10.46299/ISG.2022.MONO.TECH.4.4.3

4.3 Numerical modeling of a rotary-pulsation apparatus with rectangular holes (3D model)

The following section presents the results of numerical simulation in the working chamber RPA with rectangular holes using a 3D model.

Development of 3D geometry of the rotor-pulsation apparatus

In fig. 1 schematically shows a rotary-pulsation apparatus (RPA) for feed preparation. The boot apparatus is made in SOLID WORKS, then loaded into ANSYS Design Modeler. The FILL component is selected to simulate the mixture. Boundary conditions were set and the project was transferred to ANSYS Meshing.

Figure 2 shows the boundary conditions of RPA. At the inlet limit of the inlet, a mixture with a mass flow rate of 0.92 kg/s with a temperature of +20 ºC is loaded. The output limit of the outlet is set by pressure (pressure outlet). RPA case is made of aluminum. The temperature of +20 ºC is set on the surface of the metal case. The volume of RPA stator is $2{,}181 \cdot 10^{-3}$ m^3 (Fig. 1). The volume of RPA rotor is $3{,}798 \cdot 10^{-4}$ m^3. The rotor, on the z axis, rotates at a speed of 3000 rpm. Numerical simulations were performed three times with a constant dynamic viscosity 0,2; 1,1 and 2,1 Pa·s.

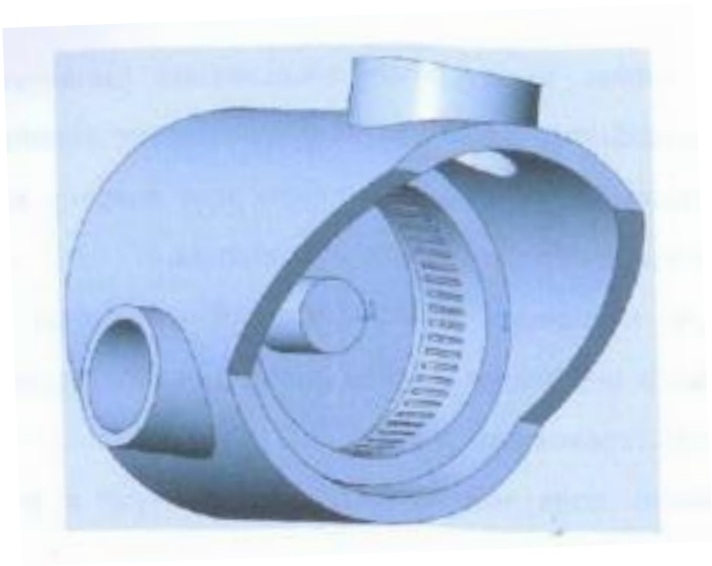


Fig. 1 General view of the rotor-stator system in RPA

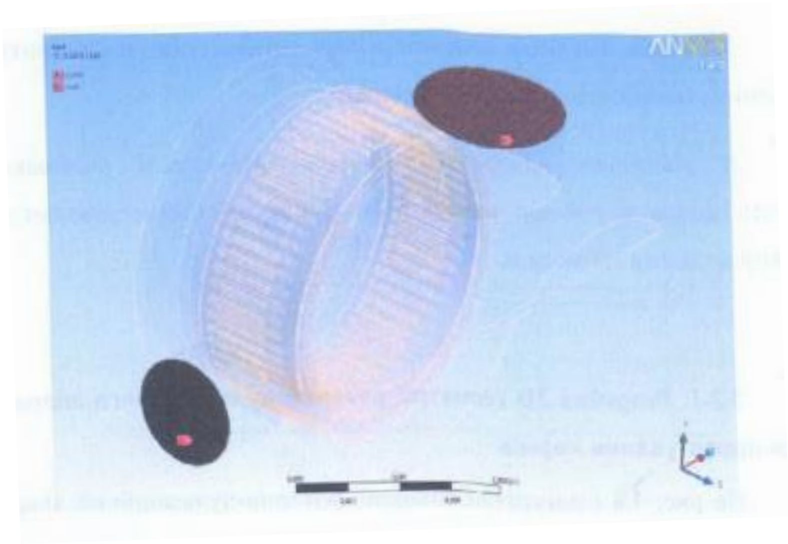


Fig. 2 Boundary conditions in RPA channels

For numerical modeling, a mixture of water and grain in the ratio 10x1 was introduced. The thermophysical properties of water, which varies with temperature, are taken from the relevant tables. The thermophysical properties of grain are considered independent of temperature and equal to: density – 770 kg/m^3; the thermal conductivity coefficient – 0,11 W/m·ºC; heat capacity – 1550 kJ/kg·ºC.

The mixture of water/grain (Y) for all required thermophysical quantities is calculated according by the following formula

$$Y = \frac{10x_1 + 1x_2}{x_1 + x_2}, \tag{1}$$

where x is the thermophysical value; 1 – the value of water; 2 – the value of grain. The thermophysical properties of a liquid grain mixture depend on the temperature.

Using the finite element method to build a 3D grid in ANSYS Meshing

Currently, Finite Elements Method FEM has found wide application in solving problems of thermal conductivity in solids and in the calculation of materials for strength. In addition, this method is used to calculate the flow of liquids or gases [252]. There are also methods Numerical that combine elements of the finite volume method and the finite element method [253, 254]. The combination of these methods allows the use of a wider range of computational grids (pyramidal, tetragonal, prismatic, polyhedral), which is necessary when solving problems with complex geometry. This approach is used by CFD packages Ansys CFX, Ansys Fluent, Star-CD, Star-CCM+, Comsol, etc.

The grid was built out in the ANSYS Meshing generator based on the Workbench platform. In the table 1 shows the setting parameters and grid results in ANSYS Meshing for further transfer of the calculation of heat and mass transfer and hydrodynamics in Ansys Fluent. The geometry is built in real size. The number of elements and faces is quite large (table 1). Due to the large size of RPA, the size of the element and the face is not increased much due to limited production and design power of the computer.

Table 1. Grid construction parameters for RPA

Parameter	Value
Grid quality indicator (orthogonal quality)	0,161
Number of elements, pcs.	4485359
Number of nodes, pcs.	4791394
Curvature angle, city	45
Method	Cut-Cell
Maximum size, m	$1 \cdot 10^{-3}$
Minimum size, m	$5 \cdot 10^{-4}$

In Fig. 3-5 shows the constructed RPA grid. The maximum grid size for the entire volume does not exceed $1 \cdot 10^{-3}$ m. These measures are used to improve the calculation of hydrodynamics. In the section YZ and YX (Fig. 4), the thickening of the grid in the grooves of the rotor and stator is clearly visible. The size of the thickening of the grid near the grooves is $5 \cdot 10^{-4}$ m (Fig. 5). Thanks to which you can better assess the quality and shortcomings of the grid itself. The number of grid elements is about 4,5 million. The quality of the grid in orthogonal quality is 0,161. Detailed results are given in table 1.

Using the finite element method, a 3D calculation grid in ANSYS Meshing software package for solving hydrodynamics and heat transfer problems in RPA was constructed. As a result of building various grids for CFD models, the most optimal and high-quality one was chosen, which allows to obtain reliable and most accurate results of heat and mass transfer processes in RPA.

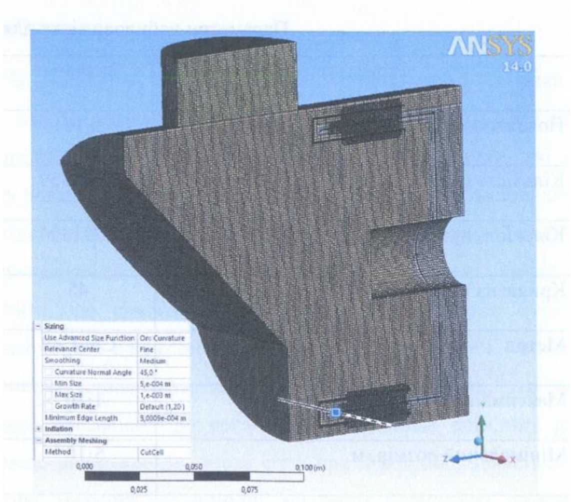


Fig. 3 The result of the constructed grid in the longitudinal section YZ

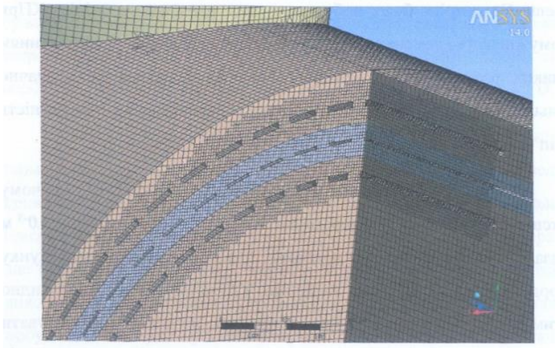


Fig. 4 The result of the constructed grid in the section of YZ and YX

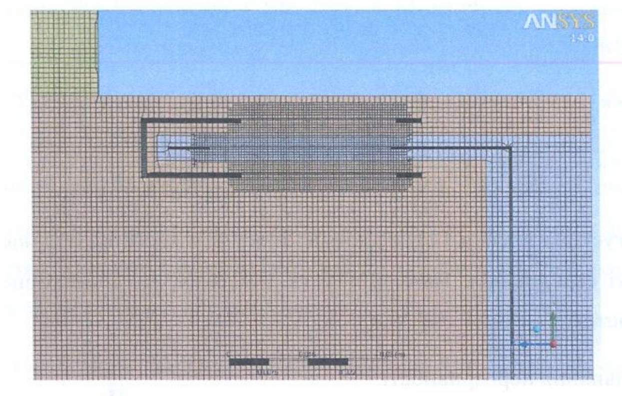


Fig. 5 The result of the constructed grid between the grooves in the longitudinal section YZ

Mathematical modeling of heat and mass transfer processes in RPA

The motion of a viscous liquid or gas is described by a system of equations, which includes the equation of continuity equation and the equation of conservation of momentum in projections on the coordinate zone. If the motion of the medium is accompanied by heat transfer, the equation of energy conservation is added to the system of these equations.

The mathematical model is based on the Navier-Stokes equation [255] and the energy transfer equation for convective flow. The standard k-ε turbulence model [256] was used in the calculations. The initial equations are given below.

Navier-Stokes equations:

$$\begin{aligned}
&\rho\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}=\rho g_x-\frac{\partial p}{\partial x}+\mu\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2},\\
&\rho\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}=\rho g_y-\frac{\partial p}{\partial y}+\mu\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2},\\
&\rho\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}=\rho g_z-\frac{\partial p}{\partial z}+\mu\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2},
\end{aligned}\tag{2}$$

where ρ – the density of the medium, kg/m^3; μ – the coefficient of dynamic viscosity of the medium, Pa·s; p – pressure, Pa; u, v, w – vector field of velocities; t – a moment of time, s.

The equation of inseparability:

$$\frac{\partial\rho}{\partial\tau}+\frac{\partial(u\rho)}{\partial x}+\frac{\partial(v\rho)}{\partial y}+\frac{\partial(w\rho)}{\partial z}=0 \tag{3}$$

Energy conservation equation:

$$\rho C_p\frac{\partial T}{\partial\tau}+u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}+w\frac{\partial T}{\partial z}=\frac{\partial}{\partial x}\lambda\frac{\partial T}{\partial x}+\frac{\partial}{\partial y}\lambda\frac{\partial T}{\partial y}+\frac{\partial}{\partial z}\lambda\frac{\partial T}{\partial z}+Q(\tau,x,y,z) \tag{4}$$

where T – the temperature at some point, K; λ – the thermal conductivity coefficient of the medium, W/m·K; C_p – specific heat capacity of the medium, J/kg·K.

Turbulence model

For the flow of coolants in the channels it is necessary to choose a model of turbulence that most corresponds to the conditions of hydrodynamic flow in these channels. One of the most common is the standard k-ε model used in Ansys Fluent software package, used in practical engineering calculations of flows in channels and was proposed by Launder and Spalding [256]. The reliability, economy and accuracy of this model for a wide range of turbulent flows explain its wide application for modeling heat transfer in channels of various configurations.

The standard k-ε model [256] is based on the model of the turbulence transfer equation of kinetic energy (k) and its dissipation rate (ε). The equation of the transport model for k is from the exact equation, and the equation of the transport model for ε is obtained for physical reasons.

In deriving the k-ε model, the flow is assumed to be completely turbulent that and the effects of molecular viscosity are negligible, i.e. the standard k-ε model is used for fully turbulent flows.

Subsequently, the standard k-ε model was improved, which increased its efficiency. There are two improved versions of this model, which are available in the software package Ansys Fluent, namely, RNG k-ε model [257] and implemented (realizable) k-ε model [258].

The transfer equation of the standard k-ε model

The kinetic energy of turbulence k and its dissipation rate ε is found from the solution of the following transport equations [259-261], which are given below:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\mu+\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\mu+\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}+G_{1\varepsilon}\frac{\varepsilon}{k}(G_k+G_{3\varepsilon}G_b)-G_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon \tag{6}$$

where G_k – the generation of kinetic energy turbulence due to velocity gradients (see equation (12)); G_b – the generation of kinetic energy turbulence (see equation (16)); Y_M – the contribution to the expansion of the compressibility fluctuations of turbulent flows in the total dissipation rate (see equation (18)); C_{1e}, C_{2e} and C_{3e} – experimental constants; σ_k and σ_e – are Prandtl turbulent numbers for k and ε, respectively. The values of S_k, S_e are defined as characteristics of the kinetic energy and the dissipation of this energy:

$$S_k=\tau_{ij}\frac{\partial u_i}{\partial x_j}-\rho\varepsilon-\mu\frac{g_i\partial\rho}{\sigma_{\hat{A}}\rho\partial x_i}, \tag{7}$$

$$S_{\varepsilon} = C_{\varepsilon 1}\frac{\varepsilon}{k} f_1 \tau_{ij}\frac{\partial u_i}{\partial x_j} + \mu C_{\hat{A}}\frac{g_i \partial \rho}{\sigma_{\hat{A}} \rho \partial x_i} - C_{\varepsilon 2} f_2 \frac{\rho \varepsilon^2}{k}. \tag{8}$$

where $C_{\varepsilon 1}$ = 1,44, $C_{\varepsilon 2}$ = 1,92, σ_{ε} = 1,3, σ_k = 1 – empirical coefficients [261]; f_1, f_2 – are variables that depend on the coefficients of dynamic and turbulent viscosity and are determined by the following formulas:

$$f_1 = 1 + \frac{0,05^3}{f_{\mu}},\ f_2 = 1 - e^{-R_T^2}. \tag{9}$$

The values of constants and parameters in this model were determined from experiments for fundamental turbulent flows, including shear flows, boundary layers, mixed layers, as well as for the decomposition of an isotropic turbulence grid. They have proven to be quite effective for a wide range of wall-bound landslides.

Modeling of turbulent viscosity

The turbulent (or vortex) viscosity μ_t calculated through k and ε as follows:

$$\mu_t = \rho C_{\mu}\frac{k^2}{\varepsilon} \tag{10}$$

where C_{μ} – constant.

The degree to which ε depends on buoyancy is determined by the constant C_{3e}. In Ansys Fluent, the value of C_{3e} is calculated by the following ratio:

$$C_{3e} = \tanh\left|\frac{v}{u}\right| \tag{11}$$

where ν – the component of the flow velocity that is parallel to the gravitational vector; u – the component of the flow velocity perpendicular to the gravitational vector. Thus, C_{3e} well be equal to 1,0 for floating shear layers, for which the main flow direction is combined with the direction of gravity. For floating shear layers that are perpendicular to the gravitational vector, C_{3e} becomes zero.

Modeling of turbulent processes in the k-ε model

The parameter G_k, is a process of kinetic energy of turbulence, which is modeled equally for standard, RNG and realizable k-ε models and is defined as:

$$G_k = -\rho \overline{u_i u_j}\frac{\partial u_j}{\partial x_i} \tag{12}$$

To estimate G_k according to Bussinec's hypothesis:

$$G_k = \mu_t S^2 \tag{13}$$

where S – the modulus of the average rate of change of the strain tensor, defined as:

$$S = \sqrt{2S_j S_{ij}} \tag{14}$$

Effects of buoyancy on turbulence in the k-ε model

Generation of turbulence due to buoyancy is determined by:

$$G_b = \beta g_i \frac{\mu_t}{\Pr_t} \frac{\partial T}{\partial x_i} \tag{15}$$

where Pr – the turbulent Prandtl number for the energy; g_i – the component of the gravitational vector in the *i*-th direction. For standard k-ε models, its value is Pr = 0,85.

The coefficient of thermal expansion β is defined as:

$$\beta = -\frac{1}{\rho} \frac{\partial \rho}{\partial T} \tag{16}$$

For ideal gases, the ratio (15) is as follows:

$$G_b = -g_i \frac{\mu_t}{\rho \Pr_t} \frac{\partial \rho}{\partial x_i} \tag{17}$$

As can be seen from the transfer equations for the parameter k (see equation (5)), the turbulence of the kinetic energy is usually increased (G_b>0) under the conditions of unstable flow stratification. For stable stratification we have a reduction of this turbulence (G_b<0). In the Ansys Fluent package, the effects of buoyancy on the generation of *k* are always taken into account if there is a non-zero gravitational field and a non-zero temperature gradient.

To account for these effects in the k-ε turbulence model in Ansys Fluent package, we use scattering parameter of the extension Y_M, which is included in *k* equation (6). This parameter is modeled according to Sarkar's proposal:

$$Y_M = 2\rho\varepsilon M_t^2 \tag{18}$$

where M_t – the turbulent Mach number, which is defined by the expression:

$$M_t = \sqrt{\frac{k}{a^2}} \tag{19}$$

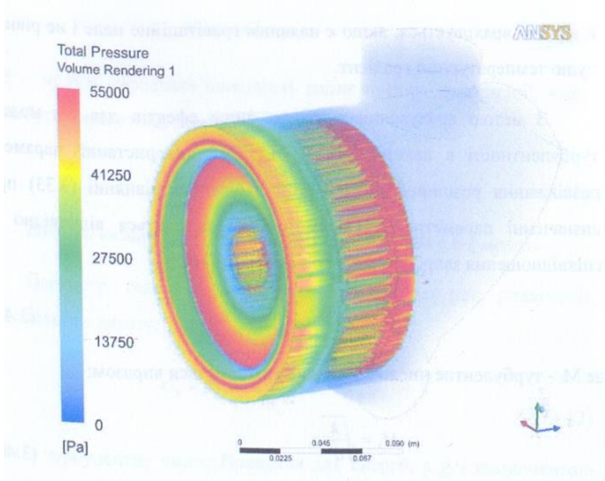
where $a = \sqrt{\gamma RT}$ – the speed of sound.

The results of numerical modeling of transfer processes in RPA

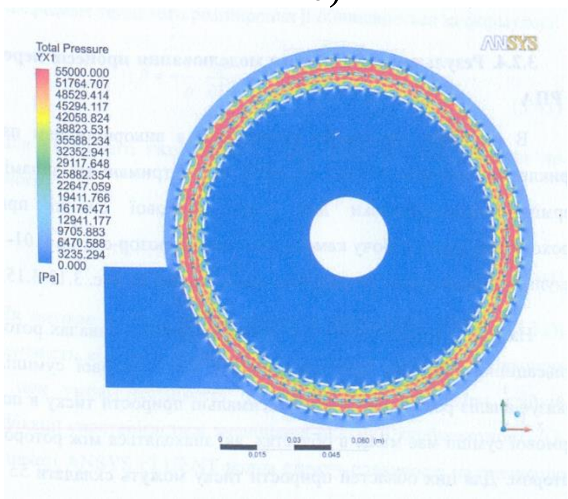
As a result of numerical calculations in Ansys Fluent application package, all dynamic and thermal characteristics of the liquid mixture during its passage through the rotor-stator system were obtained. The results of the calculations are presented in Fig. 6-8.

In Fig. 6 shows the pressure distribution in the channels of the rotary-pulsation apparatus during the flow of the mixture. As shown by the analysis of the pressure distribution, the maximum pressure increase in the flow of the mixture is observed in the zones between the rotor and the stator. In these zones, the pressure increase may be 55 kPa compared to the pressure at the inlet to the channel. This indicates a significant effect of the viscosity of the mixture on the change in pressure in the rotor-stator channels, which is accompanied by the conversion of the kinetic energy of rotation of the rotor into the thermal energy of the mixture.

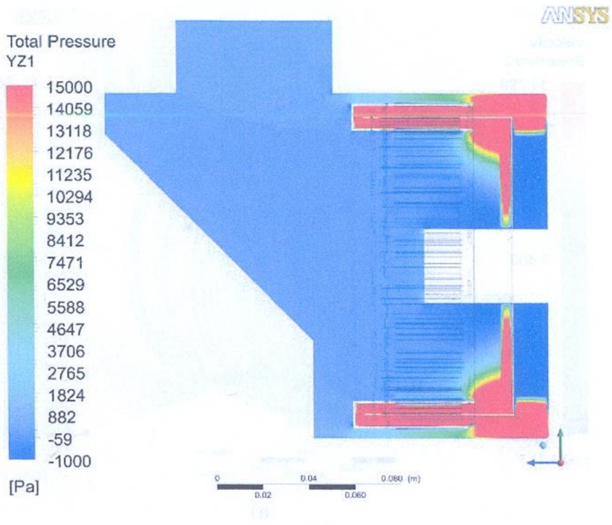
In Fig. 7 shows the velocity distribution in the channels of the working zone in rotor-stator system. The maximum values of flow rate of the mixture are observed in the channels between the rotor and the stator, where these values can exceed 7 m/s. When the liquid leaves the rotor-stator channel the flow vortices zones are formed, which turbulizes the flow in the channels and leads to intensive mixing of the mixture. The result is a homogeneous mixture.



a)

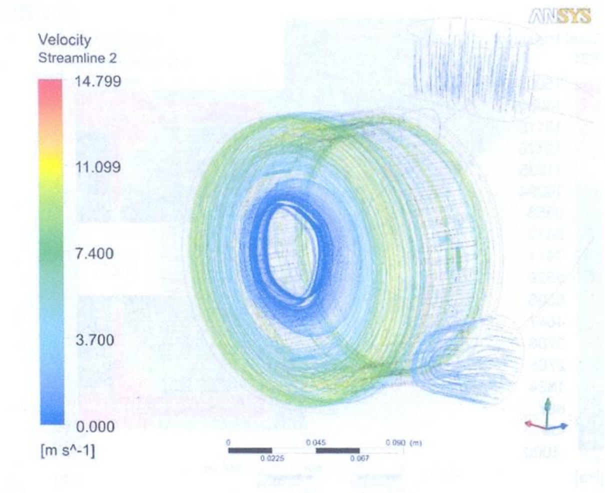


b)

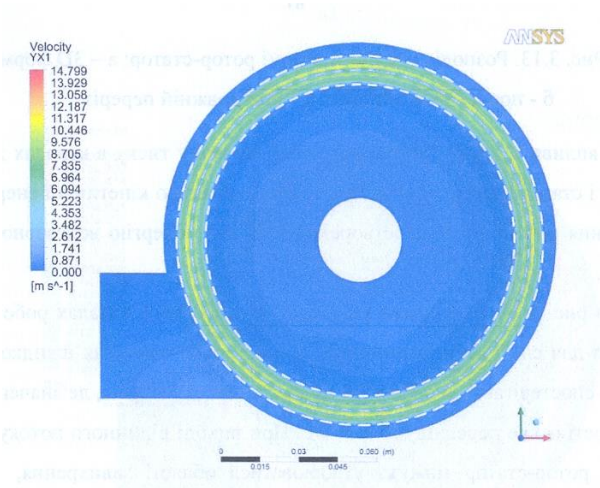


c)

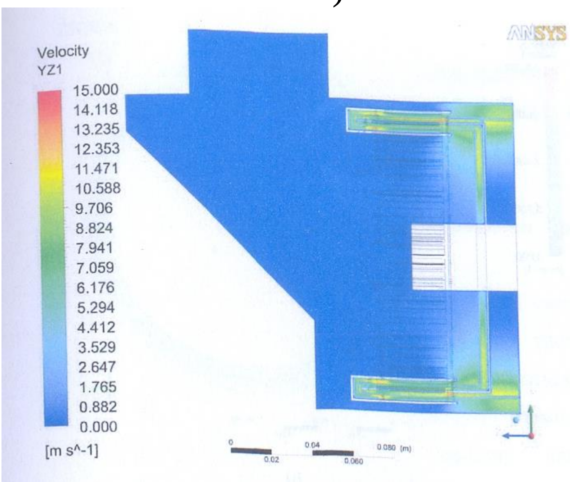
Fig. 6 Pressure distribution in the rotor-stator system: a – 3D format; b – cross section; c – longitudinal section.



a)



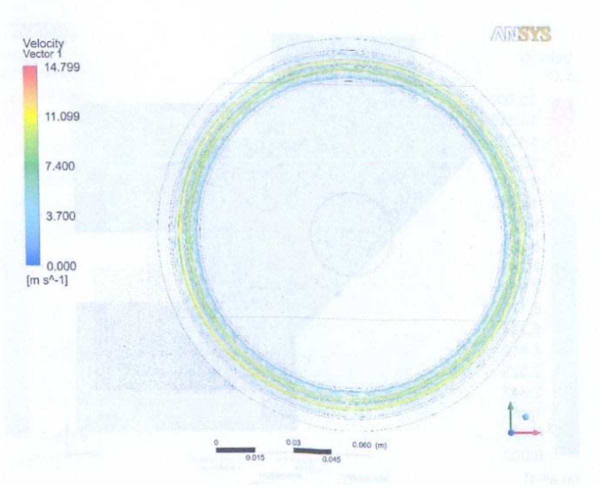
b)



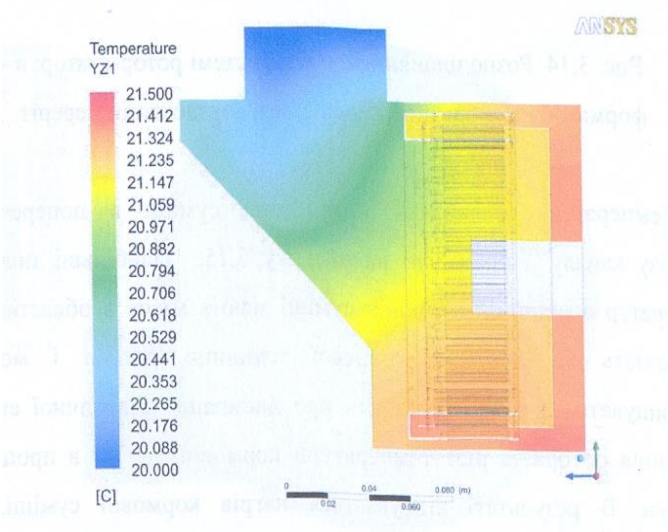
c)

Fig. 7 Velocity distribution in the rotor-stator system: a – 3D format; b – cross section; c – longitudinal section.

Temperature distributions in the liquid mixture in the cross section of the channel are shown in Fig. 8. The highest values of temperatures in the liquid mixture take place in the zones adjacent to the surface of the end plane of the rotor and can exceed 21 ºC. This indicates a dissipation of the kinetic energy of rotation of the rotor and an increase in the temperature of the mixture during its processing. As a result, the mixture is heated, which is especially important in the winter period of the year and allows you not to heat the mixture during this period.



a)



b)

Fig. 8 Temperature distribution in the mixture for longitudinal (a) and cross (b) sections in the rotor-stator system

Conclusions

1. As a result of numerical calculations in Ansys Fluent application package, all dynamic and thermal characteristics of the liquid mixture during its passage through the rotor-stator system were obtained.

2. The maximum pressure increase in the flow of the mixture is observed in the zones between the rotor and the stator. In these zones, the pressure increase may be 55 kPa compared to the pressure at the inlet to the channel.

3. The maximum values of flow rate of the mixture are observed in the channels between the rotor and the stator, where these values can exceed 7 m/s. When the liquid leaves the rotor-stator channel the flow vortices zones are formed, which turbulizes the flow in the channels and leads to intensive mixing of the mixture. The result is a homogeneous mixture.

4. The highest values of temperatures in the liquid mixture take place in the zones adjacent to the surface of the end plane of the rotor and can exceed 21 ºC. This indicates a dissipation of the kinetic energy of rotation of the rotor and an increase in the temperature of the mixture during its processing.

5. As a result of numerical simulation, the fields of pressures, velocities and temperatures in the liquid mixture were obtained, which made it possible to choose the design of the rotor-pulsation apparatus, which will be used to develop an experimental sample of such device.