of predicted KT & KQ with experimental data [1]. It shows that KT and KQ coefficients are decreasing with increasing of advance coefficients (J). Maximum efficiency is observed at J = 0.8.

Fig. 8 and Fig. 9 shows the velocity vectors at r/R= 0.70 for advance velocity 2.00m/s, 9.98rps, J=0.80 and velocity vectors at r/R= 0.75 for advance velocity 4.13m/s, 9.98rps, J=1.657 respectively. From the two figures it is clearly observed that there is no flow separation near the blade surface at every radial section, which was expected as the propeller was a well designed standard one.

Fig. 10 and Fig. 11 show the pressure distribution on surface of impeller blades in terms of pressure coefficient at advance velocity 2.00m/s, rotational speed 9.98rps & advance coefficient J=0.80 and at advance velocity 4.13m/s, rotational speed 9.98rps & advance coefficient J=1.65 respectively. The face and back are experiencing high pressure and low pressure respectively. However when propeller was operating at very low rpm it is not able to generate thrust, so a reverse trend in pressure was observed. This explains the development of thrust by propeller at high rotations whereas the propeller is contributing to resistance. It is evident that there is a concentration of high-pressure region near the leading edge of the propeller.

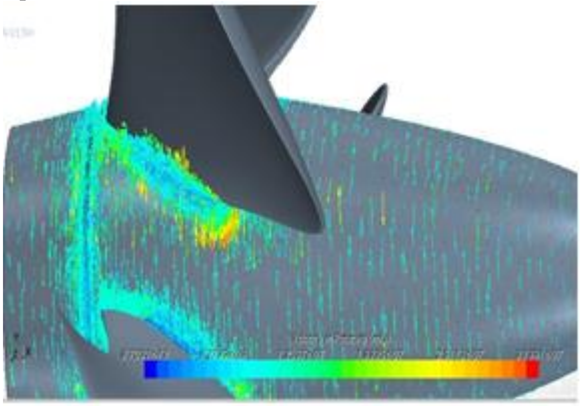


Figure 8. Velocity vectors at r/R= 0.264 for advance velocity 2.00m/s, 9.98rps, J=0.80

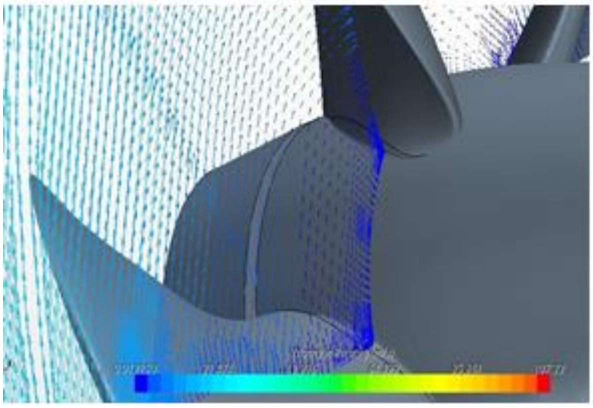


Figure 9. Velocity vectors at r/R = 0.704 for advance velocity 4.13m/s, 9.98rps, J=1.657

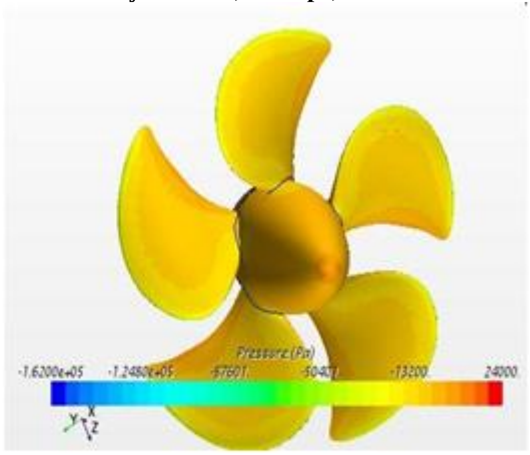


Figure 10. Pressure distribution on the surface of the blades in terms of pressure coefficient at advance velocity 2.00m/s, 9.98rps, J=0.80

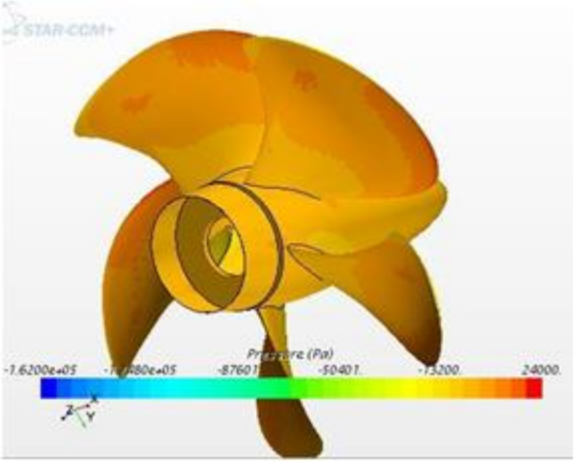
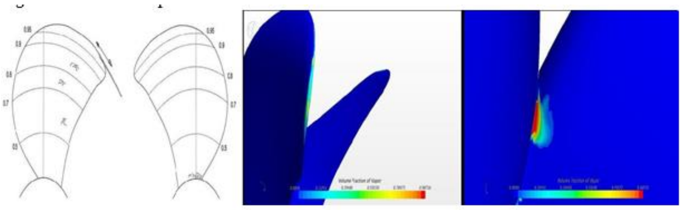


Figure 11. Pressure distribution on the surface of the blades in terms of pressure coefficient at advance velocity 4.13m/s, 9.98rps, J=1.657

Propeller Under Cavitation

When the operating pressure was lowered below the vapor pressure of surrounding liquid it simulates cavitating condition. In this condition two phases, water and water vapour are considered in simulations with STAR-CCM+. Table 6 shows the comparison between the performance of the propeller in cavitating and non-cavitating conditions. The cavitation number for this cavitating condition is 1.88, which is fairly high, and so the propeller is marginally cavitating and not heavily cavitating. Because of this only a small drop in thrust coefficient was observed in Table 6, when the torque demand was increased slightly.

Fig. 13 shows the contour of pressure coefficient in cavitation condition. When compared with pressure in distribution under non-cavitating conditions in Fig. 10 and Fig. 11, it is slightly increased in cavitating condition as shown in Fig. 13. The pressure is expected to remain constant over the cavitating part of the blade. But some change in pressure distribution is observed when propeller started cavitating. However, the phenomenon of constant pressure in the cavitating region was not observed clearly in the Fig. 10 and Fig. 11. This may be because of the fact that cavitation has just initiated or the computational solution could not capture the phenomenon properly. From this it is observed that the volume fraction is varying from 0 to 0.9. It clearly shows that water got vaporized in particular area and this particular portion of the propeller blade is made to cavitate. Thus it reduces the thrust generated by the propeller and slight increase the torque demand



(a) Experiments [1] (b) Simulations

Figure 12. Development of cavities on propeller blade and comparison between CFD and experiments [1]

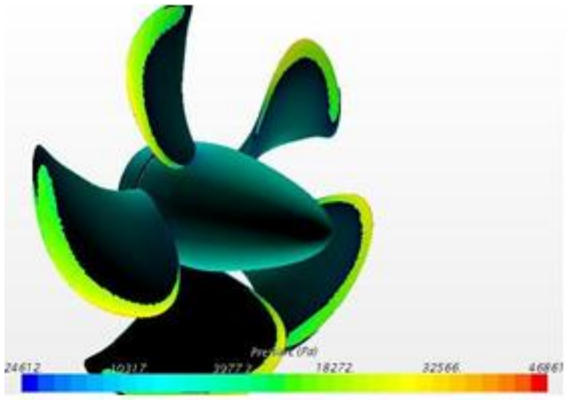


Figure13. Contour of pressure coefficient in cavitation

Conclusions

Based on foregoing analysis it is concluded that

- Computation results are in good agreement with experimental findings [1].
- Commercial CFD code like STAR-CCM+ can solve open water characteristics of propeller with reasonable accuracy. Estimations are very close to that off experimental results.
- CFD and commercial code STAR-CCM+ can be used to solve advanced phenomena like cavitation. In view of the complexities involved, the present result of cavitation and their agreement with experiment is very encouraging. However more detailed studies and validations of cavitating propeller for different cavitation numbers are to be taken up to establish reliability of CFD for this type of studies.

References

[1] Potsdam Propeller Test Case (PPTC) Open Water Tests and Cavitation tests Case 2.1 Ulf Barkmann, Potsdam Model Basin (SVA)

[2] J. Lundberg. 2009. Propelling a More Efficient Fleet[online]. Rolls-Royce Marine.http://www.ansys.com/magazine/vol3-iss2-2009/rolls-royce.pdf

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

[4] Bernad, S. 2006. Numerical analysis of the cavitating flows, Center of Advanced Research in Engineering Sciences, Romania Academy, Timisoara Branch, Romania.

[5] Senocak, I. and Shyy, W. 2001. Numerical simulation of turbulent flows with sheet cavitation, Department of Aerospace Engineering, Mechanics and Engineering Science, University of Florida, Florida.

[6] Sridhar, D, Bhanuprakash, T. V. K., and Das, H. N. 2010. Frictional resistance calculations on a ship using CFD, Int. J. of Computer Applications, Vol. 11, No.5, pp 24-31.

[7] Salvatore, F., Greco, L. and Calcagni, D. 2011. Computational analysis of marine propeller performance and cavitation by using an inviscid-flow BEM model, Second International Symposium on

Marine Propulsors, smp'11, Hamburg, Germany.

[8] Bertetta, D., Brizzolara, S., Canepa, E., Gaggero, S. and Viviani, M. 2012. EFD and CFD characterization of a CLT propeller, Int. J. of Rotating Machinery, Vol. 2012, Article ID 348939, 22 pages, doi:10.1155/2012/348939.

[9] Zhi-feng ZHU and Shi-liang FANG, 2012. Numerical investigation of cavitation performance of ship propellers, J. of Hydrodynamics, Ser. B, Vol. 24, No. 3, pp 347–353.

[10] Pereira, F., Salvatore, F., and Di Felice, F. (2004). Measurement and modeling of propeller cavitation in uniform

[11] inflow, J. of Fluids Engineering, Vol. 126, pp 671-679.
[11] Pereira J. C. F. and Sequeira, A. 2010. Propeller-flow predictions using turbulent vorticity-confinement, V European Conference on Computational Fluid Dynamics, ECCOMAS CFD 2010, Lisbon, Portugal.

Nomenclature

J Advance coefficient (=V/n.D)
KT Thrust coefficient ($=T/\rho.n^2.D^4$)
KQ Torque coefficient ($=Q/\rho.n^2.D^5$)
η_0 Open water efficiency ($=J.K_T/2.\Pi.K_Q$)
σ_n Cavitation number ($= (P-P_V)/(0.5.\rho.n.D^2)$)