SigmaFlow CFD code as a tool for predicting the wind environment around a group of buildings

V D Meshkova¹, A A Dekterev¹,², A A Gavrilov¹,² and K Yu Litvintsev¹

¹ Kutateladze Institute of Thermophysics SB RAS, 1 Lavrentyev Avenue, Novosibirsk, 630090, Russia
² Siberian Federal University, 79 Svobodny Avenue, Krasnoyarsk, 660041, Russia

E-mail: redel-vd@yandex.ru

Abstract. The paper reveals the capabilities of SigmaFlow CFD code to predict wind conditions in terms of pedestrian comfort as illustrated by a model problem. The proposed numerical model was verified by comparing with experimental data. A group of buildings consisting of low-rise buildings and a high-rise building was considered. A comparative analysis of five computational variants with different grid saturation was performed. The results of mathematical modeling allow observing the vortex flow structure that is generated when streamlining buildings.

1. Introduction

When designing residential neighborhoods and planning urban space in contemporary cities, it is necessary to take into account an important factor – the wind regime of the territory, since it is associated with a comfortable and safe stay of people. The construction of new neighborhoods has a great impact on the wind flow, which leads to irreversible changes. These changes may harm society in terms of the formation of stagnant zones, where sedimentation and accumulation of harmful impurities occur, as well as to a local increase in wind speeds to uncomfortable values. Timely information received about the wind flow speed field in the proposed built-up environment will eliminate or minimize these impacts.

Due to advances in high-speed processing of personal computers and the spread of software for numerical analysis of fluid dynamics, detailed aerodynamic information can be obtained based on Computational Fluid Dynamics (CFD).

The SigmaFlow complex was designed and is being developed at the Krasnoyarsk Branch of the Institute of Thermophysics SB RAS and the Department of Thermophysics of the Siberian Federal University. This complex is designed to study a wide range of hydrodynamic and thermophysical processes and allows performing parallel computations on contemporary multi-core processors and cluster systems. SigmaFlow allows simulating stationary and non-stationary liquid and gas flows, as well as turbulent flows using Reynolds-averaged Navier–Stokes equations (RANS) [1-3].

An important issue in any numerical study is evaluating the adequacy of numerical predictions. The verification process includes computational and physical aspects. To assess how well the mathematical model is implemented, it is necessary to prove the reliability of the results obtained. This is usually done by comparing simulation results with actual and/or experimental data.
When solving problems of hydrodynamics by a numerical method, one inevitably faces unsatisfied accuracy of the results obtained. This may be caused by many reasons but one of the most common reasons is the insufficient resolution of the computational grid.

In this paper, the effect of grid saturation on the computation results is considered.

2. Mathematical approach
To solve various problems of spatial streamlining, Reynolds-averaged systems of Navier-Stokes equations (RANS) are used, since, for large Reynolds numbers, the turbulent flow has a huge number of degrees of freedom, which makes a direct solution impossible. Various differential turbulence models are used for closure. All these models contain a different number of empirical constants, which are usually applied under certain conditions. Long-term experience of using various turbulence models for solving heterogeneous problems shows that this or that model allows obtaining acceptable results for certain flow classes.

The incompressible gas model was adopted as a mathematical model for describing the flow processes around buildings. In the spatial part of the model, the stationary flow of an incompressible medium is solved. A two-layer K-ω SST Mentor model is used for closure of the turbulent characteristics [4]. The SIMPLE-C procedure was used for numerical matching of velocity and pressure fields. The discretization of the conservation equation in the computational domain is performed by the control volume method.

3. Verification of the numerical model as illustrated by streamlining a group of buildings
The work of the Japanese Architectural Institute is considered as a model problem. A team of Japanese scientists conducted several comparative studies of the wind streamlining of a group of buildings. An experiment was conducted in a wind tunnel at the Niigata Institute of Technology for a high-rise building in an urban area [5]. The mathematical model presented above was verified based on the results of this work.

3.1. Problem statement
It was assumed that a low-rise urban quarter had an area of 40 m² and a height of 10 m to simulate a condition in which a low-rise building is completely blocked. The high-rise building has an area of 25 m² and a height of 100 m. Roads with a width of 10, 20, and 30 m are located between buildings (Fig. 1). The locations of the monitoring points of measurement are shown in Fig. 2 [5].

![Figure 1](image1.png)  ![Figure 2](image2.png)
3.2. Model parameters
For the comparative analysis, five variants of structural multi-block grids were constructed with different particularization in the area of the location of observation monitoring points. In each variant, the grid was refined by rebuilding the initial grid. As the number of calculated nodes increased, the problem dimension was becoming larger, since the entire computational domain was changing. The characteristics of the considered computation variants are shown in Table 1. The geometry of the computational domain and an example of the grid is shown in Fig. 3.

Table 1. Conditions of computation variants.

| Variant | Turbulence model | Scheme for convection terms | Computational method and time integral scheme | Grid |
|---------|------------------|-----------------------------|-----------------------------------------------|------|
| A       | $k-\omega$ SST   | UMIST TDV                   | SIMPLE-C                                      | 1 016 690 |
| B       | $k-\omega$ SST   | UMIST TDV                   | SIMPLE-C                                      | 1 343 819 |
| C       | $k-\omega$ SST   | UMIST TDV                   | SIMPLE-C                                      | 4 482 726 |
| D       | $k-\omega$ SST   | UMIST TDV                   | SIMPLE-C                                      | 4 830 841 |
| E       | $k-\omega$ SST   | UMIST TDV                   | SIMPLE-C                                      | 5 257 032 |

Figure 3. The geometry of the area under consideration with the computational grid.

3.3. Analysis of the results
Mathematical simulation allows observing the structure of the vortex flow, which is generated when streamlining buildings (Figs. 4-5). Analyzing the flow pattern, it can be seen that a vortex zone is formed behind a high-rise building. The maximum flow velocity occurs in the area where eddies breakdown from the sharp side faces of the building and equals to about 5 m/s. The dense location of low-rise buildings leads to the formation of recycling zones with low wind speeds, where the wind flow is blocked, and its further removal from the building area is difficult.

The maximum overpressure is observed in the frontal plane of a high-rise building, while the relative pressure on the upper surface and behind the streamlined body is negative (Fig. 6).
The comparison was performed by speed values. The grid variants A, B, and D were denser in the area of monitoring points. The simulation results have shown that the used particularization is not sufficient for a complete simulation of the actual flow. At the entrance to a high-rise building, the calculated speed values were underestimated in contrast to the experimental result, and at the exit, they were characterized by a large scatter (Fig. 7). In this regard, calculations were performed using two more grids C and E with a large grid concentration downward the flow along narrow streets from the entrance to the end of the area with monitoring points. The calculation results on these grids were better correlated with the experimental results and gave a good prediction of the velocity field (Fig. 8).
Figure 7. Comparison of the results of CFD computation using SigmaFlow code with experimental data (variants A, B, D).

Figure 8. Comparison of the results of CFD computation using SigmaFlow code with experimental data (variants C and E)
Discrepancies between the calculation and experimental data are mainly observed in the negative pressure phase and recirculating zones.

The relative errors for averaged velocities at the monitoring points for the profiles located along a high-rise building in the wind flow direction were calculated by the formula and are presented below (Tables 2-3) in comparison with the experimental data.

\[ \varepsilon = \frac{(v - v_{exp.})}{v_{exp.}} \times 100\% \]

Table 2. Comparison of relative velocities (m / s) and errors at monitoring points for all variants of numerical modeling (profile I).

| Monitoring points | \( v_{exp.} \) | \( A \) | \( B \) | \( C \) | \( D \) | \( E \) |
|------------------|----------------|--------|--------|--------|--------|--------|
| \( v \)          | \( \varepsilon,\% \) | \( v \) | \( \varepsilon,\% \) | \( v \) | \( \varepsilon,\% \) | \( v \) | \( \varepsilon,\% \) |
| 17               | 2.27           | 2.36   | 4      | 2.64   | 16     | 2.72   | 20     | 2.55   | 12     | 2.69   | 18     |
| 25               | 3.27           | 2.71   | -17    | 2.79   | -15    | 2.98   | -9     | 2.87   | -12    | 3.06   | -6     |
| 31               | 4.39           | 4.34   | -1     | 4.17   | -5     | 4.37   | 0      | 4.54   | 3      | 4.41   | 0      |
| 35               | 4.89           | 4.87   | 0      | 4.57   | -7     | 4.89   | 0      | 5.11   | 4      | 4.88   | 0      |
| 39               | 4.59           | 4.82   | 5      | 4.5    | -2     | 4.8    | 5      | 5.13   | 12     | 4.69   | 2      |
| 47               | 4.17           | 4.4    | 6      | 3.91   | -6     | 4.22   | 1      | 4.81   | 15     | 4.04   | -3     |

Profile I

Table 2. Comparison of relative velocities (m / s) and errors at monitoring points for all variants of numerical modeling (profile II).

| Monitoring points | \( v_{exp.} \) | \( A \) | \( B \) | \( C \) | \( D \) | \( E \) |
|------------------|----------------|--------|--------|--------|--------|--------|
| \( v \)          | \( \varepsilon,\% \) | \( v \) | \( \varepsilon,\% \) | \( v \) | \( \varepsilon,\% \) | \( v \) | \( \varepsilon,\% \) |
| 30               | 3.27           | 2.78   | -15    | 2.71   | -17    | 2.93   | -10    | 2.93   | -10    | 2.99   | -9     |
| 34               | 4.39           | 4.35   | -1     | 4.07   | -7     | 4.35   | -1     | 4.61   | 5      | 4.34   | -1     |
| 38               | 4.89           | 4.86   | -1     | 4.44   | -9     | 4.91   | 0      | 5.18   | 6      | 4.91   | 0      |
| 44               | 4.59           | 4.82   | 5      | 4.4    | -4     | 4.95   | 8      | 5.08   | 11     | 4.86   | 6      |
| 52               | 4.17           | 4.43   | 6      | 3.91   | -6     | 4.6    | 10     | 4.62   | 11     | 4.4    | 6      |

Profile II

Conclusions
The considered verification of the numerical method for calculating complex spatial flows confirms the extreme importance of this procedure since it allows determining how well a mathematical model can predict changes in the properties of the environment.

As a result of the conducted convergence studies, in the case of changes in the particularization of the initial computational grid, the optimal calculation options were found characterizing by minimal deviations from the experimental data for the considered profiles. These are C and E variants.
Based on the conducted computations, it can be concluded that the SigmaFlow CFD code and the proposed mathematical model allow predicting correctly the wind environment around a group of buildings.

Acknowledgments
The reported study was funded by RFBR, project number 19-31-90096.

References
[1] Dekterev A A and others 2003 Computational technologies 8(1) 250–5
[2] Dekterev A A, Gavrilov A A and Minakov AV 2010 Collection of articles. Modern science: research, ideas, results, technology 4(2) 117–22
[3] Dekterev A A et. al 2017 Journal of the Siberian Federal University. Technics and Technologies 10(4) 534–42
[4] Menter F R 1993 AIAA Paper 93(2906) 21
[5] Kato M and Launder B E 1993 Ninth Symposium on «Turbulent Shear Flows» (Japan) pp 10–14