ECCOMAS Congress 2016
VII European Congress on Computational Methods in Applied Sciences and Engineering
M. Papadrakakis, V. Papadopoulos, G. Stefanou, V. Plevris (eds.)
Crete Island, Greece, 5–10 June 2016

MATHEMATICAL MODELLING OF CITY AERODYNAMICS

Svetlana A. Valger[1], **Natalya N. Fedorova**[1,2], **Alexandr A. Fedorov**[1,2]

[1] Novosibirsk State University of Architecture and Civil Engineering (Sibstrin)
Leningradskaya Str., 113, Novosibirsk (Russia), swetla-ya@mail.ru

[2] Khristianovich Institute of Theoretical and Applied Mechanics SB RAS
Institutskaya Str., 4/1, Novosibirsk (Russia), {nfed, fedorov}@itam.nsc.ru

Keywords: city aerodynamics, wind actions, pedestrian comfort, turbulence, CAE, Fluent.

Abstract. *The article is devoted to the numerical simulation of turbulent air flows in the vicinity of building complexes. The simulation was performed under isothermal assumption on the basis of 3D URANS equations supplemented with the k-ω SST turbulence model. The ANSYS Fluent Software was used as the main modeling tool. At the first stage, the atmospheric flow in the neighborhood of a building of a complex shape was carried out taking into account the surrounding objects. The numerical simulation was performed under the conditions of the experiments. The 3D structure of the flow in the vicinity of the building was obtained and the comparison of the calculation results with the experimental data on the pressure coefficient distribution on the walls of the building was performed. At the second stage, we made the full-scale numerical modeling of the flow around the complex of high-rise buildings in the Frankfurt/Main, Germany, city center. According to the results of the numerical modeling the transient flow structure in the vicinity of the buildings was obtained and the mean and the fluctuating components of the wind load were estimated. The pedestrian comfortable/uncomfortable zones were also detected for this area of the city.*

1 INTRODUCTION

Aerodynamics of buildings is an area of applied research which has emerged in the recent years as an academic discipline examining the wind impacts on civil engineering structures disposed on a ground surface in the atmospheric boundary layer (ABL). In the recent years the density of urban environment tends to increase. When new high-rise objects are constructed within an already densely-built area, it leads to changes in air flow around the already existing buildings through vortex formation or higher wind velocities. The study of wind actions on buildings is one of the most important design stages since such actions affect significantly the mechanical strength of buildings. Wind actions accompanied by the acceleration due to structure vibrations under dynamic gusts may violate the normal service conditions in upper-floor premises of high-rise buildings.

The study of air flow in the vicinity of buildings is especially important for design of additional structural elements and ventilated facades. Caving of facades elements under the strong wind conditions is often observed. To prevent, specific calculation methods have to be developed which take into account the inhomogeneous distribution of wind load and the presence of local zones of negative pressure on the building walls. Another important task is to provide a comfortable environment for human activity, which requires a study of urban pedestrian comfort.

Presently, there are two approaches to solving building aerodynamics problems: the theoretical approach and the experimental one. The theoretical approach is based on rational mechanics methods, in particular, on fluid dynamics, and it applies mathematical models to describe transient turbulent air flows. For solving the initial-boundary-value problems, both theoretical methods and numerical algorithms are used. In the latter half of the XX century, the basic concepts of hydrodynamic processes in atmosphere were formulated [1 – 3] and, in this way, a certain theoretical basis for evaluating the wind actions was developed [4 – 6]. Progress in computing means and numerical methods has enabled the use of computer science in solving hydro- and gas-dynamics problems, including problems in the exterior building aerodynamics. The experimental approach is based on wind-tunnel studies [7 – 11] followed by treatment of the experimental data aimed to get empirical engineering dependences to describe the wind actions on various objects [12 – 14]. It should be noted, that the experimental modeling is time and cost consuming. Moreover, experiments are significantly limited by parameters of wind facilities. For example, it is quite difficult to reproduce the atmospheric boundary layer in the experiment. As for semi-empirical engineering methods, they permit evaluation of the mean and fluctuating wind-load components only for very simple configurations and fail to account for the interference effects between buildings as well as for complex phenomena of self-exciting oscillations and resonances arising under external dynamic actions. It leads to the necessity to develop new approaches that allow considering the interference effects in the air flow to predict wind loads and pedestrian comfort zones location. Thus it is reasonable to use methods based on computational experiments, which allow us to reduce time and cost to optimize engineering project and to dump excessive oscillations. Such methods may be based on the use of CAD/CAE software packages [15], [16].

In [17], the authors carried out the numerical simulation of turbulent flows of incompressible air in the vicinity of a plate-mounted prism of a square cross-section. The comparison with experimental data [18] on the velocity and turbulent kinetic energy (TKE) profiles was performed and satisfactory agreement was obtained. However, it becomes especially important to study the interference effects in flows around more complex configurations, for example for building complexes in cities.

In the paper, the numerical simulation of the turbulent air flows in the vicinity of the buildings complexes was performed. At the first stage, the atmospheric flow in the neighborhood of a building of a complex shape was carried out taking into account the interference with surrounding objects. The numerical simulation was performed under the conditions of the experimental data obtained in 3-AT-17.5/3 aerodynamic facility [19]. The 3D structure of the flow in the vicinity of the building was obtained and the comparison of the calculation results with the experimental data on the pressure coefficient distribution on the walls of the building was performed. At the second stage, the full-scale numerical modeling of the flow around the complex of high-rise buildings in the Frankfurt/Main, Germany, city center was made and pedestrian comfort of the area was investigated.

2 METHOD OF COMPUTATIONS

The simulation was performed under isothermal assumption on the basis of 3D URANS equations [20] supplemented with the *k-ω SST* turbulence model [21] and Kato-Launder [22] correction. To solve the initial-boundary problem, the finite-volume method and the method of splitting taking physical processes into account were used [23]. A monotonic solution was obtained using the MUSCL scheme of third-order accuracy for convective terms and the central-difference scheme of second-order accuracy for the viscous terms. For the temporal approximations, an implicit scheme of second-order accuracy was used. The ANSYS Fluent Software [15] supplemented with user-defined functions for settings the initial and boundary conditions was used.

3 AIR FLOW IN THE VICINITY OF BUILDING OF COMPLEX SHAPE

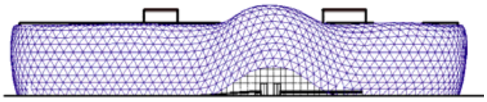
3.1 Problem statement, boundary conditions and computational grid

Let us consider a turbulent isothermal air flow in the neighborhood of a building of complex shape (Fig. 1, a, b). The numerical simulation was performed under the conditions of the experimental data obtained in 3-AT-17.5/3 aerodynamic facility [19]. The model is of 1:150 to the real scale. The main goal of the investigation was to get pressure coefficient distribution on the walls of the building of a complex shape (building *A*, fig. 1, c). Two cases were examined in the paper: when the building *A* was surrounded by another buildings (*conf. 1*) and when it was isolated (*conf. 2*). The incoming flow velocity is U_∞= 16.6 m/s and the boundary-layer thickness is δ = 0.92 m for these cases. The Reynolds number calculated by length scale δ and flow velocity U_∞ is $Re \approx 1.05 \times 10^6$.

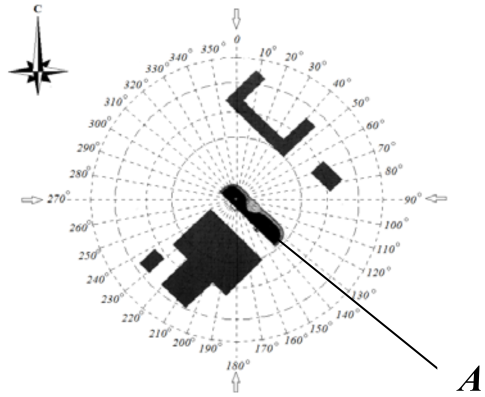
The computational domain is shown in Fig. 2, a. The characteristic diameter of the domain is about 3.5δ and the height of the domain is $\approx 2.5\delta$. The sizes of the calculation domain were chosen with regard for the scale δ so, that the artificial boundaries do not influence the calculated data [18].

In the computation, the north wind direction (α = 0º, Fig. 1, c) was modeled with the inflow velocity and TKE profiles taken from the experimental data [19]. The "no-slip" boundary condition was used for the substrate and the walls of the buildings. At the exit, the condition of constant static pressure was used, $\Delta P = P_{\text{wall}} - P_0 = 0$. At the upper boundary, we used the symmetry boundary condition, which provided the absence of flow across this boundary.

a)



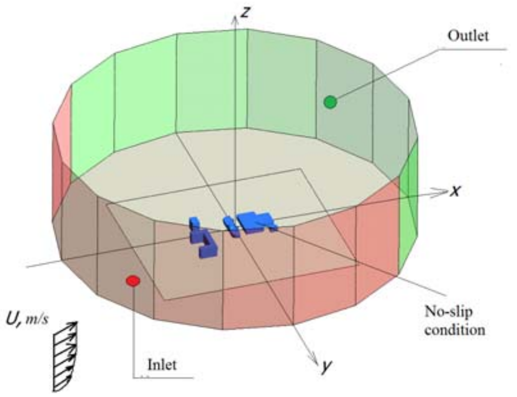
b)



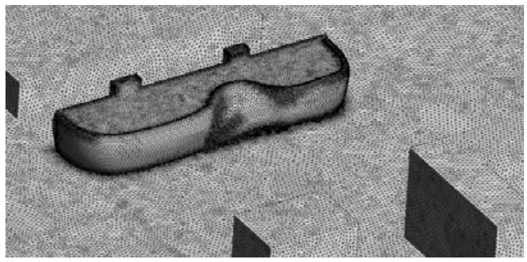
c)

Fig. 1: Building of complex shape (a), the geometrical model of the building (b, facade's view) and location of the building *A* relative to the surrounding area (c).

The computational finite-volume grid was constructed with the use of the ANSYS Meshing preprocessor. A fragment of unstructured computational grid on building surfaces is shown on Fig. 2, b. The computational finite-volume grid includes tetra elements in the outer region and the prismatic layers near the solid walls. A series of calculations using various meshes was performed for *conf. 1* to obtain the mesh-independent solution. The parameters of the grids are presented in Table 1. The most detailed numerical grid includes approx. 59 millions of cells. The non-dimensional distance to the wall at the first calculation node was y+= 1 for all cases.



a)



b)

Fig.2: The computational domain (a) and a fragment of the mesh on the building's walls (b).

No	Total number of cells, millions	Characteristic element size near building *A*, m	y+
1	≈ 9	≈ 0.013	
2	≈ 16,6	≈ 0.012	
3	≈ 26	≈ 0.01	≤ 1
4	≈ 46	≈ 0.008	
5	≈ 59	≈ 0.0066	

Table 1: Mesh characteristics.

3.2 Results and discussions

Let us consider the flow structure in the vicinity of the building *A* taking into account the surrounding buildings (*conf. 1*). In this case, an steady-state solution with residuals for all equations smaller than 10^{-3} was obtained. The flow structure is considerably influenced by the interference effects due to presence of the neighboring buildings. Under the conditions of north wind direction, the building *A* is located behind the building *B*. As we can see in Fig. 3, a, the flow separates from the edge *b1* of the building *B*. The large recirculation zone *V2* is formed behind it. That changes the angle at which the free stream comes on the front side of building *A*. Moreover, the free stream velocity is increased up to 20.5 m/s, when passing around the building *B* and zone *V2*. It leads to an increase in the pressure coefficient values on the facade *a1* of the building *A* (Fig. 3, c). We can also see a horseshoe vortex in front of the building *A*. The large separation zone is formed on the roof of the building *A*, and the negative values of pressure are observed in this zone. It is also important to note that the building *C* is located extremely close to the building *A* and it is the cause the velocity increase up to 19 m/s in the region between them.

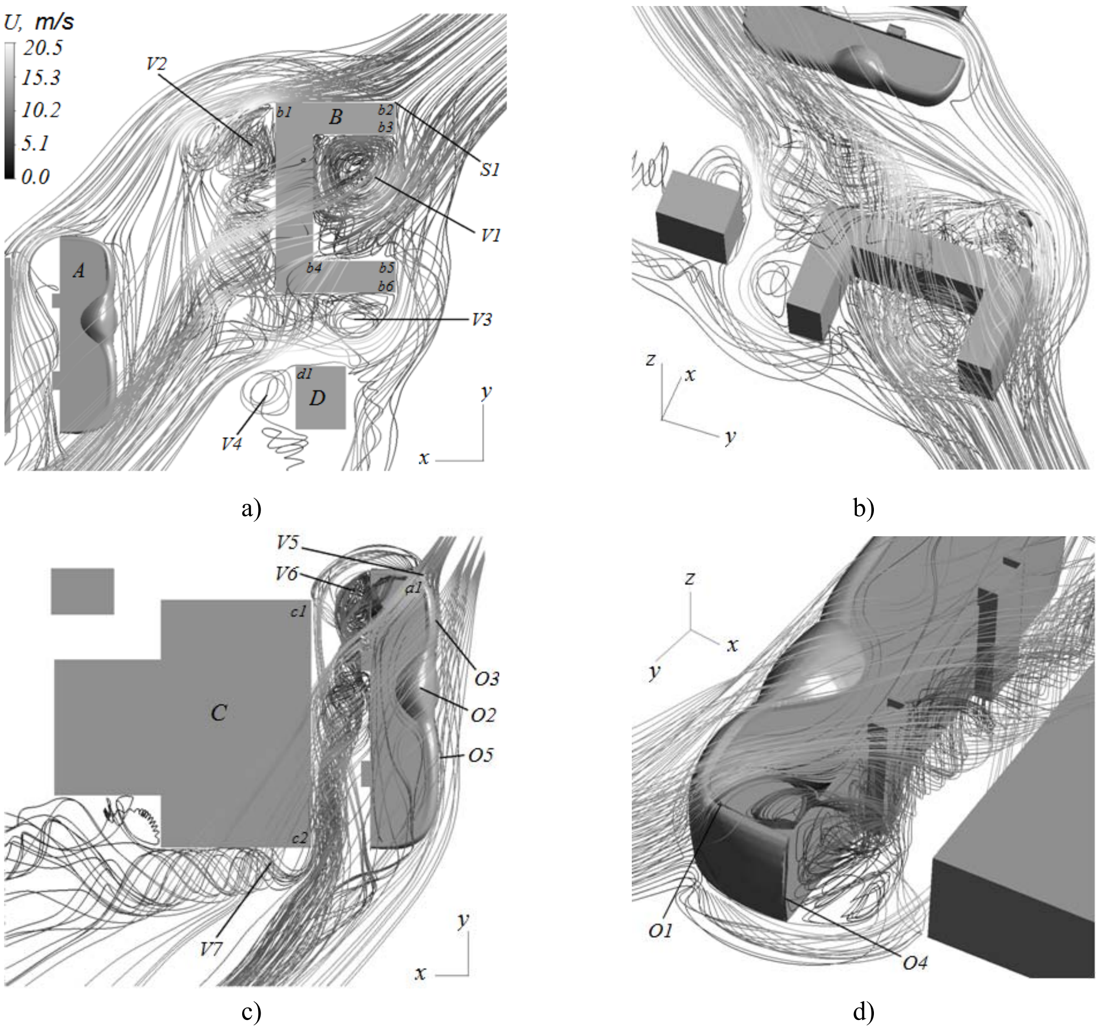
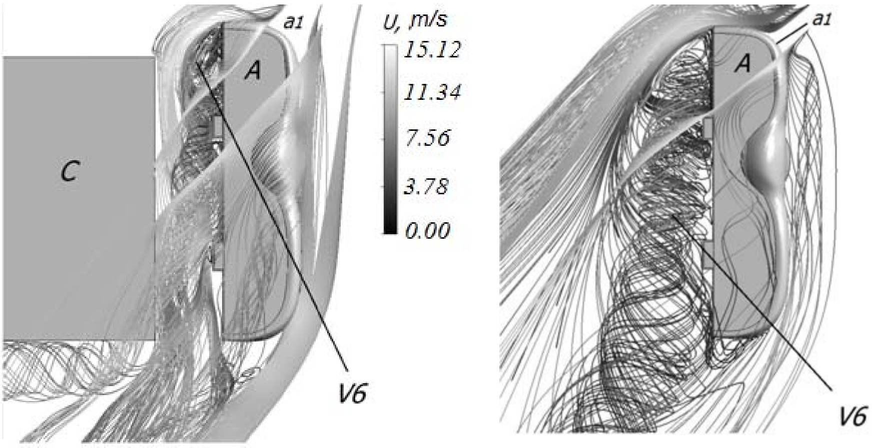


Fig. 3: Air flow structure in the vicinity of building *B* (a,b) and in the vicinity of building *A* (c,d).

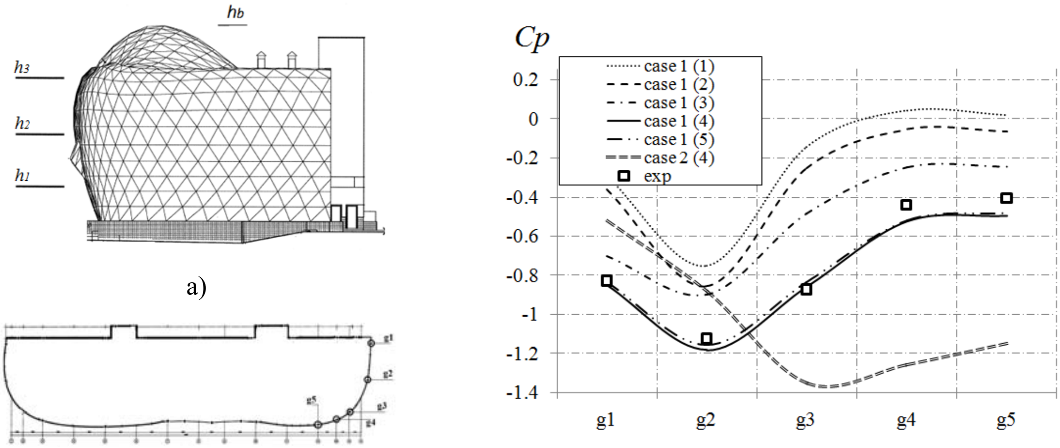
The flow structures in the vicinity of building *A* are shown on the Fig. 4 for both the configurations 1 and 2. In the second case, the inflow comes to the facade of the building *A* with a slower speed and at a smaller angle (Fig. 4, b) in comparison with the first case. The maximal values of positive pressure coefficient are less at this zone for *conf. 2* and reaches values of $C_{p,max,2}\approx 0.83$ ($\approx 0.6C_{p,max,1}$), where $C_{p,max,1}$ is the maximal values of positive pressure coefficient for *conf. 1*. The maximal absolute values of negative pressure coefficient for configuration 2 are about $C_{p,min,2}\approx$-1.68 ($\approx$ 0.73$C_{p,min,1}$).

For configuration 2 the absence of obstacles behind the building *A* also leads to the formation of an extensive recirculation zone *V6* on the leeward side. For configuration 1 zone *V6* is smaller "pressed" to the leeward side of the building *A* (Fig 4,a).



a) b)

Fig. 4: Flow structure in the vicinity of building *A* obtained from numerical simulation for *conf. 1* (a) and *conf. 2* (b).



b) c)

Fig. 5: The pressure gauges location in experiment [19] (a, b) and pressure coefficient values (c) for gauges g1 - g5 in section h_3 obtained from the experiment (exp), from the numerical simulation of *conf. 1* for various meshes (case 1 (1) - (5)) and from the numerical simulation of *conf. 2* for mesh (4) (case 2).

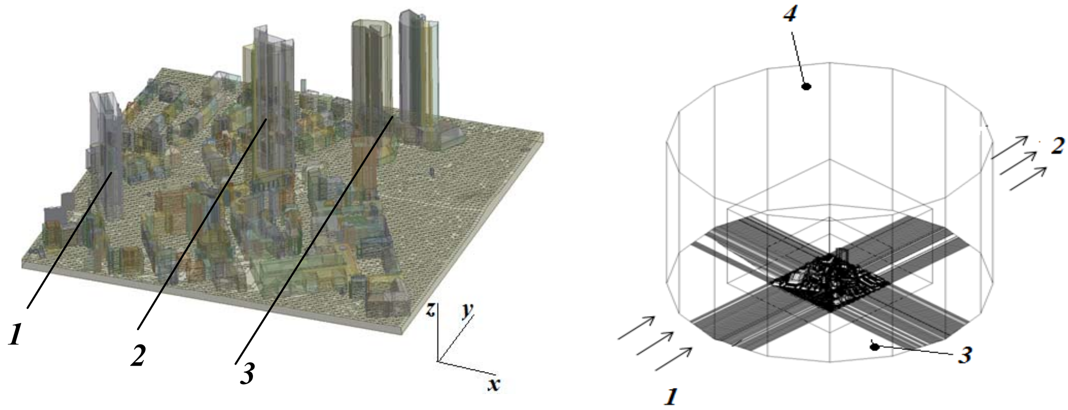
The obtained numerical results for *conf. 1* and *2* were compared with the experimental data [19] on the pressure coefficient C_p. Fig. 5 shows the values of the C_p, obtained in characteristic points (Fig. 5, b) at the section $h_3 = 0.760h_b$ (Fig. 5, a) in the experiment and in the numerical simulation for *conf. 2* on the grid No. 4 (see Table 1) and for *conf. 1* on the various grids. As we can see in Fig. 5, negative pressure coefficient values are observed for all the considered points in the section h_3, which connected with the fact, that the cross section h_3 is located in the separation zone. The good agreement with experimental data [19] was obtained for *conf. 1* on the grid No. 4 and No. 5. For isolated building (*conf. 2*), the numerical solution significantly differs from the experimental data. For *conf. 2* the separation zone on the facade is shifted due to the change in free stream flow angle. It testifies of the necessity to consider the interference effects and flow interaction with the surrounding buildings when calculating wind loads.

4 AIR FLOW IN THE VICINITY OF THE COMPLEX OF HIGH-RISE BUILDINGS

4.1 Problem statement, boundary conditions and computational grid

At the second stage, a turbulent isothermal air flow is considered in vicinity of a high-rise building complex located in Frankfurt/Main city centre (Fig. 6, a). The numerical simulation was performed using full-scale geometrical model[1]. The maximal height of the buildings is $h_{b,max}$≈186 м. The velocity in the flow core is U_∞= 25.4 m/s and the boundary-layer thickness is $\delta \approx 500$ m [24]. The freestream Reynolds number calculated by length scale $L = \delta$ and flow velocity U_∞ is $Re \approx 3.2 \times 10^8$. The computational domain shown in Fig. 6, b, has a diameter of $d \approx 10h_{bmax}$ and a height of $h \approx 6h_{bmax}$.

The inflow velocity and TKE were calculated using Monin-Obukhov theory considering the nature measurements [24] in some points on Commerz Bank walls for south-west wind direction (α = 225º). The total number of finite-volume cells is ≈33.6×10^6 for the case. The characteristic size of cells near the walls is about ≈3 m.



a) b)

Fig. 6: The geometrical model of buildings complex (a) and the computational domain (b): 1 – inflow conditions, 2 –outlet, 3 – "no-slip" wall condition, 4 – symmetry.

4.2 Results and discussions

The air flow near the buildings has a complex 3D vortex structure. In Fig. 7, the instantaneous velocity fields at the moment t=210.7 sec are presented in several horizontal cross-section. Under the conditions of south-west wind direction, the vortexes separate from the side edges of the first-located high-rise building (building 1, Fig. 6, a) and a von-Karman vortex structure is formed. It leads to velocity increasing up to ≈1.3÷1.5 $U_{h,\infty}$, where $U_{h,\infty}$ is a freestream velocity at the height h. The building 2 is located almost completely in separation zone formed behind the building 1. It causes the high negative values of pressure coefficients at the front faces of the building 2.

[1] Geobasisdaten:© Stadtvermessungsamt Frankfurt am Main, Stand 20.01.2015

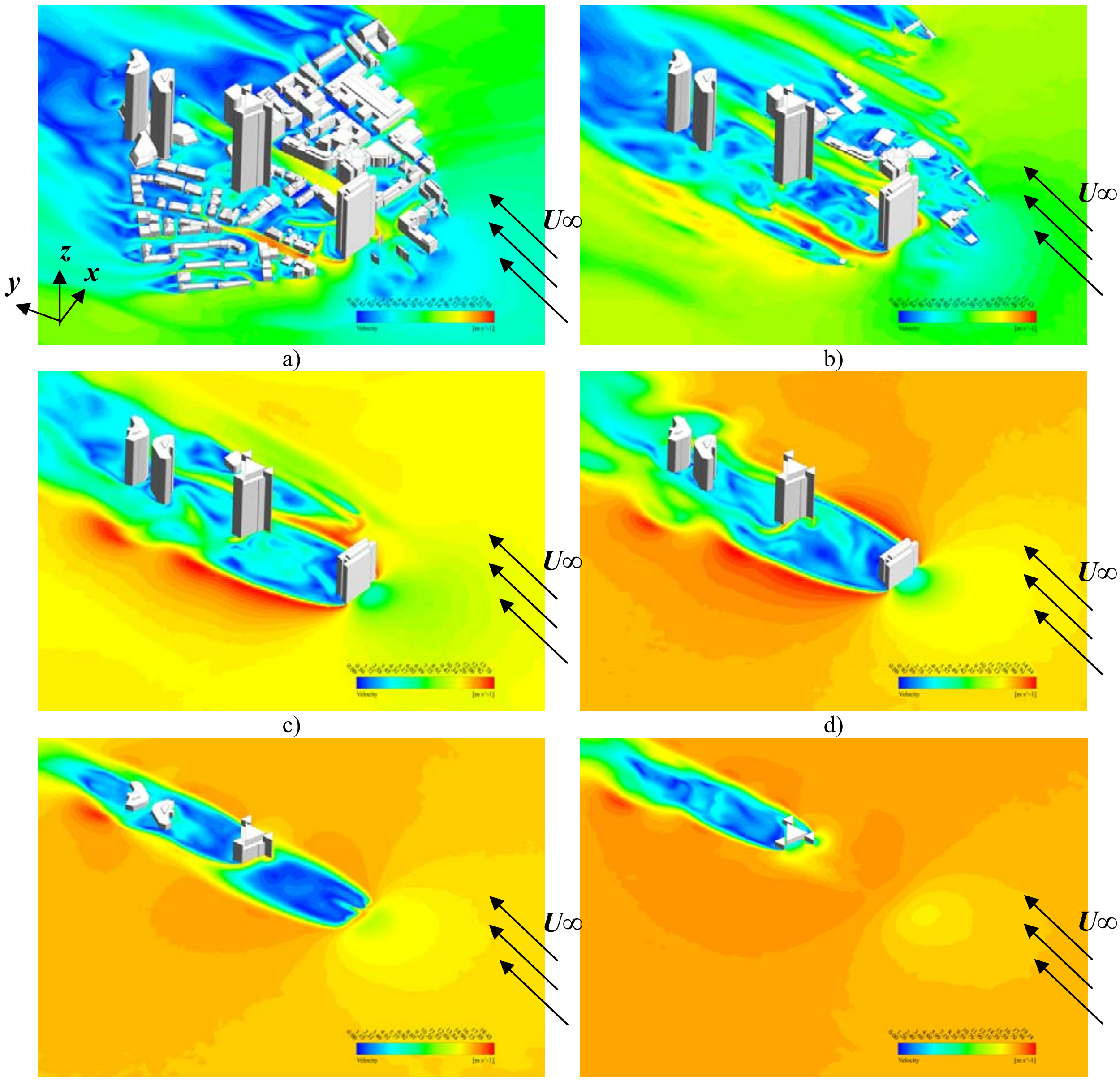
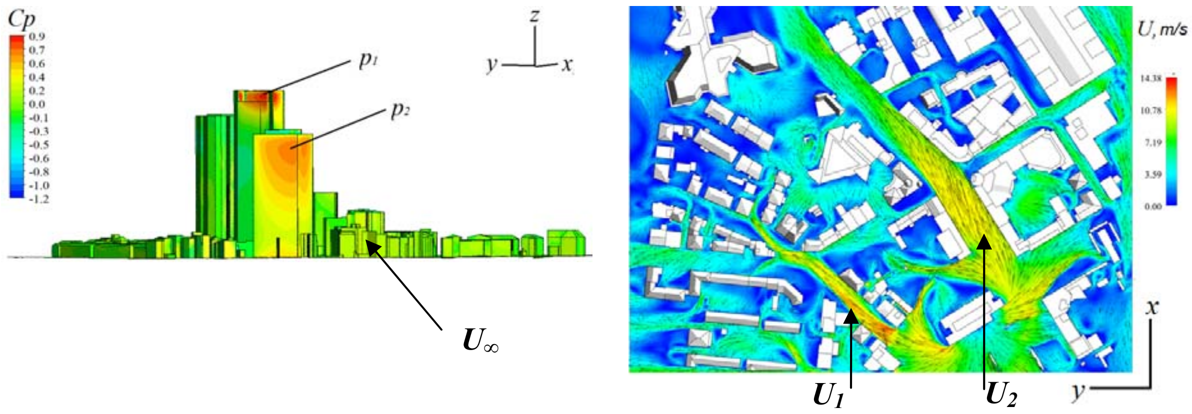


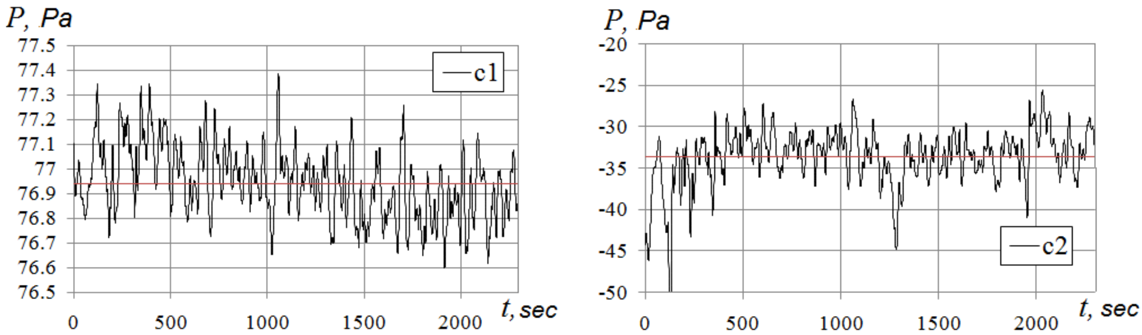
Fig. 7: The instantaneous velocity fields at the moment t=210.7 sec. for different characteristic horizontal sections: z= 8 m (a), 26 m (b), 56 m (c), 86 m (d), 146 m (e), 176 m (f).

In Fig. 8, a we can see an instantaneous distribution of the pressure coefficient C_p on the walls of the high-rise buildings. The analysis of the C_p distribution has shown that high values of wind pressure are observed in the *p2* zone where the flow comes to the front face of the building 1, and also in the *p1* zone where the flow comes to the upper part of the windward face of the building 2. To estimate the mean and the fluctuating components of the wind load on the building 1, we calculated the pressure time history in several characteristic points on its surface. The Fig. 9, a shows *P(t)* at the point *c1* (-107.8 m;-189.5 m; 60 m) located at the frontal face of building 1. The average pressure at this point is $P_{c1}\approx$76.9 Pa. Pressure fluctuations at this point are not significant ($P'_{c1}\approx$0.5 Pa). We observe a different situation at the frontal surface of building 2 (Fig. 9, b), namely, the fluctuating component of the wind load at the point *c2* (10.3 m; 46.5 m; 60 m) reaches more than 30 % from the average value ($P_{c2}\approx$-33.6 Pa; $P'_{c2}\approx$11.4 Pa).



a) b)

Fig. 8: The instantenous pressure coefficient distribution on the buildings walls at the moment t=210.7 sec. (a) and the field of velocity for horizontal section z= 2 m (b).



a) b)

Fig. 9: The static pressure - time dependence at the gauge point c1 (-107.8 m;-189.5 m; 60 m) located at the frontal surface of building 1 and the gauge point c2 (10.3 m; 46.5 m; 60 m) located at the frontal surface of building 2.

Low-rise buildings located in the neighborhood of high-rise buildings 1, 2 and 3 do not significantly influence the formation of large vortex flow structures. However, low-rise buildings form roughness of the ground surface and affect the air flow in pedestrian zones. The instantaneous velocity field at the horizontal section z=2 m is shown in Fig. 9. The maximal velocity amplitude at this section occurs at zones U_1 and U_2, that is caused by "restriction" of flow in these areas. The maximal flow velocity at the zone U_1 reaches values of about $4.8U_{\infty,h1}$, where $U_{\infty,h1} \approx 2.8$ m/s is a velocity of free stream at the height z=2м. The maximal flow velocity at the zone U_2 reaches values of about $3.5U_{\infty,h1}$. From the viewpoint of pedestrian comfort such values may be considered as unfavorable. A visualization of the flow structure near the pedestrian zones may allow us to optimize city environment taking into account street topology. It also allows us to design additional protective measures like park zones etc. to control wind flows.

5 CONCLUSIONS

- 3D computational analysis of a turbulent separated air flow in the vicinity of the building of complex shape has been performed with a high resolution of the viscous sublayer taking into account the surrounding buildings.
- The flow interference effects were described and their influence on the wind load distribution on the buildings wall was evaluated. It has been demonstrated that local separated zones on the building faces may occur and pressure coefficient may be sign-alternating with high amplitudes that must be taken into account in the design of facades.

- The comparison of the numerical results and the experimental data [19] on the pressure coefficient distribution on the buildings walls was carried out for the two configurations. Satisfactory qualitative and quantitative agreement with the experimental data was obtained for the non-isolated building location (*conf. 1*). We have observed that flow interference effects may lead to a significant increase in the wind load (up to 50%) on a building. The results obtained for the *conf. 2*, when the building was isolated, significantly underpredict the values of the wind pressure coefficient.
- We performed the 3D computational analysis of the flow around the complex of high-rising buildings in the Frankfurt/Main city center, Germany. The flow structure and pressure distribution on buildings walls were obtained. The main features of a wind flow in the vicinity of buildings were investigated and pedestrian comfortable/uncomfortable zones were detected.

6 ACKNOWLEDGMENTS

The work was supported by the Ministry of Education of the Russian Federation (Project No. 211, task No. 2014/140 for executing scientific activities within the basic part of government order); German Academic Exchange Service (DAAD) 2014/2015.

The authors would like to thank:

- Prof. Valery M. Mitasov, Novosibirsk State University of Architecture and Civil Engineering (Sibstrin), for providing the wind tunnel experimental data [19] and valuable advices on wind loads modelling;
- CADFEM International GmbH and especially Dr.-Eng. Günter Müller and Dr.-Eng. Stefan Trometer for providing geometry model of Frankfurt/Main city area and the valuable advices on using of CAE ANSYS for cityscape modelling;
- Our colleagues from IAG, Stuttgart University, Prof. Dr.-Eng. Ewald Krämer, Dr.-Eng. Uwe Gaisbauer and Dr.-Eng. Thorsten Lutz for their support of Svetlana Valger during her internship at IAG.

REFERENCES

[1] L.T. Matveev, Fundamentals of General Meteorology: Physics of Atmosphere, Gidrometeoizdat, Leningrad, 1985 (in Russian).

[2] A. Gill, Atmosphere-Ocean Dynamics, Vol. 1, Mir, Moscow, 1986 (in Russian).

[3] A. Gill, Atmosphere-Ocean Dynamics, Vol. 2, Mir, Moscow, 1986 (in Russian).

[4] E. Simiu and R. Scanlan, Wind Effects on Structures, edited by B.E. Maslov, Stroiizdat, Moscow, 1984 (in Russian).

[5] A.N. Birbraer and A.Yu. Roleder, Extreme Impacts on Buildings, St.-Petersburg Polytechnic University, St. Peters-burg, 2009 (in Russian).

[6] G. Fershing, Fundamentals of Aeroelasticity, edited by G.M. Fomin, Mashinostroenie, Moscow, 1984 (in Russian).

[7] Computational Techniques for Fluid Dynamics, Springer-Verlag, Berlin, 1988.

[8] P.N. Solyanik, Experimental Aerodynamics, Kharkov, National Aero-Space University “Kharkov Aviation Institute”, 2007.

[9] N.A. Zaks, Basics of Experimental Aerodynamics, Oborongiz, Moscow, 1953 (in Russian).

[10] S.M. Gorlin, Experimental Aerodynamics, Vysshaya Shkola, Moscow, 1970 (in Russian).

[11] S.V. Nikolaev, V.M. Ostretsov, et al. Aerodynamic tests of high-rise building models, in: Unique and Special Technologies in Construction Activities, Dom na Brestskoi, Moscow, 2005, P. 82-84 (in Russian).

[12] Construction norms and regulations, CN&R 2.01.07-85*, Loads and Impacts, FGUP CPP, Moscow, 2010 (in Russian).

[13] ENV 1991: Eurocode 1: Basis design and actions on structures. Parts 2-4: Wind action, CEN, 1994.

[14] Eurocode 1. Impacts on Building Structures, Parts 1–4, General impacts, Wind impacts, Standard-Practice Engineering Code, Minstroi Arkhitektura, Minsk, 2010.

[15] ANSYS Fluent [Electronic resource] // ANSYS Software [Official web-site],

http:// www.ansys.com

[16] SOFISTIK CFD [Electronic resource] // SOFISTIK [Official web-site], http://www.sofistik.com/

[17] S.A. Valger, N.N. Fedorova, A.V. Fedorov, Structure of turbulent separated flow in the neighborhood of a plate-mounted prism of square section. Thermophysics and Aeromechanics, 2015, T. 22. Vol. 1, P. 29-41 (in Russian).

[18] A. Mochida, Y. Tominaga, S. Murakami, R. Yoshie, T. Ishihara, and R. Ooka, Comparison of various k-ε models and DSM applied to flow around a high-rise building report on AIJ cooperative project for CFD prediction of wind environment, Wind and Structures, 2002, Vol. 5, No. 2-4, P. 227-244.

[19] The science-technical report on experimental measurements of pressure on the buildings walls of model "Biotechnopark", Kol'tcovo city, 2013 (in Russian).

[20] J. Blazek, Computational Fluid Dynamics: Principles and Applications, Elsevier, Amsterdam, 2005.

[21] Menter F.R. Two-Equation Eddy-Viscosity Turbulence Models for Engineering Application // AIAA Jour. 32, No. 8, 1994. P. 1598-1605

[22] Kato, M., Launder B.E. The modelling of turbulent flow around stationary and vibrating square cylinders // Ninth Symposium on "Turbulent Shear Flows". Kyoto, Japan. August 16-18, 1993. –P.10–14.

[23] A.J. Chorin, Numerical solution of Navier—Stokes equations, Math. Comp., 1968, P. 745–762.

[24] Hessisches Ministerium des Innern. Ergänzungserlaß zu DIN 1055, Teil 4, Ausgabe August 1986, Betr. Windlasten bei hohen Hochhäusern im Raum Frankfurt am Main, 1987. – 491 p.

[25] Berneiser A., König G. Wind Loads in City Centres Demonstrated at the New Commerzbank Building in Frankfurt/Main // LACER, 1997. – No. 2. – P. 231–244.