Calculus investigation of cavity resistant propellers. Defining cavity protraction circumstances

S V Dikiy, G V Kochnov
Moscow Institute of Physics and Technology (State University), Dolgoprudnyi, Moscow region, Russia

E-mail: dikiy.s.v@gmail.com

Abstract. One of the key purpose in water propeller design is to determine operating conditions where the cavity mode appears. They may result in dramatic thrust lack, drag rise and mechanical damage. An objective of the article is to verify potential of computer fluid dynamics program set OpenFoam in 5-bladed propeller propulsion characteristics investigation next to the cavity mode, and to study pitch and maximum thickness roles in cavity sustainability. Applied in the work method is based on solution of Reynolds equations with turbulent model closure via finite volume method. Suggested methodology proved a high conformity to experimental data.

1. Introduction

Growth of engine power on high-speed boats induces growth of their velocity. Despite this, in most cases propellers operate in near-cavity mode, only a few use cavity. That is why there is the necessity for design propellers with such parameters, which can decrease the risk of cavity rise. Thus, many negative effects caused by cavity may be eliminated, such as efficiency drop, erosion, vibration.

As its known, cavity is determined as continuity break in liquid, where at some speed the pressure falls at blade surface to the saturation pressure causing a phase transition from the liquid to vapor. Developing cavity-resistant foils consist of searching the practical ways in order to prevent too much pressure decrease at surface without efficiency decrease.

One of well-known methods of prevention of early cavity appearing is to enlarge blade area. However, it has a huge disadvantage, which is expressed in dramatic efficiency loss in non-cavity mode.

The other way to prevent early blade cavitation arise is the choice opt for cavity-resistant foils of the blade, for investigation which the special attention is paid. Study of transonic foils says that making the foil thinner, moving maximum thickness backward and changing the general form that is possible at the same $C_y$ and flow velocity to decrease maximum pressure discharge at lifting face respectively to decrease of foil drag. Transonic foils suit demands for cavity-resistant foils. Therefore they may be used for propeller development.

2. Test cases

For comparative analysis, 3 variants of one propeller were used, which differences are in twist and profile thickness.

The first is a 5-blade propeller in a flat circular covering (cylinder with an internal diameter of 120 mm) with blades consist of aviation profiles with a thickness of 20% (see Table 1). In this case, the angle of attack (AoA) of each section changed according to the linear law $\alpha(r) = a - br$ (for more details, see
Table 3). This configuration serves as a reference point for model verification in accordance with the experiment.

The second configuration differs from the first one only in the fact that the pitch in each section is kept equal to the pitch of the ending section of the blade with a linear twist (Table 4). It is expected that due to an increase in the AoA of the foils at the root, the thrust of such configuration will become larger in a similar motion condition. But at the same time, there is a danger of cavitation.

The third variant is the blade with a constant pitch, in which the profile is replaced with a thinner one (Table 2). The remaining parameters remained the same (Table 5). A thinner profile should tighten the transition to cavitation mode.

| Table 1 |
|------------------|---------|
| Initial foil (thick) | |
| Thickness (b/c) | 20% |
| Maximum thickness position (x_b/c) | 0,3 |
| Camber (l/c) | 4% |
| Maximum camber position (x_c/c) | 0,53 |

| Table 2 |
|------------------|---------|
| Initial foil (thin) | |
| Thickness (b/c) | 15% |
| Maximum thickness position (x_b/c) | 0,3 |
| Camber (l/c) | 4% |
| Maximum camber position (x_c/c) | 0,53 |

| Table 3 |
|------------------|---------|
| Linear twist propeller | |
| Number of blades Z | 5 |
| Diameter D [m] | 0,12 |
| Shaft diameter d [m] | 0,025 |
| Root pitch H_r/D | 0,7 |
| Peripheral pitch HD/D | 1,6 |
| Propeller are ratio | 0,95 |
| Foil thickness (b/c) | 20% |

| Table 4 |
|------------------|---------|
| Constant pitch propeller (thick) | |
| Number of blades Z | 5 |
| Diameter D [m] | 0,12 |
| Shaft diameter d [m] | 0,025 |
| Pitch HD/D | 1,6 |
| Propeller are ratio | 0,95 |
| Foil thickness (b/c) | 20% |

| Table 5 |
|------------------|---------|
| Constant pitch propeller (thin) | |
| Number of blades Z | 5 |
| Diameter D [m] | 0,12 |
| Shaft diameter d [m] | 0,025 |
| Pitch HD/D | 1,6 |
| Propeller are ratio | 0,95 |
| Foil thickness (b/c) | 15% |

3. Calculation methodology
In the direct formulation, the solution of the current problem requires necessarily to calculate all the blades and apply the model of overset associated with rotating blades, which increases the size of the computational domain and grid and significantly increases the complexity of the calculations.

To optimize the calculation time, it was decided to use reference frame associated with the rotor blade and search for the solution in the reverse formulating. This approach, in addition to eliminating the need to use the model of the overset meshes, allows us to consider a sliced computational domain with one blade due to usage of shift symmetry condition at the cutoff boundaries from neighboring blades (in Figure 1, it corresponds to border No. 9).

The domain and the boundary conditions are illustrated in Fig. 1, and described at Table 6.

![Figure 1. Domain boundaries](image)

### Table 6. Domain boundaries

| №  | Color mode | Name            | Conditions                      |
|----|------------|-----------------|---------------------------------|
| 1  |            | Blade           | Rigid wall:                     |
|    |            |                 | non-porous,                     |
|    |            |                 | non-slip                        |
| 2  |            | Joints          |                                 |
| 3  |            | Shaft           |                                 |
| 4  |            | Covering        |                                 |
| 5  |            | Velocity inlet  | Constant pressure               |
|    |            |                 | Constant normal velocity        |
| 6  |            | Side inlet      | Constant pressure               |
| 7  |            | Outlet          | Constant pressure               |
| 8  |            | Side outlet     | Constant pressure               |
| 9  |            | Cutoff plain    | Default                         |
|    |            |                 | Shift symmetry interface        |
3.1 Equation modification for reference frame

The principle of mechanics claims that the motion of a point in a rotating coordinate system is described as

\[ \mathbf{u}_0 = \mathbf{u}_p + \mathbf{\omega}_p \times \mathbf{r}, \]

where

- \( \mathbf{u}_0 \) – velocity in laboratory reference frame,
- \( \mathbf{\omega}_p \) – axial evolution velocity
- \( \mathbf{u}_p \) – velocity in rotating reference frame,
- \( \mathbf{r} \) – position vector in rotating reference frame.

Due to discrete continuum in the CFD software the \( \mathbf{r} \) is the position of the center of the control volume in rotating reference frame.

Than the convection in momentum equation will be

\[ \frac{D\mathbf{u}_0}{Dt} = -\nabla \left( \frac{p}{\rho} \right) + \nu \nabla \cdot \nabla \mathbf{u}_0 \]

And the right-hand part keep the form

\[ \nabla \cdot \nabla \mathbf{u}_0 = \nabla \cdot \nabla \left( \mathbf{u}_p + \mathbf{\omega}_p \times \mathbf{r} \right) = \nabla \cdot \nabla \mathbf{u}_p + \nabla \cdot \nabla \left( \mathbf{\omega}_p \times \mathbf{r} \right) = \nabla \cdot \nabla \mathbf{u}_p \]

On account of

\[ \nabla \cdot \nabla (\mathbf{\omega}_p \times \mathbf{r}) = 0 \]

Applying all transformations, expanding Lagrange’s derivation the momentum equation becomes

\[ \frac{d\mathbf{u}_p}{dt} + \nabla \cdot \left( \mathbf{u}_p \otimes \mathbf{u}_p \right) + \frac{d\mathbf{\omega}_p}{dt} \times \mathbf{r} + 2 \mathbf{\omega}_p \times \mathbf{u}_p + \mathbf{\omega}_p \times \mathbf{\omega}_p \times \mathbf{r} = -\nabla \left( \frac{p}{\rho} \right) + \nu \nabla \cdot \nabla \mathbf{u}_p \]

Accounting that angular velocity does not change neither value nor position, also supposing the flow to be stationary, the final form is

\[ (\mathbf{u}_p \cdot \nabla) \mathbf{u}_p + 2 \mathbf{\omega}_p \times \mathbf{u}_p + \mathbf{\omega}_p \times \mathbf{\omega}_p \times \mathbf{r} = -\nabla \left( \frac{p}{\rho} \right) + \nu \Delta \mathbf{u}_p \]

At the same time mass flow hold its form as

\[ \nabla \cdot \mathbf{u}_p = 0 \]

Therefore, Navier-Stokes equations got non-inertial term, that made them more complex. On the other hand, there is no mere need in direct grid motion, as well as solving time, while the physical similarity to the motion with constant rotation was preserved.

3.2 Specific volume of fluid model

Since cavitation flows are multiphase flows, it is impossible to use the equations valid for single-phase continuous media. To solve this problem, the specific volume hypothesis of each medium was applied.

In each unit of volume, the next relation is correct

\[ r_\alpha = \frac{V_\alpha}{V} \]

where the variable \( r_\alpha \) is the volume fraction of the fluid \( \alpha \) in the control volume \( V \). Then, for the entire volume, the conservation law is in the form

\[ \sum_{\alpha=1}^{n} r_\alpha = 1 \]
Thus the mixture density is counted as

$$\rho = \sum_{\alpha=1}^{n} r_\alpha \rho_\alpha$$

Taking into account that the resulting mixture is still a continuous medium it is easy to see that the Navier-Stokes equations decompose into n pieces in the form of

$$\frac{\partial}{\partial t} (r_\alpha \rho_\alpha) + \nabla (r_\alpha \rho_\alpha \mathbf{u}) = \sum_{\beta=1}^{n_i} \Gamma_{\alpha\beta}$$

$$\frac{\partial}{\partial t} (r_\alpha \rho_\alpha \mathbf{u}) + (\mathbf{u} \cdot \nabla) r_\alpha \rho_\alpha \mathbf{u} = -r_\alpha \nabla (p) + \nabla \left( r_\alpha \mu_\alpha \left( \nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) \right) + \sum_{\beta=1}^{n_i} \left( \Gamma_{\alpha\beta} - \Gamma_{\beta\alpha} \right) \mathbf{u} + \mathbf{M}_\alpha$$

Here, $\Gamma_{\alpha\beta}$ shows the flow of matter from state $\beta$ to state $\alpha$ inside the control volume, and the vector $\mathbf{M}_\alpha$, respectively, the momentum flow due to mass changes.

Thus, in addition a pair of $r_\alpha$ was added to the variables $\mathbf{u}$ and $p$ (since two media are involved in the analysis - water in a liquid state and a gaseous one), which, in view of the conservation law, can be represented in terms of one quantity $r_1$, when $r_2$ is expressed through it in the form

$$r_2(r_1) = 1 - r_1$$

### 3.3 Schnerr-Sauer’s cavity

Cavitation occurs as a result of a decrease in static pressure when flowing around a profile with an increase in local velocity pressure. This effect is similar to how, in particular, water boils at lower temperature when the atmospheric pressure decreases. This means that with a sufficiently large discharge on the profile, when the saturated vapor pressure is reached, a phase transition occurs.

For simplicity, we assume that there are always initial cavitation centers inside the liquid in the form of spheres with a certain particle density $n_0 = N/V$, but with a very small (equal to zero) radius $R$. As they move, they fall into the region favorable pressure for growth, where their volume can no longer be neglected. Then the volume occupied by the steam is

$$V_2 = N \frac{4}{3} \pi R^3$$

And than the volume fraction of vapor depends on radius as

$$r_2 = n_0 \frac{4}{3} \pi R^3$$

Dynamics of gas sphere in surrounding liquid is described with Relay-Plesset equation

$$R \frac{d^2 R}{dt^2} + \frac{3}{2} \left( \frac{dR}{dt} \right)^2 + \frac{4\nu_1}{R} \frac{dR}{dt} + \frac{2\gamma}{\rho_1 R} + \frac{p - p_d}{\rho_1} = 0$$

Where we neglect the surface tension $\gamma$ and fluid viscosity $\nu_1$ in favor of the assumption that saturated water vapor does not experience interaction with the fluid. We will do the same with acceleration $\frac{d^2 R}{dt^2}$, assuming that the bubbles will not quickly pass from the high-pressure zone to the low-pressure zone and vice versa. Then the equation is greatly simplified to

$$\left( \frac{dR}{dt} \right)^2 = \frac{2}{3} \left( \frac{p_d - p}{\rho_1} \right)$$

Here $p_d$ is saturation pressure of water at constant temperature, $p$ is static pressure of bubble surrounding water.
4. **Results**

Thrust coefficient $C_T$ is dimensionless value by rotation rate

$$C_T = \frac{T}{\rho n^2 D^4}$$

characterizing the force with which a propeller with a given ratio of motion speed and revolutions $\lambda = v_\infty / n D$ ejects a fluid behind itself

$$T \propto v - v_\infty$$

For the considered configurations, the following characteristics were obtained:

![Thrust comparison](image)

**Figure 2.** Thrust comparison

A linear twist profile is expected to have significantly lower thrust due to lower local angles of attack in the sections located in the root. For the $i$-th section

$$\alpha_i = \arctg \left( \frac{H_i}{2 \pi r_i} \right) - \arctg \left( \frac{v_\infty}{2 \pi r_i n} \right)$$

The most remarkable thing is that the deviation of the simulation results of the propeller revealed a deviation of about 4% from the data obtained earlier in the experiment.

At the same time, both propellers with a constant pitch $H$ have very close indices on all investigated steps $\lambda$. This result is consistent with the predictions of the theory. In a linear approximation, the lifting force of the profile is proportional to the curvature of the profile

$$C_y \propto h$$

For the compared configurations, the curvature is the same. A significant difference between thin and thick profiles is their sustainability to separation. On the graph, this is characterized by the deviation of the plot from the strict line at small $\lambda$. In this study, the modeling of separation effects, although carried out according to a simplified model, which does not allow us to fully evaluate its effect, but clearly demonstrates the effect. The thin profile plot on the graph has greater curvature.

Dimensionless moment coefficient

$$C_M = \frac{M}{\rho n^2 D^5}$$

defines the necessary force to maintain constant rotation.
Due to smaller angles of attack of the blade with a linear twist and its resistance is less. In a first order approximation

\[ C_x \propto C_y^2 \]

Both blades with constant pitch are expected to show a close result.

It is also worth evaluating how each configuration effectively uses engine power. To do this, we attribute the power created by changing the fluid velocity along the axis of the propeller to the power on the motor shaft.

\[ \eta = \frac{T \cdot v_\infty}{M \cdot \omega} \]

In dimensionless form

\[ \eta = \frac{C_T}{C_M} \cdot \frac{\lambda}{2\pi} \]

In comparison with the linear twist configuration, the increases in thrust and drag moment are proportional so that in the tread region \( \lambda \leq 1 \) the efficiency of the blades decreases slightly, and even increases significantly for larger \( \lambda \). In terms of dimensional characteristics, this means an increase in thrust in the area of high speeds at equal rotation rate while maintaining characteristics at lower speeds.

4.1 Cavity

For all three configurations, studies on the occurrence of cavitation were also conducted. To do this, the number of revolutions \( n \) was increased to 2000 rpm (33.3 rps), and the speed \( v_\infty \) remained 1 m/s. This is
equal to $\lambda = 0.25$. A characteristic peripheral velocity $w = \omega R = 12.5$ m/s. In this case, the cavity number of the oncoming flow, as in the previous stages

$$\chi_\infty = \frac{(p_\infty - p_d)}{\rho_1 v_\infty^2} \gg 1$$

Making Schnerr-Sauer’s equation dimensionless, we get

$$\left( \frac{d R}{dt} \right)^2 = -\frac{1}{3} \chi$$

It’s clear that the growth rate correlates with the difference between the pressure of saturated vapor and the fluid around. In the above illustrations, black colour shows $\chi \ll 1$ - the region of intensive growth, gray $\chi \approx 1$, corresponding to $R \approx \text{const}$, and white unfavorable conditions for the occurrence of cavitation. The black contour outlines the approximate boundary within which the concentration of water vapor on the surface of the blade is greater than water in the liquid state.

**Figure 5.** Distribution of cavity-friendly areas on the surface of linear-screwed blade

**Figure 6.** Distribution of cavity-friendly areas on the surface of constant-pitched blade (thick foil)

**Figure 7.** Distribution of cavity-friendly areas on the surface of constant-pitched blade (thin foil)

In the case of linear twist (Fig. 5), the cavity has the largest size of all the presented variations. It is located close to the covering ring, where $v_1 \cong w$. 
Under the same conditions, a screw with a constant pitch and a thick profile showed a complete absence of conditions for the appearance of caverns (Fig. 6). This is most likely due to the increased side flow rate of fluid along the leading edge. Further, meeting with the covering, the particle velocity decreases, raising the static pressure.

When we change the thick profile with a thinner one, a different effect plays the main role: due to the smaller radius of blunting of the leading edge, a suction force appeared, in the area of action of which cavitation was located (Fig. 7). Its detachment from the peripheral section is due to the same side flow as on the thicker profile is.

References
[1] Terentiev A G, Kirschner I N, Uhlman J S 2011 The hydrodynamics of cavitating flows (Backbone Publishing, USA) 598.
[2] Sauer J 2011 Instationaer kavitierende Stroemungen - Ein neues Modell, basierend auf Front Capturing VOF und Blasendynamik, Dissertation, Universitaet Karlsruhe, 2000 Kozhanova E R, Zakharov A A, Tkachenko I M. labpraktikum_osnov_wavelet: Certificate of State Registration of Computer Software No 2011611250
[3] Pernik A D 1966 Cavitation problems (Leningrad: Shipbuilding)
[4] Ivanov A N 1980 Hydrodynamics of developed cavitation flows (Leningrad: Shipbuilding)