UDC 519.63; 519.684

*Issakhov A.A., Baitureyeva A.R.

Faculty of Mechanics and Mathematics,
al-Farabi Kazakh National University, Almaty, Kazakhstan
*e-mail: alibek.issakhov@gmail.com

Numerical simulation of wind flow around the architectural buildings

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. In this paper considered the aerodynamics between two high-rise buildings. A mathematical model of the wind flow around obstacles was carried out. In this paper were studied existing criteria about the distance between architectural objects and obtained graphics. There were considered elements of program realization of a numerical method and the cases of non-stationary two-dimensional flow calculation algorithms. The results of test calculations and plot are presented. This mathematical model allows to precisely calculate the optimal distance between the two buildings, which will take into account the climatic features, and will preserve the natural ventilation.

Key words: Navier-Stokes equation, large eddy simulation method, splitting method, aerodynamics.

Introduction

The high rate of construction in modern cities (including the Almaty) leads to the increasing of the architectural buildings amount. Due to the increase in population and in order to save space, mostly are currently constructed high-rise buildings. This leads to disruption of the natural aerodynamics of the city. Because of this, there is an increase of gas contamination of the city, the accumulation of heavy metals in the lower atmosphere and disruption of the local climate.

Building norms and rules now used in the construction and design of buildings, do not include the aerodynamic criteria and coefficients for indicating the optimum distance between buildings of different heights. The distance between buildings is considered to be the clear distance between the outer walls and other structures. The distances, which are specified between the objects while designing the buildings, can not ensure the free movement of wind vortex. This leads to disruption of the natural air flow.

Mathematical Model

The Navier-Stokes equations, obtained from the mass conservation law and Newton's second law, are the basis for the construction of mathematical model for wind flow around technogenic obstacles.

$$\nabla \cdot \vec{\upsilon} = 0$$

$$\frac{\partial \vec{\upsilon}}{\partial t} + \vec{\upsilon} \cdot \nabla \vec{\upsilon} = -\frac{1}{\rho} \nabla p + \nu \Delta \vec{\upsilon} \qquad (1)$$

Here $\vec{\upsilon}$ – the velocity vector, t – time, ρ – the density, p – the hydrodynamic pressure fluid, $\nu = \mu / \rho$ - molecular kinematical viscosity.

To minimize the number of calculations and experimental tests and to get the optimal flow pattern, it is necessary to convert all the parameters (flow rate, length, etc.) in the dimensionless parameters [1, 3-6]. The two streams are dynamically similar if dimensionless number, which determine flows, are equal.

Dimensionless variables are described as follows: [11-13]:

$$x_i^* = \frac{x_i}{L_0}, \quad t^* = \frac{\upsilon_0 t}{L_0},$$

$$\upsilon_i^* = \frac{\upsilon_i}{\upsilon_0}, \quad p^* = \frac{p - p_0}{\rho \upsilon_0^2},$$

Hence the equation of fluid motion and the continuity equation (1) in dimensionless form:

$$\frac{\partial \upsilon_i^*}{\partial t^*} + \upsilon_j^* \frac{\partial \upsilon_i^*}{\partial x_j^*} = \frac{1}{\mathrm{Re}} \frac{\partial^2 \upsilon_i^*}{\partial x_j^{*2}} - \frac{\partial p^*}{\partial x_i^*}$$

$$\frac{\partial \upsilon_i^*}{\partial x_i^*} = 0$$

Numerical Algorithm

Consider the Navier - Stokes equation in the form of integral conservation laws for an arbitrary fixed volume Ω with boundary $d\Omega$ [5, 10]:

$$\int_\Omega \left(\frac{\partial Q}{\partial t}\right) d\Omega + \oint_{\partial\Omega} (F_i + G_i) n_i d\Gamma = 0 \quad (2)$$

Grid functions will be defined in the center of the cell and the values of flows across the border in divided cells. The volume of the cell is denoted by grid functions.

Now we perform discretization of the equation (2) by the control volume (CV) and the control surface (the CS)

$$\sum_{CV} \left(\frac{\Delta Q}{\Delta t}\right) \Delta\Omega + \sum_{CS} (F_i + G_i) n_i \Delta\Gamma = 0 \quad (3)$$

A similar representation:

$$\sum_{CV} \Delta Q \Delta\Omega + \sum_{CS} \Delta t (F_i + G_i) n_i \Delta\Gamma = 0 \quad (4)$$

For the numerical solution of equation (4) splitting scheme by physical parameters is used [2, 7-9, 12, 13]. At the first stage it is assumed that the transfer of momentum carried out only by convection and diffusion. The intermediate velocity field is found by 5-step Runge - Kutta method. At the second stage the pressure field is found based on the intermediate velocity field. Poisson equation for the pressure field is solved by Jacobi method. In a third step it is assumed that the transfer is carried out only by the pressure gradient.

I. $$\int_\Omega \frac{\vec{u}^* - \vec{u}^n}{\tau} d\Omega = -\oint_{\partial\Omega} (\nabla \vec{u}^n \vec{u}^* - \upsilon \Delta \vec{u}^*) n_i d\Gamma$$

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

III. $$\frac{\vec{u}^{n+1} - \vec{u}^*}{\tau} = -\nabla p.$$

Boundary Conditions

Two types of boundary conditions are used: Dirichlet and Neumann. For the pressure P on all borders except the output is used Neumann boundary condition. To the velocity components is used Dirichlet boundary condition. Here u_0 - the input velocity profile.

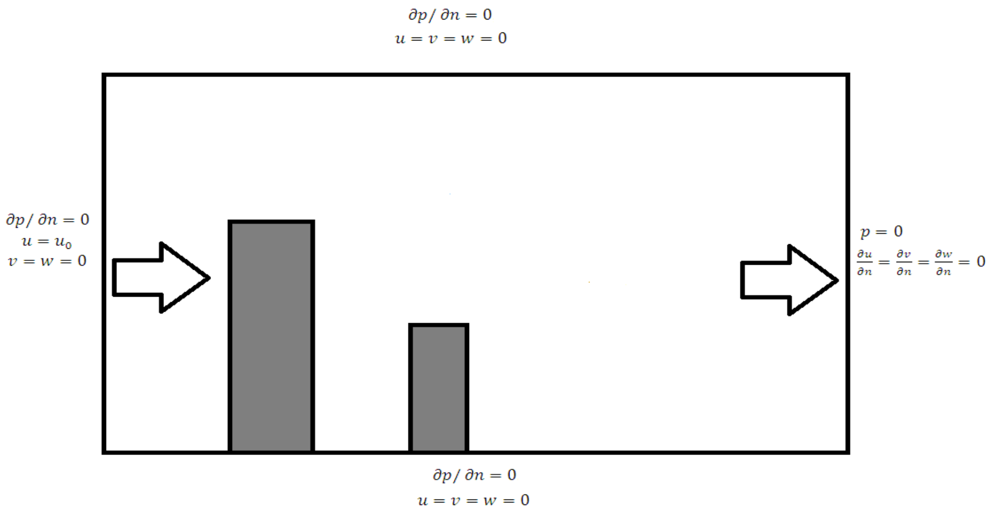


Figure 1 – Scheme of boundary and initial conditions.

Computational grid

Computational area was divided into 12 of subdomains of different sizes. Each subdomain is a grid unit, which contains a portion of a curved, irregular, unstructured grid. The number of grid points is selected in such way that oscillations wouldn't arise while solving the problem and the results would be correct at high Reynolds numbers. Thus, the computational grid contains over 100,000 control volumes. The mesh condensed between the buildings and near the streamlined surfaces of buildings. This allows to carry out more accurate calculation. Further away from the vortex zone, size control volumes increases.

Results

For the experiment, 9 floor (27 meters) and 5 floor (15 m) buildings have been considered as technogenic obstacles. Wind flow conditionally moves from high to low buildings. The following model examines "calm", according to the Beaufort wind velocities scale. This wind velocity is in the range from 0 to 0.2 m/s. Various parameters were used to find the optimum distance. They are described in Building norms and rules of RK 3.01-01-2002 [14] and other normative documents. [15]

Figures 2 - 10 illustrate the results of wind flow around the technogenic obstacles for various distances between buildings (l = 20; 25; 31; 35 m) and for different wind speeds (= 0.1; 0.2 m / s).

According to the fire protection requirements, which are set out in the document [14], the minimum distance between the buildings of 4-floor height or higher must be not less than 20 meters. However, implementation of this model showed that at such distance a wind vortex didn't appeared between the buildings. Consequently, there is no air circulation in this diapason.

a) b)

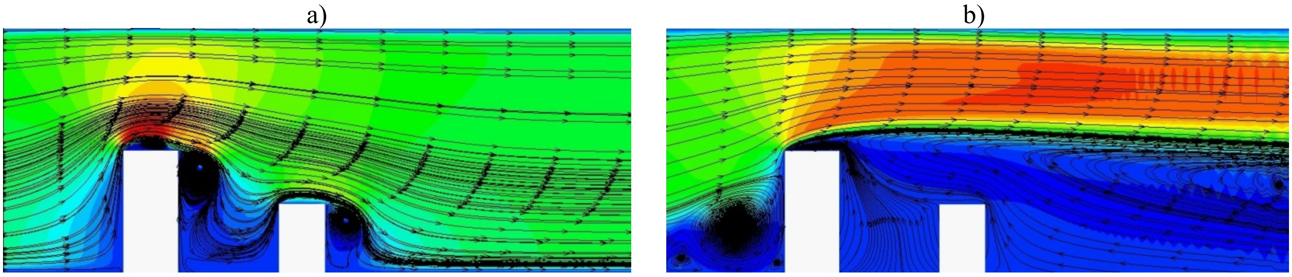


Figure 2 - The horizontal velocity components of (ν= 0.1 m/s) of flow around the technogenic obstacles with streamlines, which are calculated by using the DNS for the distance l = 20 (m) at different times (at t=0.001, t=2).

a) b)

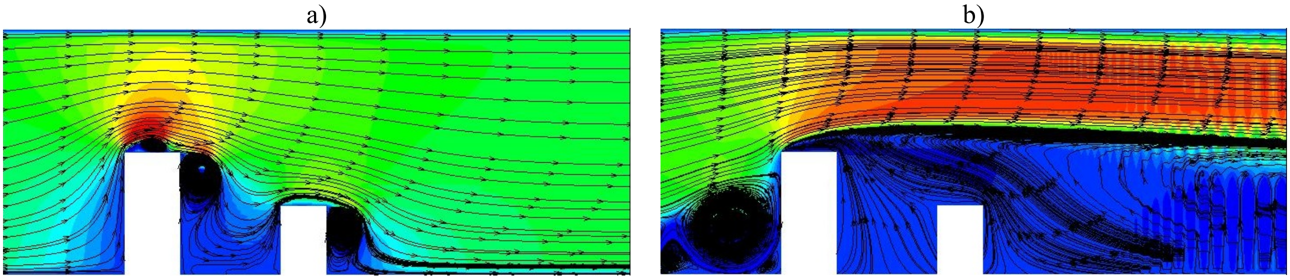
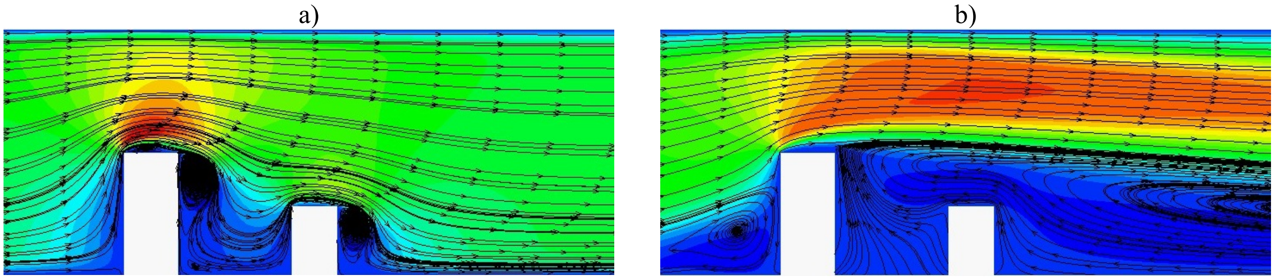


Figure 3 - The horizontal velocity components of (ν=0.2 m/s) of flow around the technogenic obstacles with streamlines, which are calculated by using the DNS for the distance l = 20 (m) at different times (at t=0.001, t=2).

In the following case the length of outbuildings (balconies, porch, etc.) has been added to the previous value and the distance of 25 m was obtained. The results showed that the vortex occurs at the initial moment, but over time it disappears, that is, wind circulation is intermittent.

a) b)



c) d)

Figure 4 - The horizontal velocity components of (v=0.1 m/s) of flow around the technogenic obstacles with streamlines, which are calculated by using the DNS for the distance l = 25 (m) at different times (at t=0.001, t=2).

a) b)

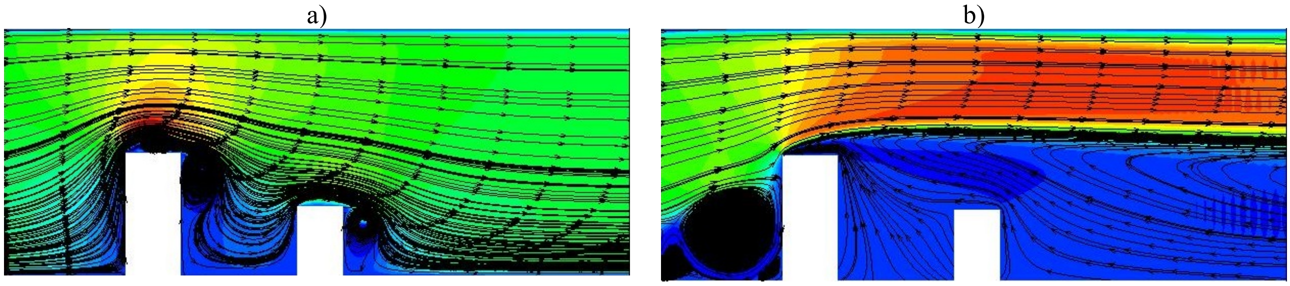


Figure 5 - The horizontal velocity components of (v=0.2 m/s) of flow around the technogenic obstacles with streamlines, which are calculated by using the DNS for the distance l = 25 (m) at different times (at t=0.001, t=2).

After that, the standard IBC (International Building Code), which is used in the United States, was considered. Here, the distance between the two buildings is calculated according to the following formula [15]:

$$\delta_{MT} = \sqrt{(\delta_{M1})^2 + (\delta_{M2})^2}$$

where δ_{MT} - the required distance, δ_{M1}, δ_{M2} - the height of the first and second buildings respectively.

The resulting distance between buildings was equal to 31 m, and satisfies all the standards specified in the national standards. The model also showed that the vortex at a given distance wouldn't arise.

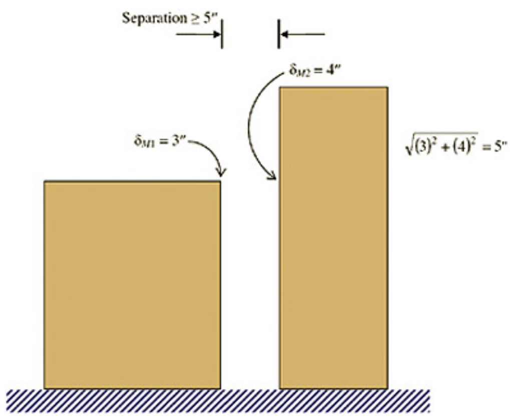


Figure 6 - The scheme of calculating the distance between the two high-rise buildings according to IBC 2009 1613.6.7 [15]

a) b)

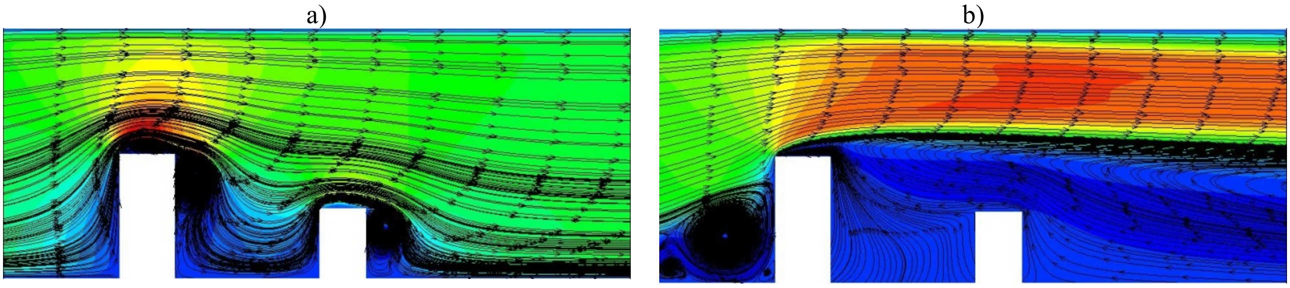
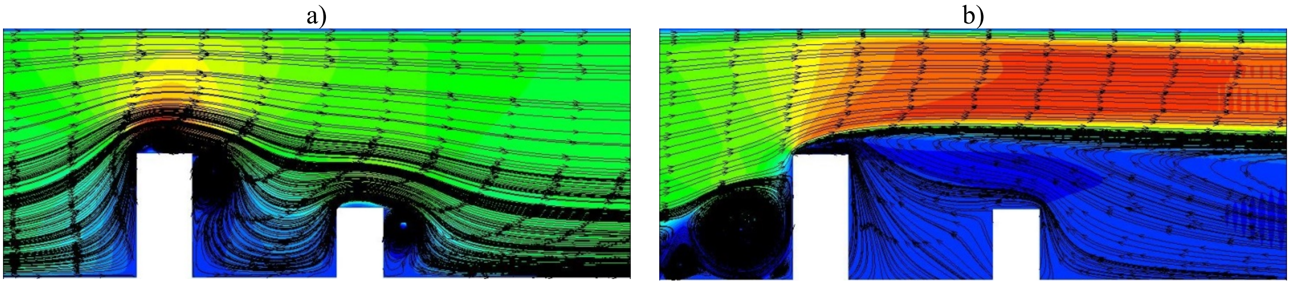


Figure 7 – The horizontal velocity components of (v=0.1 m/s) of flow around the technogenic obstacles with streamlines, which are calculated by using the DNS for the distance l = 31 (m) at different times (at t=0.001, t=2).

a) b)



c) d)

Figure 8 – The horizontal velocity components of (v=0.2 m/s) of flow around the technogenic obstacles with streamlines, which are calculated by using the DNS for the distance l = 31 (m) at different times (at t=0.001, t=2).

a) b)

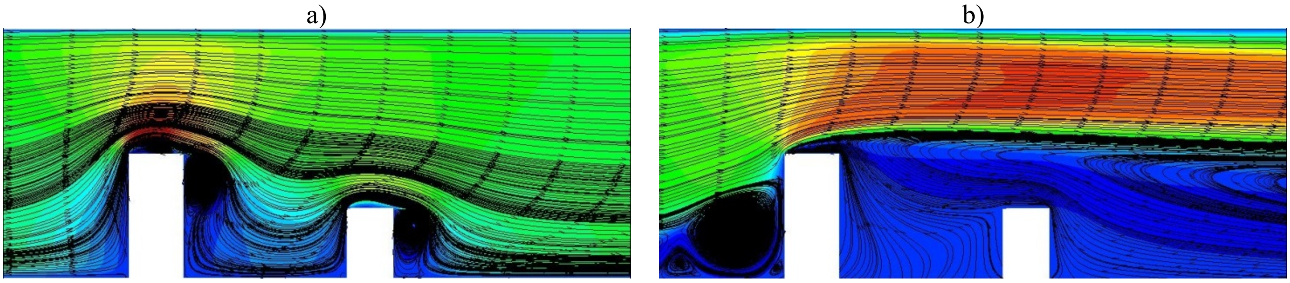


Figure 9 – The horizontal velocity components of (v=0.1 m/s) of flow around the technogenic obstacles with streamlines, which are calculated by using the DNS for the distance l = 35 (m) at different times (at t=0.001, t=2).

a) b)

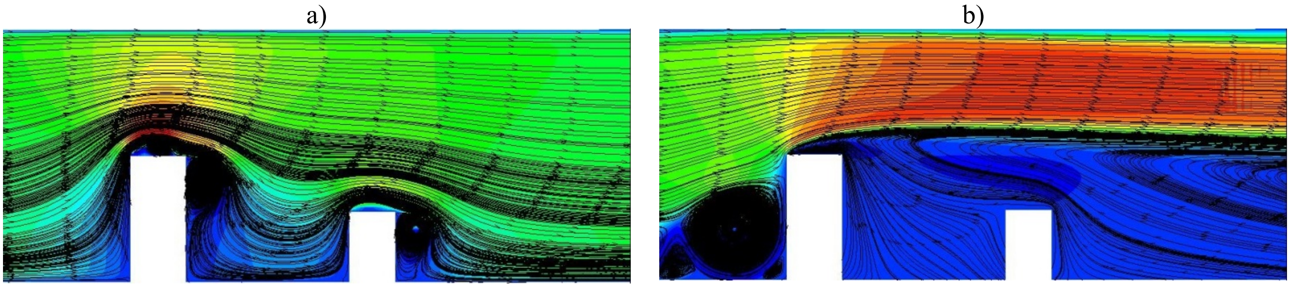


Figure 10 – The horizontal velocity components of (v=0.2 m/s) of flow around the technogenic obstacles with streamlines, which are calculated by using the DNS for the distance l = 35 (m) at different times (at t=0.001, t=0.5, t=1, t=2).

Here the length of outbuildings (balconies, porch, etc.) has been added to the previous value and obtained an approximate distance of 35 m. The model showed that the vortex produced between buildings, and therefore the natural aerodynamics between buildings is not broken.

Conclusion

In this paper, a mathematical model of wind flow around technogenic obstacles was built. It was based on full two-dimensional Navier - Stokes equations.

According to data, obtained as a result of the study, can be said, that the current standards and regulations for construction does not guarantee the necessary aerodynamics of the local areas. It should be noted that the advantages of this approach is the ability to create a model of wind flow around buildings as close as possible to reality. Thus, it is possible to assess in advance the optimality of the selected distance. This prevents the probability of choosing the wrong distance between close standing buildings, which will lead to a breach of natural circulation of air.

References

1. Anderson D., Tannehil Dzh., Pletcher R. Vychislitelnaya gidromehanika i teploobmen. M.: Mir. – 1990. – T. 1. – P. 337.
2. Belov I.A., Isaev S.A. Modelirovanie turbulentnyh techeniy: Uchebnoe posobie. // SPb: BGTU. – 2001. – P. 105.
3. Anderson J.D. Computational Fluid Dynamics. The Basics with Applications. – 1995. – P. 547.
4. Fletcher K. Vychislitelnye metody v dinamike zhidkostei. – Moskva: Mir. - 1991. – T.2. – P. 552.
5. Samarskiy A.A., Nikolaev E.S. Metody resheniya setochnyh uravneniy. – Moskva: Nauka, – 1978. – P. 592.
6. Rouch P. Vychislitelnaya gidrodinamika. – M.: Mi. – 1972. – P. 612.
7. Issakhov A. Large eddy simulation of turbulent mixing by using 3D decomposition method. // J. Phys.: Conf. Ser. – 2011. – Vol. 318. . – P. 042051.
8. Issakhov A.A. Pryamoe chislennoe modelirovanie (DNS) turbulentnyh techeniy s ispolzovaniem parallelnyh tehnologiy. Vestnik KazNU. – 2012. – № 2(73). – P .81-91.
9. Loicyanskiy L.G. Mehanika zhidkosti i gaza. // M.: GITTL. – 2003.
10. Chung T.J. Computational fluid dynamics. // CUP. – 2002.
11. Sedov L. I. Mehanika sploshnoi sredy.— M.: Nauka. – 1970. – T. 2. – P. 568.
12. Issakhov A. A., Abdibekov U.S., Zhumagulov B.T. Osnovy matematicheskogo i kompyuternogo modelirovaniya estestvenno-fizicheskih processov // "Qazaq universitety". – 2014. – P. 208.
13. Issakhov A.A. Matematicheskaya model teplovoi nagruzki na vodnuyu sredu pri razlichnyh ekspluatacionnyh moshchnostyah, Vestnik KazNU. – 2014. – Vol. 81. – No 2. – P.52-61.
14. SNiP RK 3.01-01-2002. 2.12* Gradostroitelstvo. Planirovka i zastroika gorodskih i selskih naselennyh punktov. P.7. Parametry zastroiki zhilyh i obshchestvenno-delovyh zon.
15. The International Building Code (IBC), Section 1613.6.7. – 2009.