Numerical Study on Influence of Geometry in Cavitation on Propellers

Harikrishnan Venugopal and S. Ajith Kumar

July 7, 2020

Proceedings of IMEC 2019
International Mechanical Engineering Congress (IMEC-2019),
29th November- 1st December 2019, NIT Tiruchirappalli, India.

IMEC2019-xx-xxxx

Numerical study on influence of geometry in cavitation on propellers

Harikrishnan Venugopal, Dr.Ajith Kumar S,

Department of Mechanical Engineering, Amrita Vishwa Vidyapeetham, Amritapuri, Kerala, India -690525

ABSTRACT

A CFD analysis is done to study and evaluate the variation in percentage of area under cavitation in marine propellers due to changes in geometrical parameters. The commonly used Wageningen–B series propellers are used, and several models of the propellers are made for a range of geometric properties. The study includes three important geometrical parameters for propeller design-Pitch, Blade Area Ratio and Rake angle with limits defined in the Wageningen-B series. Apart from this, another study is conducted to incorporate the effect of varying propeller speeds. The results include variation in cavitation with the above-mentioned parameters and tradeoffs that occur when design is skewed towards reducing cavitation. This result of the analysis significantly speeds up the preliminary design stages of the propeller by providing valuable information on consequences of design choices.
Keywords: Cavitation, Frozen Rotor, Propeller Geometry, Wageningen-B series

NOMENCLATURE

α = Pitch angle

D = Diameter formed by propeller blades

p = Pitch of the propeller

$\vec{\omega}$ = angular velocity

$\vec{U}$ = velocity vector, $\vec{U} = u\,\hat{\imath} + v\hat{\jmath} + w\hat{k}$

$\overrightarrow{U_r}$= relative velocity vector.

μ= viscosity of fluid

ρ= density of fluid

INTRODUCTION

In this era of modern marine vehicles, its performance is optimized for longevity and smooth operation, as well as to increase thrust and efficiency. Propeller design has undergone great changes to accommodate this advance in technology. However, cavitation effects on propeller still plays an important part in deciding the ship performance and noise generation. It causes surface damage of the propellers and affects its performance and can even be the cause of sudden failures. Many standard propeller series has been developed, Wageningen-B series being the most popular because of its focus on propeller performance and efficiency. The series relates geometrical properties to thrust created by the propeller and the overall efficiency of its operation. However, a suitable method to quantize cavitation against these geometrical properties have not been developed yet.

BACKGROUND THEORY

Cavitation

Cavitation is the phenomenon of formation of vapour cavities in a liquid when the static pressure drops below a specific value called vapour pressure. When the pressure of the liquid falls below vapour pressure, the dissolved air and water vapour molecules which form the cavitation nuclei expand and break the intermolecular H-bonds of the liquid molecules and this forms an air cavity that expands until the static pressure becomes equal to or greater than vapour pressure. Once the pressure rises beyond the vapor pressure the molecule implodes either creating micro jets of high pressure (around 800MPa) or it emits a shockwave [1][2]. These microjets impinge on a surface and can cause pitting.

Cavitation regions in propellers were first identified by Kinnas(1996)[3] by experiments in towing tanks as shown in Figure 1. He also explained in detail about the type of bubble cavities causing propeller damage. Propeller induced hull vibrations were studied by Lee,Liao and Wang(2006) [4] using CFD simulations and its result showed the detrimental effect of propeller induced vibrations on the hull .Apart from human factors it can affect the instruments in the ship and can also cause resonance failure in propeller shaft in extreme cases. In marine hydrodynamics cavitation effect is commonly seen in pumps, turbines and propellers. Here the cavitation depends on the speed of the device as well as its geometry.

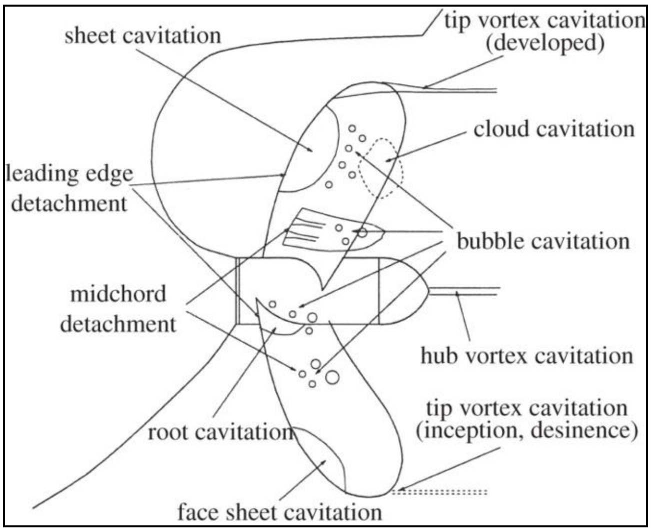


Figure 1. Propeller cavitation zones

Cavitation occurs at interfaces where the cohesive forces of the fluid are weak. Figure 1 shows the regions on the propeller that cause cavitation and these regions correspond to the fluid-solid interface.

Propeller Design

A propeller consists of two main parts: the hub and the blades. Out of these, the blades are the important part as they contribute to the efficiency and thrust of the propeller while the hub is just a structural support for the blades. The propeller blades have a highly three-dimensional surface because each section of the propeller is made to follow a helical path with the same pitch.

Gaafary, Khilani and Mustafa (2011) [5] standardized the design procedure for propellers by developing algorithms to optimize the performance of Wageningen-B series. 2D and 3D Geometric modelling of propellers were developed by Shamsi (2008) [6]. It included algorithms from which blade generation programs were developed. These algorithms were further developed and currently B-splines are used to obtain the shape of the propeller blade sections from a set of geometrical parameters. Three main geometrical parameters are explored here:

- Pitch angle (α): It is inverse tangent of the ratio of the propeller pitch to the circumference of the circle formed by the propeller blades. It is shown in Figure 2(a).

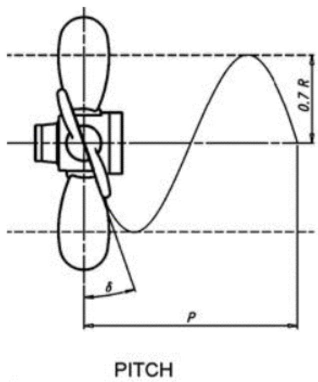
  $$\alpha = \tan^{-1}\frac{p}{\pi D} \qquad (1)$$

  In the Wageningen-B series, the Pitch-Diameter ratio is used instead of the pitch angle

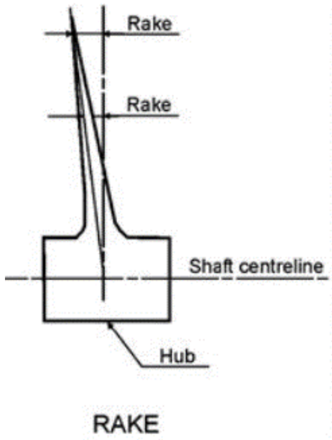
- Expanded Area Ratio (A_E/A_0): It is the total area of the blades (blade area) divided by the area formed by the propeller circle (swept area).
- Rake angle: It is the angle of inclination of the propeller blades with respect to line perpendicular to the propeller axis. It is shown in Figure 2(b).

Propellers are designed for set of design parameter values obtained from standard propeller series charts. A common series used in industry is the Wageningen-B series. Parameters like the skewing (sideways bending of the propeller blades) are not considered here as it is not included in the standard series. In the Wageningen-B series of propellers, the limits of the geometric parameters are as follows:

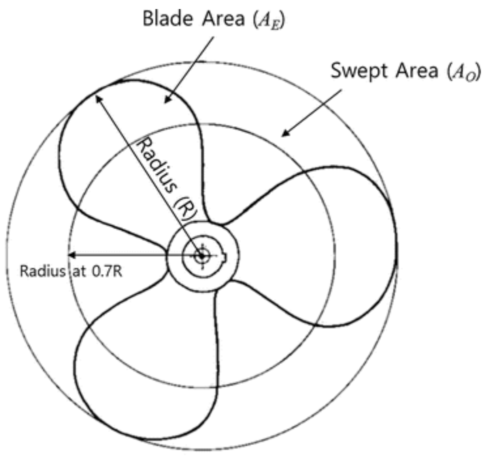
- Pitch-Diameter ratio (p/D): 0.4 to 1.2
- Expanded Area Ratio: 0.3 to 1.05
- Rake angle: 0 to 20 degrees



(a)



(b)



(c)

Figure 2: Propeller Geometry (a). Pitch (b). Rake angle (c). Blade area and swept area

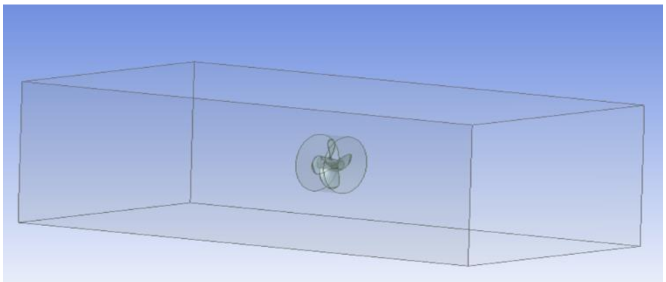


Figure 3: The frozen rotor model with a rectangular and cylindrical enclosure

NUMERICAL METHODOLODY

In the first analysis, geometry parameters used are Pitch- Diameter ratio, Expanded Area Ratio and Rake angle. Propeller diameter, speed, blade section thicknesses and hub geometry are kept constant. The second analysis keeps all the geometric parameters constant and changes the propeller speed. The analysis is done using a steady state CFD model called frozen rotor or the multiple-reference frame model.

Frozen Rotor Method

In the Frozen rotor method, we have two reference frames – one reference frame for global values and one where local coordinates are used. This model is based on the principle that a rotating object is made to be stationary provided the frame of reference rotates at the same speed as the object. This method is usually used in propeller and turbo machinery where there is both a stationary and a moving domain. When both domains are stationary, the Navier-stokes equations for the stationary reference frames are used. But when a domain rotates with respect to a stationary domain, the flow variable values have to be modified and hence Navier-Stokes equations are modified for the rotating domain to include the effects of Coriolis acceleration and centripetal acceleration [7][8]. The meshes are fixed and the propeller acts as a wall. Only the momentum equations change in the MFR model, both mass and energy conservation equations are same for all cell zones. So scalar quantities like pressure, temperature, turbulent kinetic energy etc. are transferred through cells without any change. This method has been tested to be accurate for explaining steady-state idealizations where the initial transient conditions are not of importance [7].

CFD model

Rectangular enclosure (see Figure3): Stationary domain where one face acts as inlet with velocity of 5m/s and other faces as openings with 1 atm pressure. This allows an initiating flow to the rotating domain in the cylindrical enclosure. The boundaries are distanced from the rotating domain to not affect the flow properties in the propeller trailing edge.

Cylindrical enclosure (see Figure3): Rotating domain with speed of 3000 rpm (speed variable) where all faces of the propeller are domain interfaces. The distance of the interface from the propeller is made >5% of its diameter to ensure accuracy of the frozen rotor method- [9].

Governing Equations

1.Stationary frame of reference

Navier-stokes conservation of mass:

$$\frac{\partial\rho}{\partial t} + (\nabla\rho\vec{U}) = 0 \qquad (2)$$

The continuity equation is used to check the mass conservation within a flow. For a steady flow $\frac{\partial\rho}{\partial t} = 0$ and for an incompressible flow $\rho = constant$

Navier-stokes conservation of momentum in x:

$$\frac{\partial(\rho u)}{\partial t} + \nabla\left(\rho u\vec{U}\right) = -\frac{\partial p}{\partial x} + \nabla(\mu\nabla u) \qquad (3)$$

For an incompressible, steady flow $\frac{\partial\rho u}{\partial t} = 0$ .Since the fluid is isotropic $\mu = constant.$

2.Rotating frame of reference

The absolute velocity at the stationary domain is converted to relative velocity in the rotating domain and vice versa at the interface by the following transformation formula:

$$\vec{U} = \overrightarrow{U_r} + (\vec{\omega}\times\vec{r}) \qquad (4)$$

The subscript 'r' refers to variable values in the rotating frame

Navier-stokes conservation of mass:

$$\frac{\partial\rho}{\partial t} + (\nabla\rho\overrightarrow{U_r}) = 0 \qquad (5)$$

Navier-stokes conservation of momentum in x:

$$\frac{\partial(\rho u_r)}{\partial t} + \nabla\left(\rho u_r\overrightarrow{U_r}\right) + \rho(2\vec{\omega}\times u_r + \vec{\omega}\times\vec{\omega}\times\vec{r}) = -\frac{\partial p}{\partial x} + \nabla(\mu\nabla u_r) \qquad (6)$$

In the rotating frame , two additional forces are present and hence have to be accounted for. $2\vec{\omega}\times u_r$ is the Coriolis force per unit mass and $\vec{\omega}\times\vec{\omega}\times\vec{r}$ is the centrifugal force per unit mass.

RESULTS AND DISCUSSIONS

The contour areas of cavitation have been plotted and measured by comparing the pressure of the flow near at the surface with the vapour pressure of water at ambient conditions.

Blade Area Ratio

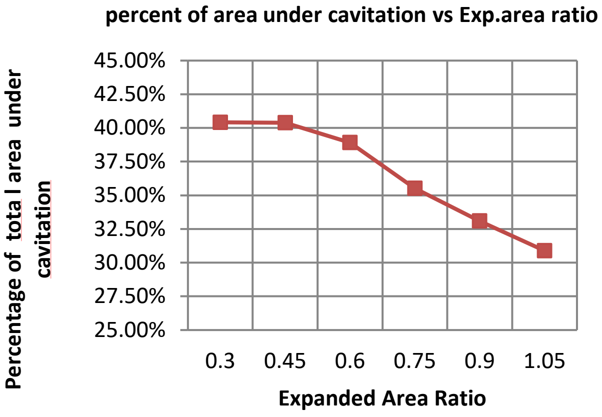


Figure 4. Percentage of area under cavitation vs Expanded area ratio

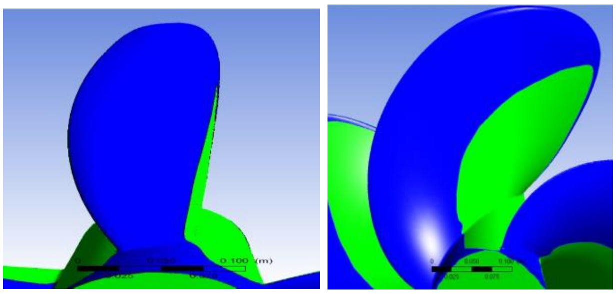


Figure 5. Cavitation region(blue) for a propeller blade back. Blade area ratio 0.3 (left). Blade area ratio 1.05 (right)

Six values of blade area ratio is analysed here , from 0.3 to 1.05.From Figure 4, we can make an inference that as blade area ratio increases percentage of cavitation decreases. This is because even though the blade area increases the cavitation regions are limited about the leading edge. This is because for larger blade sections the flow separates at the mid-chord region (region of maximum thickness) as seen in Figure 5. But lower blade area ratios are preferred due to better efficiency and performance as the frictional drag is lesser. So, the blade area should be made as little as possible but enough to control cavitation.

Propeller Pitch

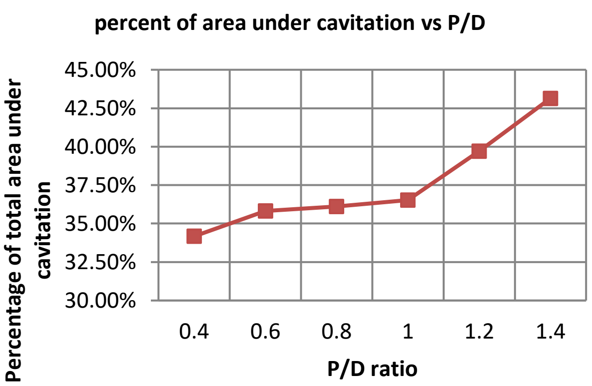


Figure 6. Percentage of area under cavitation vs Pitch-Diameter ratio

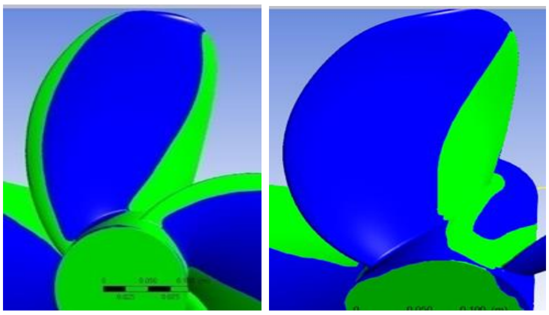


Figure 7. Cavitation region(blue) for a propeller blade back. P/D=0.4 (left) .P/D= 1.2 (right)

Propeller pitch ratio is varied from 0.4 to 1.4 with 6 values.When the P/D ratio changes, the diameter is kept constant only the pitch changes. From Figure 6 an inference is that as pitch increases, percentage of cavitation increases. At zero pitch both back and face of propeller have equal pressures but increasing the pitch increases the angle of attack and therefore the flow circulation increases along the propeller back as seen in Figure 7 and thus it will have low pressure and while the propeller face will have high pressure. Therefore, increasing the angle of attack increases the propeller thrust but care must be taken to consider the cavitation effects. There is also a sudden jump in cavitation area after P/D=1,this is because up to a certain value of angle of attack ,the hub is not affected by cavitation but we can see that after P/D=1 hub tip cavitation occurs due to increased flow rates around the hub.

Propeller Rake angle

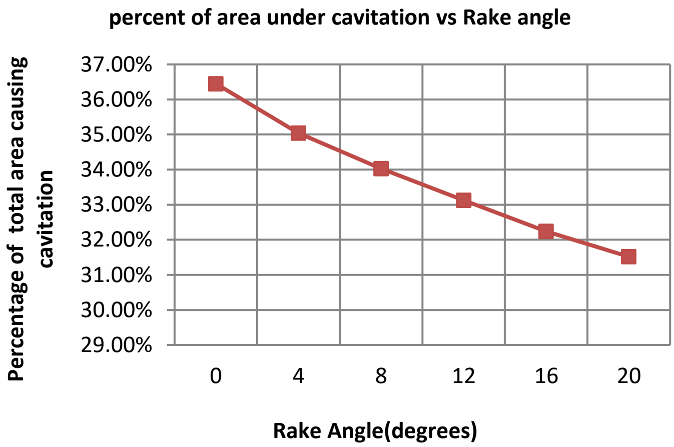


Figure 8. Percentage of area under cavitation vs Rake angle

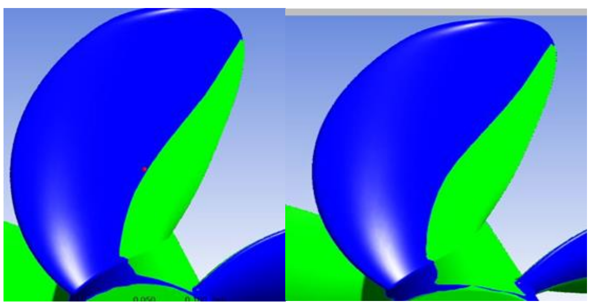


Figure 9. Cavitation region(blue) for a propeller blade back. Rake angle=0 deg (left). Rake angle=20 deg(right)

Rake angle is varied 0^0 to 20^0 in six steps and analysed. From Figure 8, we can make an inference that as pitch increases, percentage of cavitation decreases. The pressure contours for propeller back barely change, but the pressure at the root section visibly increases as the rake angle increases as seen in Figure 9. This can be because at lower rake angles the blades obstruct the normal flow direction and force the flow around the root, this effect is less in propellers with high rake angles. Higher rake angles are usually favored to reduce hull vibrations.

Propeller speed

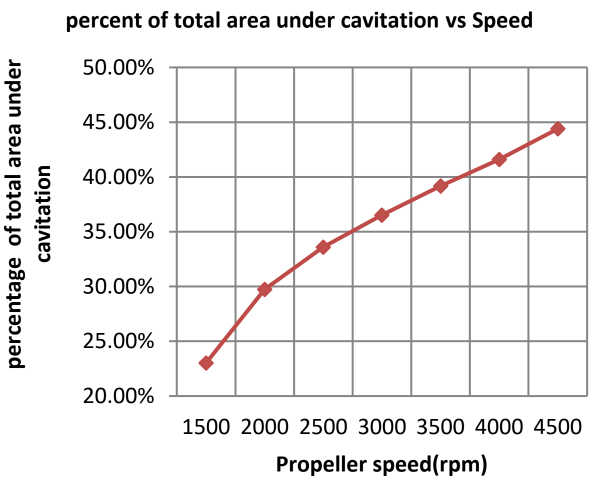


Figure 10. Percentage of area under cavitation vs propeller speed

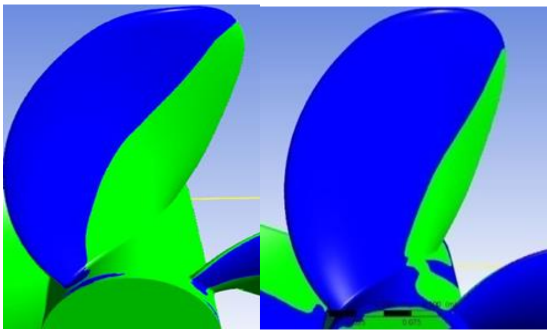


Figure 11. Cavitation region(blue) for a propeller blade back. Speed=1500 rpm (left). Speed=4500 rpm(right)

The speed is varied from 1500 rpm to 4500 rpm with 5 intermediary values and analysed. From Figure 10, we can infer that as propeller speed increases the percentage of cavitation also increases. This is not surprising as the flow velocity increases with propeller speed and thus overall pressure distribution decreases for the propeller in the propeller back and hub as seen in Figure 11, thus fueling cavitation.

CONCLUSIONS

- Increase in the blade area ratio decreases the cavitation caused by propeller but at the cost of lower performance and efficiency due to higher drag. There is a 10 % decrease in cavitation from 0.3 to 1.05 blade area ratio values.
- Propellers with high rake angles are favorable because not only does the cavitation regions decrease but also it prevents hull vibrations thus allowing larger diameter propellers, but it decreases the blade strength and the root section must be thickened. A 20-degree rake angle offers a 6% decrease in cavitation region when compared to a zero-rake propeller. Further rake angles can hamper the thrust of the propeller.
- Higher pitch angles are favored for better thrust and efficiency, but it will lead to high cavitation near the tip as well as the leading edge, so increasing the propeller diameter or the blade area ratio can counteract this effect. A 9 % increase in cavitation area is predicted for an increase in pitch ratio from 0.4 to 1.4.
- Increase of propeller speeds result in an almost linear increase in cavitation. An increase of 1 rpm gives a 0.007 % increase in cavitation area.

Further studies can be done on propellers having different blade numbers and varying blade sections. A further study on the effect of skew angle would be beneficial. Apart from this a regression analysis or factorial analysis can be carried out and a preliminary formula for cavitation can be found from the design parameters, assuming they vary linearly within the range specified in the series.

References

1. Computer study on rebound of vapour cavity bubble – W.K Soh and M.Shervani Tabar, 1992
2. Conceptual design tool for radial turbines - Elias Mikael Vagn Siggeirsson and Steinn Gunnarsson-Chalmers University, 2015
3. Theory and numerical methods for the hydrodynamic analysis of marine propulsors-Kinnas(1996)
4. Propeller induced hull vibrations-Analytical methods-Sing Kwan Lee,Min Liao and Suqiu *Wang(2006)*
5. Optimum design of B-series marine propellers-M.M. Gaafary , H.S. El-Kilani, M.M. Moustafa(2009)
6. 2D and 3D Geometric Modeling of Marine Propellers-Reza Shamsi (2008)
7. Frozen Rotor and Sliding Mesh Models Applied to the 3D Simulation of the Francis-99 Tokke Turbine with Code_Saturne -N Tonello, Y Eude, B de Laage de Meux, M Ferrand
8. ANSYS theory guide-CFX and Meshing
9. Cavitation on small commercial propellers-Hydrocomp technical report 111