CFD Methods in Architecture and City Planning

S A Valger¹ and N N Fedorova²

¹Novosibirsk State University of Architecture and Civil Engineering, Leningradskaya 113, Novosibirsk, 630008, Russia
²Khristianovich Institute of Theoretical and Applied Mechanics SB RAS, Institutskaya 4/1, Novosibirsk, 630090, Russia

Abstract. The paper provides an overview of CFD solutions for architectural design and urban planning. Aspects of numerical modelling of the aerodynamics of buildings and structures are described. Examples of the influence of turbulence models on flow parameters in the vicinity of buildings, as well as examples of influence of thermal stratification regimes on flow structure around buildings and on concentration of gaseous emissions from low sources in the vicinity of these buildings are considered. The calculation results of aeration of a city micro-district and a scenario for the arrangement of green areas to change the wind regime of the territory with the already existing development are demonstrated in the paper.

1. Introduction

Nowadays it is difficult to imagine the process of architectural planning without using software products and computational technologies. Computer-Aided Design (CAD), Computer-Aided Engineering (CAE), Building Information Modeling (BIM) and related technologies become ingrained in the range of tools of designers, architects and engineers. At the same time, the process of architectural planning and urban planning increasingly involves important application tasks at the intersection of several fields of knowledge and science. For example, design of complex building objects of high responsibility class raises problems at the intersection of applied mechanics of deformable solids and aerodynamics and hydrodynamics: assessment of the wind load pulsation component on high-rise buildings, problems of aero-elastic vibrations of bridges and building structures, interaction of shock waves with obstacles, etc. Urban planning and architectural research are also cross-disciplinary in nature. Creating a new architectural concept or a fragment of urban infrastructure now goes beyond architecture. New challenges and solutions are emerging. They are based on the pooling of knowledge of different disciplines and fields: architecture, ecology, hydro- and gas dynamics, geophysics and CFD technologies, meteorology, computational methods and systems, programming and related fields.

Today development of computer systems allows to talk about huge potential possibilities of computation models used in multi-disciplinary problems in the field of construction. Thus, the key to designing a comfortable urban environment could be a combination of knowledge in architectural urban planning and approaches to computational fluid dynamics. Full-scale and aerodynamic experiments can be complemented by virtual experiments. Virtual experiments allow to get more detailed data on characteristics of air flows, as well as to simulate multi-scale phenomena with a significant level of detail.

The problems of urban planning and architecture which can be solved with the help of modern methods of computational fluid dynamics and computational experiments can include the following:
• Modeling and evaluation of aeration of urban areas;
• Prediction of air quality and concentrations of gaseous emissions from various anthropogenic sources in the city;
• Dust assessment;
• Assessment of local climate parameters and thermal characteristics in dense urban areas;
• Control and management of heat island effects;
• Optimization of architectural solutions and arrangement of small architectural forms in terms of pedestrian comfort, etc.

In the present paper the authors discuss some aspects of computer modeling of the above-mentioned problems and give examples of their implementation for designing a comfortable urban environment on the basis of virtual computing experiments.

2. Numerical modeling of wind loads on building and structures

In the past 20 years, numerical modeling of wind loads has become an integral part of the design of complex construction projects - high-rise building complexes, buildings of unusual architectural shape, stadiums, etc. The possibility of conducting a virtual experiment makes it possible to significantly supplement the data of a full-scale experiment conducted in a wind tunnel. So, mathematical modeling of a flowing of construction objects by a wind air flow allows to receive enough detailed information on the distribution of velocity, pressure and about turbulent parameters in the flow. Moreover, using mathematical modeling, one can study the problem of flow of bluff-bodies taking into account the scale factor, which is difficult to take into account in an experiment in a wind tunnel. Indeed, the tasks of assessing aerodynamic loads on buildings and structures are quite specific from the point of view of conducting a wind tunnel experiment. This is due to the necessity of taking into account such factors as the presence of a sufficiently thick boundary layer, which thickness significantly exceeds the height of buildings and structures, and also the large sizes of buildings and structures which can reach several hundred meters.

The computational experiment allows to solve a problem of wind loading calculation on construction object taking into account real object scale, boundary layer thickness and high Reynolds numbers of an approaching flow, and also taking into account a variable wind direction and a number of additional factors. However, using CFD for estimating the parameters of wind load on buildings requires the development of correct mathematical models and methods, validated on the set of experimental data, and implemented in the form of a verified program code. A number of high-tech problems arise while creating a mathematical model describing the air flow in the vicinity of large-scale poorly streamlined bodies and their systems: description of boundary condition profiles [1-2], correct from the point of view of similarity to the atmospheric currents and sizes of the calculated area [3]; description of the velocity pulsation component in the flow through the development and adaptation of turbulence models [4-9]; mathematical description of the parameters of the viscous sublayer in order to correctly describe the separation points and the location of the separation zones [10-11]. The solution of the created mathematical model should also account for the features and limitations of the existing numerical methods, the form and size of cells of finite-volume grids, the scheme of the temporal and spatial approximation of the general equations should be correctly chosen. In works [1 – 13] various aspects of numerical modeling of wind loads on construction objects on the basis of original program codes and modern open and commercial program complexes are resulted.

In [8], the authors tested various modifications of turbulence models on the problem of the flow around a square prism under the conditions of the experimental data [14]. The Fig. 1 (a)-(e) shows the problem statement and the results of computations obtained using various modifications of the RANS models $k$-$\varepsilon$ and $k$-$\omega$, implemented in ANSYS Fluent: 1 $– k$-$\varepsilon$; 2 $– k$-$\omega$; 2.1 $– k$-$\omega$ with Production Limiter $C_{lim}=3$; 2.2 $– k$-$\omega$ model (Kato-Lauder); 3.1 $– k$-$\omega$ SST with Production Limiter $C_{lim}=10$; 3.2 $– k$-$\omega$ SST (Kato-Lauder). As can be seen from the Fig. 1 (c) and Fig. 1 (e), the choice of the turbulence model and model constants essentially influences the size and form of the main break-away zones and the distribution of the kinetic energy of turbulence in the typical sections around the prism.
Comparison of the results of calculations with the experimental data [14] shows that the RANS model adequately predicts the mean velocity profiles in the typical areas around the prism, but the simulation of TKE profiles to assess the load pulsation characteristics is possible only when calibrating the model on the basis of experimental data.

3. CFD methods for the assessment of city aeration and pedestrian comfort analysis

CFD methods in architectural design and urban planning have become more recent in use. However, the existing base of numerical methods and techniques of computer modeling for the assessment of aerodynamics of poorly streamlined bodies opens up great prospects for architects and engineers in the design of "smart" and comfortable urban spaces. In [15-18] reviews and specific examples of CFD modeling techniques are given to assess the wind conditions in pedestrian zones, taking into account the complex interference effects arising from the flow of air through the complexes of buildings. In [19] the authors performed 3D numerical modeling of aerodynamics of the complex-shaped building micro-districts. It is shown that the distribution of wind pressure on the facades of buildings and the distribution of velocities in pedestrian zones between buildings significantly depend on the interference effects arising in the flow.

Figure 1. Computational domain scheme (a, b); shapes of the separation zones (c) on the top wall of the prism (left) and on the substrate surface (right), calculated using different turbulence models; profiles of $U_\text{m}$, mean velocity (d) and TKE (e) in section X1, obtained from the experimental data [14] and from calculations [8] with various models of turbulence.
Fig. 2, 3 show an example of the calculation of aeration of the city micro-district located in Novosibirsk, Russia. Numerical modeling included the following steps: preparation of 3D models with a low level of detail (Fig. 2, a) on the basis of the original architectural solution from the Autodesk Revit Software (Fig. 2, b); design grid taking into account the peculiarities of the flow (a more detailed grid near buildings was used, elongated prismatic grid layers near the substrate walls and the building walls were constructed); creation of mathematical model on the basis of Reynolds averaged Navier-Stokes equations supplemented with k-ω SST model of turbulence; specification of boundary conditions profiles for velocity and turbulent parameters describing the behavior of the boundary layer at the entrance to the computational domain for different wind directions; selection of stable temporal and spatial approximation schemes.

As you can see from the Fig. 3 (a, b), numerical simulations provide a sufficiently detailed picture of the air flow, taking into account the flowing of obstacle systems, which can be used to design a more wind-comfortable environment.

It is necessary to notice that use of numerical methods at architectural designing allows to solve problems of the best possible layout of construction objects and their optimum spatial form. In addition, the modeling of different scenarios for the arrangement of small architectural forms and green areas allows the architect to offer new solutions for controlling and changing the wind regime of the territory with the already existing development. Fig. 3 (c, d) shows an example of the numerical calculation of the scenario for the optimization of the wind regime in the local area in front of the high-rise building due to the location of green areas.
4. Numerical modeling of heat and mass transfer in cities

The comfort of pedestrian zones and micro-districts in cities is provided not only by the wind regime, but also by microclimatic parameters, such as temperature, humidity and air composition. The composition and quality of air in cities are significantly affected by low and high sources of emissions from industry and vehicles, as well as by the climatic conditions of the region and by the topology and architecture of the building. The research papers [20-27] are devoted to studying the aspects of computer modeling of thermal regimes of the lower atmosphere and describing the effect of thermal stratification on the intensity of mixing and transfer of emissions in the air. A detailed analysis of CFD methods for describing the air flow parameters in urban canyons is presented in [24].

**Figure 3.** Mean velocity fields in the vicinity of the city area at a height of 2 m above ground level (a) and at a height of 10 m above ground level (b) in the south-west wind direction; layout of green areas in the vicinity of house B1 (c) and mean velocity field at a height of 2 m above ground level, taking into account the calculation of green areas (d).
In problems of numerical modeling of wind loads on buildings and structures, as a rule, the thermal regime of the boundary layer is not accounted for, and the assumption on the neutral stratification of the air flow is accepted. However, for the tasks of predicting emission transfer in the city air, it is necessary to consider more complex scenarios when the boundary layer is in a stable or unstable thermal state. Changes in the thermal regime of the air also affect the formation of ascending streams near high-rise buildings.

One of the important issues in the architectural design is ensuring high air quality in the presence of transport highways near the micro-districts. The study of this issue is complicated by the small scale and local nature of emissions and micro-particles transfer near roads. The verification of numerical methods for assessing air quality in cities and the development of scenarios for filtering and reducing concentrations of emissions and dust are carried out using data from measurements and experiments in wind tunnels. For example, a series of experiments on estimating the parameters of emission filtration through green spaces near a building are presented in [28].

The Fig. 4 (a)-(d) shows an example of the numerical simulation of the various thermal stratification regimes of the boundary layer in the vicinity of the prism [27]. The problem statement was chosen in accordance with the experimental data [29], where different thermal regimes of the air flow were reproduced by the boundary conditions of the inflow air temperature profile, wall and substrate temperatures.

Figure 4. Scheme of the computational domain (a, b); mean temperature $T$ at the symmetry $XZ$ plane for stable (c) and unstable (d) state.

In [29], injection of ethylene through a hole in the substrate behind the prism is researched. Numerical modeling of air flow taking into account ethylene injection [27] showed that the level of ethylene concentration in the recirculation zone behind the bluff-body significantly changes depending on the thermal stratification regime of the boundary layer.
5. Conclusions
In the paper the authors provide a short overview of CFD solutions for architectural design and urban planning. Examples of the influence of turbulence models on flow parameters in the vicinity of buildings, as well as examples of influence of thermal stratification regimes on flow structure around buildings and on concentration of gaseous emissions from low sources in the vicinity of these buildings are considered. The authors consider the calculation of aeration of a city micro-district located in Novosibirsk, Russia and describe a scenario for the arrangement of green areas to change the wind regime of the territory with the already existing development.

Acknowledgments
The work was supported by Russian Foundation for Basic Research, grant 18-08-00755 A “Numerical simulation of heat and mass transfer processes in multicomponent flows, describing mechanisms for the emission dissemination from vehicles in the vicinity of urban roads”.

The authors are grateful to P. Dubovskoy and S. Sarapulova for their contribution into the calculation of aeration of the city micro-district described in the Section 3 of the article.

References
[1] Richards P and Hoxey R 1993 J. Wind Eng. Ind. Aerod. 46 145-153
[2] Yang Yi, Gu M, Chen S and Jin Xi 2009 J. Wind Eng. Ind. Aerod. 97 88-95
[3] Revuz J, Hargreaves D and Owen J. 2010 Proc. 9th UK Conf. Wind Eng. pp 273-276
[4] Mochida A, Tominaga Y, Murakami S, Yoshie R, Ishihara T and Ooka R 2002 Wind and Structures 5 227-244
[5] Chenga Y, Liena F S, Yeeb E and Sinclair R 2003 J. Wind Eng. Ind. Aerod. 91 1301-1328
[6] Murakami S, Mochida A, Hayashi Y 1990 J. Wind Eng. Ind. Aerod. 35 87-100
[7] Isaev S A and Lysenko D A 2009 J. Eng. Phys. Thermoph. 82 492–499
[8] Valger S A, Fedorova N N, Fedorov A V 2015 Thermoph. Aeromech. 22 29-41
[9] Dubinsky S I Numerical Modeling of wind effects on tall buildings and complexes (Moscow: Moscow State University of Civil Engineering Press) p 231
[10] Abdi D S and Bitsuamlak G T 2016 Wind and Structures 22 503–524
[11] Blocken B, Stathopoulos T Carmeliet J 2007 Atmospher. Envir. 41 238-252
[12] Isaev S A, Baranov P A, Zhukova Yu V., Tereshkin A A and Usachov A E 2014 J. Eng. Phys. Thermoph. 87 112–123
[13] Blocken B, Stathopoulos T Carmeliet J and Hensen J 2016 J. Build. Perform. Simul. 4 157-184
[14] Ishihara T, Hibi K 1998 J. of Wind Eng. 76 55–64
[15] Blocken B 2015 Building and Environment 91 219-245
[16] Stathopoulos T 2006 J. Wind Eng. Ind. Aerod. 94 (11) 769-780
[17] Mochida A and Lun L.Y.F. 2008 J. Wind Eng. Ind. Aerod. 96 1498-1527
[18] Yoshiie R, Mochida A, Tominaga Y, Kataoka H, Harimoto K, Nozu T et al. 2007 J. Wind Eng. Ind. Aerod. 95 1551-1578
[19] Valger S A, Fedorova N N and Fedorov A V 2017 Thermoph. Aeromech. 24 35-44
[20] Tan J, Zheng Y, Tang X, Guo C, Li L, Song G et al. 2010 Int. J. Biometeorology 54 75-84
[21] Memon RA, Leung D Y C, Liu C-H and Leung M K H 2010 Theor. Appl. Climat. 103 441-450
[22] Toparlar Y, Blocken B, Vos P, van Heijst G J F, Janssen W D, van Hooff T et al. 2015 Building and Environment 83 79-90
[23] Murakami S, Ooka R, Mochida A, Yoshida S and Kim S 1999 J. Wind Eng. Ind. Aerod. 81 57-81
[24] Starchenko A V, Nuterman R B and Danilkin E A 2015 Numerical Study of Turbulent Flows and Pollution Transport in Street Canyons (Tomsk: Tomsk University Press) 252 pp
[25] Fernando H J S, Zajic D, Di Sabatino S, Dimitrova R, Hedquist B and Dallman A 2010 Physics of Fluids 22(5) 051301
[26] Yee E and Bitloft C A 2004 Boundary-Layer Meteorol. 111 363–415
[27] Valger S and Fedorova N 2018 AIP Conference Proceedings 2027 040043
[28] https://www.windforschung.de/CODASC.htm
[29] http://www.wind.arch.t-kougei.ac.jp/info_center/pollution/Non-Isothermal_Flow.html