International Journal for Innovative Engineering and Management Research

PEER REVIEWED OPEN ACCESS INTERNATIONAL JOURNAL

www.ijiemr.org

IJIEMR Transactions, online available on 26th Dec 2021. Link :http://www.ijiemr.org/downloads.php?vol=Volume-10&issue=Issue 12

10.48047/IJIEMR/V10/ISSUE 12/26

TITLE: Numerical Modeling of Cavitation Patterns and Their Influence on Ship Propeller Design

Volume 10, ISSUE 12, Pages: 187-193

Paper Authors **C. Syamsundar, Lakshmipathi Yerra, G. Venkatasubabaiah**

International Journal for Innovative Engineering and Management Research
PEER REVIEWED OPEN ACCESS INTERNATIONAL JOURNAL
www.ijiemr.org

Numerical Modeling of Cavitation Patterns and Their Influence on Ship Propeller Design

C. Syamsundar[1*], Lakshmipathi Yerra[1], G. Venkatasubabaiah[2]

[1]Department of Mechanical Engineering, CMR Engineering College,
Hyderabad, Telangana 501 401, India
[2]Department of Mechanical Engineering,
MVSR Engineering College, Nadergul, Hyderabad, Telangana 501 510, India
[*]Corresponding author. Tel.: +91-98408 91252, syamsundariitm@gmail.com

Abstract: The cavitation phenomenon is an unpredictable issue and intriguing subject in fluid dynamics, and the investigation of cavitation structures around a ship propeller in a cavitation water tunnel for experimentation is very complicated and consumes a lot of time. In the present paper, the consequences of cavitation structures at design and off-design testing conditions are predicted by utilizing RANS equations in ANSYS CFX. A complete three-dimensional ship propeller is demonstrated to simulate cavitation on a screw propeller. From the literature, it is evident that the most severe off-design operating conditions are not accurately anticipated. In this paper, computational analyses were carried out on various cavitation numbers at both designed and off-designing conditions to validate the experimental results. From these results, we have observed that cavitating structures and tip vortex formation on the blades were observed with good accuracy by competing with experimental results.

Keywords: Cavitation structures Computational fluid dynamics Ship propeller

1. Introduction

Today, ships can be described in different methods, but most of them have fundamentally the same propulsion (Colley 2012; Chen 2015) as shown in Fig. 1. The back side of the blade, which is in motion direction, is always at very low pressures (Colley 2012). We all know that cavitation is a multiphase complex occurrence; due to this, three different definitions are given below (Knapp et al. 1970).

- When the static pressure of liquid reaches low pressures (vapor pressures) or below it (Coutier-Delgosha et al. 2003).
- The formation phase, growth and collapse phase of bubbles in a liquid medium (Young 1989).
- The collapse will take place in liquid at high pressures (Franc and Michel 2004).

The repeated collapse of these cavitation bubbles on the blade surface causes erosion, vibration and noise. Frank et al. (2007) studied the ship propeller and concluded that, by using CFD, it is still difficult to predict cavitation at high-pressure fluctuations. Chen (2015) presented a numerical simulation by using a commercial code STAR-CCM+. Kamal et al. (2017) modeled sheet cavitation at the suction side of the propeller and found that they are

International Journal for Innovative Engineering and Management Research
PEER REVIEWED OPEN ACCESS INTERNATIONAL JOURNAL
www.ijiemr.org

close to experimental results.

The thrust force and torque produced by the propeller are represented in non-dimensional numbers, and they mainly depend on the diameter of the propeller. (D), speed of rotation (n), advance velocity (V_A), acceleration due to gravity (g), dynamic viscosity (μ), fluid density (ρ) (Chen 2015).

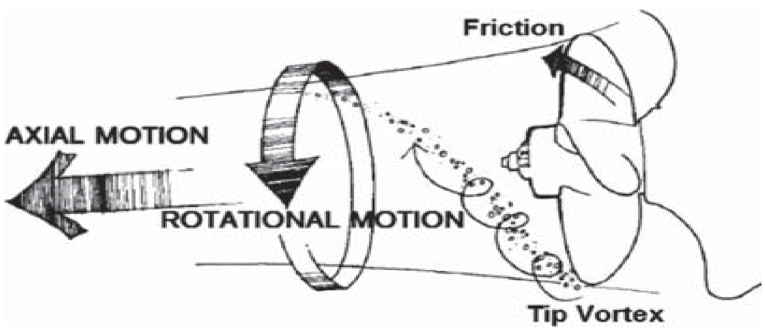

Fig. 1. Schematic diagram of a ship propeller (Chen 2015)

Therefore, the thrust can be expressed as

$$T = kD^a n^b V_A^c \rho^d \mu^e g^f \quad (1)$$

Where k is a proportionality constant, and a, b, c, d, e and f are property indexes. The *f*inal expression is

$$T = \rho n^2 D^4 . f_1\left(\frac{V_A}{nD}, \frac{nD}{v}, \frac{n^2D^2}{gD}\right) \quad (2)$$

There are three non-dimensional numbers in this equation.
Thrust coefficient k_T is defined as

$$k_T = \frac{T}{\rho n^2 D^4} = f_1\left(\frac{V_A}{nD}, \frac{nD^2}{v}, \frac{n^2D^2}{gD}\right) \quad (3)$$

Where $\frac{V_A}{nD}$ is coefficient of advance (J), $\frac{nD^2}{v}$ is Reynolds number (Re) and $\frac{n^2D^2}{gD}$ is Froude number (Fr). In general, the ship propeller is operated far away from the free surface of the liquid and doesn't produce any surface waves, so the Froude number can be ignored.

Till now, the cavitation effects have been studied by many researchers by experiments. It involves high costs due to the construction of cavitation water tunnels. Because of this, it is highly important to explore the CFD simulation techniques and cavitation models (Frank et al. 2007; Chen 2015). At the same time, in most of the papers, the most severe off-design operating conditions are not properly studied. In this paper, the CFD simulations are performed at the same experimental conditions and results are compared.

International Journal for Innovative Engineering and Management Research
PEER REVIEWED OPEN ACCESS INTERNATIONAL JOURNAL
www.ijiemr.org

2. Computational Analysis of a Ship Propeller

The ship propeller (Marine Trinket propeller) having 6 blades is modeled by using CATIA V5 R 19. The blade is modeled by taking several sections at various radii and is rotated through their respective pitch angles, as shown in Fig. 2(a). The computational domain considered for analysis is a cylindrical domain of length and diameter 23 m, which is ten times the propeller diameter and has a rotational speed of 1500 RPM as shown in Figs. 2(b) and 3.

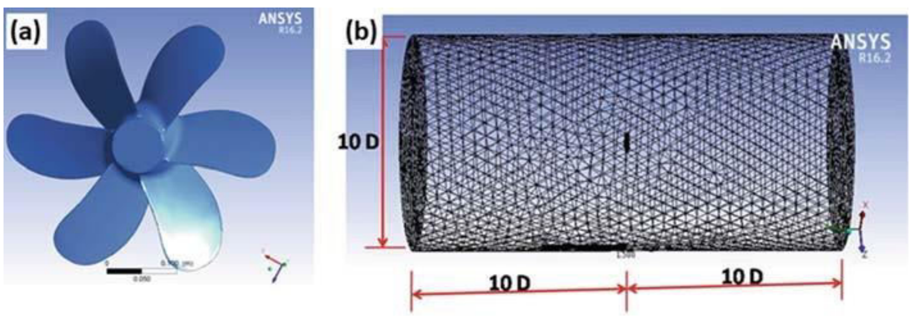

Fig. 2. (a) Modelled ship propeller, and (b) Generation of mesh refinement around the blades

In mesh generation, nearly 3 million hexahedral cells are generated, as shown in Fig. 3. Near wall $y+$ is very sensitive, and it is maintained as $y+ < 5$.

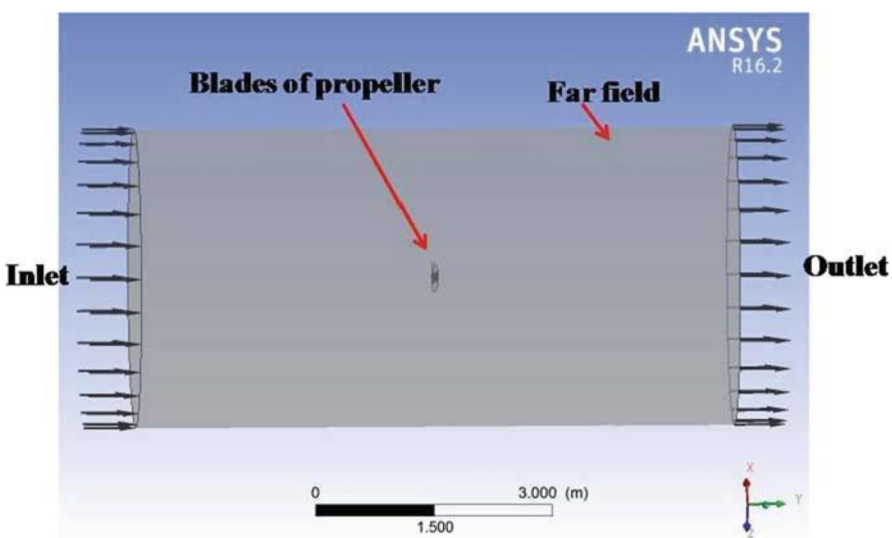

Fig. 3. Computational fluid domain considered and boundary conditions for CFD analysis

At the inlet boundary, a uniform velocity of 6.22 m/s was prescribed, and at the outlet, the atmospheric pressure was considered. Table 1 clearly shows the detailed solver control parameters for both non-cavitating and cavitating test conditions.

International Journal for Innovative Engineering and Management Research
PEER REVIEWED OPEN ACCESS INTERNATIONAL JOURNAL
www.ijiemr.org

Table 1. Details of solver control parameters

Parameters	Non-cavitating flow	Cavitating flow
Pressure and Velocity coupling:	SIMPLE	SIMPLE
Discretization scheme:	Upwind - Quadratic	First order upwind
Turbulence model:	$K-\varepsilon$	$K-\varepsilon$
Solver control:	Steady-state	Unsteady state Multiphase-Mixture 1. Water 2. Water vapor

3. Computational Results and Analysis

Computational simulations have been carried out at six different cavitation conditions (r), i.e., 7.2, 5.1, 3.7 (design condition), 2.9, 2.3, and 1.9, respectively. The corresponding inlet velocities are 5.2, 6.2, 7.2 (design condition), 8.2, 9.2 and 10.2 m/s. Theexperimental and computational results are compared at cavitation number 3.7 (Paik et al. 2013), as shown in Fig. 4, which shows a good agreement with experiments.

Figures 5 and 6(a–f) show the variation of pressure on the front and back sides of ship propellers. From these figures, we can clearly visualize that water got vaporized atparticular low-pressure regions, which causes cavitation and flow separation. By clear observation, it is also concluded that cavitation and flow separation mainly happen at the back side of the ship propeller and also at an outer blade location, as shown in Fig. 7 (a)–(f).

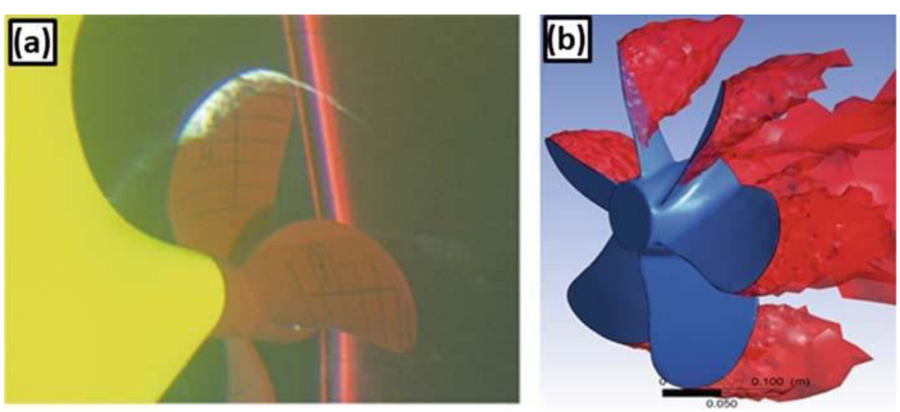

Fig. 4. Comparison between (a) Kwang-Jun Paik et al. 2013, experimental results and (b) Computational simulation outcomes at cavitation number 3.7

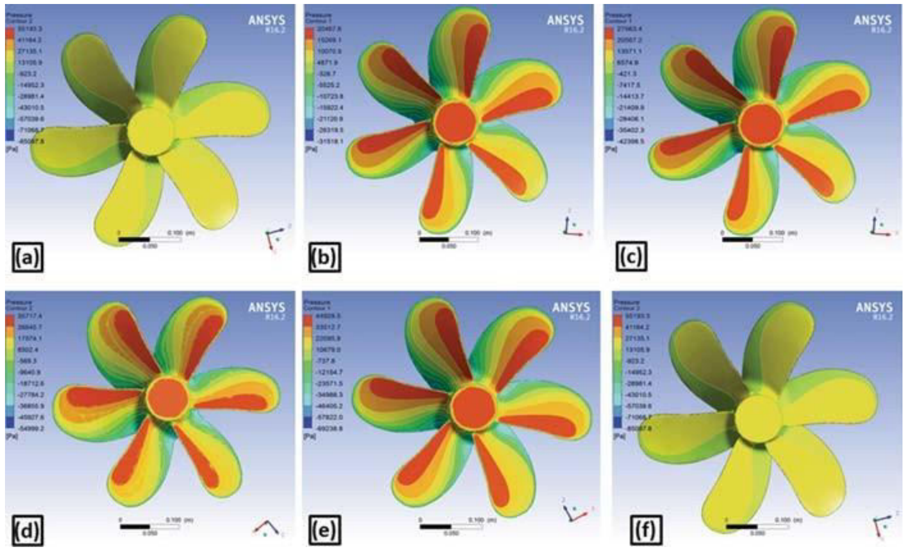

Fig. 5. Total pressure distribution at propeller inlet of cavitation numbers (a) 7.2, (b) 5.1, (c) 3.7(design condition), (d) 2.9, (e) 2.3 and (f) 1.9 at a rotational speed of 1500 RPM

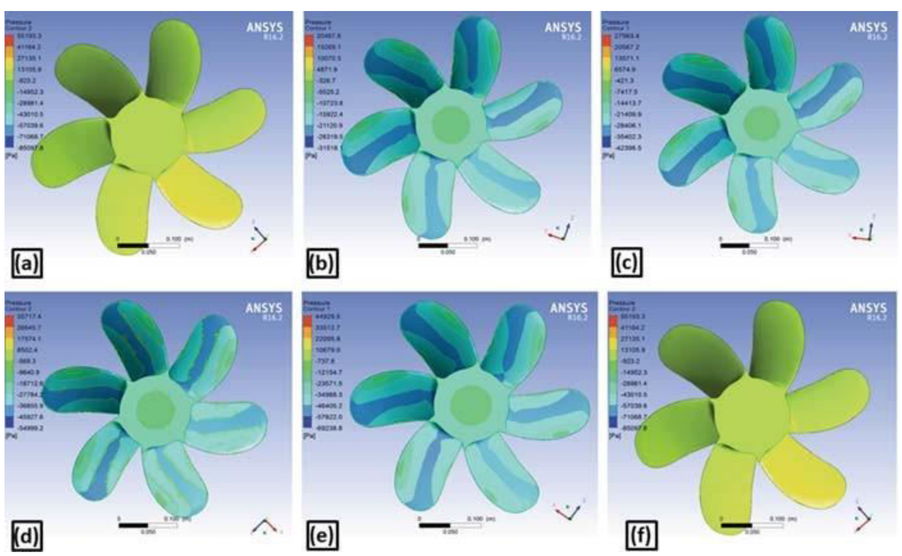

Fig. 6. Total pressure distribution on the propeller back side of cavitation numbers (a) 7.2, (b) 5.1, (c) 3.7 (operating condition), (d) 2.9, (e) 2.3, and (f) 1.9, at a rotational speed of 1500RPM

International Journal for Innovative Engineering and Management Research

PEER REVIEWED OPEN ACCESS INTERNATIONAL JOURNAL

www.ijiemr.org

Using experiments to calculate the cavity thickness is very complicated because we have to use image processing on the cavity surface. Using computational fluid dynamics, we can also obtain the cavity length for different cavitation numbers, as shown in Fig. 8(a)–(c). At higher cavitation numbers, there is a small attached cavitation observed compared with lower cavitation numbers.

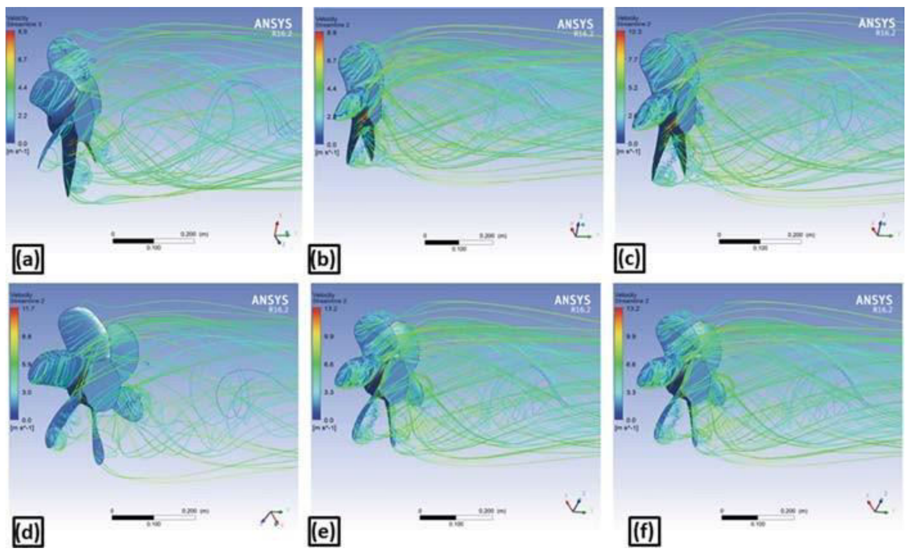

Fig. 7. Velocity streamlines of cavitation numbers (a) 7.2, (b) 5.1, (c) 3.7 (design condition), (d) 2.9, (e) 2.3, and (f) 1.9 at rotational speed of 1500 RPM

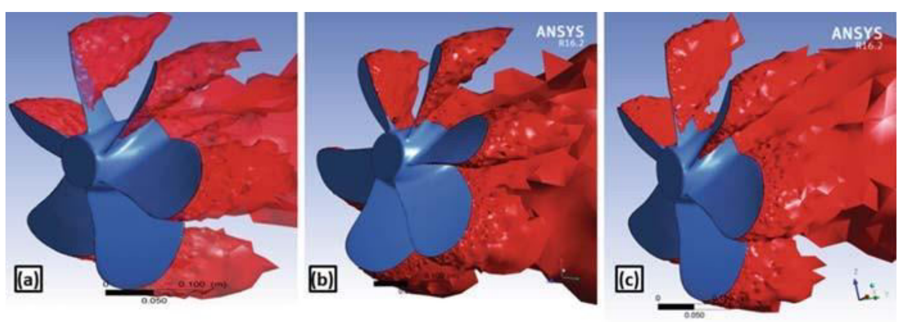

Fig. 8. Different types of cavitation structures on blades of a propeller operating at cavitation numbers (a) 7.2, (b) 3.7 (design condition), and (c) 1.9 at the rotational speed of 1500 RPM

4. Conclusions

From CFD simulations for different cavitation numbers (design and off-design condition), the main observations are;

1. Computational results are shown a very encouraging, good agreement with experiments, and they were reproducible.
2. At higher cavitation numbers, there is a small attached cavitation observed compared with lower cavitation numbers.

References

1. Morgut M, Nobile E (2012) Numerical predictions of cavitating flow around model scalepropellers by CFD and advanced model calibration

2. Sato K, Oshima A, Egashira H, Takano S (2009) Numerical prediction of cavitation and pressure fluctuation around the marine propeller

3. Kamal IM et al. (2017) A CFD RANS cavitation prediction for propellers

4. Ghose JP, Gokarn RP (2004) Basic ship propulsion. Allied Publishers Pvt. Limited

5. Knapp RT, Daily JW, Hammitt FG (1970) Cavitation. McGraw-Hill Book Company, London

6. Coutier-Delgosha O, Reboud JL, Delannoy Y (2003) Numerical simulation of the unsteady behavior of cavitating ows. Int J Numer Methods Fluids 42:527–548

7. Young F (1989) Cavitation. Imperial College Press, London

8. Franc JP, Michel JM (2004) Fundamentals of Cavitation. Kluwer Academic Publisher, Dordrecht

9. Colley E (2012) Analysis of flow around a ship propeller using open FOAM, pp 1–8

10. Sipilä T (2012) RANS analyses of cavitating propeller flows, Thesis for the degree of Licentiate of Science in Technology, Aalto University School of Engineering

11. Frank T, Lifante C, Jebauer S, Kuntz M, Rieck K (2007) CFD simulation of cloud and tip vortex cavitation on hydrofoils. In: 6th international conference on multiphase flow, ICMF 2007, Leipzig, Germany, 9–13 July 2007, pp 1–5

12. Chen Z (2015) CFD investigation in scale effects on propellers with different blade area ratio, Master thesis, Aalesund University College, Norway