

Numerical Simulation of Wind Loads on High-Rise Buildings

A.A. Antipin¹, V.N. Alekhin¹, S.N. Gorodilov¹, S.V. Khrantsov¹, A.S.Noskov¹

¹Ural Federal University named after the first President of Russia B.N. Yeltsin,
620002 Ekaterinburg, Russia

Abstract

The paper presents the results of experimental study and numerical simulation of external air flow around the cylindrical body with the diameter of 8 mm, placed on the way of free submerged jet of air. The cylinder which has simple geometry form was used due to the need to achieve the generality of the outcomes. Also it was used due to the complexity of experimental measurements for objects which have larger dimensions and more complex shapes. Testing of various turbulence models was performed. The obtained data was compared with the experimental measurements. Parameters of the numerical model which makes it possible to minimize the divergence of the numerical computations were derived.

Information on finished designing projects in which the results of the present research were used is presented in the final part of the paper.

Introduction

Wind loads on high-rise buildings are referred to the basic load group. Most design works considerably deal with aerodynamic analysis. Expert assessment of the standard values of wind pressures for buildings higher than 200 meters (corresponding to wind region VI, Russia) shows that the wind pressure can be more dangerous to the overall strength than the nine-point earthquake. According to active codes of practice [4, 6, 15], since aerodynamic issues do not sufficiently reflect wind effects on high buildings one has to apply experimental and numerical methods which include: monitoring and field measurements, wind tunnel tests and numerical simulation.

Civil Engineering Institute of Ural Federal University named after the first President of Russia B.N. Yeltsin steadily deals with design and deformation monitoring of a number of high-rise buildings in Ekaterinburg (Figs. 1, 2, 3, 4, 5), [2, 3]. Both wind pressure calculation and blow-through in wind tunnels of the building models were performed. Numerical and experimental data convergence was found to be satisfactory (within 10-30%).



Figure 1



Figure 2



Figure 3

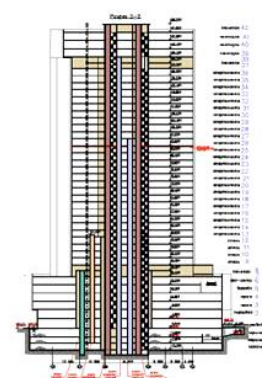


Figure 4

Computational Model

Numerical simulation is performed using finite element method applied in the software package ANSYS. The computational model is the numerical analogue of wind tunnel. The high-rise buildings and surrounding objects are placed in the domain, whose sizes are selected so that air flow on its boundaries is not affected by the buildings placed in it. The example of such domain is shown in Fig.5. A simulated high-rise building is on the left; the domain with the building proper and surrounding objects in it is on the right.

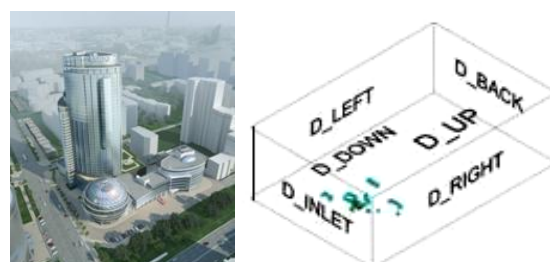


Figure 5

The size of the computational domain in vertical, lateral and longitudinal flow directions is conditioned by the simulated site development and boundary conditions. In the experience of testing in wind tunnels the building height H is assumed to affect up to a distance of $10H$. This height can be recommended to be an essential requirement to the model. According to test calculations made by the Institute of Architecture AIJ, Japan [1], the size of the computational domain in the vertical flow direction for an isolated building should be not less than $5H$.

Discussion

Analysis of wind pressure calculations proves the problem of data accuracy to be unsolved. Paper [16] shows that one fails to consider the essential parameters like scale factor, rigidity parameter, etc. while testing high-rise building models in wind tunnels. In addition, the agreement between experimental data and actual pressure values remains vague in computational analysis.

Diverse semi-empirical turbulence models are currently developed [8, 17]. Meanwhile each model is acceptable only in a limited number of simulation cases. This resulted from significant differences in turbulence internal structure arising from varied fluid flow conditions. A unified turbulence model valid for all feasible simulation cases has not been developed yet. Two parameter semi-empirical turbulence models ($k-\epsilon$, $k-\omega$ etc.) are mainly practiced [17]. The given models enable relatively simple flows (e.g. fluid flow in a pipeline) to be predicted with an adequate accuracy but in the simulation of more complicated cases they provide qualitatively and quantitatively inadequate results [10].

For outward fluid flow near-wall boundary layers are essential as Reynolds numbers change in a wide range and partial flow laminarization may arise there. For example, in the flow past a cylindrical body one can observe several fluid flow conditions: the boundary cylinder layer is laminar when Reynolds numbers are less than a critical number ($Re < Re_{cr}$). Separation of laminar boundary layer occurs in the frontal side of a cylinder with separation angle $\varphi = 82^\circ$ [14]. Thus, proper mathematical formulation of fluid laminarization effects in near-wall layers is of great importance. To this end, authors [17] designed and proposed the transition turbulence model to be applied in the case of significant Reynolds number decreasing. The application of the suggested model along with the model of turbulence SST allows to predict more accurately fluid behavior in a boundary layer in various fluid conditions both subcritical and supercritical.

Numerical simulation and experimental determination of wind velocities when flowing past a cylindrical body were carried out in order to estimate the accuracy of resulting values. The research task was testing and verification with respect to experimental measuring results of varied semi-empirical turbulence models in the case of outward flow past a cylindrical body placed in the passage of a freely submerged air stream.

Experimental Unit

Most researchers are known to devote their work to this problem [13-15]. The following scheme of an experimental unit was chosen on the basis of their research results (Fig. 6).

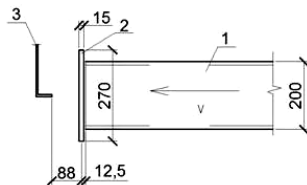


Figure 6

Wind from nozzle 1 flows over a round cylinder 2. Prandtl tube 3 fixed on the measuring plane is placed at a distance of 88 mm from the cylinder centre 2. Measurements are done with coordinate spacer changing its position on the horizontal axis "Y" every 5 mm and vertical axis "Z" every 10 mm respectively. Fig. 7 shows axis designation

adopted in the paper.

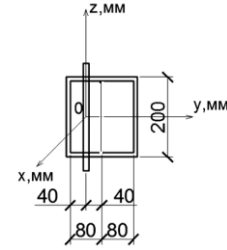


Figure 7

Computer Model of a Cylinder

Tetrahedral finite element mesh was used. Calculations were done for several mesh options with varied finite element dimensions (Fig. 8). The options of mesh parameters are listed in Table 1. Figure 9 shows the computational domain.

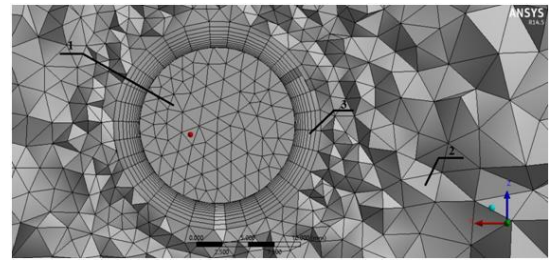


Figure 8

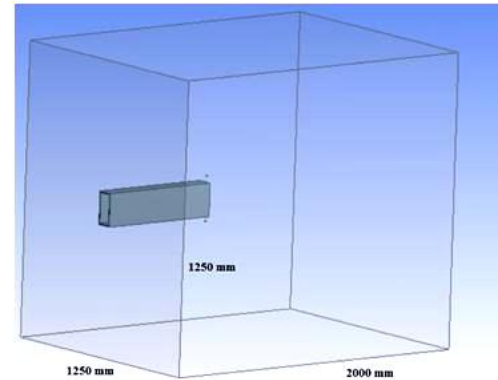


Figure 9

Options	Mesh parameters				
	Cell dimensions of the mesh, mm		Number of boundary layers	Boundary layer thickness, mm	Enlargement factor
	Cylinder	Domain			
1	1	5	10	0.1	1.2
2	0.5	2	10	0.1	1.2
3	0.1	2	10	0.1	1.2
4	0.3	3	5	0.1	1.2

Table 1. The options of the finite element mesh.

The main model SST ratios [9] are given below.

The equation for turbulent kinetic energy k and turbulence frequency ω :

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho \bar{U}_j k}{\partial x_j} = P_k - \beta^* \rho \omega k$$

$$+ \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) \quad (1)$$

$$\begin{aligned} \frac{\partial \rho \omega}{\partial t} + \frac{\partial \rho \bar{U}_j \omega}{\partial x_j} &= \frac{\alpha}{\nu_t} P_k - \beta^* \rho \omega k + \\ &+ \beta \rho \omega^2 \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) \\ &+ (1 - F_1) 2 \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \end{aligned} \quad (2)$$

where ρ – density; y – the distance to the surface; \bar{U}_j – flow velocity.

$$P_k = \min(\mu_t S^2, 10 \beta^* \rho \omega k) \quad (3)$$

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k}, \Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \quad (4)$$

$$S = \sqrt{2 S_{ij} S_{ji}} \quad (5)$$

$$\sigma_k = \frac{1}{\frac{F_1}{\sigma_{k1}} + \frac{(1-F_1)}{\sigma_{k2}}} \quad (6)$$

$$\sigma_\omega = \frac{1}{\frac{F_1}{\sigma_{\omega 1}} + \frac{(1-F_1)}{\sigma_{\omega 2}}} \quad (7)$$

where $\sigma_{k1} = 1,176$; $\sigma_{k2} = 1$; $\sigma_{\omega 1} = 2$; $\sigma_{\omega 2} = 1,168$

$$\nu_t = \frac{a_1}{\max(a_1 \omega, S F_2)} \quad (8)$$

where ν_t – turbulent viscosity;

$$\alpha = \alpha_1 F_1 + \alpha_2 (1 - F_2) \quad (9)$$

$$\arg_1 = \min \left(\max \left(\frac{\sqrt{k}}{\beta^* \omega y}, \frac{500 \mu}{\rho y^2 \omega}, \frac{4 \rho \sigma_{\omega 2} k}{C D_{k\omega} y^2} \right) \right) \quad (10)$$

The positive part of transverse diffusion terms is calculated according to:

$$C D_{k\omega} = \max \left(2 \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}; 10^{-10} \right) \quad (11)$$

Turbulence viscosity is calculated by the equation:

$$\mu_t = \min \left[\frac{\rho k}{\omega}; \frac{a_1 \rho k}{S F_2} \right] \quad (12)$$

with constant $a_1 = 0.31$ and blending function.

$$F_2 = \tanh(\arg_2^2) \quad (13)$$

$$\text{where } \arg_2 = \left[\max \left(\frac{2 \sqrt{k}}{\beta^* \omega y}; \frac{500 \nu}{y^2 \omega} \right) \right]^2$$

Blending functions F_1 and F_2 are given so that to define whether the point in question is inside or outside the surface boundary layer.

Consequently, functions F_1 and F_2 possess the following limit values:

$$F_1 = 0, \text{ off the surface } \Rightarrow k - \varepsilon \text{ model}$$

$$F_1 = 1, \text{ inside the boundary layer at surfaces } \Rightarrow k - \omega \text{ model}$$

$$F_2 = 0 \Rightarrow \text{combination } k - \varepsilon \text{ and } \Rightarrow k - \omega$$

$$F_2 = 1 \Rightarrow \text{SST model}$$

Results

Numerical analysis results show that option 4 (Table 1) is found to be optimal. Figure 10 shows the comparison of experimental data of wind velocities for four turbulence models.

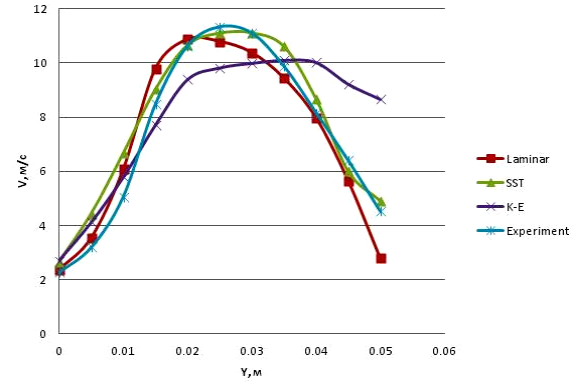


Figure 10

Thus, the results being in agreement with experimental data can be obtained by means of the model SST.

The results of numerical investigation of a cylinder model are applied in the computation of wind pressures upon high-rise buildings (Figs. 1-5). The computational results were compared with those of the blow-through in wind tunnels. Based on the comparison of these results, the convergence of wind pressures obtained numerically as well as experimentally proves to be satisfactory. Maximum computational wind pressures are close in the range of 10-25%, though pressure distribution-height graphs may slightly differ.

A 52-storied building “Iset Tower”, Ekaterinburg (Fig.1) was taken for the calculation of wind loads. The height of the building is 209 m. It consists of two main blocks: the residential part (52 above-ground stories) and the underground part with parking and technical facilities. The cross-section of circular superstructure is shown in Fig.11. The surface of the circular part of the building is ribbed, the rib height being 700 mm.

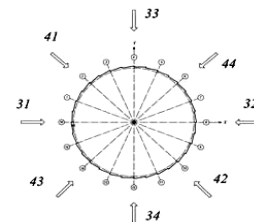


Figure 11

The blow-through in the wind tunnel of the building was performed by the company «WACKER INGENIEURE», Germany. Since the scale of the model was small - 1/380, it necessitated specification of wind pressure distribution on the surfaces of the building and its impact on the framework.

Finite element mesh with tetrahedral cells and prisms in the wall regions (“Wall” – like boundary conditions) were employed. The finite element size near the boundaries of the domain is 100 m, near the surface of the building is 0.5 m. Figure 12 shows the finite element mesh near the surface of the building.

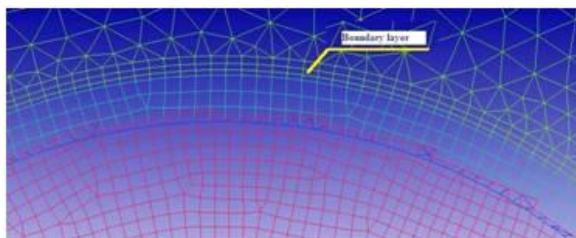


Figure 12

The characteristic form of wind pressure distribution along the building is shown in Fig. 13; the pattern of distribution of velocities at a height of 58.5 m is in Fig. 14. The convergence of computational and experimental results is found to be satisfactory.

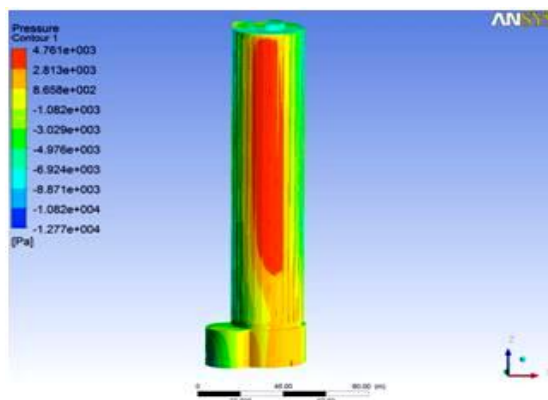


Figure 13

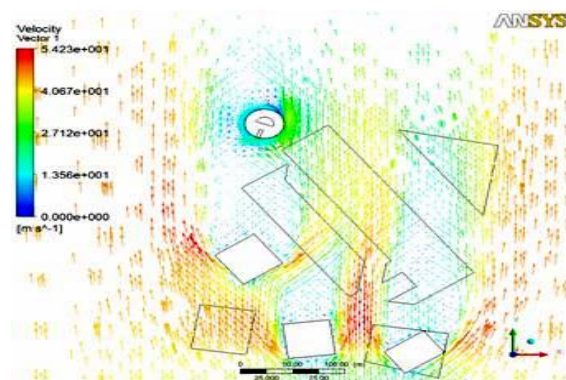


Figure 14

Conclusions

Numerical analysis of wind pressure on high-rise buildings reveals the following:

- The results of numerical analysis and experimental wind determination are qualitatively similar.
- The difference of computational and experimental values is mainly due to varied computational models

of buildings used in natural oscillation frequency tests. Maximum wind loads obtained numerically are in good agreement with those obtained experimentally.

From the research results one can conclude that in the design of high-rise and unique buildings both aerodynamic experiments and numerical analysis are required.

References

- [1] AIJ Recommendations for Loads on Buildings. *Architectural Institute of Japan*, Japan. 1996.
- [2] V.N. Alekhin, A.A. Antipin, S.N. Gorodilov. *Analysis of wind impacts on the high-rise building “Iset Tower”*. *Applied Mechanics and Materials*, 281, 639-644.
- [3] Vladimir Alekhin, Aleksey Antipin, Sergey Gorodilov, Sergey Khramtsov. *Numerical simulation of wind loads on high rise buildings. Proceedings of 13th International Conference on Construction Applications of Virtual Reality*.-London: 2013. 620-628.
- [4] British Standard, Loadings for Buildings - Part 2: Code of Practice for Wind Loads, *Building and Civil Engineering Sector Board*, UK, 1995.
- [5] Dongke Sun, John S. Owen, Nigel G. Wright, Kai Fan Liaw. Fluid-structure interaction of prismatic line-like structures, using LES and block-iterative coupling. *Journal of Wind Engineering and Industrial Aerodynamics* 96 (2008) 840–858.
- [6] Eurocode 1: Basis design and action on structures. Part 1: "Basis design". ENV 1991 1, // *CEN*, 1994. 232 p.
- [7] J. H. Ferziger, M. Peric. Computational methods for fluid dynamics, *3rd edition*, Springer, 2002. 423p.
- [8] P.G. Frik. *Turbulence: models and approaches*. Perm, Russia, 1998.
- [9] R.B. Langtry, F.R. Menter, S.R. Likki and Y.B. Suzen. A correlation-based transition model using local variables - Part II: Test Cases and industrial applications. *Journal of Turbomachinery*, 128(3), 423-434, 2004.
- [10] F.R. Menter. Review of the shear-stress transport turbulence model experience from an industrial perspective. *International Journal of Computational Fluid Dynamics*, Vol. 23 No. 3, 2009. P. 305 – 316.
- [11] Menter F.R., Kuntz M., Langtry R. Ten years of industrial experience with the SST turbulence model // *Turbulence: Heat and Mass Transfer 4: Proc. Intern. Conf.* Begell House, Inc. 2003. - 8.
- [12] Sridhar Muddada, B.S.V. Patnaik An assessment of turbulence models for the prediction of flow past a circular cylinder with momentum injection. *J. Wind Eng. Ind. Aerodyn.* 98 (2010) 575–591.
- [13] Francesco Ricciardelli Effects of the vibration regime on the spanwise correlation of the aerodynamic forces on a 5:1 rectangular cylinder. *J. Wind Eng. Ind. Aerodyn.* 98 (2010) 215–225.
- [14] H. Schlichting. *Boundary layer theory. Translation from German into Russian*. Moscow, 1974. 712 p.
- [15] SNiP 2.01.07-85 Loads and effects. *State Committee of Russia — Moscow: SUE LAC*, 2001.—44 p.
- [16] J. Wacker, R. Friedrich, E.J. Plate, U. Bergdolt. Fluctuating wind load on cladding elements and roof pavers. *J. of Wind Engineering and Industrial Aerodynamics*, 38, 1991, 405-418.
- [17] David C. Wilcox. *Turbulence modeling for CFD*, DCW Industries, California, 1994. 460 p.