Indian Journal of Science and Technology, *Vol 9(45), DOI: 10.17485/ijst/2016/v9i45/104693, Decemeber 2016*
*ISSN (Print) : 0974-6846*
*ISSN (Online) : 0974-5645*

# Performance Evaluation of Marine Propeller using Numerical Simulation

**Veeranagouda Patil[1], H. R. Purushothama[1], A. Manjunatha[2] and Vijay Kumar Mishra[2]**

[1]Siddaganga Institute of Technology, Tumkur– 572103, Karnataka, India; veereshvpatil@gmail.com
[2]Contra Volts Infotech Pvt Ltd, Bangalore – 560016, Karnataka, India; manjunatha@contravolts.com

## Abstract

In present paper, the performance of the marine propeller under non-cavitating condition is evaluated numerically using Computational Fluid Dynamics (CFD) approach. The first objective of this paper is to review physics involved in bubble dynamics and prediction and control of cavitation that have been suggested in the literature. These mechanisms are evaluated with observations that are available from marine propeller, where cavitation has leads to erosion damage on the marine propeller. A second objective is to review physical model and its characteristics for the prediction of the risk of cavitation erosion. A detailed phenomenon of cavitation and description of the process leading to tip vortex cavitation and its effects on marine propeller performance is hypothesized.

**Keywords:** Cavitations, Marine Propeller, Numerical Simulation, Steady State Analysis

## 1. Introduction

Cavitation is phenomenon of change of phase from liquid to gaseous state. This phase change occurs due to vapour bubble formation. If the water pressure drops below the vapour pressure of surrounding fluid then that leads to cavitation. Behaviour of this phenomenon is periodic, means, initially vapour will form then it will grow into bigger size and collapse at high pressure region. This kind of phenomenon occurs usually in marine propeller, water pump and pipes of power plant. Cavitating flow leads to damage of material, vibration in a body, loss of power and efficiency. Cavitation involved in phase transition from liquid to gaseous state, which is as explained below.

### 1.1 Phase Transition

Water usually exist in three phases; solid, liquid and gas. The phase change from liquid and the phase change from vapor to water is called condensation. Cavitation and boiling phenomenon are different. If the phase change occurs due to increase in temperature is boiling. If the phase change occurs due to decrease in pressure, the phenomenon is called cavitation. In marine propeller, evaporation occurs by sudden drop of local pressure in water due to dynamic action of propeller. Cavitation is violent behavior of fluids due to its nature of rapid phase change process, which involved in the condensation and evaporation of cavitating bubbles. Then bubble will form small cavity based on vapor, this cavity again grows and collapse at high pressure region. The growth and collapse of bubbles emit very high pressure. Flow velocity is also causes cavitation if this increased by decreasing pressure by any dynamic action. Behavior of cavitation is described by the parameter is called cavitation number. It is defined by the following equation[1].

$$\sigma = \frac{(pa - pv)}{(1/2\ \rho\ v2)} \tag{1}$$

### 1.2 Bubble Dynamics

Bubbles are formed from small bubble filled with gas or vapor or it may combination of both. The region where dent is present, that region forms vapor cavity. The dust particle forms a nucleus to form a cavity. Hence cavitation does not incept in pure liquid[1].

### 1.3 Tip Vortex Cavitation

Tip vortex cavitation is one among the types of cavitation which occurs at blade tip and hub. Usually occurs near blade tip and hub, this type cavitation initially behind the

** Author for correspondence*

tip of the blade region and unattached to the blade tip, because velocity is increasing and pressure is dropping further. With lower cavitation number it will move to blade tip and attaches to it. Due to any dynamic action if there is any further reduction in pressure around blad surface, the tip vortex cavitation will cover little more area[2].

## 1.4 Cavitation Effects

Cavitating flow leads to damage of material, vibration in a body, loss of power and efficiency. Cavitation not only occurs due to drop in pressure but also occurs due to dissolved gases, dust, and dent present in water. Hence prediction and control of cavitation is very important aspect for rotating bodies in liquid. With development of high computational power made feasible to simulate this complex phenomenon. Numerical study on cavitating flow has been carried out from 1960 onwards for different propeller geometries using Reynolds Averaged Navier Strokes (RANS) solver. A study shows, CFD gives good understanding in comparison with experimental one, but requires more CPU-hour. Earlier days propeller performance were predicted by lifting line theory and later on, with advanced technology of computability makes to simulate cavitating flows effectively. In the present paper SC/Tetra v12 (Software Cradle Co., Ltd) CFD software is used.

R. Arazgaldi1, A. Hajilouy1 et al,2009 studied the behaviour of marine propeller in cavitating and non-cavitating flow condition both numerically and experimentally and suggested physical model to predict cavitation. The physical model is based on the full cavitation model by Singhal et.al. This model includes phase change, bubble dynamics and pressure fluctuations and also non-condensable gases[3].

Bin Ji, Xian-wu Luo, Yu-lin Wu, et al 2010, employed RANS, and simulation has carried out with turbulence model k-ω SST. In this simulation unsteady behaviour of cavitation is predicted numerically. Author observed that Due to the non-uniform wake inflow and gravity effect, there is an occurrence of periodic changes of cavitation inception, growth, shrinking etc. near the tip of the propeller. And also it has been observed that large pressure fluctuations near the propeller during operation[4].

Bio Haung et.al, 2010 evaluated unsteady characteristics of cavitating flows with different cavitation models and proposed density modify based cavitation model due to its compressibility nature[5].

Keita Fujiyama et.al, 2011, performance of the marine propeller using numerical simulation is studied in this paper. CFD code SC/Tetra v9 was used for the numerical analysis. To capture tip vortex cavitation fine mesh was used behind the tip region with $y^{+}<=30$. Author observed from the results that thrust and torque coefficient decreases in increase in advance coefficient. On the other hand the open water efficiency increases as advance coefficient increases[6].

## 1.5 Cavitation Physical Model

To calculate cavitating flows physical model is required, in order to predict better tip vortex cavitation and cavity pattern, proper model is selected for numerical simulation. From literature survey it is found that Singhal's full cavitational model gives better results. Characteristics of this model are discussed below[7].

This model considers formation of bubble, collapse of bubble and non-condensable gas:

$$\rho = \frac{dP(P+Pc)\rho}{K(1\text{-}Y\text{-}Y_g)P*(T+T_0)\rho_g + RY(P+Pc)T\rho_g + Y_g P(P+Pc)} \tag{2}$$

In full cavitation model condensation and vaporization term are solved in following equation.

$$\frac{\partial(\rho Y)}{\partial t} + \frac{\partial(\rho u_j Y)}{\partial x_j} = R_e\text{-}R_c \tag{3}$$

The evaporation term;

$$R_e = C_e\left(\frac{\sqrt{K}}{\sigma}\right)\rho_1\rho_v\sqrt{\frac{2}{3}\frac{P_v-P}{\rho_1}}\,(1\text{-}Y\text{-}Y_g) \quad \text{if } P < P_v \tag{4}$$

$$R_c = C_c\left(\frac{\sqrt{K}}{\sigma}\right)\rho_1\rho_1\sqrt{\frac{2}{3}\frac{P-P_v}{\rho_1}}\,(Y) \quad \text{if } P > P_v \tag{5}$$

Where,

$C_e = 0.02$ and $C_c = 0.01$ are model constant

$$P_v = P_a + 0.39\rho\frac{K}{2} \tag{6}$$

| Nomenclature | |
|---|---|
| D | Propeller diameter |
| J | Advance coefficient $J=\frac{Va}{nD}$ |
| N | Rotational speed |
| T | Thrust |
| Q | Torque |
| Z | Number of blades |

## 2. Numerical Methodology

SC/Tetra v12 commercial CFD software has been used for the numerical simulation, which employs finite volume method on unstructured mesh. The simulation is performed with RNG k-epsilon turbulence models. Performance of the marine propeller under non-cavitating condition is evaluated for different advance coefficient. Performance of marine propeller is measured in terms of thrust coefficient and torque coefficient. Mathematically, thrust coefficient and torque coefficients are as follows[6].

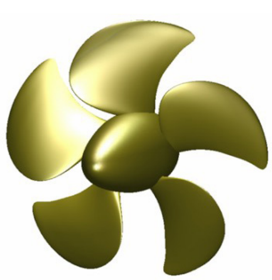
$$K_T = \frac{T}{\rho n^2 D^4} \tag{7}$$

$$K_Q = \frac{Q}{\rho n^2 D^5} \tag{8}$$

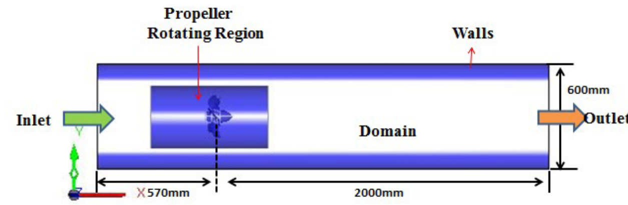
## 3. Computational Methodology

### 3.1 Geometric Modelling

A controllable pitch type of smp11 propeller model is used for the numerical analysis. Same experimental geometry is used for non-cavitating flow analysis. Propeller with five bladed having diameter D = 0.250m, skew angle 18.8 and pitch ratio at 0.7R 1.635. Thrust coefficient is evaluated for three advance coefficient. Geometry and computational domain are as same as experimental towing tank set up. Figure 1(a) shows Potsdam Propeller Test Case (PPTC) of smp11 and Figure 1(b) and Table 1 shows geometric model for cavitating and non cavitating cases. Rotating region is modelled with moving element condition and assigned specified speed of the propeller. Stationary and rotating parts are connected discontinuously using discontinuous mesh approach. Inlet is given with inflow velocity and outlet with static pressure zero Pascal. Free slip condition to walls of the computational domain[6].



**(a)**




**(b)**

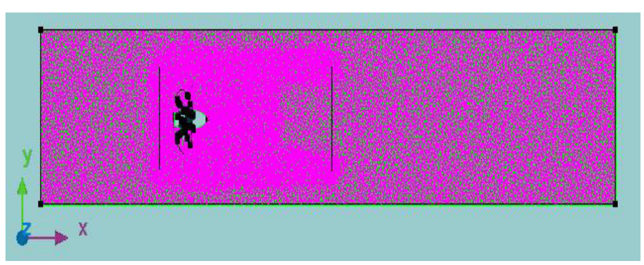
**Figure 1.** **(a)**. PPTC propeller. **(b)**. Computational domain (VP1304 Dipl.-Ing. H.-J. Heinke).

**Table 1.** Test conditions[6]

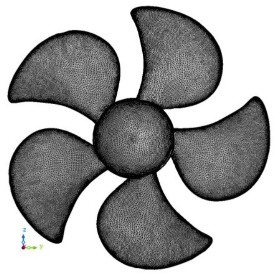
| Test Cases | | Case 1 | Case 2 | Case 3 |
|---|---|---|---|---|
| Rate of revolutions (1/s) | n | 24.897 | 24.986 | 25.014 |
| Water Temperature (degC) | T | 23.2 | 23.2 | 23.2 |
| Water Density (kg/m$^3$) | $\rho$ | 997.44 | 997.4 | 997.37 |
| Vapor Pressure (pa) | Pv | 2818 | 2818 | 2869 |
| Kinematic viscosity of water (m$^2$/s) | v | 9.34e-07 | 9.34e-07 | 9.73e-07 |
| Advance coefficients | J | 1.019 | 1.269 | 1.408 |
| Inlet Velocity (m/s) | $V_A$ | 6.365 | 7.927 | 8.805 |
| Thrust coefficient (non cavitating) | $K_T$ | 0.387 | 0.245 | 0.167 |
| Cavitation Number | σ | 2.024 | 1.428 | 2.000 |

### 3.2 Mesh Generation

The octree is generated initially; it converted into unstructured tetrahedral mesh using advance front method. Polygon feature was used for more mesh refinement. The region around tip of blade is refined more to capture pressure and viscous forces. The number of tetrahedral mesh elements was about 18 million. Prism layer are inserted to capture boundary layer phenomenon to maintain $y^+ <= 30$. Control of $y^+$ gives better results. The convergence criteria for all residual 1e-5 were used in the present study (Figure 2(a),(b)).




**(a)**

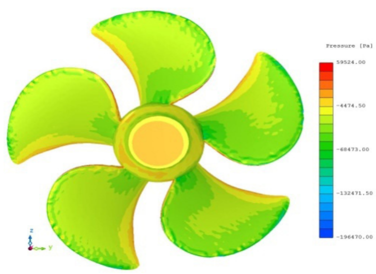


(b)

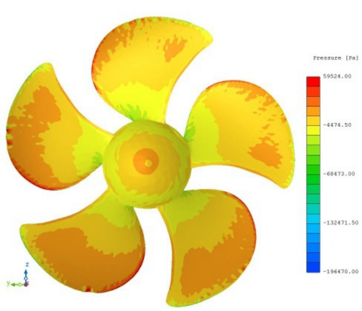
Figure 2. (**a**). Mesh Sectional view. (**b**) Mesh generation.

## 4. Results

Figure 3(a) Shows pressure distribution at pressure side of the blade surface and Figure 3(b) shows pressure distribution at suction side of the blade surface. Water at which cuts the blades surface at Blade tip, root and hub are with more pressure compared to other regions.



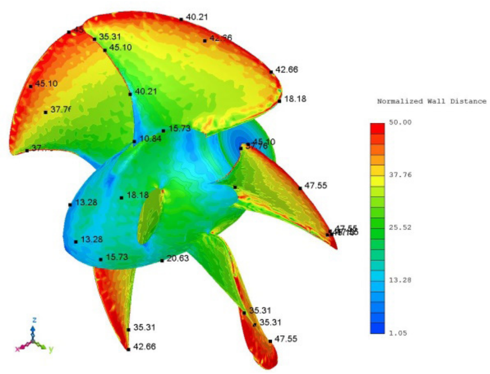
(a)



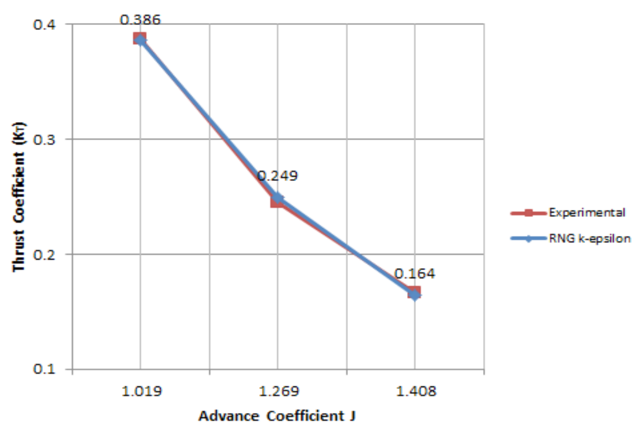
(b)

Figure 3. Pressure distribution at pressure side of the blade surface (b). Pressure distribution at suction side of the blade surface.

Figure 4(a) shows $y^+$ value for the current numerical simulation i.e 28.63 which indicates quality of the mesh elements are good to predict better results with RNG k-epsilon model. Figure 4(b) shows comparison of propeller characteristics in terms of thrust coefficient at different advance ratio between experimental and simulation results, shows are in good agreement. And also observed that thrust coefficient reduces as the advance coefficient increases. Thrust coefficient was calculated at advance ratio of 1.016, 1.269, and 1.408, as shown in Table 2, and percentage of error of difference in thrust coefficient is also tabulated. From Table 2 it is observed that numerical result shows good agreement with experiment result (Experimental results are available in literature survey[8]). Percentage of error between experimental and analysis result is less than 2% in all three cases.



(a)




(b)

Figure 4. (**a**). Normalized wall distance (**b**) Propeller performance.

Table 2. Predicted non-cavitating performance of propeller[10]

| Results | Thrust Coefficient ($K_T$) | | | Error (%) | | |
|---|---|---|---|---|---|---|
| | Case 1 | Case 2 | Case 3 | Case 1 | Case 2 | Case 3 |
| Experimental (non-cavitating) | 0.387 | 0.245 | 0.167 | | | |
| Analysis (RNG k-epsilon) | 0.386 | 0.249 | 0.164 | -0.2 | 1.6 | -1.82 |

## 5. Conclusions

A detailed description of the cavitation phenomenon and its process leading to tip vortex cavitation erosion is discussed. The detailed description is based on experiments and observations published in open literature. Steady state numerical simulation has been carried out on PPTC propeller. The simulation was carried under non-cavitating flow condition for different advance coefficient. Numerical result shows good agreement with experimental result. Research and study shows that RNG k-epsilon model gives better predictions due to its near wall turbulence effects. And also this model has better reattachment flow capability. Hence this model is chosen for cavitating flow analysis in future work.

## 6. Acknowledgements

I wish to express my deep sense of gratitude to Dr. H.R. Purushotham, Associate Professor, Department of Mechanical Engineering, SIT, Tumkur, for his suggestions and guidance given by him during entire period of this dissertation. I wish to express my sincere and heartfelt gratitude to Directors, Contra Volts Infotech Pvt. Ltd., Bangalore, for providing resource and support. I wish to express my deep sense of gratitude to Mr. Manjunatha A, Team Lead, Contra Volts Infotech Pvt. Ltd., Bangalore, for his supervision, through provoking discussions, and suggestions.I wish to express my regards and gratitude to CRADLE support, for his valuable suggestions and guidance.

## 7. References

1. Tuomas Sipilä VTT 2012 RANS analyses of cavitating propeller flows. ISSN 2242-1203. Available from: http://www.vtt.fi/publications/index.jsp
2. Eur Ing. Jeffrey Casciani-Wood HonFIIMS. An introduction to propeller cavitation. Feature article 2015 Jan14 by News Hound.
3. Arazgaldi1 R, Hajilouy A, Farhanieh B. Experimental and Numerical Investigation of Marine Propeller Cavitation. SCIENTIA ERANICA, Transaction B: Mechanical Engineering. 2009 Dec; 16(6):525-33.
4. Bin Ji 1, Xian-wu Luo, Yu-lin Wu, Shu-hong Liu, Hong-yuan Xu, Akira Oshima. Numerical investigation of unsteady cavitating turbulent flow around a full scale marine propeller. Elsevier Science Ltd 9th International Conference on Hydrodynamics 2010 Oct. 11-15, Shanghai, China.
5. Biao Huang, Guo-yu Wang, Hai-tao Yuan. A cavitation model for cavtating flow simulation. Elsevier Science Ltd, 9th International Conference on Hydrodynamics 2010 Oct. 11-15, Shanghai, China.
6. Keita Fujiyama, Chuel-Ho Kim, Daisuke Hitomi. Performance and Cavitation Evaluation of Marine Propeller using Numerical Simulations. Second International Symposium on Marine Propulsors smp11, 2011 Jun, Hamburg, Germany.
7. SC/Tetra user guide v. 12. Software Cradle Co., Ltd., Japan.
8. Hans Jürgen Heinke. Potsdam Propeller Test Case (PPTC) Cavitation Tests, Potsdam Model Basin (SVA), The Second International Symposium on Marine Propulsors 2011 – smp'11, April 2011, VP1304, Report 3753.