ISSN 2518-1726 (Online),
ISSN 1991-346X (Print)

ҚАЗАҚСТАН РЕСПУБЛИКАСЫ
ҰЛТТЫҚ ҒЫЛЫМ АКАДЕМИЯСЫНЫҢ

ӘЛЬ-ФАРАБИ АТЫНДАҒЫ
ҚАЗАҚ ҰЛТТЫҚ УНИВЕРСИТЕТІНІҢ

ХАБАРЛАРЫ

ИЗВЕСТИЯ

НАЦИОНАЛЬНОЙ АКАДЕМИИ НАУК
РЕСПУБЛИКИ КАЗАХСТАН

КАЗАХСКИЙ НАЦИОНАЛЬНЫЙ
УНИВЕРСИТЕТ ИМЕНИ АЛЬ-ФАРАБИ

NEWS

OF THE NATIONAL ACADEMY OF SCIENCES
OF THE REPUBLIC OF KAZAKHSTAN

AL-FARABI KAZAKH
NATIONAL UNIVERSITY

ФИЗИКА-МАТЕМАТИКА СЕРИЯСЫ

◆

СЕРИЯ ФИЗИКО-МАТЕМАТИЧЕСКАЯ

◆

PHYSICO-MATHEMATICAL SERIES

4 (320)

ШІЛДЕ – ТАМЫЗ 2018 ж.
ИЮЛЬ – АВГУСТ 2018 г.
JULY-AUGUST 2018

1963 ЖЫЛДЫҢ ҚАҢТАР АЙЫНАН ШЫҒА БАСТАҒАН
ИЗДАЕТСЯ С ЯНВАРЯ 1963 ГОДА
PUBLISHED SINCE JANUARY 1963

ЖЫЛЫНА 6 РЕТ ШЫҒАДЫ
ВЫХОДИТ 6 РАЗ В ГОД
PUBLISHED 6 TIMES A YEAR

АЛМАТЫ, ҚР ҰҒА АЛМАТЫ, НАН РК ALMATY, NAS RK

«ҚР ҰҒА Хабарлары. Физика-математикалық сериясы».

ISSN 2518-1726 (Online), ISSN 1991-346X (Print)

Меншіктенуші: «Қазақстан Республикасының Ұлттық ғылым академиясы» РҚБ (Алматы қ.)
Қазақстан республикасының Мәдениет пен ақпарат министрлігінің Ақпарат және мұрағат комитетінде 01.06.2006 ж. берілген **№5543-Ж** мерзімдік басылым тіркеуіне қойылу туралы куәлік

Мерзімділігі: жылына 6 рет.
Тиражы: 300 дана.

Редакцияның мекенжайы: 050010, Алматы қ., Шевченко көш., 28, 219 бөл., 220, тел.: 272-13-19, 272-13-18,
www:nauka-nanrk.kz / physics-mathematics.kz

Типографияның мекенжайы: «Аруна» ЖК, Алматы қ., Муратбаева көш., 75.

«Известия НАН РК. Серия физико-математическая».

ISSN 2518-1726 (Online), ISSN 1991-346X (Print)

Собственник: РОО «Национальная академия наук Республики Казахстан» (г. Алматы)
Свидетельство о постановке на учет периодического печатного издания в Комитете информации и архивов Министерства культуры и информации Республики Казахстан **№5543-Ж**, выданное 01.06.2006 г.

Периодичность: 6 раз в год.
Тираж: 300 экземпляров.

Адрес редакции: 050010, г. Алматы, ул. Шевченко, 28, ком. 219, 220, тел.: 272-13-19, 272-13-18,
www:nauka-nanrk.kz / physics-mathematics.kz

Адрес типографии: ИП «Аруна», г. Алматы, ул. Муратбаева, 75.

News of the National Academy of Sciences of the Republic of Kazakhstan. Physical-mathematical series.

ISSN 2518-1726 (Online), ISSN 1991-346X (Print)

Owner: RPA "National Academy of Sciences of the Republic of Kazakhstan" (Almaty)
The certificate of registration of a periodic printed publication in the Committee of information and archives of the Ministry of culture and information of the Republic of Kazakhstan N 5543-Ж, issued 01.06.2006

Periodicity: 6 times a year
Circulation: 300 copies

Editorial address: 28, Shevchenko str., of. 219, 220, Almaty, 050010, tel. 272-13-19, 272-13-18,
www:nauka-nanrk.kz / physics-mathematics.kz

Address of printing house: ST "Aruna", 75, Muratbayev str, Almaty

N E W S
OF THE NATIONAL ACADEMY OF SCIENCES OF THE REPUBLIC OF KAZAKHSTAN
PHYSICO-MATHEMATICAL SERIES
ISSN 1991-346X
Volume 4, Number 320 (2018), 48 – 57

UDC 532.517.4.519.6

***A. Issakhov, A. Abylkassymova, M. Sakypbekova**

al-FarabiKazakh National University,Almaty, Kazakhstan
*e-mail: alibek.issakhov@gmail.com

APPLICATIONS OF PARALLEL COMPUTING TECHNOLOGIES FOR MODELING OF THE WIND FLOW AROUND THE ARCHITECTURAL OBSTACLES WITH THE VERTICAL BUOYANCY FORCES

Abstract. Taking into account the high rate of construction in the modern big cities, it is very important to save the natural aerodynamics between the buildings. It is necessary to explore the ventilation of space between architectural structures, making a preliminary prediction before construction starting. The most optimal way of evaluating is to build a mathematical model of air flow. This paper presents numerical solutions of the wind flow around the architectural obstacles with the vertical buoyancy forces. An incompressible Navier-Stokes equation is used to describe this process. This system is approximated by the control volume methodand solved numerically by the projection method. The Poisson equation that is satisfying the discrete continuity equationsolved by the Jacobi iterative method at each time step.For check correctness of mathematical model and numerical algorithm is solved test problem. The numerical solutions of the backward-facing stepflow with the vertical buoyancy forces, which was compared with the numerical results of other authors. This numerical algorithm is completely parallelized using various geometric domain decompositions (1D, 2D and 3D). Preliminary theoretical analysis of the various decomposition methods effectiveness of the computational domain and real computational experiments for this problem were made and the best domain decomposition method was determined. In the future, a proven mathematical model and parallelized numerical algorithm with the best domain decomposition method can be applied for various complex flows with the vertical buoyancy forces.

Keywords: domain decomposition method, flow around the architectural obstacles, backward-facing step flow, projection method, vertical buoyancy forces, mixed convection.

1 Introduction

The increased pace of construction in modern large cities and, in particular, Almaty, leads to a tightening of architectural structures. Due to the increase in the population of cities and to save space, mostly high-rise multi-storey buildings are being built. As a consequence, this entails such consequences as a violation of the natural aerodynamics of the city, which in turn leads to increased gas contamination of the city, the accumulation of heavy metals in the lower atmosphere, and to the violation of the local climate. The building codes and norms currently used in the construction and design of buildings do not contain aerodynamic criteria and coefficients indicating the optimal distance between buildings of different heights. When determining these standards, various natural and climatic features are taken into account, such as wind loads, insolation, etc. Fire safety requirements are also taken into account. However, the above-mentioned documents do not take into account the factor of natural aerodynamics of space between neighboring buildings. The distance between buildings and structures is considered to be the distance between the outer walls or other structures. As a result, when designing, the distances between building objects are laid, which can not provide free movement of the wind vortex, which leads to a disturbance of the natural air flow. In this thesis, a model of aerodynamics between two high-rise buildings is considered. This mathematical model allows you to accurately calculate the optimal distance between the two buildings, which will take into account the climatic features and will preserve the natural purge.

In many technical flows of practical interest, like flow divisions, with the sudden expansion of geometry or with subsequent re-joining, are a common occurrence. The existence of a flow separation and recirculation area has a significant effect on the performance of heat transfer devices, for example, cooling equipment in electrical engineering, cooling channels of turbine blades, combustion chambers and many other heat exchanger surfaces that appear in the equipment.

Many papers are devoted to the motion of a fluid with separation and reconnection of flows without taking into account the buoyancy forces. The importance of this process is indicative of the number of papers where special attention was paid to building equipment [1-3] and developing experimental and theoretical methods for detailed study of flows with separation regions [4-7]. An extensive survey of isothermal flows in fluid flows is given in papers [10-12]. Heat transfer in the flows has been investigated by many authors, like Aung [11, 12], Aung et al. [13], Aung and Worku [14], Sparrow et al. [15, 16] and Sparrow and Chuck [17]. However, published paperson this topic do not take into account the strength of buoyancy force on the flow stream or the characteristics of heat transfer. These effects become significant in the laminar flow regime, where the velocity is relatively low, and when the temperature difference is relatively high. Ngo and Byon [18] studied the location effect of the heater and the size of the heater in a two-dimensional square cavity using the finite element method. Oztop and Abu-Nada [19] numerically investigated natural convection in rectangular shells, partially heated from the side wall by the finite volume method.

In this paper considered the influence of buoyancy forces on the flow and heat transfer characteristics in individual flows. Numerical solutions for a laminar mixed convective airflow (Pr=0.7) in a vertical two-dimensional channel with a backward-facing step to maintain the buoyancy effect are shown in Figure 1. Numerical results of interest, such as velocity and temperature distributions, re-binding lengths and friction coefficients are presented for the purpose of illustrating the effect of buoyancy forces on these parameters [20].

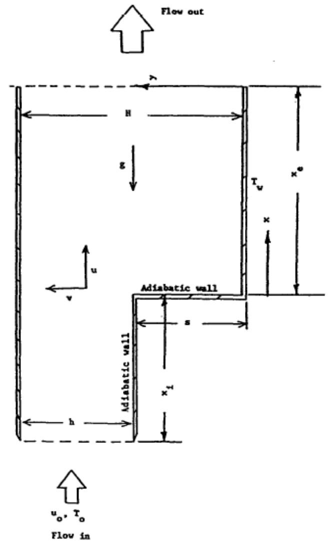

Figure 1 - Schematic representation of the backward-facing step flows.

2. Mathematical formulation of the problem

Consider a two-dimensional laminar convective flow in a vertical channel with a sudden expansion behind the inverse step of height s, as shown in Fig. 1. The straight wall of the channel is maintained at a uniform temperature equal to the temperature of the inlet air T_0. The stepped wall below the stage is heated to a uniform temperature, which can be adjusted to any desired value T_w. The upper part of the stepped wall and the reverse side is installed as an adiabatic surface. The inlet length of the channel x_i and the outlet lower length x_e of the channel are appropriate dimensions. These lengths are assumed to be infinite, but the simulation domain is limited by the length $L_e = x_e + x_i$. The smaller section of the channel before the projection has a height, and the large section below the stage has a height $H = h + s$. Air flows up the channel with mean velocity u_0 and uniform temperature T_0. The gravitational force g in this problem is considered to act vertically downwards.

To describe this physical problem, was used assumption about constant properties, and was used the Boussinesq approximation. This system of equations in an immense form can be written in the form:

$$1)\quad \frac{\partial U}{\partial X} + \frac{\partial U}{\partial Y} = 0 \tag{1}$$

$$2)\quad \frac{\partial U}{\partial t} + U\frac{\partial U}{\partial X} + V\frac{\partial U}{\partial Y} = -\frac{\partial P}{\partial X} + \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right) + \frac{Gr}{\mathrm{Re}^2}\theta \tag{2}$$

$$3)\quad \frac{\partial V}{\partial t} + U\frac{\partial V}{\partial X} + V\frac{\partial V}{\partial Y} = -\frac{\partial P}{\partial Y} + \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}\right) \tag{3}$$

$$4)\quad \frac{\partial \theta}{\partial t} + U\frac{\partial \theta}{\partial X} + V\frac{\partial \theta}{\partial Y} = \frac{1}{\mathrm{Pr}\,\mathrm{Re}}\left(\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2}\right) \tag{4}$$

The dimensionless parameters in the equations given above are defined by the formula: $U = u/u_0$, $V = \upsilon/u_0$, $X = x/s$, $Y = y/s$, $\theta = (T - T_0)/(T_w - T_0)$, $P = p/\rho_0 u_0^2$, $\mathrm{Pr} = \nu/\alpha$, $\mathrm{Re} = u_0 s/\nu$, $Gr = g\beta(T_w - T_0)s^3/\nu^2$, where α – the temperature diffusion, ν – the kinematic viscosity, and β – the thermal expansion coefficient are estimated at the film temperature $T_f = (T_0 + T_w)/2$.

Boundary conditions:

(a) Inlet conditions: At the point $X = -X_i$ and $1 \le Y \le H/s$: $U = u_i/u_0$, $V = 0$, $\theta = 0$.

where u_i is the local distribution of velocities at the inlet, which is assumed to have a parabolic profile and u_i/u_0 an average inlet velocity, that is, given by formula $u_i/u_0 = 6[-y^2 + (H+s)y - Hs]/(H-s)^2$

(b) Outlet conditions: At the point $X = X_e$ and $0 \le Y \le H/s$: $\partial U/\partial X = 0$, $\partial^2\theta/\partial X^2 = 0$, $\partial V/\partial X = 0$.

(c) on the top wall:At the point $Y = H/s$ and $-X_i \le X \le X_e$: $U = 0$, $V = 0$, $\theta = 0$.

(d) on the wall of the upper stage: At the point $Y = 1$ and $-X_i \le X < 0$: $U = 0$, $V = 0$, $\partial\theta/\partial Y = 0$.

(e) on the wall of the lower stage: At point $X = 0$ and $0 \le Y \le 1$: $U = 0$, $V = 0$, $\partial\theta/\partial X = 0$.

(f) on the wall below the stage: At the point $Y = 0$ and $0 \le X \le X_e$: $U = 0$, $V = 0$, $\theta = 1$.

The last term on the right-hand side of equation (2) is the contribution of the buoyancy force. The length of the downstream flow from the simulation area was chosen to be 70 steps (X_e=70). The upper length of the design area was chosen to be 5 steps (i.e. X_i=5), and the velocity profile at the input area was set as parabolic profile, like $u_i/u_0 = 6[-y^2 + (H+s)y - Hs]/(H-s)^2$, and temperature was chosen as uniform T_0.

3. The numerical algorithm

For a numerical solution of this system of equations, the projection method is used [21-24]. The equations are approximated by the finite volume method [21-23]. At the first stage it is assumed that the transfer of momentum is carried out only through convection and diffusion, and an intermediate velocity field is calculated by the fourth-order Runge-Kutta method [21-23]. At the second stage, according to the found intermediate velocity field, there is a pressure field. The Poisson equation for the pressure field is solved by the Jacobi method. At the third stage it is assumed that the transfer is carried out only due to the pressure gradient. At the fourth stage, the equations for the temperature are calculated by the fourth-order Runge-Kutta method.

$$I.\quad \int_{\Omega}\frac{\vec{u}^{*}-\vec{u}^{n}}{\Delta t}d\Omega=-\oint_{\partial\Omega}\left(\vec{u}^{n}\,\vec{u}^{*}-\frac{1}{\mathrm{Re}}\nabla\vec{u}^{*}\right)n_{i}d\Gamma-\int_{\Omega}\frac{Gr}{\mathrm{Re}^{2}}\theta d\Omega,$$

$$II.\quad \oint_{\partial\Omega}(\nabla p)d\Gamma=\int_{\Omega}\frac{\nabla\vec{u}^{*}}{\Delta t}d\Omega,$$

$$III.\quad \frac{\vec{u}^{n+1}-\vec{u}^{*}}{\Delta t}=-\nabla p,$$

$$IV.\quad \int_{\Omega}\frac{\theta^{*}-\theta^{n}}{\Delta t}d\Omega=-\oint_{\partial\Omega}\left(\vec{u}^{n}\,\theta^{*}-\frac{1}{\mathrm{Re\,Pr}}\nabla\theta^{*}\right)n_{i}d\Gamma,$$

4. Parallelization algorithm

For numerical simulation was constructed a computational meshby using the PointWise software. The problem was launched on the ITFS-MKM software using a high-performance computing. This numerical algorithm is completely parallelized using various geometric domain decompositions (1D, 2D and 3D). Geometric partitioning of the computational grid is chosen as the main approach of parallelization. In this case, there are three different ways of exchanging the values of the grid function on the computational nodes of a one-dimensional, two-dimensional, and three-dimensional mesh. After the domain decomposition stage, when parallel algorithms are built on separate blocks, a transition is made to the relationships between the blocks, the simulations on which will be executed in parallel on each processor. For this purpose, a numerical solution of the equation system was used for an explicit scheme, since this scheme is very efficiently parallelized. In order to use the domain decomposition method as a parallelization method, this algorithm uses the boundary nodes of each subdomain in which it is necessary to know the value of the grid function that borders on the neighboring elements of the processor. To achieve this goal, at each compute node, ghost points store values from neighboring computational nodes, and organize the transfer of these boundary values necessary to ensure homogeneity of calculations for explicit formulas.

Data transmission is performed using the procedures of the MPI library [25]. By doing preliminary theoretical analysis of the effectiveness of various domain decomposition methods of the computational domain for this problem, which will estimate the time of the parallel program as the time T_{calc} of the sequential program divided by the number of processors plus the transmission time $T_p = T_{calc} / p + T_{com}$. While transmissions for various domain decomposition methods can be approximately expressed through capacity:

$$\begin{aligned}
T_{com}^{1D} &= t_{send}\,2N^{2}x2\\
T_{com}^{2D} &= t_{send}\,2N^{2}x4p^{1/2}\\
T_{com}^{3D} &= t_{send}\,2N^{2}x6p^{2/3}
\end{aligned}\tag{5}$$

where N^3 – the number of nodes in the computational mesh, p - the number of processors (cores), t_{send} – the time of sending one element (number).

It should be noted that for different decomposition methods, the data transmission cost can be represented as $T_{com}^{1D} = t_{send}\,2N^{2}xk(p)$ in accordance with the formula (5), where $k(p)$ is the proportionality coefficient, which depends on the domain decomposition method and the number of processing elements used.

At the first stage, one common program was used, the size of the array from start to run did not change, and each element of the processor was numbered by an array of elements, starting from zero. For the test simulation is used well known problem – 3D cavity flow. Despite the fact that according to the theoretical analysis of 3D decomposition is the best option for parallelization (Figure 3), computational experiments showed that the best results were achieved using 2D decomposition, when the number of processes varies from 25 to 144 (Figure 3).

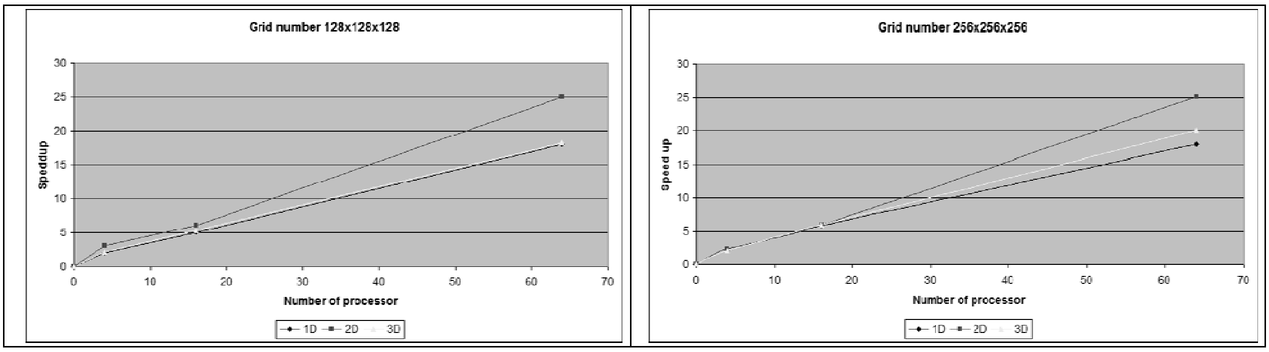

Figure 2 - Speed-up for various domain decomposition methods of the computational domain

Based on the preliminary theoretical analysis of the graphs, the following character can be noted. The simulation time without the interprocessor communications cost with different domain decomposition methods should be approximately the same for the same number of processors and be reduced by T_{calc} / p . In fact, the calculated data show that when using 2D decomposition on different computational grids, the minimal cost for simulation and the cost graphs are much higher, depending on the simulation time, on several processors taken T_{calc} / p .

To explain these results, it is necessary to pay attention to the assumptions made in the preliminary theoretical analysis of efficiency for this task. First, it was assumed that regardless of the distribution of data per processor element, the same amount of computational load was done, which should lead to the same time expenditure. Secondly, it was assumed that the time spent on interprocessorsending's of any degree of the same amount of data is not dependent on their memory choices. In order to understand what is really happening, the following sets of computational simulations test were carried out. For evaluation, the sequence of the first approach was considered when the program is run in a single-processor version, and thus simulates various geometric domain decomposition methods of data for the same amount of computation performed by each processor.

5. Numerical results for test problem

Geometric parameters are indicated in Figure 1: channel length L=75, channel height H=2, step height S=1. Numerical results were obtained for the dimensionless numbers Re=50, Pr=0.7 and Gr=19.1 [20].

Figure 3 shows the comparison of the longitudinal velocity profile with the numerical data of Lin et al. [20] at the point $x/x_f = 0.5$, where $x_f = 2.91$. Figure 4 shows the comparison of temperature profiles with the numerical data of Lin et al. [20] at the point $x/x_f = 0.5$, where $x_f = 2.91$. It can be seen from the figures that the mathematical model and the numerical algorithm which is used in this paper is coincided with the numerical results obtained by Lin et al. [20]. Figure 5 shows the streamlines and the horizontal velocity contour for dimensionless numbers Re=50, Pr=0.7 and Gr=19.1.

Figure 6 shows the temperature profile for dimensionless numbers Re=50, Pr=0.7 and Gr=19.1. For a better understanding of this process from figures 6-8 can be seen the development of the backward-facing step flow with vertical buoyancy force: the initiation and process of the development of the region of flows reconnection with taking into account the buoyancy forces.

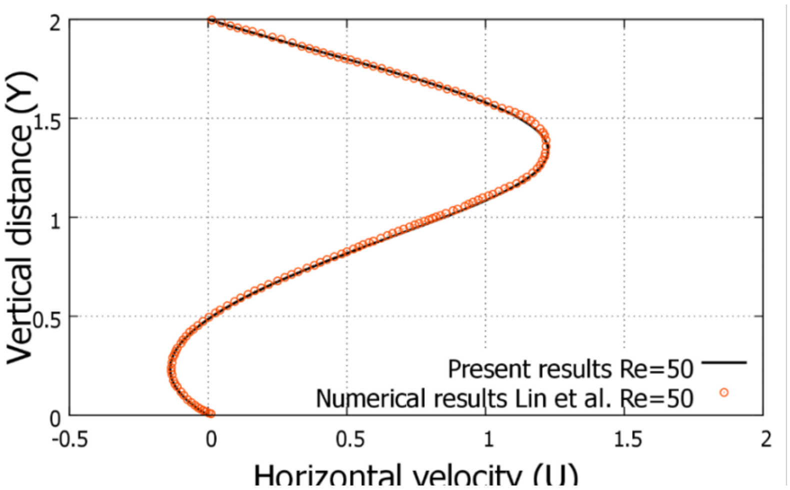

Figure 3 - Velocity profile with vertical buoyancy forces for dimensionless number Re=50, $\Delta T = 1°C$, $x/x_f = 0.5$, where $x_f = 2.91$.

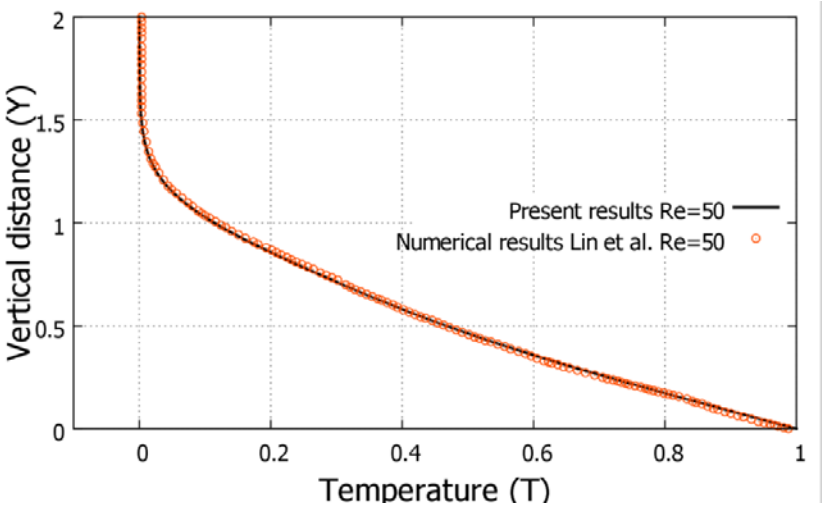

Figure 4 - Temperature profile with vertical buoyancy forces for dimensionless number Re=50, $\Delta T = 1°C$, $x/x_f = 0.5$, where $x_f = 2.91$.

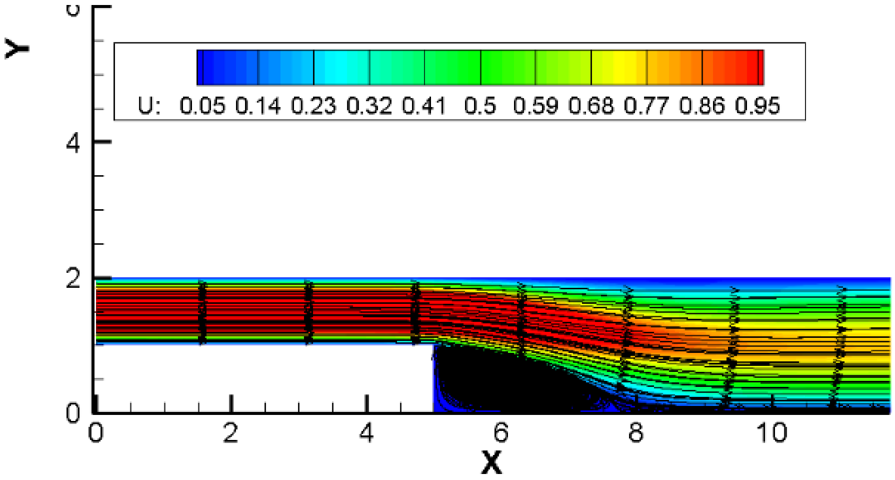

Figure 5 - The contour of the horizontal velocity component with streamlines for dimensionless numbers Re=50, Pr=0.7 and Gr=19.1.

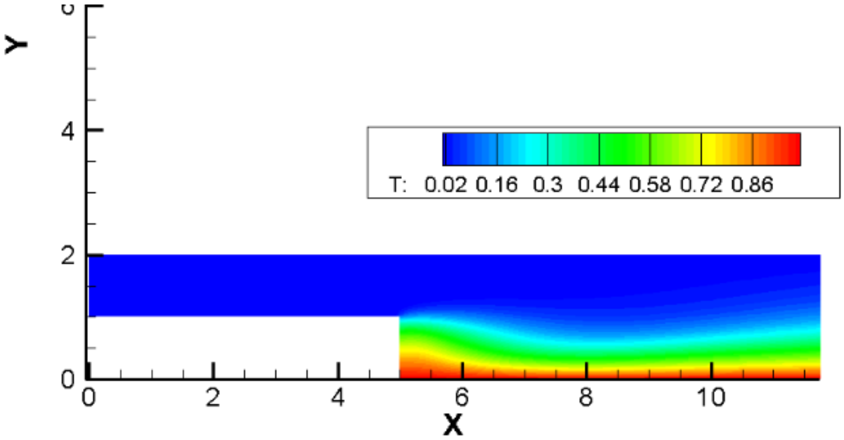

Figure 6 - Temperature contour for dimensionless numbers Re=50, Pr=0.7 and Gr=19.1

6. Numerical results for real problem

For the real problem considered a man-made obstacle 9 floors (27 m) and 5 floors (15 m) buildings. The wind flow is conventionally moving from the high building side to the low one. The following models consider the calm, according to the Beaufort wind speed scale. The speed of wind is in the range from 0 to 0.2 m/s. To find the optimal distance, various parameters prescribed in the above-mentioned standards were used.

According to the fire protection requirements specified in Building norms and regulations of the Republic of Kazakhstan 3.01-01-2002, 2.12 *, the minimum distance between houses with a height of 4 floors or more must be at least 20 meters. However, having constructed this model, a result was obtained, showing that at such a distance between the buildings, there was no wind, therefore, air circulation does not occur in this interval.

After that, the IBC (International Building Code) standard used in the USA was considered, where the distance between two buildings is calculated according to the following formula: $\delta_{MT} = \sqrt{(\delta_{M1})^2 + (\delta_{M2})^2}$, where δ_{MT} - required distance, δ_{M1}, δ_{M2} - height of the first and second buildings, respectively.

For the calculations, the area was divided into the 14 design subareas of different sizes. Each subregion is a grid block, which contains a part of a curvilinear, uneven, unstructured grid. When creating a grid, the number of points is chosen in such a way that no oscillations occur in the solution of the problem and the results are correct for large values of the Reynolds number. Thus, the constructed grid contains more than 100 000 control volumes.

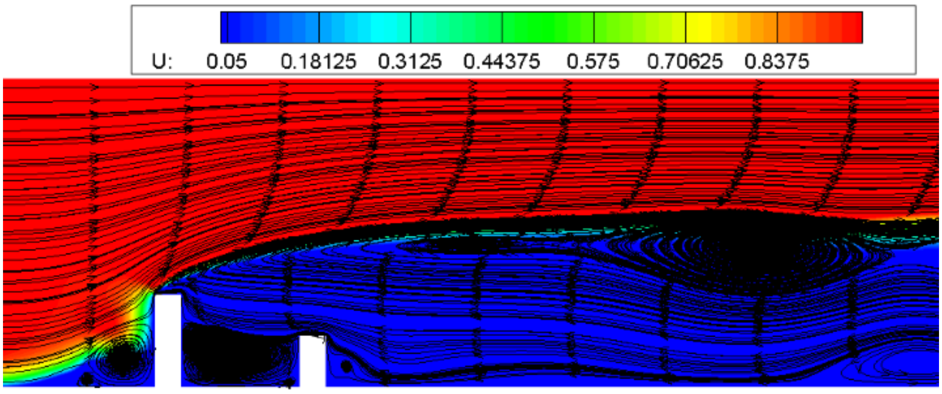

Figure 7 - Horizontal velocity-component and streamlines with buoyancy force for Pr=0.7 and $\Delta T = 1°C$

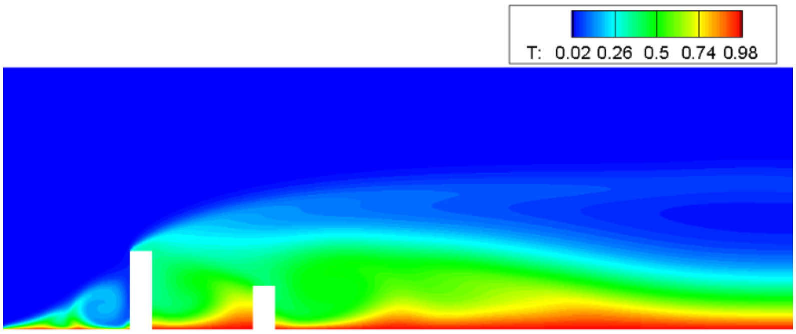

Figure 8 - Temperature component with buoyancy force for Pr=0.7 and $\Delta T = 1°C$

In the interval between the buildings and near the streamlined surfaces of buildings, the grid is thickened, i.e. the sizes of control volumes decrease. This allows for more accurate simulations. As the grid is removed from the vortex zone and the dimensions of the control volumes increase the flow around. Therefore, in areas that are not of great interest in solving a given problem, net catch is smaller. And in the most important zones for the model, the number of nodes is greater, which allows obtaining more accurate results.

In figure 7 shows horizontal velocity-component and streamlines with buoyancy force for Pr=0.7 and $\Delta T = 1°C$. In figure 8 can be seen temperature component with buoyancy force for Pr=0.7 and $\Delta T = 1°C$. The distance obtained for existing buildings was 31 m and satisfied all the standards specified in the republican standards. This model also showed that a vortex does not occur at a given distance. In the following case, the length of extensions to the buildings (balconies, porches, etc.) was added to the previous value and an approximate distance of 35 m was obtained. This model showed that a vortex in the gap occurs and therefore the natural purging between buildings is not broken.

7. Conclusion

Numerical studies of the laminar flow were carried out by the zone of joining the flows behind the backward-facing step with taking into account the buoyancy forces. This gave a deeper insight into the internal flow behind the backward-facing step and the processes of flows reconnection under the influence of temperature effects, which in turn gave an idea of the further appearance of secondary zones. The distance from the ledge to the canal boundary is 4 times the channel height, for a more detailed study of the backward-facing step flows withtaking into account the buoyancy forces [20]. The numerical data of the velocity distribution showed the formation of a primary reattachment zone of backward-facing step flows. To numerically solve the Navier-Stokes equations system, the projection methodwas used. From numerical results can be seen that the realized numerical method gives a small error in comparison with the numerical results of other authors [20] for the dimensionless numbers Re=50, Pr=0.7 and Gr=19.1.

After testing the numerical algorithm with the buoyancy forces, for the real problem considered a man-made obstacle with 9 floors (27 m) and 5 floors (15 m). In this problem, the wind speed was regarded as calm, according to the Beaufort wind speed scale (from 0 to 0.2 m/s). According to the data obtained as a result of the numerical simulation taking into account the buoyancy forces, it can be said that the current standards and rules for construction do not guarantee the required aerodynamics of the terrain.

Also in this paper is used a parallel algorithmto obtain fast numerical results. This parallel algorithm is based on one-dimensional, two-dimensional and three-dimensional domain decomposition method. The numerical results from the 3D cavity flow test problem, which used 1D, 2D and 3D domain decomposition method showed that 3D domain decomposition is not time-consuming compared to 2D domain decomposition, for the number of processors that does not exceed 250, and 3D domain decomposition has more time-consuming software implementation and the use of 2D domain decomposition is sufficient for the scope of the problem. That's why for backward-facing step flow with vertical buoyancy force is used 2D domain decomposition. It should also be noted that setting the boundary conditions is an important process. In the future, this mathematical model and a parallel numerical algorithm can be applied to various complex flows taking into account the buoyancy forces.

REFERENCES

[1] Abbott D.E., Kline S.J. (**1962**) Experimental investigations of subsonic turbulent flow over single and double backward-facing steps,J. Basic Engng. 84:317.

[2] SebanrA. (**1964**) Heat transfer to the turbulent separated flows of air downstream of a step in the surface of a plate,J. Heat Transfer.86:259.

[3] Goldsteinr J., Eriksenv L., Olsonr M., Eckerte R.G. (**1970**) Laminar separation, reattachment and transition of flow over a downstream-facing step,J.Basic Engng. 92:732.

[4] Durst F., Whitelawj H. (**1971**) Aerodynamic properties of separated gas flows: existing measurements techniques and new optical geometry for the laser-Doppler anemometer,Prog. Heat Mass Transfer.4:311.

[5] Gosmana D., Punw M. (**1974**) Lecture notes for course entitled: 'Calculation of recirculating flow, Heat Transfer Rep. 74:2.

[6] Kumara, Yajnikk S. (**1980**) Internal separated flows at large Reynolds number, J. Fluid Mech. 97:27.

[7] Chiang T.P., Tony W.H., Sheu, Fang C.C. (**1999**) Numerical investigation of vortical evolution in backward-facing step expansion flow, Appl. Math. 23:915-932.

[8]. Armaly B. F. and Durst F.(**1980**) Reattachment length and recirculation regions downstream of two dimensional single backward facing step. In Momentum and Hear Transfer Process in Recirculating Flows, ASME HTDVol. 13, pp. l-7. ASME, New York.

[9]. Eaton J. K. and Johnson J. P., (**1981**) A review of research on subsonic turbulent flow reattachment, AIAA J. 19:1093-1100.

[10]. Simpson R. L.(**1981**) A review of some phenomena in turbulent flow separation, J. Fluid Engng 103:528-530.

[11]. Aung W. (**1983**) An experimental study of laminar heat transfer downstream of backsteps, J. Heat Transfer 105:823-829.

[12]. Aung W., (**1983**) Separated forced convection, Proc. ASMEIJSME Thermal Enana Joint Con/.. 2: 499-515. ASME. New York.

[13]. Aung W., Baron A. and Tsou F. K., (**1985**) Wall independency and effect of initial shear-layer thickness in separated flow and heat transfer, Int. J. Hear Mass Transfer, 28:1757-1771.

[14]. Aung W. and Worku G., (**1986**) Theory of fully developed. combined convection including flow reversal, J. Hear Transfer, 108:485-488.

[15]. Sparrow E. M., Chrysler G. M. and Azevedo L. F.(**1984**) Observed flow reversals and measured-predicted Nusselt numbers for natural convection in a one-sided heated vertical channel, J. Heat Transfer, 106:325-332.

[16]. Sparrow E. M., Kang S. S. and Chuck W .(**1987**) Relation between the points of flow reattachment and maximum heat transfer for regions of flow separation. Int. J. Heat Mass Transfer, 30:1237-1246.

[17]. Sparrow E. M. and Chuck W., (**1987**) PC solutions for heat transfer and fluid flow downstream of an abrupt, asymmetric enlargement in a channel, Numer. Heat Transfer, 12:1940.

[18]. Ngo I., Byon C., (**2015**) Effects of heater location and heater size on the natural convection heat transfer in a square cavity using finite element method, J. Mech. Sci. Technol. 29 (7):2995.

[19]. Oztop H. F., Abu-Nada E. (**2008**) Numerical study of natural convection in partially heated rectangular enclosures filled with nanofluids, Int. J. Heat. Fluid Fl., 29(5):1326 - 1336.

[20]. Lin J.T., Armaly B.F., Chen T.S. (**1990**) Mixed convection in buoyancy-assisting, vertical backward-facing step flows, International Journal of Heat and Mass Transfer. 33(10): 2121-2132.

[21]. Issakhov A., (**2015**) Mathematical modeling of the discharged heat water effect on the aquatic environment from thermal power plant // International Journal of Nonlinear Science and Numerical Simulation, 16(5):229–238, doi:10.1515/ijnsns-2015-0047.

[22]. Issakhov A., (**2016**) Mathematical modeling of the discharged heat water effect on the aquatic environment from thermal power plant under various operational capacities // Applied Mathematical Modelling, 40(2):1082–1096 http://dx.doi.org/10.1016/j.apm.2015.06.024.

[23]. Issakhov A. (**2017**) Numerical Study of the Discharged Heat Water Effect on the Aquatic Environment from Thermal Power Plant by using Two Water Discharged Pipes. International Journal of Nonlinear Sciences and Numerical Simulation, 18(6):469–483.

[24]. Chorin A.J. (**1968**) Numerical solution of the Navier-Stokes equations, Math. Comp.22: 745-762.

[25]. Karniadakis G. E., Kirby II R. M. (**2000**) Parallel Scientific Computing in C++ and MPI: A Seamless Approach to Parallel Algorithms and their Implementation. CambridgeUniversityPress, p. 630.

А. Исахов, А. Абылкасымова, М. Сакыпбекова

Казахский национальный университет им. аль-Фараби, Алматы, Казахстан

ПРИМЕНЕНИЕ ТЕХНОЛОГИЙ ПАРАЛЛЕЛЬНЫХ ВЫЧИСЛЕНИЙ ДЛЯ МОДЕЛИРОВАНИЯ ВЕТРОВОГО ПОТОКА ВОКРУГ АРХИТЕКТУРНЫХ ПРЕПЯТСТВИЙ С ВЕРТИКАЛЬНЫМИ СИЛАМИ ПЛАВУЧЕСТИ

Аннотация. Принимая во внимание высокие темпы строительства в современных крупных городах, очень важно сохранить естественную аэродинамику между зданиями. Необходимо исследовать аэродинамику между архитектурными сооружениями, сделав предварительное прогнозное исследование перед началом строительства. Наиболее оптимальным способом оценки является построение математической модели воздушного потока. В настоящей работе представлены численные решения ветрового потока вокруг архитектурных препятствий с вертикальными силами плавучести. Для описания этого процесса используется несжимаемое уравнение Навье-Стокса. Эта система аппроксимируется методом контрольного объема и решается численно методом расщепление по физическим параметрам. Уравнение Пуассона, удовлетворяющее уравнению дискретной непрерывности, решаемому итерационным методом Якоби на каждом временном шаге. Для проверки правильности математической модели и численного алгоритма решена тестовая проблема. Численные решения обратного потока с вертикальными силами плавучести, которые сравнивались с численными результатами других авторов. Этот численный алгоритм полностью распараллеливается с использованием различных методов геометрических декомпозиций (1D, 2D и 3D). В работе проделан предварительный теоретический анализ различных методов декомпозиции вычислительной области и реальных вычислительных экспериментов для этой задачи, и был определен наилучший метод разложения доменов. В будущем для различных сложных потоков с вертикальными силами плавучести может быть применена проверенная математическая модель и параллельный численный алгоритм с наилучшим методом декомпозиции доменов.

Ключевые слова: метод декомпозиции доменов, обтекание архитектурных препятствий, обратный поток, метод проецирования, вертикальные силы плавучести, смешанная конвекция.

А. Исахов, А. Абылкасымова, М. Сакыпбекова

әл-Фараби атындағы Қазақ Ұлттық университеті, Алматы, Қазақстан

ПАРАЛЛЕЛЬДІ ЕСЕПТЕУ ТЕХНИКАСЫНЫҢ СӘУЛЕЛЕНУ КЕДЕРГІЛЕРІНІҢ АЙНАЛАСЫНДА ЖЕЛДІҢ АҒЫНЫН ҮЛГІЛЕУ ҮШІН ҚОЛДАНУ

Аннотация. Қазіргі заманғы ірі қалаларда құрылыстың жоғары қарқынын ескере отырып, ғимараттар арасындағы табиғи аэродинамиканы сақтау өте маңызды. Архитектуралық құрылыс арасындағы аэродинамиканысын зерттеу қажет, құрылысты бастамас бұрын алдын-ала болжамды зерттеуді жасау керек.Бағалаудың оңтайлы әдісі ауа ағынының математикалық моделін құру болып табылады. Бұл мақалада тік қозғалыс күштерімен архитектуралық кедергілер айналасында желдің ағынының сандық шешімдері көрсетілген. Бұл процесті сипаттау үшін сығылмайтын Навье-Стокс теңдеуі қолданылады. Бұл жүйе ақырлы көлем әдісімен жуықталып, физикалық параметрлер бөліну әдісімен сандық түрде шешілді. Дискретті үзіліссіз теңдеуін қанағаттандыратын Пуассон теңдеуі әр кезеңде Якоби итеративті әдісімен шешіледі. Тексеру үшін математикалық модельдің дұрыстығын және сандық алгоритмді сынау мәселесі шешіледі. Басқа авторлардың сандық нәтижелерімен салыстырылған тік күшінің күштерімен артқа бағытталған қадамдық ағынның сандық шешімдері қарастылды. Бұл сандық алгоритм әртүрлі геометриялық доменді декомпозиция (1D, 2D және 3D) арқылы толығымен параллельленеді. Сандық доменнің әртүрлі декомпозиция әдістерінің алдын-ала теориялық анализі жәнеосы мәселе бойынша нақты есептеу эксперименттер жасалды және ең тиімді домен декомпозиция әдісі анықталды. Келешекте тік қаттылық күштерімен әртүрлі күрделі ағындар үшін дәлелденген математикалық модель және үздік домен декомпозиция әдісімен параллельді сандық алгоритм қолданылуы мүмкін.

Түйін сөздер: домендідекомпозиция әдісі, архитектуралық кедергілердің ағымы, артқа қарай жүретін қадамдық ағын, проекциялау әдісі, тік күші күші, аралас конвекция.

МАЗМҰНЫ

СОДЕРЖАНИЕ

* * *

CONTENTS

* * *

Publication Ethics and Publication Malpractice in the journals of the National Academy of Sciences of the Republic of Kazakhstan

For information on Ethics in publishing and Ethical guidelines for journal publication see http://www.elsevier.com/publishingethics and http://www.elsevier.com/journal-authors/ethics.

Submission of an article to the National Academy of Sciences of the Republic of Kazakhstan implies that the described work has not been published previously (except in the form of an abstract or as part of a published lecture or academic thesis or as an electronic preprint, see http://www.elsevier.com/postingpolicy), that it is not under consideration for publication elsewhere, that its publication is approved by all authors and tacitly or explicitly by the responsible authorities where the work was carried out, and that, if accepted, it will not be published elsewhere in the same form, in English or in any other language, including electronically without the written consent of the copyright-holder. In particular, translations into English of papers already published in another language are not accepted.

No other forms of scientific misconduct are allowed, such as plagiarism, falsification, fraudulent data, incorrect interpretation of other works, incorrect citations, etc. The National Academy of Sciences of the Republic of Kazakhstan follows the Code of Conduct of the Committee on Publication Ethics (COPE), and follows the COPE Flowcharts for Resolving Cases of Suspected Misconduct (http://publicationethics.org/files/u2/New_Code.pdf). To verify originality, your article may be checked by the Cross Check originality detection service http://www.elsevier.com/editors/plagdetect.

The authors are obliged to participate in peer review process and be ready to provide corrections, clarifications, retractions and apologies when needed. All authors of a paper should have significantly contributed to the research.

The reviewers should provide objective judgments and should point out relevant published works which are not yet cited. Reviewed articles should be treated confidentially. The reviewers will be chosen in such a way that there is no conflict of interests with respect to the research, the authors and/or the research funders.

The editors have complete responsibility and authority to reject or accept a paper, and they will only accept a paper when reasonably certain. They will preserve anonymity of reviewers and promote publication of corrections, clarifications, retractions and apologies when needed. The acceptance of a paper automatically implies the copyright transfer to the National Academy of Sciences of the Republic of Kazakhstan.

The Editorial Board of the National Academy of Sciences of the Republic of Kazakhstan will monitor and safeguard publishing ethics.

Правила оформления статьи для публикации в журнале смотреть на сайтах:

www:nauka-nanrk.kz

http://www.physics-mathematics.kz

ISSN 2518-1726 (Online), ISSN 1991-346X (Print)

Редакторы *М. С. Ахметова, Т.А. Апендиев, Д.С. Аленов*
Верстка на компьютере *А.М. Кульгинбаевой*

Подписано в печать 08.06.2018.
Формат 60x881/8. Бумага офсетная. Печать – ризограф.
5,5 п.л. Тираж 300. Заказ 4.

Национальная академия наук РК
050010, Алматы, ул. Шевченко, 28, т. 272-13-18, 272-13-19