Numerical simulation of wind flow around building complex with different software approaches

V D Olenkov¹, I V Lazareva², A D Biryukov³

¹Department "Urban planning, engineering networks and systems", South Ural State University, 76, Lenin Avenue, Chelyabinsk 454080, Russia
²Department "Territorial bases of urban planning", Central Institute for Research and Design of the Ministry of Construction and Housing and Communal Services of the Russian Federation, 29, Vernadsky Ave., Moscow 119331, Russia
³Department "Building production and theory of construction", South Ural State University, 76, Lenin Avenue, Chelyabinsk 454080, Russia

E-mail: olenkovvd@susu.ru

Abstract. Numerical simulation of wind flow around 4-building complex in hot climate conditions is described in order to assess a wind comfort in the courtyard area. The solution of the air flow around the multiply building complex geometry including the using of various numeral simulation software packages is presented in this article. The article also provides the subsequent comparison of the results obtained from specialized and simplified software packages (ANSYS FLUENT, Autodesk Flow design). Use of the numeral simulation in conditions of technogenic relief and complicated construction is relevant at the present time. This method allows significantly reducing the costs of undoing mistakes and to speed up the problem solving. An explicit demonstration of the results is also a very important criterion for choosing the methods and numeral simulation software. Use of modern methods of CFD-data visualization in real time can greatly simplify the interpretation of simulation results for non-professional users.

1. Introduction

Planning and design issue of large cities, in connection with environmental and climatic factors is relevant for the urban planning theory and practice. This problem is related to increasing urbanization. The air pollution just as the wind and heating comfort is an acute problem for the modern city inhabitants living in conditions of high-density. The Urban Science advanced applications also concern the regions with a hot climate. An average annual temperature in these regions is about 20 ºC. This climate is rather common, and it is more related to southern and near the equator countries [1].

Degradation of microclimate in modern cities is explained by the fact that the area climate specificities were not carefully taken into account during design phase. One of the most effective and sustainable methods of microclimate improvement and regulation in hot countries is mainstreaming of aeration at the beginning of buildings and architectural complexes design. It is necessary to forecast and calculate the impacts of taken design decisions that were based on the natural and technological landscape analysis [2].

Aeration is the process of natural and regulated ventilation of the cities territories and other human settlements [3]. There are various methodologies approaches to study and forecast the aeration
schemes of residential development. These methodologies were highlighted in several works that demonstrated the theoretical and experimental researches of the aeration processes of various territories [4,5].

The most progressive approaches to the research of aeration regime and microclimate forecasting for the architectural complexes are based on methods of numerical simulation and computer analysis of air masses movement in the area studied [6,7].

Its main objective is to resolve the tasks of CFD simulation (computational fluid dynamics) with the aid of software complexes [8]. At present, there are many software solutions which are intended for computer simulation of wind influence on buildings and structures by the CFD methods. Some complexes focus on narrowly defined tasks, while others imply a more general approach.

In addition, computer modeling of the experiments with the use of different software packages is an interesting and relevant issue. This process also involves comparison of results and evaluations of their matches taking into account the calculation time, and optimal choice.

This article describes the solution of a particular problem of simulation the air flow around the architectural complex of buildings in the city of Erbil (Iraqi Kurdistan) in order to review its optimal location as well as to assess the quality of temperature comfort and parameters of courtyard microclimate.

2. Selection of software, and description of the initial conditions and experimentation methodology.

2.1. The software review and selection variants for experimentation.

In this paper, the simulation will be implemented through two distinct software complexes. There are relatively large numbers of CFD packages exist currently on the market. These software packages have different specializations and characteristics. The major solutions of CFD simulation were reviewed in Broekhuizen’s work in 2016 [9]. It is possible to select the two main software packages through their capabilities for the urban planning application as the professional and simplified variants: *Ansys fluent* as a reference method of simulation due to highest prevalence and well described calculation methods; *Autodesk Flow Design* as a software which is oriented on less advanced researchers. This software also implies quicker experimentation cycles and simple configuration with acceptable. These complexes were selected for simulation and qualitative comparison of results.

2.2. The baseline data review for numerical experiment.

Experimentation methodology can be described in terms of the following stages: 1) Obtaining of baseline data on building and environmental parameters; 2) Modelling of building geometry in two versions; 3) Adjust settings for each software package; 4) Obtaining and visualization of results; 5) Qualitative image comparison and numerical values analysis; 6) Definition of desired outcomes and verification of conformity.

The blueprints of the building located in Erbil (Kurdistan) were used as the baseline data. This construction is an architectural complex comprising a group of four combined buildings. It is necessary to figure out a forecast microclimate in the residential area between the buildings considering a person's height, and using the method of experimental modeling [10-12]. Evaluation the level of wind comfort in this zone is needed as well. Building orientation was defined according to plan. It helped to construct the original model in the right way. And after that the data were superimposed on the wind rose.

The geometrical dimensions of buildings with a dominant wind direction mark for this point were taken from the plan, and are shown in figure 1-a. On the basis of given baseline data, the starting point for experiment will be the following values of the experiment parameters: geometrical dimensions in the plan – 117 meters x 95 meters; predominant wind direction is Northern; average wind speeds - 6
meters per second. Further, the generalized structural model was made in the 3D graphic software. This model also can be used as a reference model (in the original scale for all experiments; figure 1-b).

![Figure 1](image)

Figure 1. a) The graphical footprint of building is displaying the predominant wind direction. b) 3-D model of building

2.3. The course of experiment & baseline setup

2.3.1 For ANSYS Fluent case.

Let’s consider the first reference case with the use of ANSYS Fluent software package. Calculation for this software was carried out according to the following algorithm:

1. The preparation of the calculation model:
   - Optimization of the polygon model with the aim of creating a solid-state parametric model.
   - Importing of the polygon model into the Space Claim (solid-state modelling editor).
   - Building a geometric area of the air flow around the building for creating computational domain.
   - Generation of mesh model based on the constructed geometry in ANSYS MESHING module.
   - Setting up the boundary and initial conditions, physical model of calculation – processing.

2. Solution of problem in the calculation module.

3. Viewing and evaluation of results - post-processing.

Construction of calculating area is based on the rules described in the article written by Poddaeva[13]. The purpose is to reduce the impact of the region boundaries on calculation. Center of coordinates remain in the center of the researched block of buildings. The block of buildings was rotated 45 degrees in order to be compatible with the researched wind rose of windy object.

\[
\begin{align*}
A & \geq 5H_{\text{max}} \text{ distance from the center of the object to the edge of the calculating area.} \\
B & \geq 5H_{\text{max}} \text{ distance from the center of the object to the Inlet boundary.} \\
C & \geq 15H_{\text{max}} \text{ the distance from the center of the object to the Outlet boundary.} \\
H_{\text{max}} & \text{ The largest dimensional object size.}
\end{align*}
\]

Thus, calculating area is constructed with the following sizes: width - 1100 meters; length – 2100 meters; height – 600 meters (figure 2).

Quality of the finite element mesh construction is crucial in accurate calculation results achievement. This quality is important during the process of finite element modeling.

Mesh density was regulating by indication of the maximum value for finite element edge, while the experiment was carried out. (Max Face Size Parameter). Set point for parameter was 10 meters. The mesh density was increased (Face Sizing Function) because of the need for accurate calculation near the buildings. The maximum edge size was 2 meters (*near the buildings). As a result the finite element mesh consisting of 4925989 components was obtained.
Quality of the finite element mesh construction is crucial in accurate calculation results achievement. This quality is important during the process of finite element modeling.

Mesh density was regulating by indication of the maximum value for finite element edge, while the experiment was carried out. (Max Face Size Parameter). Set point for parameter was 10 meters. The mesh density was increased (Face Sizing Function) because of the need for accurate calculation near the buildings. The maximum edge size was 2 meters (*near the buildings). As a result the finite element mesh consisting of 4925989 components was obtained.

Experience of research in the wind tube demonstrates that it is necessary to follow the geometric, cinematic and dynamic similarity during the experiment [14-16]. The experiment reproduces completely and geometrically the real building sizes. Cinematic and dynamic similarity of an oncoming wind air flow is provided by the ANSYS software package and Fluent Flow analysis. The conditions for a change speed profile and height in calculation module ANSYS Fluent are set through the connection to boundary conditions of data tables including the rate of speed (csv format) [17]. Average flow speed is 6 meters per second. The flow is directed at an angle to building in accordance with the wind direction to wind rose.

2.3.2. For Autodesk Flow Design case.
Let’s consider the setup process for modelling a similar task using the Autodesk Flow Design. This software package supports dynamic modelling of gas processing in real time. The software also allows viewing results as a cutting plane and three-dimensional demonstration with the help of supporting particles or visual presentation of air flows.

This software relates to the «meshless» type. This type provides number of advantages in speed and simplicity of work. Universal approach of visualization was chosen for comparison of all results of purging. This visualization is presented in the form of counter plane.

Calculation is carried out according to the following algorithm: 1) Preparation of polygonal model in the original scale; 2) Downloading models in program; 3)Setup of parameters for virtual wind tube according to recommended speed – 6 meters per second; 4)Setting the initial conditions, choice of purging area sizes. It is unnecessary to comply with the requirements concerning the size of calculating area in accordance with ANSYS, but it is recommended to maintain the significant distances around model to minimize the impact of boundary areas.

The following settings for setup were chosen using the empirical selection (on the basis of analysis of area sizes & cell size & modelling speed): size of the modeling area around the object: 1600x1000x300 meters; size of the voxel: 3.3 meters on the side.

3. Results and analysis of simulation data for verification of environmental comfort in the built-up area
During the course of numerical simulation in Ansys Fluent the following results were obtained: contour scheme displaying 10 levels of wind speed (parameter contours count) in a horizontal projection at the level of 2 meters above the ground with a range limit of visualized speeds from 0 to
12 meters per second. The velocities which are at key points relative to the centre of courtyard were obtained with the aid of point measurement. It helped to measure the flow transformation coefficients inside the building block. The measurement results are given in the table 1 below.

| X  | Y  | Z  | Wind speed | Transformation coefficients | Average wind speed | Transformation coefficients |
|----|----|----|------------|-----------------------------|-------------------|-----------------------------|
| 0  | 2  | 0  | 4,46       | 0,74                        |                   |                             |
| 20 | 2  | 0  | 2,03       | 0,34                        |                   |                             |
| -20| 2  | 0  | 1,46       | 0,24                        | 2,43              | 0,40                        |
| 0  | 2  | 20 | 1,27       | 0,21                        |                   |                             |
| 0  | 2  | -20| 2,93       | 0,49                        |                   |                             |

Unlike Ansys Fluent, the results in the second case can be obtained at any time as a flat image, animated vectors, or video. In this case, a variant of the same plane was chosen with a cut at a height of 2 meters. Character visualization in this software package is different from that in the other packages, but is sufficiently informative. Figure 3. shows a snapshot of the results of modeling the flow velocity at a height of 2 m for both packages overlaid with mirror horizontal separation in graphics editor.

Flow Design does not have visualization settings for the contours and their quantities, as well as the built-in tool for point retrieval of characteristics by coordinates, therefore, the speed values can be estimated only visually. In view of the difficulty of characterization, it was decided to reduce the simulation area to the immediate surroundings of the building in an area of reduced detalization.

![Figure 3.](attachment:image.png)

**Figure 3.** a) Autodesk flow Design output at the top; b) Fluent output at the top.

Taking into account the assumption of a slight effect of external boundaries on the flow rate inside the building, it is possible to average the values at similar points. From calculating the discreteness of the color scale and the price of dividing 1.37 units, we obtain multiple values for the characteristic points. The data obtained were recorded in table 2.

| X  | Y  | Z  | Wind speed | Transformation coefficients | Average wind speed | Transformation coefficients |
|----|----|----|------------|-----------------------------|-------------------|-----------------------------|
| 0  | 2  | 0  | 3,79       | 0,63                        |                   |                             |
| 20 | 2  | 0  | 1,37       | 0,22                        |                   |                             |
| -20| 2  | 0  | 3,79       | 0,63                        | 2,6               | 0,31                        |
| 0  | 2  | 20 | 2,71       | 0,45                        |                   |                             |
| 0  | 2  | -20| 1,37       | 0,22                        |                   |                             |
The results of Ansys Fluent were used as a reference result to check the quantitative characteristics of the speed of wind flow between buildings. According to table 1 the average speed of the wind in the area between the buildings can be considered to be more than 3 m/s. At the same time, according to [14,18], a speed of up to 4 m/s can be considered a comfortable wind speed in a residential zone. Consequently, the simulation results can be assumed that the wind speed in this zone is considered comfortable for people, it is worth noting that with height variation average velocity increases significantly, and in the field of flow separation near the wall and can reach high impulses to 8-10 m/s.

Also, for a qualitative analysis of the suitability of simulation results in flow design, the images were combined in a graphical editor and each placed along the axis of symmetry with different location, Fluent results are places at the bottom (Figure 3-a) and at the Flow Design [19] at Figure 3-b. These images clearly show a significant visual correspondence of the nature of the flows, especially in the areas of the borders of abrupt changes in velocity (contrast transition), which repeat exactly the contours in both images. It should be noted that for a detailed analysis of the flow rate in the interval between buildings, the equivalent setting of the zone parameters is not enough, due to the low detail of the color contours in the zone of interest.

4. Findings

In this article, the practical implementation of setting up a computer model of air streams wrap around the complex of four buildings of complex shape was considered. Mathematical modeling and simulation were performed on the source data at real scales using different modeling software packages and the data obtained were combined on the same graphic image to reveal the visual similarity of the graphic image of the flow map. In summary the following conclusions can be drawn: after analyzing the flow rate using classical CFD modeling tools, it was concluded that the wind level is comfortable in the built-up area; at the same time, a similar result could be obtained using a much simpler Autodesk Flow Design package; flow design modeling package was the easiest to set up and allows you to quickly get the initial estimated results sufficient to quickly test hypotheses at an early stage, followed by modeling in professional packages. These experiments and datasets may be useful for further research on methods of simplified modeling of wind comfort in town planning tasks for non-professional CFD users [20].

References

[1] Shevtsov K K 1986 Building Design for Areas with Special Climatic Conditions (Moscow) p 231
[2] Olenkov V D 2017 Accounting for wind regime of an urban development in town planning with the use of computer simulation technologies Bulletin of the South Ural State University. Ser. Construction Engineering and Architecture 17(4) 21–27
[3] Serebrovskiy F L 1985 Aeration of Population Settlements (Moscow: Stroyizdat) 172
[4] Olenkov V D and Kolbin D S 2011 A city’s aeration mode and its effect on urban planning Construction and education: a collection of scientific papers (Yekaterinburg: UrFU) 14 71–74
[5] Kolbin D S and Olenkov V D 2011 Studying the wind regime with the purpose of aeration and wind protection of urban territories PNRPU Construction and Architecture Bulletin 1 36–39
[6] Toparlar Y, Blocken B, Maiheu B and van Heijst G J F 2017 A review on the CFD analysis of urban microclimate Renewable and Sustainable Energy Reviews 80 1613–40
[7] Calautita J K, Hughesa B R, Chaudhrya H N and Ghani S A 2013 CFD analysis of a heat transfer device integrated wind tower system for hot and dry climate Applied Energy 112 576–91
[8] Kalmikov A, Dupont G, Dykes K and Chan C P 2010 Wind power resource assessment in complex urban environments: MIT campus case-study using CFD analysis. Lawrence Berkeley National Laboratory Retrieved from https://escholarship.org/uc/item/27r989mg
[9] Broekhuizen I 2016 Integrating Outdoor Wind Simulation in Urban Design: A Comparative Study of Simulation Tools and their Benefits for the Design of the LTU Campus in Luleå Dissertation

[1] Abdulbasit A, Norhati I, Sabarinah Sh A and Josmin Y 2015 Thermal performance analysis of courtyards in a hot humid climate using computational fluid dynamics CFD method Procedia - Social and Behavioral Sci. 170 474–83

[11] Blackmore P 2011 Wind Microclimate around Buildings DG 520 (Watford: BRE Press)

[12] Janssen W D, Blocken B and van Hooff T 2013 Pedestrian wind comfort around buildings: comparison of wind comfort criteria based on whole-flow field data for a complex case study Build Environ 59 547

[13] Poddaeva O I and Dunichkin I V 2017 Architectural-Building Aerodynamics Vestnik MGSU vol 12(6) (105) 602–609

[14] Serebrovskiy F L 1985 Aeration of Population Settlements (Moscow: Stroyizdat) p 172

[15] Gorlin S M 1970 Experimental aeromechanics (Moscow) p 423

[16] Retter E I 1984 Architectural Aerodynamics (Moscow: Stroyizdat) p 294

[17] Poddaeva O I, Kubenin A S and Churin P S 2017 Architectural-building aerodynamics Textbook 2nd ed (Moscow) p 88

[18] Olenkov V D and Puzyrev P I 2014 Numerical simulation of wind load on a unique building Academic Bulletin URALNIIPROEKT RAACS. Ser. Building science 4 87–89

[19] Daemei A B, Khotbehsara E M, Nobarani E M and Bahrami P 2019 Study on wind aerodynamic and flow characteristics of triangular-shaped tall buildings and CFD simulation in order to assess drag coefficient Ain Shams Engineering J. DOI: 10.1016/j.asej.2018.08.008.

[20] Afkhamiaghda M 2017 The application of using computational fluid dynamic (CFD) to modern building design

Acknowledgments
The work was supported by Act 211 Government of the Russian Federation, contract No. 02.A03.21.0011.