Verification of the results of numerical modeling of the developed cavitation in a cramped flow by experimental data

A.Yu. Radzyuk[1], E.B. Istyagina[1], T.A. Pianykh[1], M.P. Baranova [1,2], I.I. Grishina[1]

[1]Siberian Federal University
[2] Krasnoyarsk State Agrarian University

The existing mathematical models of fluid flows accompanied by cavitation describe quite accurately the flow behavior in wide ranges of input parameters, however, the multiparameter and variety of problems to be solved require experimental confirmation of the calculation results based on the selected models. The article presents the results of calculations based on modeling the flow of a quasi-homogeneous mixture by the Navier-Stokes equation, and their comparison with measurements on an experimental setup with a similar geometry and flow parameters.

Keywords: hydrodynamic stand, cavitation, mathematical modeling, supercavitational flow around, Rayleigh-Plesset equation

Interest in an in-depth study of two-phase flows is due to the wide practical use of process liquids moving at high speeds in channels that have obstacles in the form of shutoff valves and variable channel cross-sections. These conditions contribute to the formation of cavitation flow regimes that affect the stability and reliability of the technological installation as a whole. Theoretical studies of the quasi-homogeneous fluid flow, supplemented by an experimental study, contribute to the improvement of a reliable theory of cavitation flows that adequately describes the real processes.

The scientific literature devoted to this problem is extensive and can be conditionally divided into theoretical [1-4] and experimental works [5-7], characterizing the variety of tasks facing researchers. The complex nature of the described processes (the simultaneous existence of boundary dynamics, phase transition, and a strong change in density), which is reflected in the mathematical formulation of the problem, necessitates the use of experimental data as reliability criteria.

Mathematical model

To simulate the flow in a cavitation installation [8], an approach was used similar to that used by the authors earlier [9, 10] and based on modeling the flow of a quasi-homogeneous mixture by the Navier – Stokes equation.

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

The problem of cavitation fluid flow was considered in a stationary formulation and included the solution of the equation of through flow and the momentum conservation equation.

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

The empirical constants in the transfer equation of turbulence kinetic energy k and energy dissipation ε had the following values: $\sigma_k = 1,0$; $\sigma_\varepsilon = 1,3$; $C_1 = 1,44$; $C_2 = 1,92$, $C_\mu = 0,09$.

The phenomenon of cavitation involves the determination of interphase mass transfer, when the mass is transferred from one phase to another. This applies to both heterogeneous and homogeneous multiphase models.

Transfer equation of proportion by volume

$$(1)$$

where is the density of the mixture of steam and gas is the volume fraction of steam and gas; , - evaporation and condensation rates.

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

$$(2)$$

where RB is the radius of the bubble, pv is the pressure in the bubble (it is assumed that this is the pressure of saturated vapor at the temperature of the liquid), p is the pressure in the liquid surrounding the bubble, is the density of the liquid, and is the surface tension coefficient between the liquid and the vapor. Neglecting terms of the second order and surface tension, equation (2) is reduced to the form:

$$(3)$$

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

$$(4)$$

and the rate of change in the mass of the bubbles:

$$(5)$$

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

$$(6)$$

and the total rate of interphase mass transfer per unit volume is:

$$(7)$$

This expression was obtained [9] from the assumption of bubble growth (evaporation). Taking into account the condensation, equation (7) can be generalized as follows:

$$(8)$$

where F is an empirical coefficient that takes into account the differences in the rates of condensation and evaporation. During calculation, the radius of the cavitation nuclei Rnuc will replace the radius of the bubble RB.

The evaporation process begins in the cavitation cores, which most often are microbubbles of non-condensing gases. When the volume fraction of steam increases, the density of the cavitation core should decrease accordingly, therefore, for vaporization, equation (8) will change as follows:

(9)

where *rnuc* is the volume fraction of cavitation cores.

For calculations, the following values of empirical constants were taken:

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

(10)

To solve the system of equations, the ANSYS CFX software product was used.

The system of equations (2) - (5), (9) is closed by the following boundary conditions: at the entrance to the working section, the speed is set, the volume fractions of steam and water are 0.0 and 1.0, respectively; outlet pressure. The flow sticking effect was taken into account; the velocity on the walls was zero.

Numerical experiment

The effect of the flow velocity on the pressure obtained at the exit of the working section, the formation of the cavity length and the volume fraction of steam in a two-phase flow in the range of moving fluid velocities were numerically studied.

For modeling, a segment of the working area with an angular size of 30 ° was selected (Fig. 1). The study area is supplemented with a direct part to prevent the formation of reverse flows at the outlet of the diffuser.

Figure 1. Modeled area

The used grid had a thickening in the area of decreasing the cross section and, correspondingly, high speeds (Fig. 2). The number of nodes is 132781, the total number of elements is 698205.

Figure 2. The thickening of the grid in the cone area

The comparison of the results of numerical research and measurements on experimental setup

To assess the accuracy of the obtained results using the mathematical model and their adequacy to the experimental results, measurements were made of the cavity length and pressure at the outlet of the diffuser on experimental setup at similar flow rates.

A comparison of the calculated and experimental data of the dependence of the cavity length and pressure on velocity is shown in Fig. 3 and 4.

Figure 3. The comparison of the cavity length

The qualitative dependence of the length of the cavitation cavity on the flow velocity is physically justified and is in satisfactory agreement with the experimental results. As for the quantitative

characteristics, due to the blurred outlines of the cavity it is not possible to accurately determine its dimensions by visual means. Because of this, it is the design characteristics that become priority.

Fig. 4. The comparison of the calculated and measured pressure at the outlet from the diffuser

The calculated and experimental values of pressure in the expanding part of the working section are more consistent with values of velocity of 0.8 - 1.0 m / s. Under natural conditions, with an velocity increase at the entrance to the working section in the diffuser, it increases significantly, which leads to a decrease in pressure.

Figure 5 shows the obtained data on the concentration of the volume fraction of steam in the stream flow rates at the inlet to the calculated section from 0.85 to 1.3 m / s.

Fig. 5. The volume fraction of steam in the stream for different speeds at the entrance to the experimental section

With an increase in the flow velocity, the zone of the two-phase area, due to the development of cavitation processes, increases, which is observed visually in the experimental setup.

The discussion of the results

The obtained simulation results quite reliably describe the processes of cavitation occurring in a restricted flow. From Fig. 3-5 it can be seen that the nature of the dependencies constructed from the calculated results is correlated with the data obtained in the experiments. However, the calculated cavern lengths are slightly larger than the cavern lengths observed in the experimental setup at the same velocities at the entrance to the working section. This discordance can be explained by the imperfection of the mathematical model and the need to adjust empirical constants.

The multiparameter task of motion of a quasi-homogeneous mixture necessitates the introduction of a number of simplifying assumptions when solving it. This inevitably leads to errors in the obtained results. It is possible to minimize them when using certain software products by varying software empirical constants and a thorough analysis of real flow motion models.

The proposed methodology suggests the possibility of improving the numerical experiment in these directions without significantly increasing the estimated time to obtain the results. The considered algorithm allows analyzing the hydrodynamic state of cavitation flows in a wide range of operating parameters and resistance configurations.

The obtained data can be used to compare and assess the reliability of the results of calculations of supercavitation flows based on modified models, and also be of independent interest in the analysis of cavitation phenomena in technological installations.