3D modelling of cavitation structures on a ClarkY foil

Dorota Homa[1] **and Włodzimierz Wróblewski**[2]

[1] *Institute of Power Engineering and Turbomachinery, Silesian University of Technology*
Konarskiego 18, 44-100 Gliwice, Poland
e-mail: dorota.homa@polsl.pl

[2] *Institute of Power Engineering and Turbomachinery, Silesian University of Technology*
Konarskiego 18, 44-100 Gliwice, Poland
e-mail: wlodzimierz.wroblewski@polsl.pl

Abstract

The article concerns numerical investigation of water flow over a foil. Due to pressure decrease on the upper side of the foil the vaporization process occurr. Next, the vapour bubbles collapse near the trailing edge of the foil. The process of forming and collapsing of vapour bubbles in liquid flow is called cavitation. Two mathematical models of cavitation were compared in the article: Schnerr & Sauer model and Kunz model. The calculations were performed in OpenFOAM software, with interPhaseChangeFOAM solver. 3D structural mesh was used. The changes in time of vapour phase volume in the domain were monitored and compared for both models.. The distributions of vapour phase along the foil, as well as the changes in direction perpendicular to the flow were described. The courses of pressure in monitor points located on foil were also a part of comparison.

Keywords: cavitating flow, cavitation models, computational fluid dynamics, flow over foil, multiphase flow

1. Introduction

Cavitation phenomenon is very important issue when designing and exploiting pumps' systems. As the working under cavitating condition can lead to serious damage of the blades and walls of the rotor, it is useful to provide numerical simulations which can assess the risk of cavitation appearance at defined flow condition.

This paper includes comparison of two popular cavitation models – Schnerr & Sauer and Kunz model. They are widely used in both commercial and open source CFD codes. The aim of this study is to compare simulation results obtained while using both this models in case of cavitating flow over a ClarkY foil.

2. Models description

The investigated models are single-fluid cavitation models. In these models the flow is threated as the mixture of two phases and the conservation equations (mass and momentum) for the mixture are solved. In this simulation no slip between the phases was assumed. To calculate fraction of gaseous phase mass conservation equation of vapour is solved [1].

$$\frac{\partial \alpha \rho_v}{\partial t} + \nabla(\alpha \rho_v u) = R_e - R_c \tag{1}$$

where: α – vapour volume fraction, ρ_v – vapour density, t – time, u – velocity, R_e, R_c – source terms.

The difference between models is in determining source terms R_e and R_c. In Kunz model they are defined empirically. Kunz used free stream velocity and time t_∞ - derived from free stream velocity and characteristic length [2]:

$$R_e = \frac{C_e \rho_v \alpha \min(0, p - p_s)}{0.5 \rho_l u_\infty^2 t_\infty} \tag{2}$$

$$R_c = \frac{C_c \rho_v \alpha^2 (1-\alpha)}{t_\infty} \tag{3}$$

where: p – pressure, p_s – saturation pressure, ρ_l – liquid density, u_∞ – free stream velocity, t_∞ - free stream time, Ce, Cc – models coefficients.The coefficients Ce and Cc are assumed according to the flow type.

In Schnerr & Sauer model the source terms are derived from Rayleigh-Plesset (RP) equation, which describes the dynamics of vapour bubbles. The RP equation is simplified, surface tension is omitted, as in formula below [3]:

$$\frac{Dr_B}{Dt} = \sqrt{\frac{2}{3}\frac{p_B - p}{\rho_l}} \tag{4}$$

The final formulas for source terms [4] is:

$$R_e = \frac{\rho_l \rho_v}{\rho} \alpha(1-\alpha) \frac{3}{r_B} \sqrt{\frac{2}{3}\frac{p_s - p}{\rho_l}} \tag{5}$$

$$R_c = \frac{\rho_l \rho_v}{\rho} \alpha(1-\alpha) \frac{3}{r_B} \sqrt{\frac{2}{3}\frac{p - p_s}{\rho_l}} \tag{6}$$

where: r_B – radius of vapour bubble, ρ – mixture density.

The radius of bubble r_B is derived from vapour volume fraction α and n_b - number of bubbles per volume of liquid, as shown in equation 7 [4]:

The presented work was supported by the Polish National Science Centre founds within the project UMO-2016/21/B/ST8/01164.

$$r_B = \left(\frac{\alpha}{1-\alpha} \frac{3}{4\pi} \frac{1}{n_B} \right)^{\frac{1}{3}} \tag{7}$$

3. Simulation setup

The investigated foil was ClarkY. Chord of the foil was equal to c=70 mm. The foil was placed at distance of 4c from the inlet and 6c from the outlet. The height of the channel was 2.7c. The simulations were performed at constant inlet velocity equal to 10 m/s. The pressure at the outlet was at the level of 72 kPa, which corresponded to the flow conditions at which sheet cavitation can be observed [5]. One side plane of the domain was set to be wall boundary conditions, the opposite plane – symmetry boundary condition. Upper and lower planes of the domain were walls as well. The calculations were performed in OpenFOAM open source code with the solver interPhaseChangeFOAM. The calculations were transient, with time step 0.5 ms. The flow was assumed to be isothermal. The turbulence model chosen was k-ω SST. The following models parameters were chosen: for Kunz model C_e = 20 000, C_c = 1000; for Schnerr&Sauer model $n_B = 1.6 \times 10^{13}$, $r_B = 10^{-6}$ m [6]. First calculations concerned flow of pure water, without cavitation.

4. Results of simulations

The one of parameters that describes the cavitating flow is frequency of vapour structures forming and collapsing. To determine this frequency the vapour volume in the domain was monitored for both cavitation models. The course of this parameter during calculations is showed in Fig. 1.

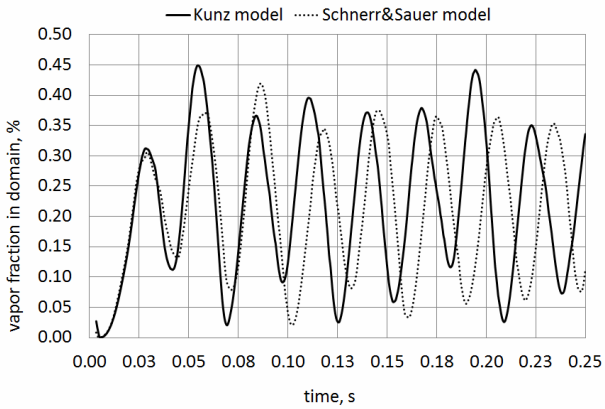

Figure 1: Vapour volume in the domain in time

The calculations lasted for 0.25 s. For Kunz model eight periods of changes were captured, for Schnerr & Sauer model – eight and a half. The frequency of changes for Kunz model was equal to 35.7 Hz, for Schnerr & Sauer model 34.3 Hz. The highest value of vapour volume achieved was slightly higher for Kunz model – 0.46% of the domain. For Schnerr & Sauer model this parameter was 0.419%, which is 10% less.

Figure 2 shows isometric view of the cavitation structures at the peak value of vapour volume in the domain. For both models it was observed that the structures are more sharpened as they were further from the wall (in direction perpendicular to the direction of water flow). The structures tended to attach to the wall, which caused difficulty in determining the period of changes for the whole structure. The collapse of cavitation clouds near the wall occurred less frequently than collapses in the middle or at the opposite end of the computational domain. The strong unsteady flow was observed near trailing edge of the foil, which was also observed during experiments [5]. The changes of pressure in time for both models were also monitored. For Kunz model peaks of pressure were higher, up to 6 bar, when in case of use of Schnerr & Sauer model, the highest peak of pressure achieved 3.7 bar. In both cases the greatest changes of pressure were observed at the end of the profile – near trailing edge.

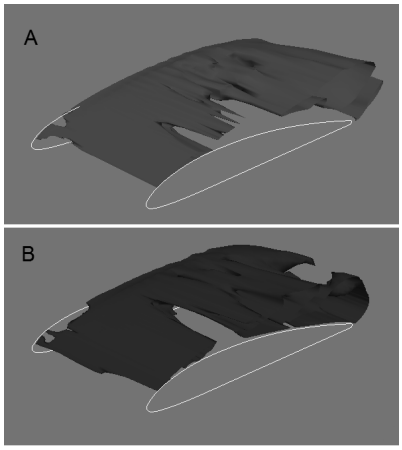

Figure 2: Isometric view of the cavitation structures A) Kunz model; b) Schnerr & Sauer model

5. Conclusions

The two investigated models: Kunz model and Schnerr & Sauer model provided similar results in case of sheet cavitation on foil. For Kunz models the obtained volume of vapour was higher. The frequency of changes for vapour structure was also higher for Kunz model. The cavitating flow features can be observed only in 3D simulations, as distribution of vapour volume fraction along the foil is strongly dependent on distance from the wall the foil is attached to. Simulations affirmed the region near trailing edge, where collapses of cavitation cloud appeared, is exposed to sudden pressure changes, of amplitude even up to 6 bar.

References

[1] Puffary, B., Numerical Modelling of Cavitation, *Design and Analysis of High Speed Pumps*, 2006.

[2] Kunz, R., A preconditioned Navier-Stokes method for two-phase flows with application to cavitation prediction, *Computers & Fluid*, Vol 29, p. 849-875, 2000.

[3] Brennen, C. E., *Cavitation and Bubble Dynamics*, Oxford University Press, 1995.

[4] Schnerr, G. H., Sauer, J., Physical and Numerical Modelling of Unsteady Cavitation Dynamics, *Fourth International Conference on Multiphase Flow*, New Orleans, Louisiana, 2001.

[5] Wang, G., Senocak, I., Shyy, W., Ikohago, T., and Cao, S., Dynamics of attached turbulent cavitating flows, *Progress in Aerospace Sciences*, Vol.37, p. 551 – 581, 2001.

[6] Bensow, R. E., Bark, G., Simulating cavitating flows with LES in OpenFOAM, *5th European Conference On Computational Fluid Dynamics*, 2010.