Modeling of heat transfer in a vertical channel at bubble flows by means a DNS method

Vladimir V. Chudanov, Anna E. Aksenova, Valerii A. Pervichko

Abstract—This paper deals to simulation heat transfer in a vertical channel at bubble flows by means DNS method. The effective numerical CFD algorithm for 3D calculation of two-phase flows with selection of interface boundary explicitly and calculation of surface tension forces is presented. The main attention of this paper is focused on verification of numerical technique with usage of the numerical tests and experiments.

Keywords—Heat transfer, intensification bubble flows, CFD, DNS.

I. INTRODUCTION

IN spite of long history of studies on two-phase and vapor-liquid flows there is still no satisfactory computational approach for determining the moving of different phases and behavior of two-phase mixtures. One of the reasons of such a situation is the absence of clear and detailed picture of bubbles interaction both with the carrying medium and with each other. The problems are complex enough for direct theoretical investigation because of a lot of uncertain parameters, and moreover, the situation is significantly more complicated for non-stationary interaction conditions.

In this connection, the development of computational methods and algorithms for solving the problems of heat and mass transfer in incompressible media taking into account surface tension forces, as well as development of three-dimensional CFD codes based present-day standards of calculation techniques and numerical methods are very important.

One of the most frequent two-phase CFD flows is bubble flow. The regime is realized in developed liquid boiling on variously oriented surfaces, two-phase flow in vertical and horizontal channels. The method of direct numerical modeling being elaborated and used today, need the thorough testing on available experimental and analytical data in literature, in particular on the tests describing steady motion of single gas bubble and bubble group in large volume of incompressible liquid. The experiments on description of a bubble rising in various channel forms (cylinder, slit channels) and effects of bubbles interaction during their coalescence and fragmentation are also important. Because the development of the tests system (matrix) for verification of the developed software for direct modeling of bubble flows and generalization of results as regimes map for emerging bubble or bubble group are very important.

In this paper the features of DNS numerical methodology developed in IBRAE RAS for modeling of bubble flows and verification of this methodology are presented. The numerical technique has a high speed, which equals 10^{-5} seconds per one mesh node and time step and that allows to carry out CFD calculations using personal computer and petascale cluster.

II. CFD NUMERICAL TECHNIQUE

This section deals with the new methods and algorithms for solving of 3D heat and hydrodynamics in incompressible fluids. Among them we consider two different numerical algorithms for solving of incompressible fluid dynamics, the monotone multi-dimensional schemes of a TVD-type for solving of advection equation and the effective algorithm for solving of the elliptical equation for pressure correction.

For modelling of fluid behaviour outside of a bubble the Navier-Stokes equations for an incompressible fluid together with the energy equation are used [1-7].

For modelling of gas behaviour inside of a bubble the model of compressible fluid at low Mach numbers is offered [8].

For observation behind of two-phase system interface the following models are used: the modified level set method for direct numerical simulation of a two-phase system as a fluid-gas; the monotone multi-dimensional advection scheme with small scheme diffusion (using of sub-grid simulation); fast numerical algorithm for solving of the pressure equation with a variable density [5].

The developed algorithms, methods and software were tested on a wide set of the tests [4, 6, 7].

A. Numerical technique for solving Navier-Stokes equations

To simulate incompressible CFD problems, the time-dependent incompressible Navier-Stokes equations in the

V. V. Chudanov is with the Nuclear Safety Institute (IBRAE) of Russian Academy of Sciences, 52, Bolshaya Tulskaya street, Moscow, 115191, Russia (corresponding author to provide phone: +7-495-955-22-34; e-mail: chud@ibrae.ac.ru).

A. E. Aksenova is with the Nuclear Safety Institute (IBRAE) of Russian Academy of Sciences, 52, Bolshaya Tulskaya street, Moscow, 115191, Russia (corresponding author to provide phone: +7-495-955-22-34; e-mail: aks@ibrae.ac.ru).

V. A. Pervichko is with the Nuclear Safety Institute (IBRAE) of Russian Academy of Sciences, 52, Bolshaya Tulskaya street, Moscow, 115191, Russia (corresponding author to provide phone: +7-495-955-22-34; e-mail: valper@ibrae.ac.ru).

primitive variables [1-7] coupled with the energy equation are used:

$$\begin{cases} \dfrac{d\rho\bar{v}}{dt} = -\operatorname{grad} P + \operatorname{div} \nu \operatorname{grad} \bar{v} + \rho g + \text{CSF} \\ \\ \qquad\qquad \operatorname{div} \bar{v} = 0 \end{cases} \tag{1}$$

$$\frac{\partial(\rho h)}{\partial t} + \operatorname{div}(\rho v h) = \operatorname{div}(k \operatorname{grad} T)$$
$$h = \int_{0}^{T} c(\xi) d\xi \tag{2}$$

Basic features of developed numerical algorithm [1-7] incorporated in the present approach are the following:

Numerical implementation of operator-splitting scheme is performed as the predictor-corrector procedure with correction for the pressure:

$$\rho \frac{v^{n+1.2} - v^{n}}{\tau} + (C(v) - \operatorname{div} \nu \operatorname{grad})v^{n+1/2} + \tag{3}$$
$$+ \operatorname{grad} p^{n} - \text{CSF}^{n} = 0,$$

$$\operatorname{div}_{h}\left(\frac{1}{\rho} \operatorname{grad}_{h} \delta p\right) = \frac{1}{\tau} \operatorname{div}_{h} v^{n+1/2}, \tag{4}$$

$$v^{n+1} = v^{n+1/2} - \frac{\tau}{\rho} \operatorname{grad}_{h} \delta p .$$

Moreover the scheme with iterations is developed. In this case set of equations in view of iterations on $\hat{v}^{s}$ looks like:

$$\begin{cases} \rho\left(\dfrac{v^{n+1/2} - v}{\tau}\right) + (C(v) - \operatorname{div} \nu \operatorname{grad}) \bar{v} = \rho g + \text{CSF} \\ \\ \left(\dfrac{\hat{v}^{s} - v^{n+1/2}}{\tau}\right) \operatorname{grad} \rho + \\ \\ + \rho \operatorname{div}\left(\dfrac{- v^{n+1/2}}{\tau}\right) = -\operatorname{div} \nu \operatorname{grad} \tilde{P}^{s} \\ \\ \operatorname{div} v^{s} = 0 \end{cases} \tag{5}$$

Taking into account last line in (5) is obtained the following equation for correction of a velocity

$$\hat{v}^{s+1} = v^{n+1/2} - \tau \cdot \operatorname{grad} \tilde{P}^{s} / \rho . \tag{6}$$

For solving of convection problem the regularized nonlinear monotonic operator-splitting scheme is developed [3]. The special approximation of convection terms $C(v)$ are

employed in order to derive the discrete convective operator, which is skew-symmetrical and does not give any contribution to the kinetic energy (i.e. energetically neutral).

So, this scheme provides the second order in space and the first one in time. The algorithm is stable at a large enough integration step by time.

Details of validation presented approach on a wide set of both 2D and 3D tests are reported in [1-7].

B. Fast solver for Pressure Equations

For solving of the elliptical equations with variable coefficients, for example Poisson equation for pressure correction δp,

$$\operatorname{div}_{h}\left(\frac{1}{\rho} \operatorname{grad}_{h} \delta p\right) = \frac{1}{\tau} \operatorname{div}_{h} \mathbf{v}$$

conjugate gradient method is applied commonly which occupies 90 % from time of the central processor per one calculated step. Therefore searching algorithms requiring a smaller time amount of the CPU on one calculated step is important. The Richardson iterative method with Chebyshev's set of parameters using preconditioner FFT solver for Laplace's operator can serve by an alternative to conjugate gradient method.

The application of this approach for solving of the elliptical equations with variable coefficients allows to reach fifty multiple acceleration in comparison with a usually used method of conjugate gradients [4].

C. Curvature and tension surface

The capillary source term appearing in momentum equation can be approximated by Chella и Vinals [8]

$$f \approx -\left|\nabla \chi\right|^{2} K \cdot \kappa n$$

where n denotes the unit vector normal to the interface, K is a positive constant, and κ represents the local curvature defined by

$$\kappa = -\nabla \cdot n = -\nabla \cdot \left(\frac{\nabla \chi}{\left|\nabla \chi\right|}\right).$$

Integrating equation over the interfacial area (ds) yields the continuous surface force (CSF) model of Brackbill [9]

$$\int \sigma \kappa \delta(x - x_{f}) n ds$$

where σ is the surface tension, assumed to be constant across the entire thickness of the interfacial sub-layer, and $\delta(x - x_{f})$ represents a Dirac pulse with x_{f} being the

instantaneous location of the interface. The delta function appears in the above expression because use was made of

$$\nabla \chi = \int \delta\left(x - x_f\right) n\,ds .$$

The surface tension σ reflects the excess of capillary energy concentrated at the interface per unit surface area caused by the variation in χ across the interfacial sub-layer.

D. Monotone numerical scheme for the advection equation

Let's consider the following scheme [3]

$$\frac{y^{n+1} - y^n}{\tau} + \bar{a}^n y_{\bar{x}}^n = 0, \quad n = 0,1,\dots \quad (7)$$

with nonlinear factor

$$\bar{a}^n = a^n \left(1 + \frac{h}{2} \frac{y_{\bar{x}x}^n}{y_{\bar{x}}^n}\right) \quad (8)$$

The non-linear difference scheme (8) will be monotone, if

$$\tilde{a}^n \geq 0 \quad (9)$$

$$\max_i \tilde{a}_i^n \frac{\tau}{h} \leq 1, \quad n = 0,1,\dots \quad (10)$$

From (9)-(10) follows, that the violation of a monotonic is observed in a neighbourhood of extremum of the difference solution.

Let's present (8) in more convenient aspect as

$$\tilde{a}^n = a^n \chi^n \quad (11)$$

where the factor χ equals

$$\chi^n = \left(1 + \frac{h}{2} \frac{y_{\bar{x}x}^n y_{\bar{x}}^n}{\left|y_{\bar{x}}^n\right|^2 + \gamma h^2 \left|y_{\bar{x}x}^n\right|^2}\right) \quad (12)$$

with parameter γ. Padding (regularizing) summands χ^n from (12) have the third order on h.

Thus regularized scheme (7), (11), (12) approximates an one-dimensional equation of advection in non-divergent form with the second order on space.

Let's formulate conditions of a monotonicity of the scheme (7), (11), (12). Factor $\tilde{a}^n$ (condition (9)) will be non-negative at $\gamma = 0.25$. Steps of a grid should satisfy to a condition (10), which with taking into account (11), (12) is equivalent to an inequality

$$\max_i a_i^n \frac{\tau}{h}\left(1 + \frac{1}{4\gamma}\right) \leq 1, \quad n = 0,1,\dots \quad (13)$$

Thus for regularized scheme (11), (12) under minimum admissible value of parameter $\tilde{a}^n$ the maximal admissible time step should be calculated in accordance with (13).

III. VALIDATION ASPECT

The first test case "Rise of a spherical cap bubble in a stagnant liquid" could usually be considered as a very preliminary one for validation of a new numerical method. However, this selected case deserves special attention for the result not only consists in a final shape of the bubble (that is nevertheless a real criterion of comparison) but also in a precise build-up of the bubble velocity, starting from rest, exhibiting an overshoot before reaching its final asymptotic value.

In figure 1 the comparison of numerical predictions with the data [10] on growth rate of a bubble versus time is shown.

As it is possible to see from figure 1 the good coincidence on dynamics and form of a trajectory of bubble growth velocity is observed.
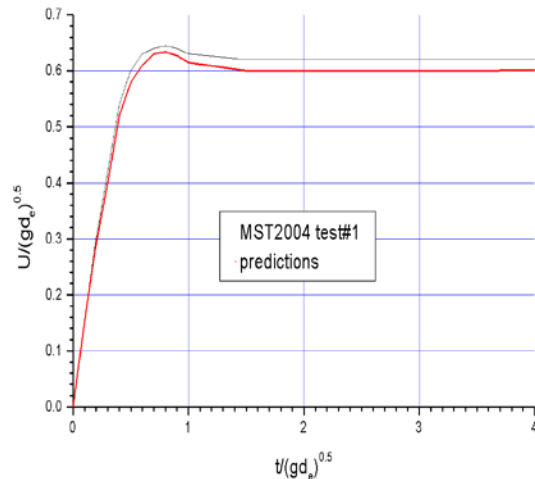

Fig. 1 rise of a spherical cap bubble in a stagnant liquid

The second test case "Trajectory of the center of mass of a raising single bubble" [11] could be considered also as a analytical test for validation of a new numerical method.

An experimental trajectory of the center of mass of a raising single bubble [11] is presented at Figure 2a.

This trajectory (see Fig. 2a) is helically due heterogeneity hydrodynamic resistance at displacement velocity vector relative to the planes of symmetry of the bubble. Numerical predictions of trajectory of the center of mass of a raising single bubble [11] are presented at Figure 2b and are in a good agreement with experiment.

The third test case "Distribution of the gas phase in height" [12] could be considered also as an analytical test for validation of a new numerical method.
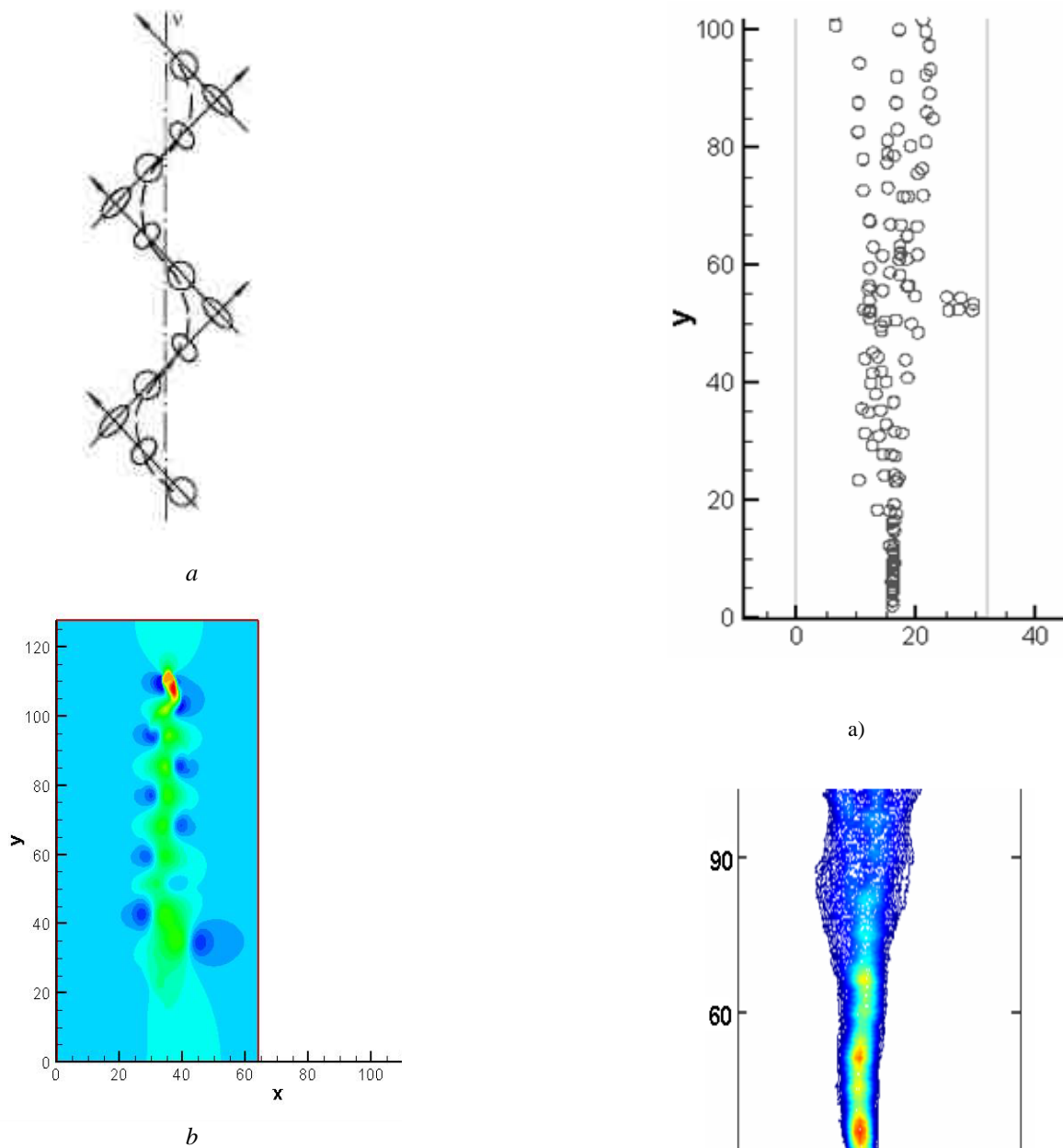
a

b

Fig. 2 raising air bubbles in a water-filled planar gap: a - trajectory [11]; b - numerical trajectory of a single bubble, raising in a quiescent water

Figure 3 shows the distribution of the gas phase in height compared to the calculation made for the conditions of this experiment [12]. In the experiments and calculations there is a noticeable sampling distribution of the gas phase in the initial section ascent bubbles with increasing distance from the source of generation of bubbles this unevenness is relaxed , which is associated with a more random distribution of bubbles across the channel. These examples show only part of the results, there is a detailed database for each case considered, and the possibility of the developed method experimental study of bubble flows and processing of experimental data allow to significantly expanding this base.

a)

b)
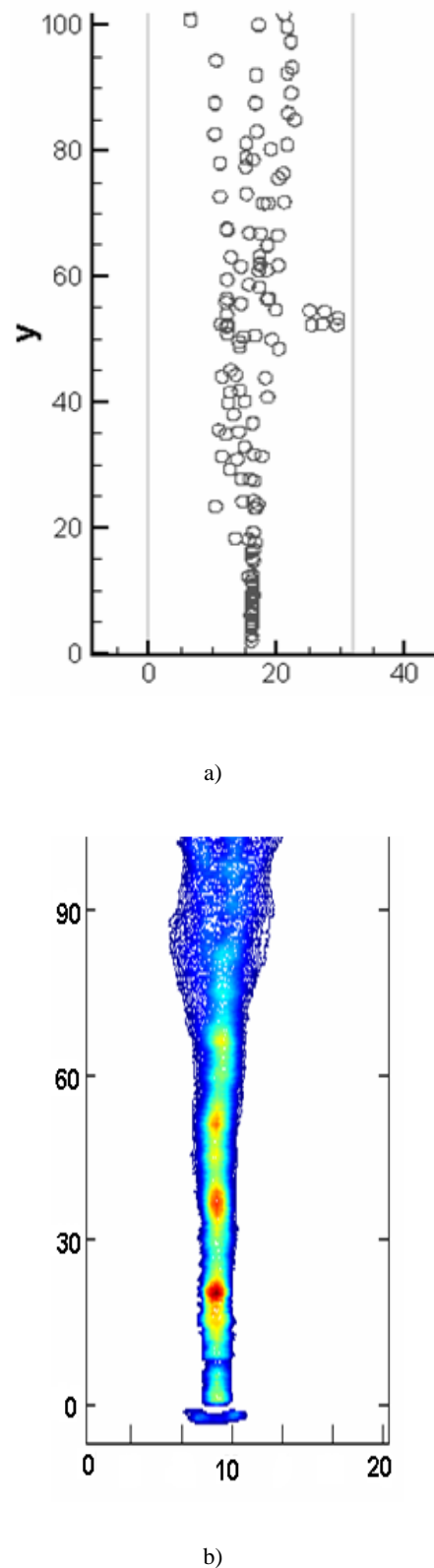
Fig. 3 field relative volume concentration of the gas phase (gas content) in a flat gap: a - numerical predictions; b - experiment [12]

IV. HEAT TRANSFER IN A VERTICAL CHANNEL

The developed approach was applied for the simulation of heat transfer in a vertical channel at bubble flows, occurring in industrial practice.

For example, the homo- and heterogeneous nucleation of a new phase significantly affects the efficiency of heat transfer processes.

Phenomena caused by special properties of the surface layers of liquids (compared to the bulk) are called by surface's phenomena.

The most common and important property of these layers is an excess free energy $F = \sigma S$, where σ - is an interfacial tension; S - is an area of the interface. Surface phenomena are most pronounced flow in heterogeneous systems with high surface interface, i.e. in dispersed systems. Spontaneous surface phenomena occur due to the decrease of the surface energy of the system.

They may be caused by decreasing the total surface of the system either by reducing the surface tension at the interface. Phenomenon of coalescence, i.e. merging droplets in the emulsions (gas bubbles or foams) at direct contact, refers to a class of surface phenomena associated with the reduction of the total surface area.

By increasing the gas flow rate, both phases can be observed as a continuous emulsions and vibrating counter flows. A time-varying oscillatory mode where large waves moving further in the direction of flow imposed on the other wavy annular dispersed flow pattern-thick films of liquid on the walls, also called emulsion flow regime. Emulsion flow also occurs at the input of the vertical channel. This is a different interpretation of the emulsion is irregular currents and the area near the entrance of a long channel, which eventually develops slug flow pattern.

Below are the results of numerical calculation for the intensification problem of heat transfer near wall, which is widely used in power plants, as well as in heat exchangers for various purposes.

A characteristic feature of heat transfer near wall is that the heat transfer intensifies only near wall region (near the border), where there is a maximum transverse temperature gradient. Heat exchangers with the intensification of heat exchanger at the boundaries have high energy efficiency. Put them into practice contributes to solving the problems of energy efficiency, in particular by reducing energy consumption to ensure circulation of the coolant circuits of heat exchangers.

Let us consider a lifting of the cold liquid flowing in the vertical tube with hot walls. In such geometry enter a jet of bubbles realized which moving along one of the hot walls is a similar as to shown in Figure 4.

In other words, the bubbles follow one another on an oscillating trajectory and carry away the layers of the liquid, forming a turbulent layer. As a result, during the motion along the wall of the bubbles (at the velocity fluid of 50 to 400

mm/c) breaking of dynamic and thermal boundary layer occurs.

At figure 4 the temperature fields at the velocity fluid 100 and 200 mm/c are presented.
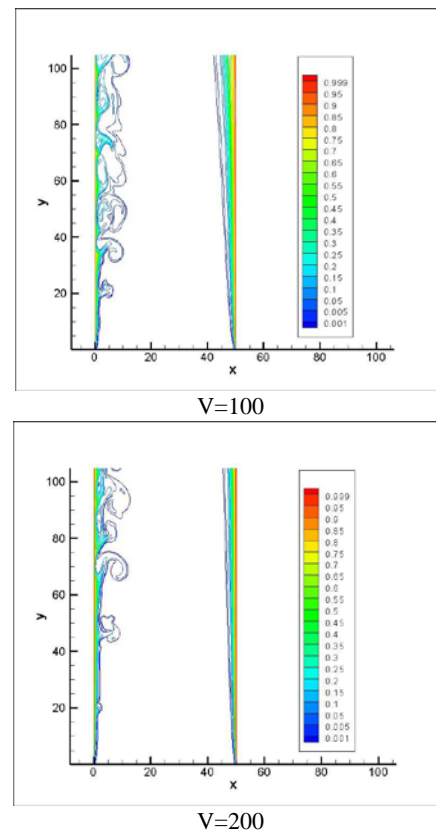

V=100

V=200

Fig. 4 the temperature field

At figures 5 and 6 the velocity fields at same values of the velocity fluid are presented.

A map of the materials (bubbles) is presented at figure 7. A dynamic boundary layer is destroyed on the left wall from rising bubbles along this wall.

This leads to the intensification of heat transfer in two or three times, as indicated in figure 8, where shows a graph of heat flow as a function of coolant velocity in the pipe.

A ratio of Nusselt numbers (Nusselt bubbles to Nusselt number) depending upon the Reynolds number in the pipe is determined by the Reynolds number bubble size (1 mm). Reynolds number is defined by following equation

$$\mathrm{Re} = \frac{V d_{bubble}}{\nu},$$

where v is a velocity fluid, d_{bubble} is a bubble diameter, d=1mm; ν is a kinematic viscosity, $\nu = 1$ mm^2/c. The initial velocity of bubble is equal 0 mm/c.
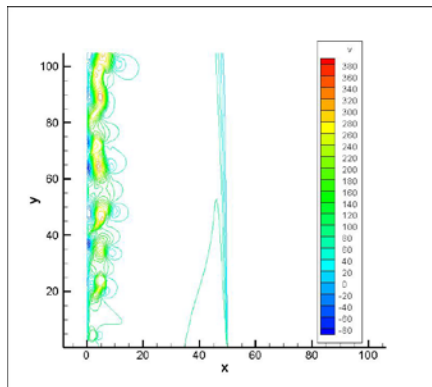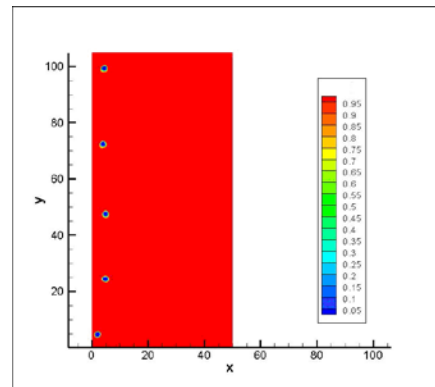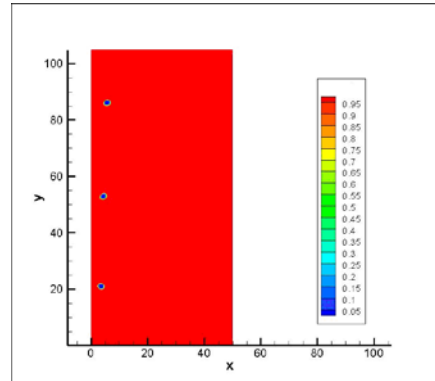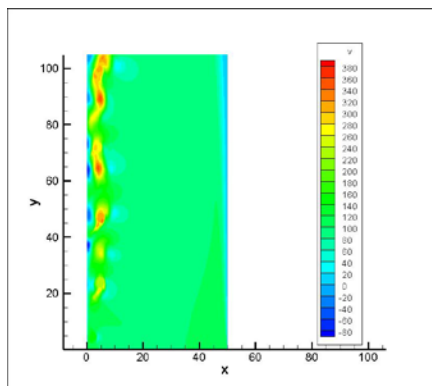
V=100
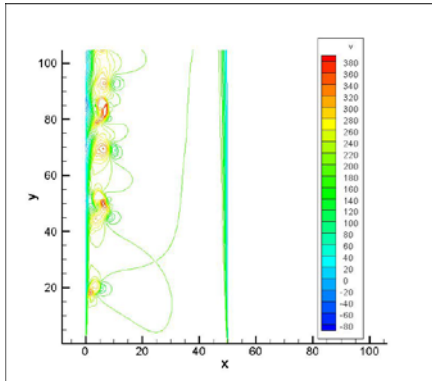
V=100

V=200
Fig. 5 the velocity field (isolines)

V=200
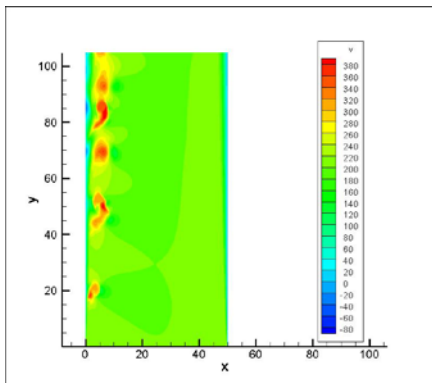Fig. 7 the map of materials

V=100

Fig. 8 the ratio of heat fluxes vs Reynolds number

V=200
Fig. 6 the velocity field

V. CONCLUSION

To solve the problems of computational fluid dynamics developed effective finite difference numerical algorithms.

Developed algorithms are applied to the equations of heat and mass transfer, namely the Navier-Stokes equations with the energy equation in natural variables for incompressible and weakly compressible flows at low Mach numbers.

Testing was performed on a set of numerical and experimental tests, which were demonstrated a good agreement between the numerical predictions and experimental data.

Computational experiments are much less expensive than the experimental work, which may allow to obtain the

necessary correlations to determine the heat transfer coefficient for tasks such heat and mass transfer , where the experimental data have a high degree of uncertainty.

NOMENCLATURE

CSF — continuous surface tension forces

$C(v)v$ = $\frac{1}{2}[(v, \text{grad})v + \text{div}(vv)]$ -convective term

c — - specific heat
g — - gravity acceleration
h — - enthalpy
f — - capillary source
k — - thermal conductivity
Nu_{bubble} — - left boundary of heat flux perturbed by raising bubble(bubble's Nusselt number)
Nu — - not perturbed right boundary of heat flux (Nusselt number)
p — - pressure normalized by the density
P — - pressure
x_f — - instantaneous location of the interface
T — - temperature
$\bar{v}$ — - Velocity, (v_1, v_2, v_3)
t — - time

Greek Symbols

χ — - indicator function
$\delta(x - x_f)$ — - delta function of Dirac
κ — - local curvature
σ — - surface tension
ν — - viscosity
ρ — - density

REFERENCES

[1] V.V. Chudanov, et al., "Current Status and Validation of CONV2D&3D code", in *OECD/CSNI Report* NEA/CSNI R(98), Germany, Garching, 1998, pp. 223–234.

[2] V.V. Chudanov, et al., "A multi-block orthogonal grid generation using CAD system", in *SANDIA REPORT* №SAND 99-2288, Lake Tahoe, USA, 1999, pp. 179–189.

[3] V.V. Chudanov, et al., "Non-linear regularized difference schemes for multidimensional advection equations", in *J. of computing mathematics and of mathematical physics*, 2001, Vol. 40, Nos. 6, pp. 900–907.

[4] V.V. Chudanov, et al., "3D unified CFD approach to thermalhydraulic problems in safety analysis", in IAEA Report, Use of Computational Fluid Dynamics (CFD) Codes for Safety Analysis of Reactor Systems, Including Containment, IAEA-TECDOC-1379, Pisa (Italy), 2002.

[5] V.V. Chudanov, et al., 2005. "3D unified CFD approach to modeling of bubble phenomena", in *Proc. 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11)*, Popes' Palace Conference Center, Avignon, France, October 2-6, 2005.

[6] V.V. Chudanov, et al., "Grid Office for orthogonal grid generation with CAD for CFD calculations", in *Proc. TCN CAE meeting*. Lecce, Italy, October 5-8, 2005.

[7] V.V. Chudanov, et al., "Grid Office for CFD Pre and Post Processing", in *Proc. of IMACS'2005 World Congress,* Paris, FRANCE, July 11-15, 2005.

[8] R. Chella and J. Vinals, "Mixing of a two-phase fluid by cavity flow", in *Phys. Rev.* E 53/4, 3832, 1996.

[9] J. U. Brackbill, D. B. Kothe, C. Zemach, „A continuum method for modelling surface tension", *J. Comp. Phys.,* 1992, Vol. 100, pp. 335.

[10] *J. Multiphase Science and Technology [MST]*, Vol. 16, Nos. 1-3, pp. 1–82.

[11] B.E. Nakoryakov, A.V. Gorin, Heat transfer in two-pahse systems [in Russian], Thermal Physis Institute SB RAS Novosibirsk, 1994, 431 p.

[12] V. Meledin, V. Pavlov, O. Tsvelodub, N.Yavorsky, „Gravity-Capillary Waves on the Surface of a Liquid Dielectric". *Doklady Physics*. 2005, Vol. 50, Nos. 8, pp. 426—430.