Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 2, 2014, vol. LX
article No. 1985

Dorota HOMA[*]

COMPARISON OF DIFFERENT MATHEMATICAL MODELS OF CAVITATION

POROVNÁNÍ RŮZNÝCH MATEMATICKÝCH MODELŮ KAVITACE

Abstract

Cavitation occurs during the flow when local pressure drops to the saturation pressure according to the temperature of the flow. It includes both evaporation and condensation of the vapor bubbles, which occur alternately with high frequency. Cavitation can be very dangerous, especially for pumps, because it leads to break of flow continuity, noise, vibration, erosion of blades and change in pump's characteristics. Therefore it is very important for pump designers and users to avoid working in cavitation conditions. Simulation of flow can be very useful in that and can indicate if there is risk of cavitating flow occurrence. As this is a multiphase flow and quite complicated phenomena, there are a few mathematical models describing it. The aim of this paper is to make a short review of them and describe their approach to model cavitation. It is desirable to know differences between them to model this phenomenon properly.

Abstrakt

Kavitace nastává během průtoku při lokálním poklesu tlaku na saturační tlak v závislosti na teplotě průtoku. Zahrnuje odpařování a kondenzaci par bublin, které se střídavě vyskytují s vysokou frekvencí. Kavitace může být velmi nebezpečná, zejména pro čerpadla, protože vede k přerušení kontinuity průtoku, hluku, vibraci, erozi lopatek a změně vlastností čerpadla. Proto je pro designéry čerpadel a uživatele velmi důležité vyvarovat se práci v kavitačních podmínkách. V tom může být velmi užitečná simulace proudění, která může určit, zda existuje riziko výskytu kavitačních toků. Toto je vícefázový tok a jde o docela komplikované jevy, kde existuje několik modelů, které je popisují. Cílem tohoto příspěvku je vytvořit jejich krátký přehled a popsat jejich přístup k modelování kavitace. Je žádoucí, znát rozdíly mezi nimi, pro správné modelování tohoto jevu.

Keywords

Cavitation modelling, Flow over a foil, Cavitation models, Computational Fluid Mechanics, Multiphase Flow modelling

1 INTRODUCTION

The equation that governs the behavior of fluid flow is called the Bernoulli equation. It is a formula derived from energy conservation equation. It states that the sum of head of pressure, velocity and elevation of the streamline must remain constant during the flow when neglecting the hydraulic losses. It means that if there are no changes of elevation, acceleration of the fluid causes the decrease of pressure. When the decrease reaches the critical value, dependent on the temperature of the fluid, the vapor bubbles are created. Then, if the velocity still increases, the bubbles grow as the vaporization on their surface proceeds. When the flow starts to slow down, the pressure rises and

[*] M.Sc. Eng., Institute of Power Engineering and Turbomachinery, Faculty of Power and Environmental Engineering, Silesian University of Technology, Konarskiego 18, Gliwice, tel. (+480) 32 237 1546, e-mail dorota.homa@polsl.pl

these bubbles collapse. In the area around them the pressure is very low (as the density of liquid phase is much higher than the density of vapor phase). The next bubbles are created. The alternately vaporization and condensation happen at high frequency. The collapses of bubbles generate pressure waves which can seriously damage the walls of the channel. The simplest example of cavitation is flow through the convergent-divergent nozzle or flow around a foil in some condition that enables cavitation to happen. In pumps' exploitation cavitation is undesirable. It leads to erosion, generate noise and vibration. It also influences pump characteristic. On the other hand, cavitation can be very useful for example in sludge disintegration to intensify the biogas production or in the navy to produce high speed torpedo as cavitation can reduce friction. In the face of the importance of this phenomenon the ability to simulate it properly is definitely necessary. It is quite complicated regarding to the fact that this is multiphase flow which includes both evaporation and condensation.

2 CAVITATION MODELLING

2.1 Multiphase modelling

Multiphase modelling includes modeling of two or more phases in flow. We can distinguish a few general regimes of multiphase flows [1]:

- Gas–liquid and liquid–liquid
- Gas–solid
- Liquid-solid

Cavitation is an example of gas–liquid multiphase flow regime. In detail – it is bubbly flow, which is characterized of discrete gaseous bubbles flow in the continuous fluid. The physical models of cavitation can be gathered into two main groups: Two–fluid models and One-fluid models [2].

2.2 Two–fluid models

The aim of these models is to solve the conservation equations of discrete phase and continuous phase separately. It can be done by using one of two methods:

- Euler–Euler approach, which stands for solving the conservation equation of each phase by focusing on a single location and observing its flow passing by.
- Euler-Lagrange approach, which include solving the conservation equations of continuous phase by Euler method and the conservation equation of discrete phase along the trajectory of a single bubbles.

These methods are exact but they cost a lot of computation time in case of vapor volume fraction higher than some critical value. According to [2], this value is about 4×10^{-4}, which means that in case of simulating cavitating flow in a nozzle or over the foil the other approach can be applied.

2.3 One–fluid models

The one–fluid models assume that the conservation equations of the mixture of vapor and liquid are solved. As the cavitating flow is most often assumed to be isothermal, only the mass and momentum conservation equation are taken into account. The properties of the mixture are described as follows:

$$\rho = \alpha\rho_v + (1-\alpha)\rho_l \tag{1}$$

$$(\rho u) = \alpha(\rho u)_v + (1-\alpha)(\rho u)_l \tag{2}$$

where:

ρ, ρ_v, ρ_l, – mixture, vapor and liquid density, respectively [$kg\ m^{-3}$],

(ρu), $(\rho u)_v$, $(\rho u)_l$, – mixture, vapor and liquid momentum [$kg\ m^{-2}\ s^{-1}$],

u – mixture velocity [$m\ s^{-1}$],

α – vapor volume fraction [-].

In the one-fluid models group the following methods to obtain the solution are used [2]:

- Two-equation models. In these models the slip between the phases is assumed. Apart from the mixture conservation equations, the two additional equations are solved: conservation equation of liquid or vapor.
- One-equation models. These models assume no slip between the phases, the one additional equation is vapor mass conservation equation:

$$\frac{\partial \alpha \rho_v}{\partial t} + \nabla(\alpha \rho_v u) = R_e - R_c \tag{3}$$

where:

R_e, R_c, – source terms [$kg\ m^{-3}\ s^{-1}$],

The models of 1 equation group differ from each other by the method of determining the source terms.

- Zero-equation models. Using these models means that the conservation equations of mixture are solved, no additional conservation equation is added, but the barotropic state law has to be determined. The function describing the density changes due to the pressure is defined.

In this paper the one-equation models will be described in detail, as they are most popular in case of simulating cavitating flow in turbomachinery.

2.4 One-equation models

The source terms in mass conservation equation of vapor can be determined empirically or by means of Rayleigh-Plesset equation. Rayleigh-Plesset equation describes dynamics of the spherical bubbles in the continuous fluid [3].

$$R_B \frac{D^2 R_B}{Dt^2} + \frac{3}{2}\left(\frac{DR_B}{Dt}\right)^2 = \left(\frac{p_B - p}{\rho_l}\right) - \frac{2S}{\rho_l R_B} \tag{4}$$

where:

R_B – bubble radius [m],

t – time [s],

p – pressure [Pa],

p_B – pressure in the bubble [Pa],

S – surface tension [$N\ m^{-1}$],

In one - equation models the simplifier form is used. The surface tension is neglected, as well as second order terms. Then, the Rayleigh-Plesset equation is modified to:

$$\frac{DR_B}{Dt} = \sqrt{\frac{2}{3}\frac{p_B - p}{\rho_l}} \tag{5}$$

The examples of one-equation models:

Kunz model. In Kunz model the source terms R_e and R_c are determined empirically [4].

$$R_e = \frac{C_{dest}\rho_v \alpha \min(0, p - p_v)}{0.5\rho_l u_\infty^2 t_\infty} \tag{6}$$

$$R_c = \frac{C_{prod}\rho_v \alpha^2 (1-\alpha)}{t_\infty} \tag{7}$$

where:

C_{dest}, C_{prod} – empirical coefficients [-],

t_∞ – mean flow timescale [s],

u_∞ – free stream velocity [$m\ s^{-1}$],

p_v – vaporization pressure [Pa],

Mean flow timescale is derived from characteristic dimension and free stream velocity. The empirical coefficients are dependent on the type of the flow (flow over a foil, flow over a blunt body etc.), therefore it is needed to know them from the literature or to determine them by experiment.

Singhal model. In this model the Rayleigh-Plesset equation is used. The formulas for R_e and R_c are as follows [5]:

$$R_e = F_{vap}\frac{\max\left(1.0, \sqrt{k}\right)(1-f_v)}{\sigma}\rho_l \rho_v \sqrt{\frac{2}{3}\frac{p_v - p}{\rho_l}} \tag{10}$$

$$R_c = F_{cond}\frac{\max\left(1.0, \sqrt{k}\right) f_v}{\sigma}\rho_l \rho_v \sqrt{\frac{2}{3}\frac{p_v - p}{\rho_l}} \tag{11}$$

$$p_v = p_{sat} + \frac{1}{2}(0.39\rho_m k) \tag{12}$$

where:

F_{vap}, F_{cond} – coefficients, equal to 0.02 and 0.01 accordingly [-],

f_v – vapor mass fraction [-],

k – turbulent kinetic energy [$m^2\ s^{-2}$],

σ – surface tension coefficient of the liquid [-],

p_{sat} – saturation pressure [Pa],

Sighal proposed to make some correction due to the turbulent pressure fluctuations. Therefore the vaporization pressure p_v is derived from saturation pressure by the equations (12). The source coefficients are function of liquid, vapor and mixture density.

Zwart–Gerber–Belamri model (ZGB). This model also uses Rayleigh-Plesset equation. The formulas for source terms are different from these proposed by Singhal: [6]

$$R_e = F_{vap}\frac{3\alpha_{nuc}(1-\alpha)\rho_v}{R_B}\sqrt{\frac{2}{3}\frac{p_v - p}{\rho_l}} \tag{13}$$

$$R_c = F_{cond}\frac{3\alpha\rho_v}{R_B}\sqrt{\frac{2}{3}\frac{p_v - p}{\rho_l}} \tag{14}$$

where:

F_{vap}, F_{cond} – coefficients, equal to 0.01 and 50 accordingly [-],

α_{nuc} – nucleation site volume fraction, equal to $5\text{x}10^{-4}$ [-],

R_B – bubble radius, 10^{-6} [m],

In ZGB model the bubble radius is a parameter and has to be set. Authors also assumed that if vapor volume fraction increases the nucleation site density must decrease. They proposed to replace α in equation (13) with $\alpha_{nuc}(1-\alpha)$. The nucleation site volume fraction has to be set. The source terms are function of vapor and liquid density.

Schnerr and Sauer model. The formulas for source terms [7]:

$$R_e = \frac{\rho_v \rho_l}{\rho_m} \alpha(1-\alpha) \frac{3}{R_B} \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_l}} \tag{15}$$

$$R_c = \frac{\rho_v \rho_l}{\rho_m} \alpha(1-\alpha) \frac{3}{R_B} \sqrt{\frac{2}{3} \frac{p - p_v}{\rho_l}} \tag{16}$$

$$R_B = \left(\frac{\alpha}{1-\alpha} \frac{3}{4\pi} \frac{1}{n_b} \right)^{\frac{1}{3}} \tag{17}$$

where:

n_b – number of bubbles per volume of liquid [-],

R_B – bubble radius, 10^{-6} [m],

The source terms are function of mixture, liquid and vapor density, like in Singhal model. Authors proposed relationship between the bubble radius, vapor volume fraction and number of bubbles per volume of liquid. The values of R_B and n_b are set before the calculations.

3. COMPARISON OF KUNZ AND SCHNERR & SAUER MODELS

To show the influence of choice of the cavitation model a short test case was performed using two different models: Kunz and Schnerr & Sauer model. The flow over a Clark-Y foil was selected as the test case. It was 2D model with 3D grid. The calculations were performed by means of OpenFoam source code. Some of the settings are shown in table 1. The time step was set to be adaptive, it was changing according to the Courant number (Courant number less than 1).

Tab. 1 Test case set - up

Chord	70 mm	Side walls bc	symmetry
Angle of attack	8°	Upper/lower wall bc	wall
Type of elements	hexahedra	Inlet velocity	10 m/s
Number of grid elements	315 000	Turbulence intensity at the inlet	5%
Cavitation number	0.8	Turbulence length scale	0.0013 m
Turbulence model	k-ω SST	Reynolds number	700 000
Heat transfer model	isothermal	Temperature	20°C
Solver	interPhaseChangeFoam	Outlet pressure	42220 Pa

bc – boundary condition

Tab. 2 Models parameters

Kunz		Schnerr and Sauer	
Cprod	1000 [8]	n_b	1.6×10^{13}
Cdest	20000 [8]	R_B	10^{-6}

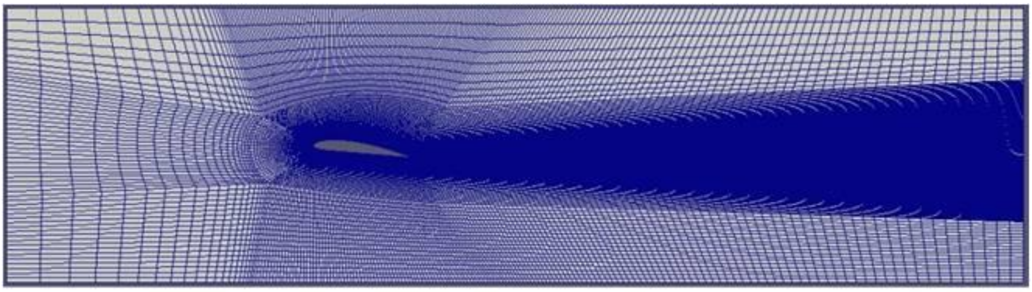

Fig. 1 Side view of grid

In these set of parameters the cloud cavitation regime should be observed. The results of the experiment can be found in [9].The frequency of cloud cavitation for these parameters are equal to 20 Hz. (50 ms for one cycle). It is characteristic of attached front portion and unsteady rear region. In the beginning of the cycle, the cavity starts to grow near the leading edge. It develops and travels downstream the foil bubbles moving with a clockwise rotation. After about the midpoint of the cycle the massive vortex shedding appears. Due to large-scale vortex dynamics the higher pressure near wall region occurs and cavitating flow is pushed away from the wall. A re-entrant flow in the wall region is induced toward the upstream. When it reaches the vicinity of leading edge, the existing cavitating flow detaches from the wall and a new cavitating flow structure forms there.

The results of the calculations are shown in figure 2.

The results show that the chosen cavitation model has strong influence on the pattern of cloud cavitation and its changes in time. In case of both cavitation models the cloud cavitation starts near the leading edge and grows in time. The cycles of grow and collapse can be observed. The strong unsteady cavitating flow in rear region also occurs. For Kunz cavitation model the time of the one cycle of cavitation cloud is 44 ms. In case of Schnerr and Sauer model this time is equal to 49.5 ms, which is closer to the measurement data. The re-entrant flow can be observed in both cases. While calculating with Kunz model it starts from 27.5 ms (close to midpoint of the cycle), with Schnerr and Sauer model – at 44 ms. In case of Schnerr and Sauer model after creating the cavity near the leading edge the strong fluctuations happen in rear region of the foil. Then the cavity enlarges and reaches far area from the foil. The cloud is much bigger and longer than in case of Kunz model use.

To answer which model is better for this set of parameters the further investigation is needed (more cycles included, different cavitation types and angles of attack, drag and lift force changes in time comparison).

4 CONCLUSIONS

Cavitation is a multiphase flow which includes both vaporization of the liquid phase and condensation of vapor bubbles. It is caused by the local pressure depression, which reaches the saturation pressure. Modelling cavitation can be done by two-fluid models (Euler-Euler, Euler-Lagrange methods) and one-fluid models (zero, one, two–equation models). In one-equation model group there is no slip condition between the phases. The modelling effort focuses on the vapor transport equation source terms. These source terms can be derived empirically (Kunz) or from the Rayleigh-Plesset equation (Singhal, Zwart-Gerber-Belamri, Schnerr and Sauer model). To show the meaning of cavitation model the flow over a hydrofoil was simulated using OpenFoam. The boundary conditions were set to values which enabled cloud cavitation to occur. The cavitation model has strong influence on the results: the pattern of flow and the changes in time were different.

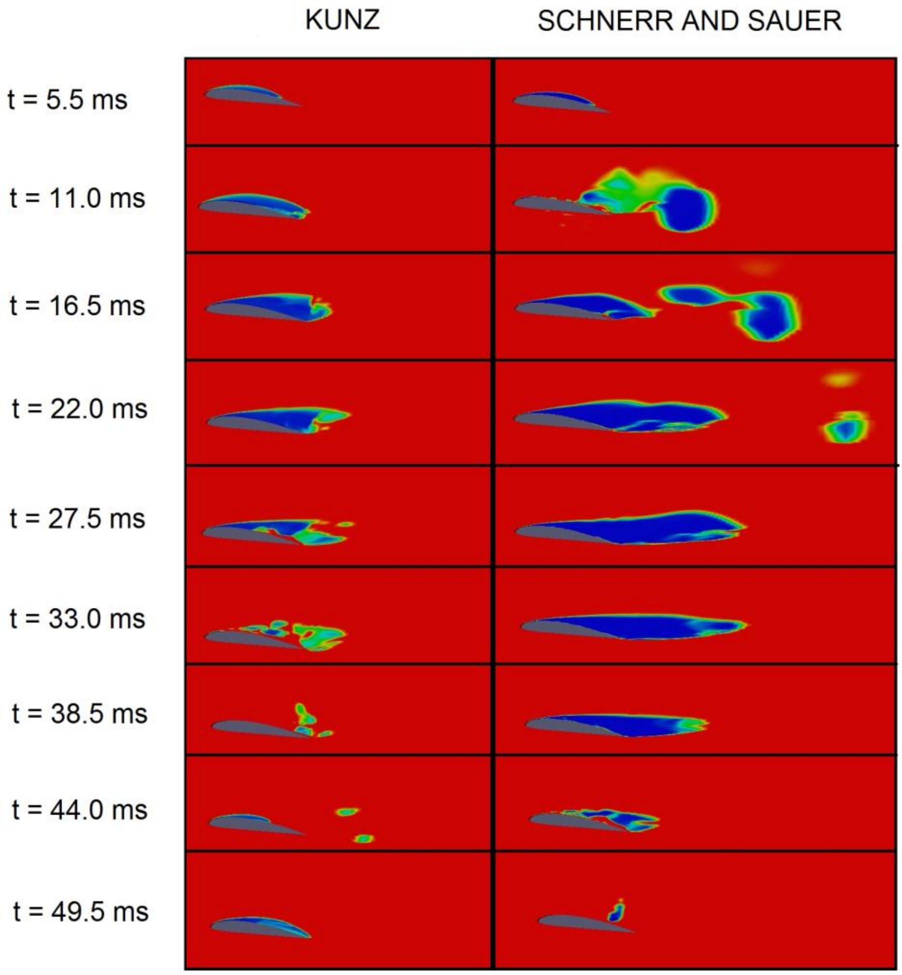

Fig. 2 Changes of cavitation structures in time (liquid volume fraction).

REFERENCES

[1] Fluent User's Guide

[2] PUFFARY, B. Numerical Modelling of Cavitation. *Design and Analysis of High Speed Pumps* 2006. pp. 3-1 – 3-54.

[3] BRENNEN, C. E *Cavitation and Bubble Dynamics*. Oxford University Press. 1995, pp. 48 – 54. ISBN 0-19-509409-3.

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

[5] SINGHAL A. K., LI H. Y., ATHAVALE M. M. & JIANG Y., Mathematical Basis and Validation of the Full Cavitation Model. In *Proceedings of ASME FEDSM'01*. New Orleans, Louisiana2001, pp. 156-158. ISBN 12-34-567-89.

[6] ZWART P. J., GERBER A. G. & BELAMRI T., A Two-Phase Flow Model for Predicting Cavitation Dynamics. In *Proceedings of Fifth International Conference on Multiphase Flow*. Yokohama, Japan 2004,

[7] SCHNERR G. H. & SAUER J., Physical and Numerical Modeling of Unsteady Cavitation Dynamics, In *Proceedings of Fourth International Conference on Multiphase Flow*. New Orleans, USA, 2001,

[8] ROOHI E., ZAHIRI P. & PASANDIDEH-FARD M., Numerical Simulation of Cavitation around a Two –Dimensional Hydrofoil Using VOF Method and LES Turbulence, In *Proceedings of the Eighth International Symposium on Cavitation (CAV 2012)*

[9] WANG G., SENOCAK I., SHYY W., IKOHAGO T. & CAO S., Dynamics of attached turbulent cavitating flows, *Progress in Aerospace Sciences,* 2001, vol. 37 p. 551 – 581.