

MATHEMATICAL MODELLING OF WIND ACTION ON CIVIL STRUCTURES USING ANSYS FLUENT

Svetlana A. Valger ¹, Natalya N. Fedorova 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@itam.nsc.ru

Key Words: *bluff bodies, wind actions, turbulence, CAE, ANSYS Fluent.*

The priority in building and structural design is to ensure the construction safety, reliability and suitability for exploitation. In structural design, particular attention should be paid to determine the dynamic wind actions, which can lead to wind resonance. When new high-rise objects are constructed within an already densely-built area, it involves changes in aerodynamics of the already existing constructions through vortex formation or higher wind velocities.

In the present day, the simplified calculation method described in Russian state regulatory act [1] is used to determine wind loads on structures. According to this method, mean and fluctuating components of wind load and wind load peak values should be considered. To calculate wind load according to [1], knowledge of the flow parameters such as integral aerodynamic coefficients of external and internal pressures, skin friction, drag, shear force, torque, are required. But these parameters are known only for some typical structure configurations.

For more complex structure configurations, [1] prescribe to perform large-scale modelling in aerodynamic facilities. This modelling requires a lot of time and finance resources. Thus it is reasonable to use methods based on computational experiments, which allow us to reduce time and finance resources and solve the problem of excessive oscillations damping. There are various CAD/CAE systems for computational experiments, but there is no unified method to calculate wind loads. There is a lot of issues to be studied in this sphere, for example turbulence modelling, interference effects between buildings etc.

In the paper, the numerical investigation of unsteady incompressible air flows in a vicinity of bluff bodies and their complexes was performed using the ANSYS Fluent [2].

During the first stage, a series of 2D and 3D calculations was conducted for the test cases [3] (a turbulent flow around cylinder, prism, two prisms etc.). Particular attention was paid to the choice of turbulence model, the types of boundary conditions and the creation of the computational grid using the methods of geometric and dynamic adaptation. The vortex shedding frequency was obtained for all test cases. The peculiarities of unsteady flows around bluff bodies and their complexes considering interference effects were investigated. The computational results were compared to the experimental data from the literature by flow parameter fields and the Strouhal number. The satisfactory agreement was achieved.

In fig.1 the results of calculation performed using the conditions from experiment [4] are presented. Numerical study of unsteady air flow around the prism was performed by means of 3D RANS-based approach closed by $k-\omega$ SST model under constant temperature. The finite volume scheme based on solving the equation for pressure was used, as well as MUSCL

scheme of the third-order was used to approximate the derivatives of the spatial variables. In the computations, 3D effects were revealed in the air flow around the prism. A vortex zone is formed behind the prism. The velocity maximal value $V \approx 6.7$ m/s in the vicinity of vortex shedding from prism side edges is observed. The maximal relative static pressure is observed in the top part of prism frontal plane. A negative value of relative static pressure appears

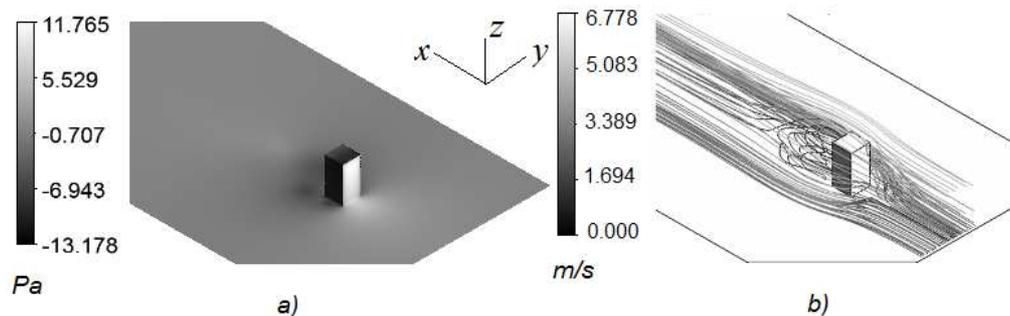


Fig. 1. Computed field of relative static pressure (a) and velocity field and streamlines (b)

on the prism top face. The computational results have good agreement with the exp. data [4] per mean velocity and turbulence kinetic energy distributions in several cross sections (fig. 2).

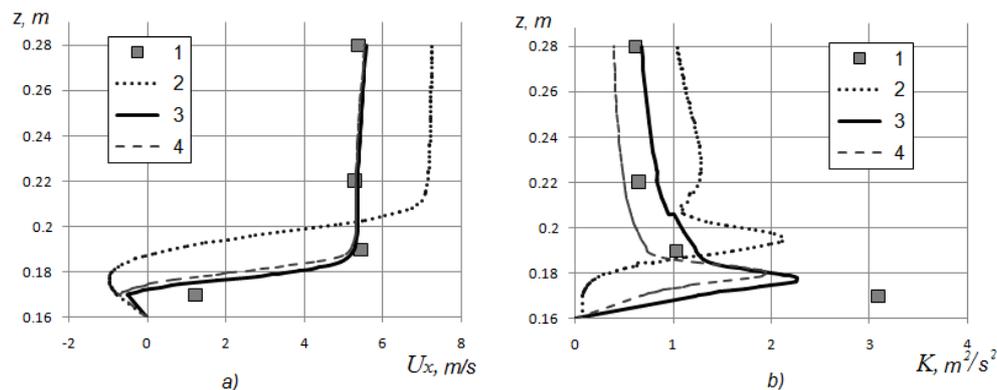


Fig. 2. Experimental and computed velocity (a) and turbulence kinetic energy (b) distributions in cross section $x = 0$ m (in the center cross section over the prism): 1 - exp., 2 - 2D calc., 3 and 4 - 3D calc.

At the next stage, the full-scale numerical investigation of air flow around constructions was conducted taking into account the atmospheric boundary layer. The results of simulations were compared to result obtained by the method [1].

REFERENCES

- [1] Building Codes and Regulations. SNIIP 2.01.07-85*. Actions on structures. Moscow: FGUP CPP, 2010. (in Russian)
- [2] www.ansys.com
- [3] Valger S.A., Fedorov A.V., Fedorova N.N. Simulation of Incompressible Turbulent Flows in Vicinity of Bluff Bodies with ANSYS Fluent // Computational Technologies, 2013. Vol. 18. No. 5. P. 27-40 (in Russian)
- [4] Yoshihide Tominagaa, Akashi Mochida, Ryuichiro Yoshiec, Hiroto Kataokad, Tsuyoshi Nozue, Masaru Yo-shikawaf, Taichi Shirasawa AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings // Journal of Wind Engineering and Industrial Aerodynamics, 2008. P. 1749 - 1761