

Numerical simulation of supercavitation in the constrained flow

A Yu Radzyuk^{1*}, V A Kulagin¹, E B Istyagina¹, T A Pyanykh¹ and M V Kolosov¹

¹ Siberian Federal University, 79 Svobodny Prospect, Krasnoyarsk, Russia

*E-mail: d327@mail.ru

Abstract. Mathematical modeling of the liquids movement, accompanied by the phenomenon of cavitation, is developing intensively, but the supercavitational flow regime has a number of features compared to the localization of the cavity on the streamlined body. The article presents a general approach to the mathematical modeling of the supercavitational flow, describes the used mathematical model. The geometry selected for modeling, the initial parameters and the physical constants characterizing the process are given. Using ANSYS CFX, the flow around a cone installed in a round tube at different flow velocities was calculated. The analysis of the obtained results was carried out, and directions for further research were outlined.

Determining the parameters of the cavitation flow depending on the geometry of the object under study, the type of liquid and flow parameters are important for the practical applications. Various aspects of the calculation of developed cavitation flows are considered in many publications [1-5], for now there are not enough data to illustrate the influence of geometry on the dimensions and shape of cavities, in particular, the results of supercavitation modeling, when the cavity is completely located behind a streamlined obstacle.

1. Mathematical model

Among the variety of theoretical approaches to the analysis of cavitation flows, an approach was chosen that was implemented in ANSYS CFX and was based on modeling the motion of a quasi-homogeneous mixture by the Navier-Stokes equation.

To study the flow, a homogeneous two-phase flow model was used, which is considered as a homogeneous mixture of the gas phase (vapor and air from cavitation seed and dissolved air from the flow) and water.

Thermophysical parameters of the mixture φ_m (mixture density ρ_m and dynamic viscosity μ_m) by their volume fractions, r were determined by the expression:

$$\varphi_m = r_f \varphi_f + (1 - r_f) \varphi_g, \quad (1)$$

where f is liquid, g is steam and gas.

The problem of the cavitation flow of a liquid was considered in a stationary formulation and included the solution of the following equations:
continuity equation:

$$\frac{\partial}{\partial x_j}(\rho_m u_j) = 0, \quad (2)$$

momentum conservation equation:

$$\frac{\partial(\rho_m u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\mu_m + \mu_T) \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) \right]. \quad (3)$$

Here ρ_m is the density of the mixture; \mathbf{u} – local speed of the mixture; μ_m , μ_T – coefficients of dynamic and turbulent viscosity of the mixture; δ_{ij} – Kronecker symbol, $\delta_{ij} = 0 (i \neq j)$, $\delta_{ij} = 1 (i = j)$.

Two new variables are added to the equations of motion taking into account the $k - \varepsilon$ turbulence model: k represents the kinetic energy of turbulence and is defined as the dispersion of velocity oscillations, m^2/s^2 ; turbulent vortex dissipation ε – the rate at which velocity fluctuations dissipate, m^2/s^3 .

The transport equations for the kinetic energy of turbulence k and energy dissipation ε were as follows:

$$\frac{\partial(\rho_m u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu_m + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + G - \rho_m \varepsilon, \quad (4)$$

$$\frac{\partial(\rho_m u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\mu_m + \frac{\mu_T}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + C_1 \frac{\varepsilon}{k} G - C_2 \rho_m \frac{\varepsilon^2}{k}, \quad (5)$$

where G – the rate of generation of turbulence, which is determined according to the equation:

$$G = \tau_{ij} \frac{\partial u_i}{\partial x_j}, \quad (6)$$

where τ_{ij} is the Reynolds stress tensor:

$$\tau_{ij} = -\overline{\rho_m u'_i u'_j} = \mu_T \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2\rho_m k \delta_{ij}}{3}. \quad (7)$$

The coefficient of turbulent viscosity μ_T is determined by the formula:

$$\mu_T = \frac{\rho_m C_\mu k^2}{\varepsilon}. \quad (8)$$

The empirical constants had the following meanings: $\sigma_k = 1.0$; $\sigma_\varepsilon = 1.3$; $C_1 = 1.44$; $C_2 = 1.92$; $C_\mu = 0.09$.

Transfer equation for volume fraction:

$$\frac{\partial(\rho_g r_g u_j)}{\partial x_j} = \dot{m}_{fg} + \dot{m}_{gf}, \quad (9)$$

where r_g is the volume fraction of steam and gas; \dot{m}_{fg} , \dot{m}_{gf} – evaporation and condensation rates.

The phenomenon of cavitation involves the determination of interfacial mass transfer, when the mass is transferred from one phase to another. This applies to both inhomogeneous and homogeneous multi-phase models.

Cavitation processes in the model used are considered on the assumption of thermal equilibrium at the interface. The rate of vapor formation and condensation in the CFX software module is determined from the Rayleigh-Plesset equation describing the growth of a gas bubble in a liquid:

$$R_B \frac{d^2 R_B}{dt^2} + \frac{3}{2} \left(\frac{dR_B}{dt} \right)^2 + \frac{2\sigma}{\rho_f R_B} = \frac{p_v - p}{\rho_f}, \quad (10)$$

where R_B is the bubble radius, p_v is the pressure in the bubble (this is assumed to be the saturated vapor pressure at the temperature of the liquid), p is the pressure in the liquid surrounding the bubble, ρ_f is the density of the liquid, and σ is the coefficient of surface tension between the liquid and vapor. Neglecting terms of the second order and surface tension, equation (10) is reduced to the form:

$$\frac{dR_B}{dt} = \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_f}}. \quad (11)$$

The rate of change in the volume of bubbles is as follows:

$$\frac{dV_B}{dt} = \frac{d}{dt} \left(\frac{4}{3} \pi R_B^3 \right) = 4\pi R_B^2 \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_f}}, \quad (12)$$

and the rate of change of mass of bubbles:

$$\frac{dm_B}{dt} = \rho_g \frac{dV_B}{dt} = 4\pi R_B^2 \rho_g \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_f}}. \quad (13)$$

If N_B is the number of bubbles per unit volume, then their volume fraction r_g can be expressed as:

$$r_g = V_B N_B = \frac{4}{3} \pi R_B^3 N_B \quad (14)$$

and the total interfacial mass transfer rate per unit volume is:

$$\dot{m}_{fg} = N_B \frac{dm_B}{dt} = \frac{3r_g \rho_g}{R_B} \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_f}}. \quad (15)$$

This expression was obtained [6, 7] from the assumption of bubble growth (evaporation). Given condensation, equation (15) can be summarized as follows:

$$\dot{m}_{fg} = F \frac{3r_g \rho_g}{R_B} \sqrt{\frac{2}{3} \frac{p_v - p}{\rho_f}} \operatorname{sgn}(p_v - p), \quad (16)$$

where F is an empirical coefficient taking into account differences in condensation and evaporation rates. In the calculation, the bubble radius R_B will be replaced by the radius of the cavitation nucleus R_{nuc} .

The evaporation process begins in cavitation nuclei, most often being microbubbles of non-condensable gases. When the volume fraction of steam increases, the density of the cavitation nucleus should decrease accordingly, therefore, for vaporization, equation (16) will change as follows:

$$\dot{m}_{fg} = F \frac{3r_{nuc}(1-r_g)\rho_g}{R_{nuc}} \sqrt{\frac{2}{3} \frac{|p_v - p|}{\rho_f}} \operatorname{sgn}(p_v - p), \quad (17)$$

where r_{nuc} is the volume fraction of cavitation seed.

For the calculations, the following empirical constants were taken: $R_{nuc} = 10^{-6}$ m; $r_{nuc} = 0.0001$; $F_{vap} = 50$, $F_{cond} = 0.01$.

The volume fraction of a phase can vary from zero to one, depending on the space occupied in the two-phase flow. Phases must completely fill the entire volume, therefore

$$\sum_{i=1}^2 r_i = 1. \quad (18)$$

Thus, the task is reduced to solving equations (2) - (5), (9).

2. Geometry and Boundary Conditions

In Figure 1 shows the geometry for which the cavitation flow was calculated. In the working section, 500 mm long and 30 mm in diameter, on a cylindrical base with a diameter of 10 mm and a length of 10 mm, a cavitator was installed in a truncated cone with a angle of 53° , a base diameter $d = 15$ mm and a height of 5 mm. The model is axisymmetric, in order to minimize the amount of calculation time and to preserve the accuracy of the results obtained, a 15° segment was chosen as the calculation area.

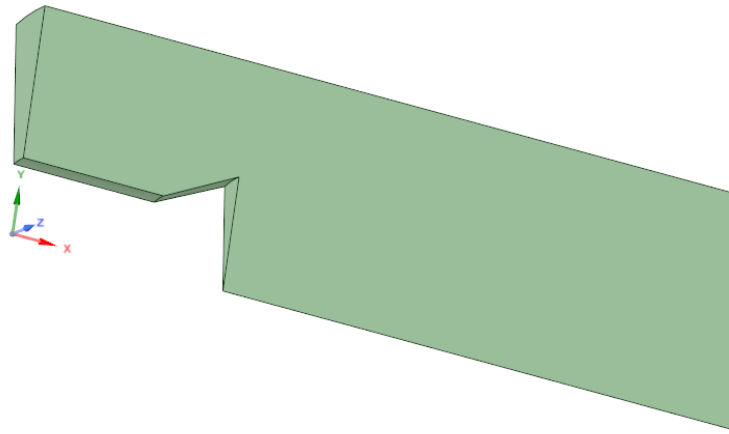


Figure 1. Geometry of the model (input section and cavitator).

The chosen geometry of the cavitator significantly affects the shape and the formation of the dimensions of the cavity and its shape.

To close the system of equations (2) - (5), (9), the following boundary conditions were accepted: at the entrance to the working section - velocities from 15 to 25 m/s; volume fractions of steam and water are 0.0 and 1.0, respectively; at the outlet – pressure 100 kPa. The sticking effect of the flow was taken into account, and the velocity on the walls was zero. The calculations were performed for water with a temperature of 25 °C.

The numerical solution of the problem was carried out using the control volume method using the ANSYS CFX program. The computational grid consisted of 50000 cells.

3. The results of numerical research

In Figure 2 - 5 show the obtained velocity fields, pressures and fractions of steam in the mixture, as well as the distribution of kinetic energy and dissipation rate for the flow rate at the inlet of 19 m/s.

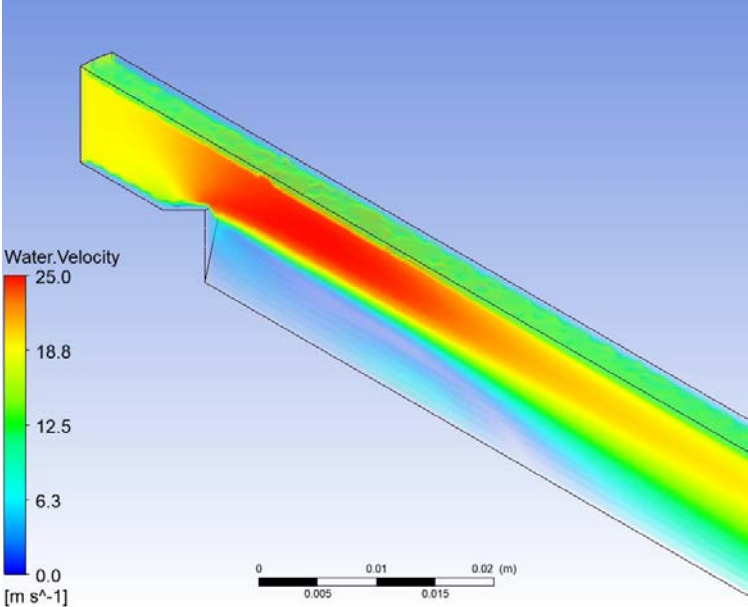


Figure 2. Velocity field.

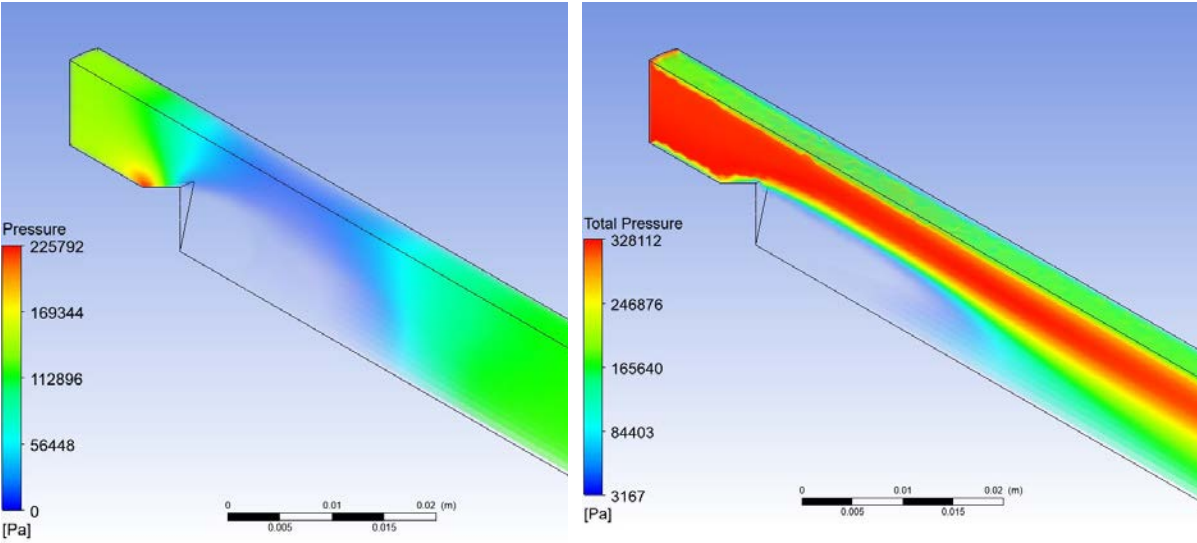


Figure 3. Static (left) and full (right) pressure.

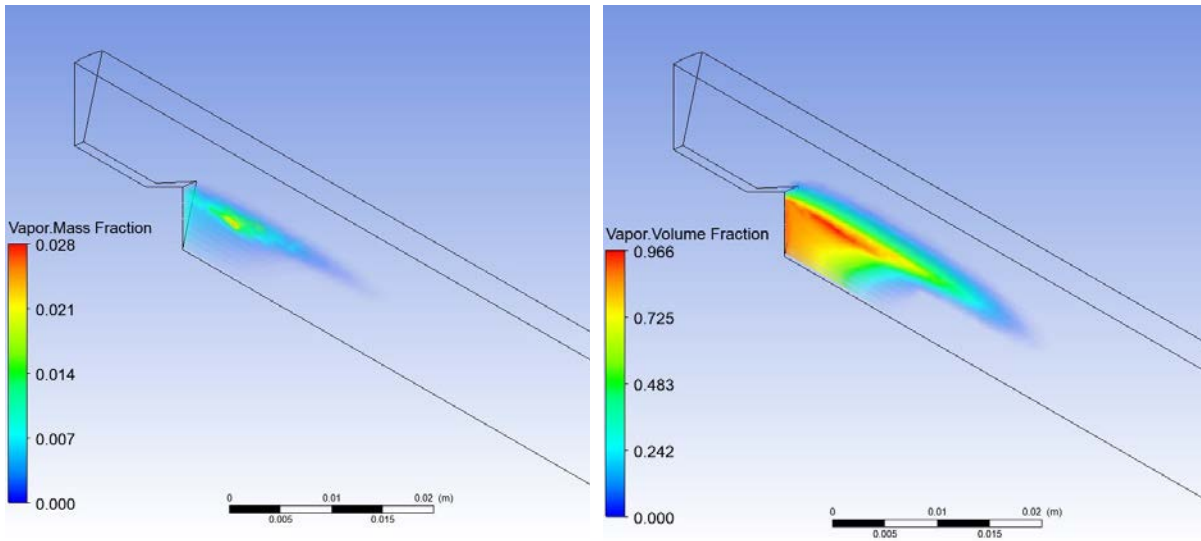


Figure 4. The proportion of steam in the stream: mass (left) and volume (right).

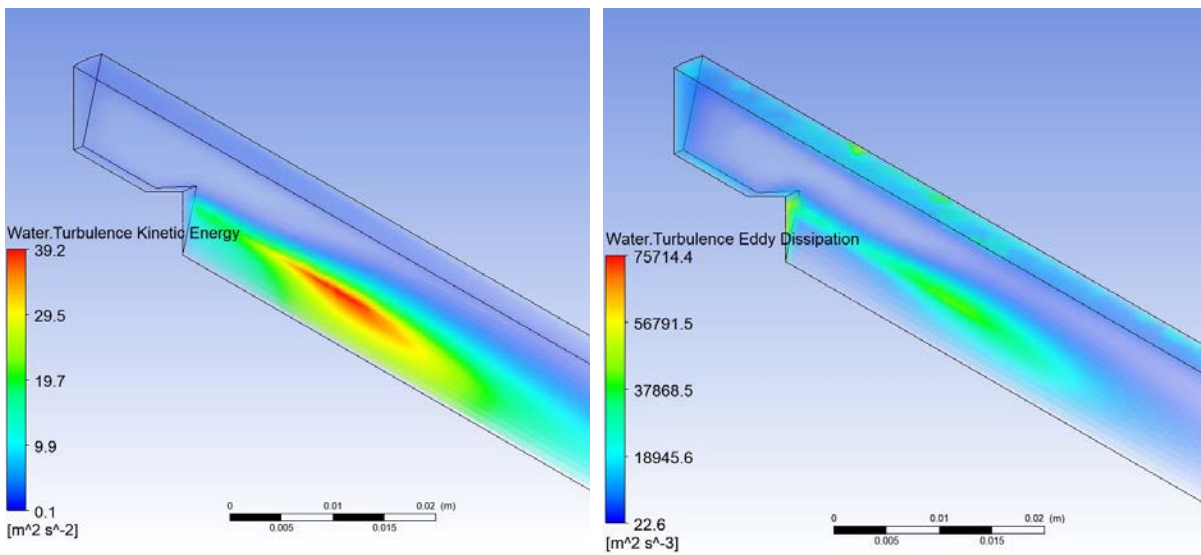


Figure 5. Kinetic energy (left) and dissipation rate (right).

In Figure 6 shows the results of calculations of the volume fraction of steam in the stream for speeds of 16; 17; 18; 19; 19.5; 20 and 25 m/s.

The calculated cavern length for the given conditions is shown in Figure 7.

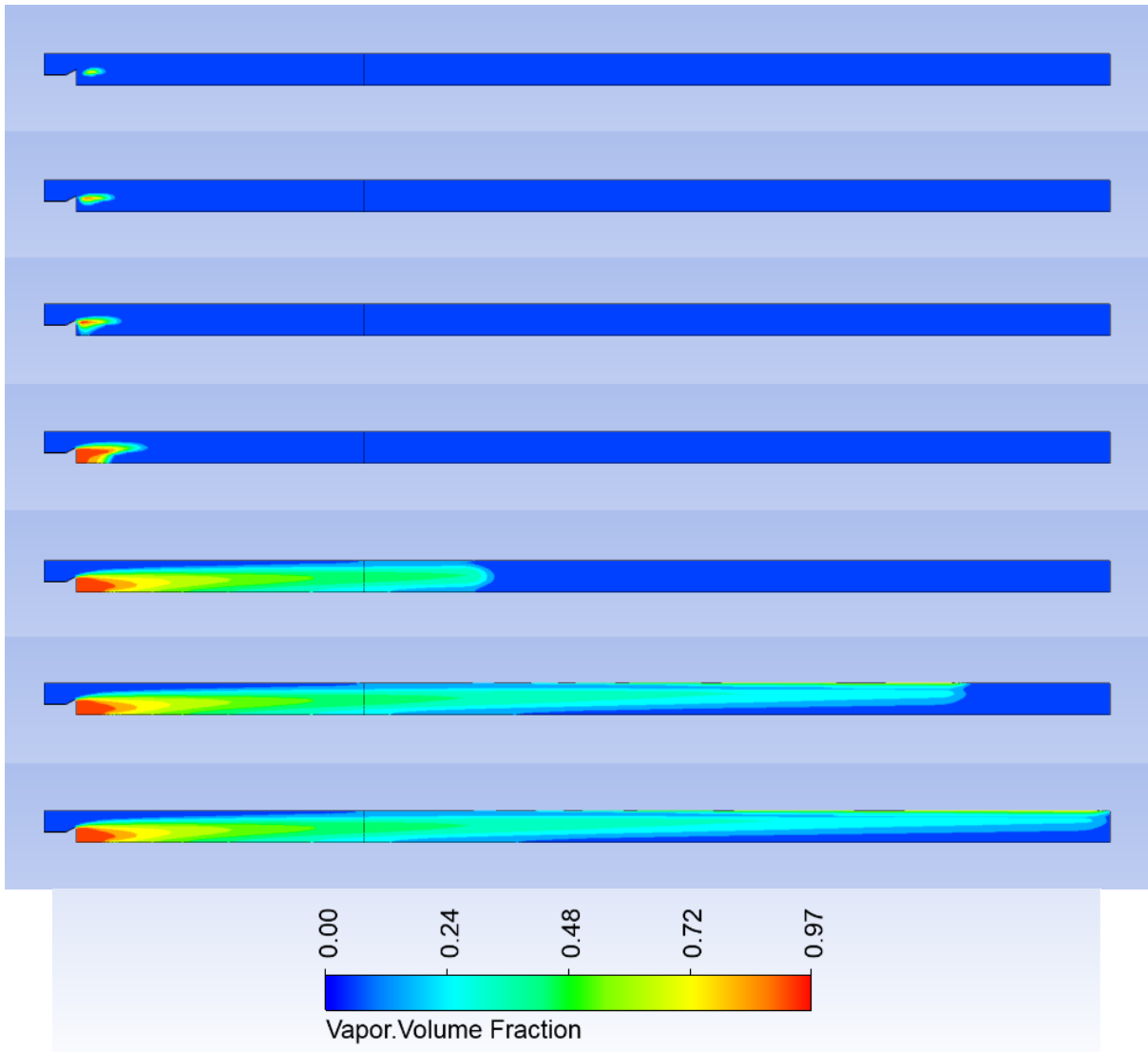


Figure 6. The volume fraction of steam in the stream for speeds of 16; 17; 18; 19; 19.5; 20 and 25 m/s.

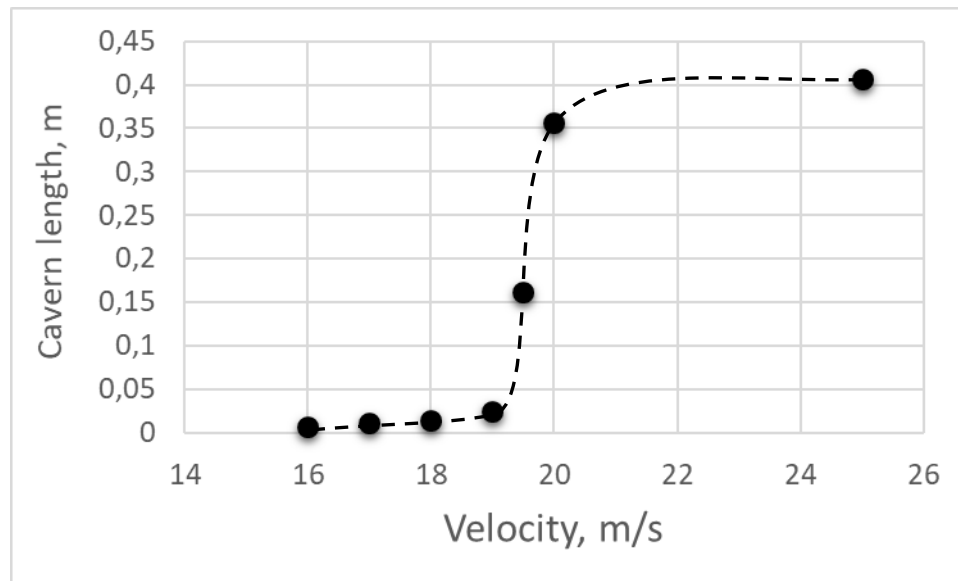


Figure 6. Estimated cavern length.

4. The discussion of the results

The simulation results quite reliably illustrate the cavitation processes occurring in a constrained flow, however, this basic approach to modeling the cavitation flow, implemented in ANSYS CFX, has limitations when calculating supercavitation processes: when calculating modes with a length longer than its width does not imply the formation of a free interface between water and steam, which is observed in a real experiment [9]; moreover, the volume fraction of steam in the calculations has not reached 1.0.

The obtained data can be used to compare and evaluate the accuracy of the results of calculations of supercavitation flows based on modified models.

Acknowledgment

The reported study was funded by RFBR and the government of Krasnoyarsk region according to the research projects № 18-48-242001, 18-41-242004 & 18-41-242008.

References

- [1] Kulagin V A, Pyanykh T A Research of Cavitating Flows by Methods of Mathematical Simulation *Journal of Siberian Federal University. Engineering & Technologies 1* (2012 5) 57-62
- [2] Ivanitsky G K 2011 Numerical modeling of bubble cluster behavior in hydrodynamic cavitation processes *Collection of scientific articles modern science: research, ideas, results, technology 2* (7) pp 52–58
- [3] Dobroselsky K G 2013 The method of investigation of the transverse flow around a cylinder in the hydrodynamic tube *Vestnik NSU Serie: Physics 8*(4) pp 110–117
- [4] Karakulin E A 2010 Study of the dynamics of the occurrence, distribution, growth and closure of bubble cavities and supercovers in a pipe with degassed liquid with water hammer *Vestnik of Samara State Aerospace University. 1* (21) pp 63–74
- [5] Panov L V 2014 *Numerical modeling of cavitation viscous fluid flow in hydro turbines dis. cand. phys.-mat. sciences* (Novosibirsk) p 135
- [6] Zwart P J, Gerber A G, Belamri T 2004 *A Two-Phase Flow Model for Predicting Cavitation*

Dynamics International Conference on Multiphase Flow 152 p 45–56

- [7] Bakir F, Rey R, Gerber A G, Belamri T and Hutchinson B 2004 Numerical and Experimental Investigations of the Cavitating Behavior of an Inducer *Int J Rotating Machinery* 10 pp 15–25
- [8] Ivanov A N 1980 *Hydrodynamics of developed cavitation flows*. (Leningrad: Shipbuilding) p 237
- [9] Kulagin V A, Radzyuk A U, Istyagina E B, Pinykh TA Experimental stand for the study of cavitation flow regimes. *IOP Conference Series: Materials Science and Engineering* vol 450. 2018. <https://iopscience.iop.org/article/10.1088/1757-899X/450/3/032023/meta>