TECHNICAL TRANSACTIONS 9/2017
CZASOPISMO TECHNICZNE 9/2017

MECHANICS

DOI: 10.4467/2353737XCT.17.156.7168

Mariusz Domagała (domagala@mech.pk.edu.pl)
Institute of Applied Informatics, Faculty of Mechanical Engineering, Cracow University of Technology

Hassan Momeni
Department of Mechanical and Marine Engineering, Western Norway University of Applied Sciences

# CFD SIMULATION OF CAVITATION OVER WATER TURBINE HYDROFOILS

---

# SYMULACJE CFD ZJAWISKA KAWITACJI PRZY OPŁYWIE PROFILI TURBIN PŁYWÓW MORSKICH

**Abstract**

Tidal turbines are gaining more and more interest as a source of renewable energy. Energy from tidal currents may be extracted in a similar way as wind energy but tidal turbines have to meet much higher requirements. Cavitation phenomena, which are difficult to predict, have great influence on hydrodynamic efficiency. This paper presents the use of CFD simulation for prediction of cavitation during flow over typical NACA airfoils.

**Keywords:** CFD, cavitation

**Streszczenie**

Turbiny wykorzystujące pływy morskie są coraz częściej wykorzystywane jako źródła czystej energii. Energię pływów morskich można uzyskać w podobny sposób jak w przypadku energii wiatrowej, ale turbiny wykorzystujące pływy morskie muszą spełnić znacznie bardziej rygorystyczne wymagania. Jednym z czynników mających wpływ na efektywność hydrodynamiczną turbin jest zjawisko kawitacji. Artykuł przedstawia wykorzystanie metod CFD do symulacji zjawiska kawitacji przy opływie typowych profili lotniczych.

**Słowa kluczowe:** CFD, kawitacja

## 1. Introduction

Renewable energy is becoming a real alternative to traditional energy sources. Tidal energy looks very promising due to its predictability and high energy density. Extracting power from tidal flow can be made in a very similar way to extracting wind energy, using turbines with horizontal or vertical axis. Commercial applications of tidal turbine are already available. Contrary to wind energy, tidal turbines have to meet much higher requirements. Moreover, one of the most important issues for extracting tidal energy is a proper selection of hydrofoil, which may be a typical airfoil or tailor made. One of the most important problems for tidal turbine is cavitation, which is very difficult to predict, and which may have significant influence on turbine efficiency or even cause turbine blade damages. The nature of cavitation cause sudden phase change, which disturbs flow and may cause blades wear. Cavitation is a complex problem investigated from many years, whose fundamental physics is presented in studies [1, 2]. The basic principle of cavitation is a sudden pressure drop below the saturation pressure at given temperature. Other studies show that also contaminations and liquid aeration may have influence on nucleation formation. Despite the rapid development of simulation tools, including Computational Fluid Dynamics (CFD), cavitation modeling is still a complex task. This paper presents an attempt to predict cavitation during flow over typical hydrofoils (NACA 4418, 4416 and 4412) with the use of Ansys CFX and a CFD code.

## 2. Mathematical model of cavitation

Mathematical description of cavitation has been a subject of studies for many years. Generally, the tendency of flow to bubbles formation may be expressed as the following cavitation number:

$$c_a = \frac{p - p_v}{0.5\rho U^2} \tag{1}$$

The Rayleigh-Plesset Formula is a common approach which describes bubble dynamics:

$$R_b \frac{d^2 R_b}{dt^2} + \frac{3}{2}\left(\frac{dR_b}{dt}\right)^2 + \frac{2\sigma}{\rho R_b} = \frac{p_v - p}{\rho} \tag{2}$$

After neglecting surface tension and the term of second order, the above eq. has the following form:

$$\frac{dR_b}{dt} = \sqrt{\frac{2}{3}}\left(\frac{p_v - p}{\rho}\right) \tag{3}$$

The rate of changes of bubble volume is as follows:

$$\frac{dV_b}{dt} = 4\pi R_b{}^2 \sqrt{\frac{2}{3}\left(\frac{p_v - p}{\rho}\right)} \tag{4}$$

The rate of changes of bubble mass is as follows:

$$\frac{dm_g}{dt}=4\pi R_b{}^2\rho_g\sqrt{\left(\frac{2}{3}\frac{p_v-p}{\rho}\right)} \tag{5}$$

The number of bubbles $N_b$ per unit volume $r_g$ is expressed by:

$$r_g=\frac{4}{3}\pi R_b^2 N_b \tag{6}$$

The total interphase mass transfer per unit volume is as follows:

$$\dot{m}_{fg}=3\frac{r_g\rho_g}{R_b}\sqrt{\frac{2}{3}\frac{p_v-p}{p_f}} \tag{7}$$

After including condensation, the above expression has the following form:

$$\dot{m}_{fg}=3F\frac{r_g\rho_g}{R_b}\sqrt{\frac{2}{3}\frac{|p_v-p|}{p_f}}\operatorname{sgn}(pv-p) \tag{8}$$

And finally, the vapor transport equation has the form of:

$$\frac{\partial}{\partial t}(\alpha\rho_v)+\nabla\cdot(\alpha\rho_v\mathbf{V})=R_e-R_c \tag{9}$$

where:

$R_b$ – bubble radius,
$p_v$ – vapor pressure,
$p$ – pressure of liquid surrounding the bubble,
$\rho$ – liquid density,
$\rho_g$ – vapor density,
$\sigma$ – surface tension,
$U$ – liquid velocity,
$\alpha$ – vapor volume fraction,
$n$ – bubble number,
$R$ – phase change rate,
$f_v$ – vapor mass fraction,
$f_g$ – non-condensable gases.

## 3. CFD simulation

Flow simulations of hydrofoils were conducted for 2D models for 0 degree of angle of attack. Geometry of hydrofoils were created using a 3D CAD Creo Parametric system and a set of points with coordinates [3]. Models of three profiles: NACA 4418, 4416 and 4412 were created. An example of NACA 4418 is presented in Fig. 1.

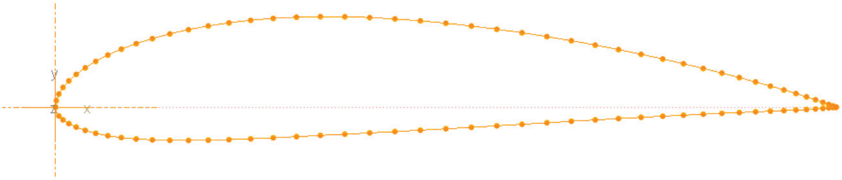

Fig. 1. Set of points used for creating geometry of NACA 4418

Grids for simulation were used using quadrilateral cells with edges aligned to create a profile. Figures 2 and 3 show the grid used in simulations. To avoid problems with the influence of boundary conditions on the flow over the foil, rules for creating computational domains for external flow were used.

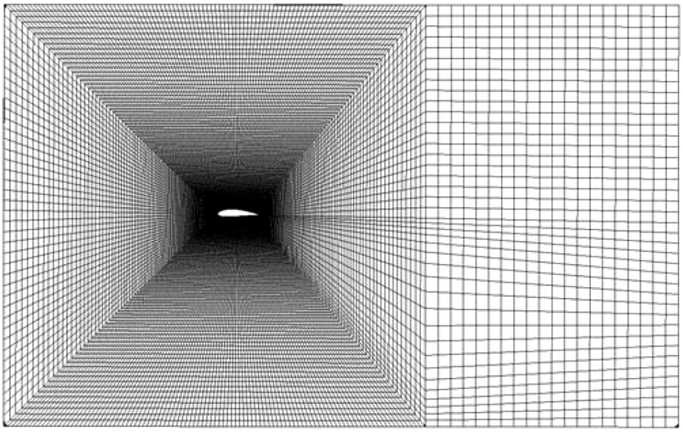

Fig. 2. Grid for NACA 4418 foil

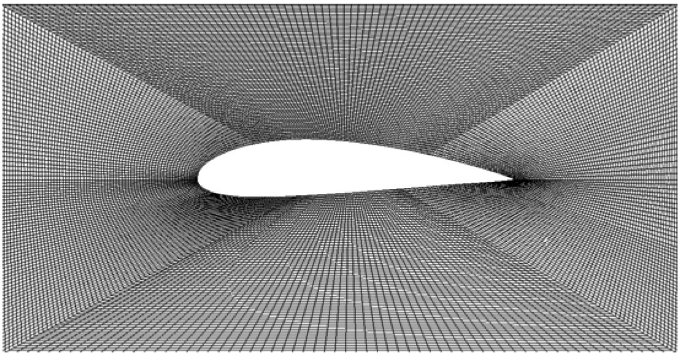

Fig. 3. Grid at the vicinity of foil

CFD simulations were conducted in the Ansys CFX code as two phase flows (water and water vapor) without interphase heat exchange for conditions corresponding to 5 m water depth and two water velocities of 5 m/s and 8 m/s. Cavitation and turbulent flow is random by nature, but for simplification simulations were conducted for steady state conditions with constant water properties for typical sea water. Calculations were conducted for the same conditions for all foils. It was also assumed that both phases are homogenous and no solid or air (vapor or air) particles are presented.

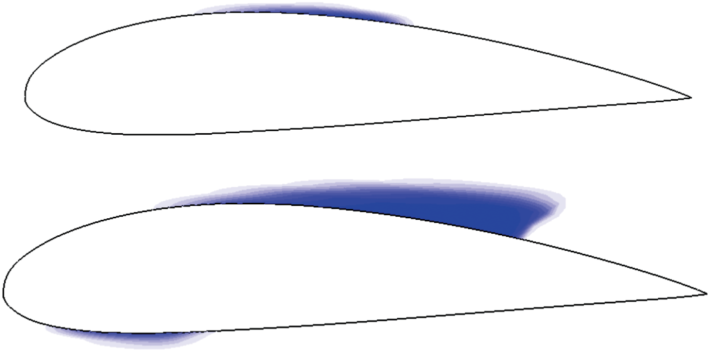

Fig. 4. NACA 4418 vapor volume fraction for 5 m/s and 8 m/s water flow

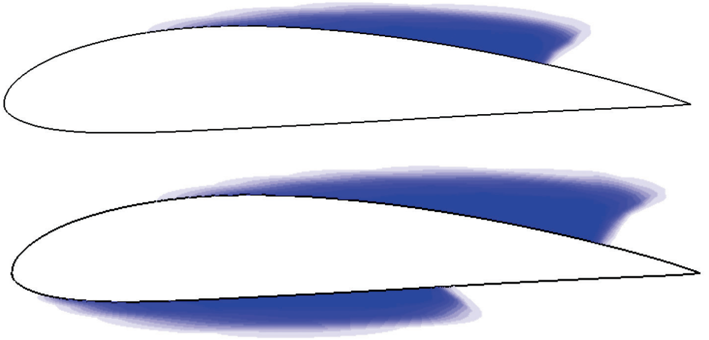

Fig. 5. NACA 4416 vapor volume fraction for 5 m/s and 8 m/s water flow

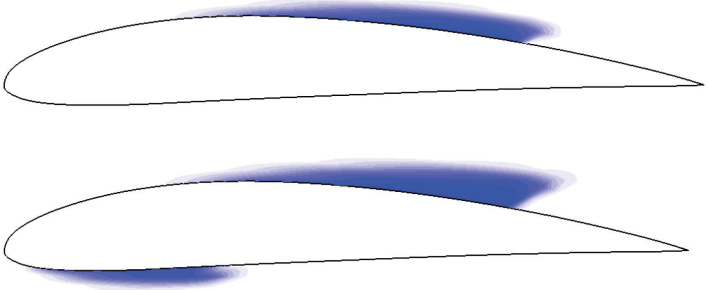

Fig. 6. NACA 4412 vapor volume fraction for 5 m/s and 8 m/s water flow

The results presented above for the selected foils show that nucleation forms in the areas with the highest velocity, which was to be expected. Simulation shows that foil shape has significant influence on the formulation of cavitation. Among three investigated foils, NACA 4418 has a much lower tendency for formulation cavitation.

## 4. Conclusions

Cavitation is a key issue for designing tidal turbines which describes hydrodynamics efficiency. It is difficult to predict, but by using CFD methods it is possible to reduce the probability of its appearance. Therefore, CFD methods may be treated as a very effective tool for designing tidal turbines.

## References

[1] Brennen C.E., *Cavitation and bubble dynamics,* Oxford University Press, 1995.

[2] Szkodo M., *Mathematical description and evaluation of cavitation erosion resistance of materials,* Journal of Materials Processing Technology, 2005.

[3] http://airfoiltools.com (access: 24.02.2017).

[4] Bahaj A.S., Batten W.M.J., Chaplina J.R, *Experimental verifications of numerical predictions for the hydrodynamic performance of horizontal axis marine current turbines,* Renewable Energy 32, 2007, 2479–2490.

[5] Mehmood N., Liang Z., Khan J., *Diffuser Augmented Horizontal Axis Tidal Current Turbines,* Research Journal of Applied Sciences, Engineering and Technology 4(18), 2012, 3522–3532.

[6] *CFD for Wind and Tidal Offshore Turbines*, E. Ferrer, A. Montlaur (eds.), Springer Tracts in Mechanical Engineering, ISSN 2195-9870 (electronic), Springer 2015.

[7] Bir G.S., Lawson M.J., Li Y., *Structural Design of a Horizontal-Axis Tidal Current Turbine Composite Blade,* ASME 30th International Conference on Ocean, Offshore, and Arctic Engineering Rotterdam, Netherlands, June 19–24, 2011.

[8] Zeiner-Gundersen D.H., *Turbine design and field development concepts for tidal, ocean, and river applications,* Energy Science and Engineering 2015, 3(1): 27–42.

[9] Domagała M., *Simulation of cavitation in jet pumps,* Technical Transactions, 1-M/2013.

[10] Domagała M., Momeni H., *Proceedings of the 8th International Conference on Business and Technology Transfer (ICBTT 2016),* Magdeburg, Germany.