Increasing Boiling Fluid Flowing Efficiency from Motive Nozzles of Two-Phase Ejectors

Serhii Sharapov 1, Oleh Chekh 1, Danylo Husiev 1, Volodymyr Klymenko 2 and Oleksiy Shaparenko 3

1 Technical Thermophysics Department, Sumy State University, 2, Rymsky-Korsakov st., 40007 Sumy, Ukraine;
2 Department of Mathematical Analysis and Optimization Methods, Sumy State University, 2, Rymsky-Korsakov st., 40007 Sumy, Ukraine;
3 Department of Applied Materials Science and Technology of Structural Materials, Sumy State University, 2, Rymsky-Korsakov st., 40007 Sumy, Ukraine

d.husiev@kttf.sumdu.edu.ua

Abstract. The article contains the possibility of increasing boiling fluid flowing efficiency from expanding channels. This process takes place in the motive flow nozzle of a liquid-vapor ejector, working on the principle of thermal stream compression. Efficiency increasing by profiling the diffuser part of the nozzle. Modern industry uses nozzles, which are like de Laval nozzles, with straight walls of the diffusers. The authors suggest paying closer attention to profiling these nozzles, which might increase their efficiency and improve their gas-dynamic characteristics. For comparison, we choose a channel of a traditional form (with straight walls of the diffuser) and a channel of parabolic shape. The article contains a mathematical model to calculate the process of flowing the boiling fluid from the authors-designed channels – the peculiarities of this model that appear after changing the geometry of its streaming part. We obtain comparative analysis calculation results based on the mathematical model and the Ansys CFX workflow model. As a result of numerical calculation using the authors mathematical model and modelling in the Ansys CFX software package, it concludes that the parabolic shape of the diffuser is the most favourable. In the boiling process, the liquid central core is boiling at the optimum distance from the nozzle throat, and the flow of a stable vapor structure with the required pressure value for each regime forming at the outlet.

1. Introduction

Technological processes under vacuum are increasingly using in modern industry. But, not all processes realizing in atmospheric pressure conditions. Also, with pressure below atmospheric, we obtain a higher quality of final products, for example, in vacuum metallurgy, the amount of impurities in steel that are present during casting under atmospheric pressure is significant [1]. Main areas of application: vacuum cooling systems for biodiesel production, installations for pumping vapor-air mixture from steam turbines condensers, vacuum systems of evaporation plants, heat pumps, heating installations, etc. [2, 3].

Using the primary flow energy is one of the ways to create a vacuum of a secondary flow in vapor jet ejectors. This type of jet apparatus uses widely in many industries. It has a simple design and reliable operation. Wherein vapor jet ejectors have some significant disadvantages associated with imperfections in their workflow [4, 5].

The application of a new type of two-phase jet apparatus becomes topical, but the workflow complexities hamper their development. Recently, international research in this industry has become popular [6].
The most promising type of two-phase jet apparatus is a liquid-vapor ejector, which operates on the principle of thermal jet compression [7–9]. The principle of jet thermocompression is to use the working fluid subcooled to saturation as an active flow. Fluid starts boiling in the throat and the nozzle diffuser. The nozzle's outlet section forms a supersonic jet of a dispersed vapor-droplet structure, which injects a passive flow. The jet has a high volumetric vapor content. Compared with vapor ejectors, it has a positive effect on workflow efficiency.

The primary flow in the diffuser of the motive nozzle is the main factor determining the efficiency of a liquid-vapor ejector. Theoretical and experimental studies of the motive nozzles, with geometry close to a de Laval nozzle, were carried out. The motive nozzle has a straight wall of the diffuser geometry. Studies show the high efficiency of the process of evaporation, in which the motive nozzle velocity ratio reaches 92–97% [10, 11]. However, negative factors occur in the motive nozzle with such diffuser geometry. There is a flow separation from the channel walls, shock waves and condensation shocks, and an unstable velocity profile over the channel area [12, 13].

The above disadvantages of the motive nozzle design lead to profiling its flow part to improve the gas-dynamic characteristics.

2. Numerical Investigation

2.1. Motive nozzle design

The authors studied the issue of profiling boiling liquid nozzles. This research allowed us to determine the most appropriate diffuser forms: the logarithmic form, the parabolic form, the elliptical form, and the form calculated by the Vitoshinsky formula (Fig. 1).
Fig. 1 Motive nozzle design:
   a) – straight wall nozzle [7], b) – logarithmic form,
   c) parabolic form, d) – elliptical form, e) Vitoshinsky form

2.2. Mathematical Model

Calculations are carrying out according to the authors method. It is basing on the mathematical model for the vacuum operation regime, which is an improved model of V. N. Marchenko for the compressor operation regime [9].

This model is basing on a dynamic model of the flow of a metastable superheated fluid. The model use the method of indirect determination of averaged flow parameters. These are the distribution of phase temperatures, velocities, vapor mass fraction, and discrete phase sizes.

The essence of this method is the joint consideration of experimental data [14]. This method includes flow rate, jet pulse, static pressure distribution, and flow visualization. Balance equations: the state of thermally metastable vapor-droplet medium, mass conservation (taking into account the phase transition), total enthalpy (first law of thermodynamics), amount of movement (impulse), entropy production (second law of thermodynamics). And the solution, together with the equations of vapor mass fraction, the channel geometry calculation, and the tangential stress on the wall.

3. CFD Simulation

3.1. Initial parameters

Software ANSYS CFX is used to simulate the flow of the nozzle. The standard system of equations Navier – Stokes is adopted as a mathematical model. The k-ε model is used to simulate turbulence. It is a joint solution of the Navier-Stokes equations for time-averaged constituent variables and additional equations for determining the pulsation components:
   — continuity equation (mass conservation);
   — equation of motion (conservation of momentum);
   — transfer equation for turbulent kinetic energy;
   — equation dissipation of turbulent kinetic energy.

The Rayleigh – Plesset equation describes boiling kinetics in the diffuser of the nozzle [15]:

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

where $R_B$ - vapor bubble radius; $\sigma$ - coefficient of surface tension between liquid and vapor; $p_v$ - fluid density; $p_v$ - saturated vapor pressure; $p_f$ - fluid pressure; $p$ - fluid pressure around the bubble.

The nozzle is axisymmetric. The calculation mesh generation is carried out automatically. It consists of 30,000 cells approximately and is compacted in places of flow restructuring (Fig. 2).
3.2. Working fluid properties
Water using as a working fluid. Properties of water are in the ANSYS-CFX internal material database industrial standard IAPWS. Water boils while passing through the primary flow nozzle and in characteristic sections has the operating parameters shown in Table 1.
The regimes are selected in Table 1 based on the main branches of the application of two-phase ejectors. The first regime uses in binary steam turbine installations for small-scale power generation, the second regime - in installations for biodiesel production, the third - in heat pump installations for heating systems, the fourth - in vacuum evaporation installations.

3.3. Boundary conditions
Boundary conditions (Fig. 3) are defined as "Inlet" with "Bulk Mass Flow Rate", "Wall" without roughness, and "Outlet" with "Average Static Pressure". The ambient pressure is assumed to be equal to atmospheric pressure under normal conditions 101325 Pa.

| Regime | $p_{in}$, bar | $t_{in}$, °C | $p_{out}$, bar |
|--------|---------------|--------------|---------------|
| 1      | 35            | 203          | 1             |
| 2      | 20            | 178          | 0.742         |
| 3      | 10            | 150          | 0.502         |
| 4      | 3             | 110          | 0.289         |

4. Results and discussion
The mathematical model proposed by the authors makes it possible to calculate an active flow nozzle with straight diffuser walls. Therefore, to assess the feasibility of profiling, we simulated the process of a nozzle of various geometry in the Ansys CFX software.
Figures 4–20 show the results of numerical simulation of various geometry of the diffuser of the nozzles. The dependences of changes in parameters, such as pressure, velocity, Mach number, and vapor mass fraction, are determined. The efficiency of the process of the active fluid flow from these nozzles is also determined. The results of the comparative analysis are given for the first regime. On the other regimes, the results are similar.

4.1. Numbering

Figure 4 presents the results of the CFD simulation in the ANSYS CFX software the pressure distribution along with the nozzle. Figure 4, shows the pressure distribution along the length of the nozzle with straight walls depending on the initial parameters (Table 1).

Figure 5 is a graph of pressure distribution along the nozzle length, which shows the calculation results according to the method described in [5, 9] and according to the CFD simulation data obtained.

Figures 4 and 5 show that the required pressure is reached at the nozzle outlet in all regimes. But with decreasing the initial parameters of the active flow in the nozzle diffuser, occur faster evaporation of the liquid central core. This effect has a positive impact on the completion of the boiling process.

Figures 6–8 show the CFD simulation results of pressure distributions along the length of nozzles with different diffusers (Fig. 1, b–e), depending on the initial parameters (Table 1).
Fig. 5 Pressure distribution in straight-wall nozzle:
– – – – numerical investigation results, ————– CFD simulation results, ● 1st regime, ▲ 2nd regime, ◆ 3rd regime, × 4th regime)

Fig. 6 Pressure distribution in nozzles (Fig. 1, b–e) for the 1st regime
The evaporation of the central core of the fluid occurs most rapidly in a nozzle with an elliptical form (Fig. 6, c). However, possible flow rotating in the diffuser, associated with a sudden expansion after the throat (Fig. 7, a). In the Vitoshinsky nozzle, the evaporation of the central core of the liquid occurs at a significant distance from the throat (Fig. 6, d). This effect adversely affects the process of boiling.

As shown in Figure 8 in the nozzles shown in Figure 1, b – d, the evaporation of the central core of the liquid occurs faster than that of a nozzle with straight walls of the diffuser (Fig 1, a).

4.2. Velocity and Mach number distribution
Figure 9 shows the results of a CFD simulation of a velocity change along the length of a nozzle with straight walls (Fig. 1, a) depending on the initial parameters (Table 1).

Figures 10 and 11 are graphs of the dependence of the velocity distribution and the Mach number along the nozzle length. It presents the calculation results according to the method in [5, 9] and according to the data from the CFD simulation.
Figures 9 and 10 show that with a pressure decrease in all regimes, the velocity at the nozzle outlet decreases. The faster the evaporation of the central core of the liquid, the faster the stabilization of the velocity profile.

As shown in Figure 11, a supersonic flow regime is achieving in each of the regimes, and the lower the nozzle outlet pressure, the greater the Mach number. The difference in the graphs of the Mach number in the calculation method and CFD simulation is explained by the fact that the calculation method contains the law for changing the Mach number for a two-phase mixture, and in ANSYS CFX the Mach numbers for individual phases are considered separately.
Figures 12–14 show the results of the CFD simulation of speed variations along the length of nozzles with different geometry of the diffuser (Fig. 1, b–e) depending on the initial parameters (Table 1). As shown in Figure 12 and Figure 13, the velocity profile stabilizes most quickly in a nozzle with an elliptical form of the diffuser (Fig. 12, c, Fig. 13, a). This effect is due to the longitudinal coordinate of the evaporation of the liquid central core. The speed of 420 m/s is reached at the nozzle outlet. In nozzles with a logarithmic and parabolic form, the velocity stabilization is approximately the same (Fig. 13 b, c). The velocity value at the outlet of these nozzles is approximately 380–400 m/s.
In the Vitosinsky nozzle, there is a central core of the fluid in the nozzle outlet. And there is a significant imbalance in the velocity profile (Fig 12, d, Fig. 13, d). It affects the speed of the nozzle outlet, which is 260 m/s.

In the logarithmic and parabolic nozzles, shown in Figure 1, b – d, the optimal velocity is 350–400 m/s, as shown in Figure 14. Speed reduction at the nozzle outlet is better for boiling and heat and mass transfer processes at the phase boundary.

4.3. Vapor Mass Fraction Distribution

Figure 15 shows the CFD simulation results of vapor mass fraction distribution along the nozzle length with straight walls (Fig. 1, a) depending on the initial parameters (Table 1).

Figure 16 shows the graphs of the vapor mass fraction distribution along the nozzle length. According to the method in [5, 9], the calculation results according to the data obtained from the CFD simulation.

When the initial parameters of the active flow at the nozzle inlet decrease, then the mass vapor fraction at the outlet also decreases. This effect indicates the incompleteness of heat and mass transfer processes between the vapor and liquid phases (Figure 15 and Figure 16).

Figures 17–20 presents the results of a CFD simulation of a vapor mass fraction distribution along the length of the nozzles with different geometry of the diffuser (Fig. 1, b – e) depending on the initial parameters (Table 1).

At nozzles with a logarithmic and parabolic form of the diffuser, the optimal vapor mass fraction in the nozzle outlet is 0.45–0.48 kg/kg (see Figures 17–19).
The elliptical nozzle and the Vitosinsky nozzle have many features. This sharp increase presence in the vapor mass fraction due to the feature of the geometry of these nozzles.

Thus, in a nozzle with an elliptical form, an abrupt change in the channel geometry (sudden expansion) occurs in the initial section (Fig. 18, a). The vapor mass fraction in the nozzle outlet increases and equals 0.505 kg/kg. At the nozzle with the Vitosinsky form, this area is closer to the nozzle outlet (Fig. 19, d), but the form is concave, leading to a decrease in the outlet vapor mass fraction (0.32 kg/kg).
**Fig. 17** Vapor mass fraction distribution in nozzles (Fig. 1, b–e) for the 1st regime

**Fig. 18** Vapor mass fraction distribution in nozzles (Fig. 1, b–e) for the 1st regime

**Fig. 19** Vapor mass fraction distribution in nozzles (Fig. 1, b–e) for the 1st regime
Fig. 20 CFD simulation results of velocity distribution in nozzles for the 1st regime: –––––––– straight wall nozzle, –––––––– logarithmic form, –––––––– parabolic form, –––––––– elliptical form, –––––– Vitoshinsky form

In the nozzles shown in Figure 1, b–d, the most suitable vapor mass fraction consists at the level of 0.45–0.48 kg/kg (see Figure 20), which achieve in the logarithmic and parabolic form nozzles.

4.4 Efficiency of Nozzles

As a criterion determining the efficiency of fluid outflow from the nozzle, using the nozzle velocity coefficient $\phi_a$. It is the ratio of the velocity at the nozzle outlet $v_a$ to the isentropic velocity at the nozzle outlet $v_{as}$:

$$\phi_a = \frac{v_a}{v_{as}}$$  \hspace{1cm} (1)

Figure 21 shows the dependences of the nozzle velocity coefficient on the level of expansion of the working fluid in the nozzle $p_{in}/p_{out}$. 
Figure 21 shows that profiling the nozzles in which the boiling liquid flows has a positive effect. This results in an increase in the speed coefficient from the values $\phi_a = 0.92$–0.97 to the values $\phi_a = 0.95$–0.98. Elliptical nozzles have the highest rates of speed.

5 Conclusions
The authors made a numerical analysis and CFD simulation in the ANSYS CFX software nozzles of various geometry of the diffuser. As a result, we can draw the following conclusions:

1. Profiling the diffuser of the nozzle in which the boiling liquid flows has a positive effect on the nature of the flow process. It is possible to reduce flow separation from the channel walls, shock waves, and condensation shocks and obtain a stable velocity profile over the channel area.

2. The parabolic form is the most favorable because the central core of the liquid is boiling at the optimum distance from the nozzle throat, and the flow of a stable vapor structure with the required pressure value for each regime is formed at the outlet. Speed value at $w_a = 380$–$400$ m/s with the optimum vapor mass fraction in the range $x_a = 0.45$–0.48 kg/kg.

3. Profiling the diffuser of the nozzle increases the efficiency of the flow process. An increase in the velocity coefficient from a value $\phi_a = 0.92$–0.97 for a nozzle with straight walls to the value $\phi_a = 0.95$–0.98 for a nozzle with a parabolic diffuser it proves.

4. Vitoshinsky form nozzle is not suitable for the flow of boiling liquid. It is strongly narrow, and the evaporation of the central core of the fluid occurs at a significant distance from the throat. It also negatively affects the velocity profile and the vapor mass fraction at the nozzle outlet.

5. References
[1] Silla, H. Chemical process engineering. design and economics. New York–Basel, Marcel Decker Inc, 2003. – 483 p.
[2] Sharapov S. O. Application of jet thermal compression for increasing the efficiency of vacuum systems [Text] / S. O. Sharapov, V. M. Arsenyev, V. M. Kozin // IOP Conference Series: "Materials Science and Engineering". – 2017. – Vol. 233.

[3] Sharapov, S., Husiev, D., Panchenko, V., Kozin, V., & Baha, V. (2020). Analysis of the possibility of using R718 for a heat pump of a heating system based on a liquid-vapor ejector. Eastern-European Journal of Enterprise Technologies, 6(8 (108), 39–44. https://doi.org/10.15587/1729-4061.2020.217274

[4] Austin, F., Street, H., Applied chemical process design. Front Cover. Plenum Press, 1978 – Science – 294 p.

[5] Power, Robert B., Steam Jet Ejectors for the Process Industries. McGraw–Hill Chemical Engineering, 1993. – 498 p.

[6] Shurchkova Ju. Adiabatnoe vskipanie. Prakticheskoe ispol'zovanie – Kyiv: Naukova dumka, 1999. – 228 p.

[7] Marchenko V. Strujnaja termokompressornaja ustanovka (STK): naznachenie, princip dejstvija, termodinamicheskaja model' i rezul'taty raschetnyh issledovanij parametrov rabochego processa – Visnik Sums'kogo derzhavnogo universitetu. Serija Tehnichni nauki, 2004. – 50–60 pp.

[8] Sharapov, S., Starchenko, M., Protosenko, M., Panchenko, V., Kovtun, V. (2019). Improvement of the vacuum cooling system for biodiesel production. Journal of Engineering Sciences, Vol. 6(1), pp. F6-F11.

[9] Marchenko V. Perspektivy primeneniya i rezul'taty chislennogo modelirovaniya strujnoj termokompressornoj ustanovki – Visnik Sums'kogo derzhavnogo universitetu. Serija Tehnichni nauki. – 2005. – 36–40 pp.

[10] Tonkonog V Fazovye perehody v potokе zhidkosti – Trudy Kazanskogo gos. Tehn. Universiteta im. A. N. Tupoleva. – 2004.

[11] Chang, Paul K. Separation of Flow. Pergamon, 1970 – 796 p.

[12] Chekh O., Sharapov S., Prokopov M., Kozin V., Butrymowicz D. Cavitation in nozzle: the effect of pressure on the vapor content. – Advances in Design, Simulation and Manufacturing II, pp 522-530 https://doi.org/10.1007/978-3-030-22365-6_52

[13] Sharapov S. Experimental investigation of liquid-vapor ejector with conical mixing chamber / S. Sharapov, V. Arsenyev, V. Kozin // Technology audit and production reserves. – 2016. - №4/1 (30). – P. 50-55.

[14] Liao Y. and Lucas D. 2015 3D CFD simulation of flashing flows in a converging-diverging nozzle Nucl. Eng. Des. 292 149-163.

[15] ANSYS INC.: CFX–Solver Theory Guide, pp. 390 (2009)