10.2478/tvsb-2013-0014

Transactions of the VŠB – Technical University of Ostrava

No. 2, 2013, Vol. XIII, Civil Engineering Series

paper #14

Vladimíra MICHALCOVÁ[1], Sergej KUZNĚTSOV[2], Stanislav POSPÍŠIL[3]

MODELS OF LOAD ON BULDINGS FROM THE EFFECTS OF THE FLOW FIELD

Abstracts

Article describes two different approaches of the solution of benchmark solution of bluff aerodynamic, which is the solution of wind pressures upon the cube exposed to the effects of air flow field. Physical modeling is carried out at the wind tunnel of the Institute of Theoretical and Applied Mechanics in Telč whereas numerical modeling is performed at the Faculty of Civil Engineering, VSB Technical University of Ostrava of software using Ansys Fluent.

Keywords

Aerodynamics, wind tunnel, CFD.

1 INTRODUCTION

Modelling a flow around low objects of non-aerodynamic shapes brings many problems [3], [4] and this applies to both numerical and physical simulations. The aim of the paper is to compare results of the physical and numerical modelling of an air-flow around an object of the shape of a cube with an edge of 0.24 m. It represents the so-called Silsoe cube with a scale of 1:25 that has gradually become a standardized experimental element in the field of building aerodynamics. The reason for this choice is the possibility of using informative data from measurement in the tunnel to assess the final results of both approaches [1].

A smooth flow field with a constant vertical velocity profile of 13.5 m.s^{-1} and a turbulence intensity of approx. 1 % was modelled within a physical experiment in the CET wind laboratory http://cet.arcchip.cz/. The object of evaluation is the pressure load of the model due to the effects of a flow field. If is defined here using the dimensionless external pressure coefficient c_{pe} that is the ratio of the static pressure and the dynamic pressure related to the reference point:

$$c_{pe,i} = \frac{p_i}{p_{dyn}} = \frac{p_{ci} - p_{ref}}{1/2 \cdot \rho \cdot u_{ref}^2}, \tag{1}$$

where:

p_{ref} is the static pressure at the reference point [Pa];

p_{ci} is the resulting static pressure on the surface of the object at the i-th point [Pa];

p is the static pressure on the surface of the object at the i-th point recalculated in relation to the reference pressure [Pa];

[1] Ing. Vladimíra Michalcová, Ph.D., Department of Structural Mechanics, Faculty of Civil Engineering, VŠB-Technical University of Ostrava, Ludvíka Podéště 1875/17, 708 33, Ostrava-Poruba, Czech Republic, phone: (+420) 597 321 313, e-mail: vladimira.michalcova@vsb.cz.

[2] Prof. Ing. Sergej Kuznětsov, Ph.D., ÚTAM AV ČR, v.v.i (ITAM AS CR, v.v.i.), Centre of Excellence Telč, Batelovská 485, 588 56 Telč, Czech Republic, e-mail: kuznetsov@itam.cas.cz.

[3] Doc. Ing. Stanislav Pospíšil, Ph.D., ÚTAM AV ČR, v.v.i. (ITAM AS CR, v.v.i.), Prosecká 809/76, 190 00 Praha, Czech Republic, e-mail: pospisil@itam.cas.cz.

u_{ref} is the x-component of velocity at the reference point [m.s^{-1}]; and

ρ is the air density ρ =1.225 [kg.m^{-3}].

The external pressure coefficient c_{pe} has been measured, calculated, and evaluated in two sections perpendicular to each other and in a horizontal section. The scheme of the model with 36 sampling points is shown in Figure 1.

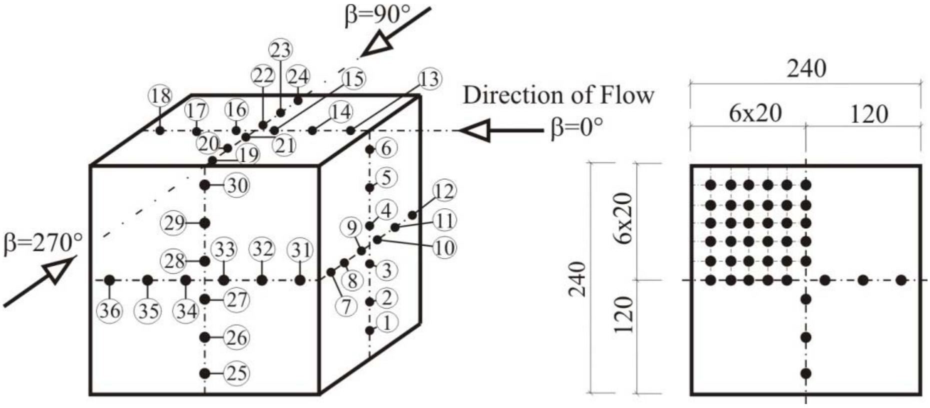

Fig. 1: Scheme of the model

2 PHYSICAL EXPERIMENT

The experiments were carried out in the climatic wind tunnel of the Centre of Excellence Telč, the Institute of Theoretical and Applied Mechanics of the AS CR. The testing section of the aerodynamic section has a rectangular cross-section of 1.9 m (width) × 1.8 m (height). The total length of the coming flow part of the aerodynamic section is 11.0 m, including the turbulent boundary layer development part with 9 m of length. If necessary, a simulation of the atmospheric boundary layer with the required characteristics is modelled using elements such as networks, so-called Counihan's generators, barriers, and floor plates with different roughness.

The model of a cube was placed in the centre of the rotary table (with a diameter of 1.76 m) in the work part of the aerodynamic section of the tunnel. The cube was made of transparent Plexiglass with a wall thickness of 5 mm; the drainage points have a diameter of 0.5 mm and are equipped with connectors in the shape of a metal tube with an inside diameter of 1 mm. All connectors of the drainage points are connected using a silicone tube (with a length of 1 m and a diameter of 2 mm) with the measuring device Scanivalve Corp. DSA 3217 for sensing the pressure (Fig. 2). The data and conversion to digital values were collected in the acquisition system (DEWETRON) with the sampling frequency equal to 1 kHz.

Fig. 2: The model in the measuring section and a detail of the measuring section

The results of the pressure coefficients are shown as the function of air flow direction $\beta = 0\ldots360°$ and can be recorded as $c_{pe,i}=f(\beta)$. The interval of model rotation on the vertical axis $\Delta\beta$ was 10°. The experiments were carried out with Reynolds number $Re = 2.1\cdot10^5$ [-] that corresponded to an air flow velocity of 13.5 $m.s^{-1}$.

3 NUMERICAL MODELLING

Numerical models suitable for this type of problem can be divided into two categories:

- RANS models and
- models for anisotropic turbulence – RSM and LES and its combinations with RANS models.

3.1 RANS models

These are statistical turbulence models that are based on the method of the time-averaging (Reynolds Averaged Navier-Stokes equations) of turbulent flow variables and on the following procedure for time-averaging balance equations describing a turbulent flow. They use so-called Boussinesq's hypothesis that uses a simplified expression of Reynolds stresses. To calculate the problem, the Spalart Allmaras, Standard k-ε, RNG k-rchε, and SST k-ω models were used respectively. They are based on isotropic turbulence modelling. They differ from each other in defining the so-called turbulent dynamic viscosity, a variable that expresses complex functional relations of the state of a flowing fluid and the position of a point being considered. An advantage of RANS models is the lower requirements for the calculation of the area grid density, the possibility of modelling a stationary problem, and the ability to make a rapid calculation. Unfortunately, they are less suitable for solving a flow around structures of non-aerodynamic shapes because strong anisotropic turbulence is created in the surroundings of the object. A calculation area of a size of 1.9 m (width) × 1.8 m (height) × 4.5 m (length) was created for RANS models.

The monitored object was placed at a distance of 1m from the entrance to the domain. $1\text{x}10^6$ tetra cells are used to create the network of the examined domain according to Fig. 3 in which the high density in the surroundings of the object as well as the uniform longitudinal strip required for solving correctly the boundary condition on the side walls are apparent.

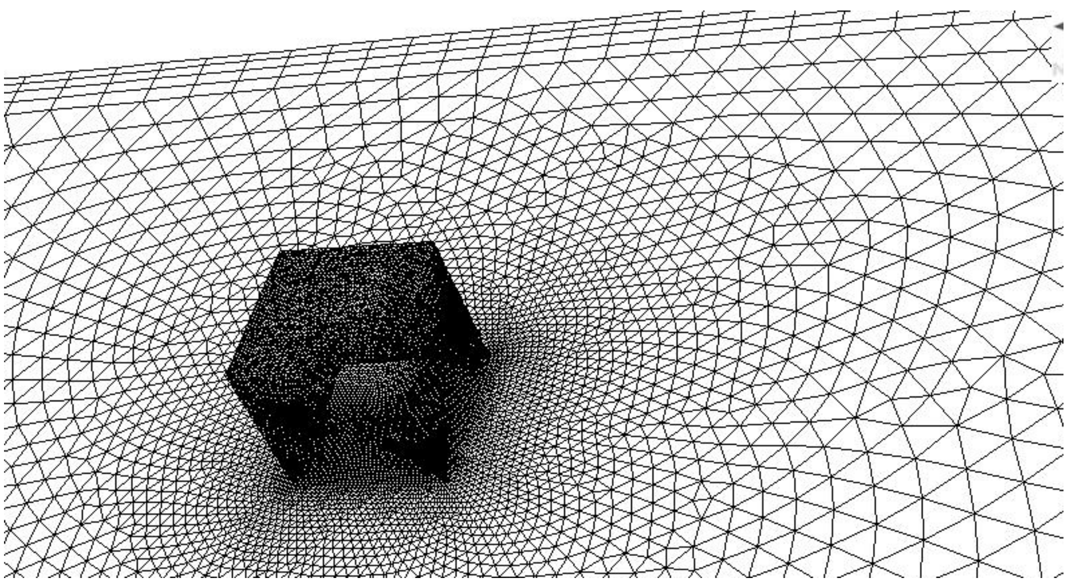

Fig. 3: Tetra grid for the RANS and SAS models: Horizontal plane in the middle of the height of the object

3.2 Models for anisotropic turbulence

These models are based on the principle of modelling anisotropic turbulence, which better reflects the examined action. However, they require higher quality and density of the network and the problems must be solved in a non-stationary way, so that the computational times are significantly longer. An exception is the RSM model. It enables to solve a problem in a stationary way, but it is very sensitive and frequent problems with convergence occur when it is used. It was not used for the calculations presented here. Newly developed hybrid models that are a combination of the LES [2] and RANS methods, namely the SAS and ELES models, were used to solve the problem. Their correct combination enables to reduce significantly the number of cells i the calculation area and thereby reduce significantly the computational time, even though the transmission of variables at the interface of the areas also partially extends the computational time.

ELES models large vortex structures in the disordered flow field area (in this case the surroundings of the object being flowed around) using a direct simulation (LES) and in the area in which an ordered isotropic flow can be expected using the RANS method. It requires the precise definition of the interface, which allows for preparing networking better. A new calculation domain with the dimensions 1.9 m (width)× 1.8 m (height) × 5.0 m (length) with combined grids (Fig. 4) was created for this calculation. The base is a polyhedral cell. The defined domain in the surroundings of the object for a direct simulation using the LES method then consists of a thick grid of the hexagonal cells. This domain with a length of 1m begins at the distance at 0.2 m in front of the object and its transverse dimensions exceed the perimeter of the cube being flowed around by 0.2 m on each of its sides. Figure 5 shows the difference in the density of the grid as well as the tendency how the sizes of the polyhedral cells gradually increase with the increasing distance from the object.

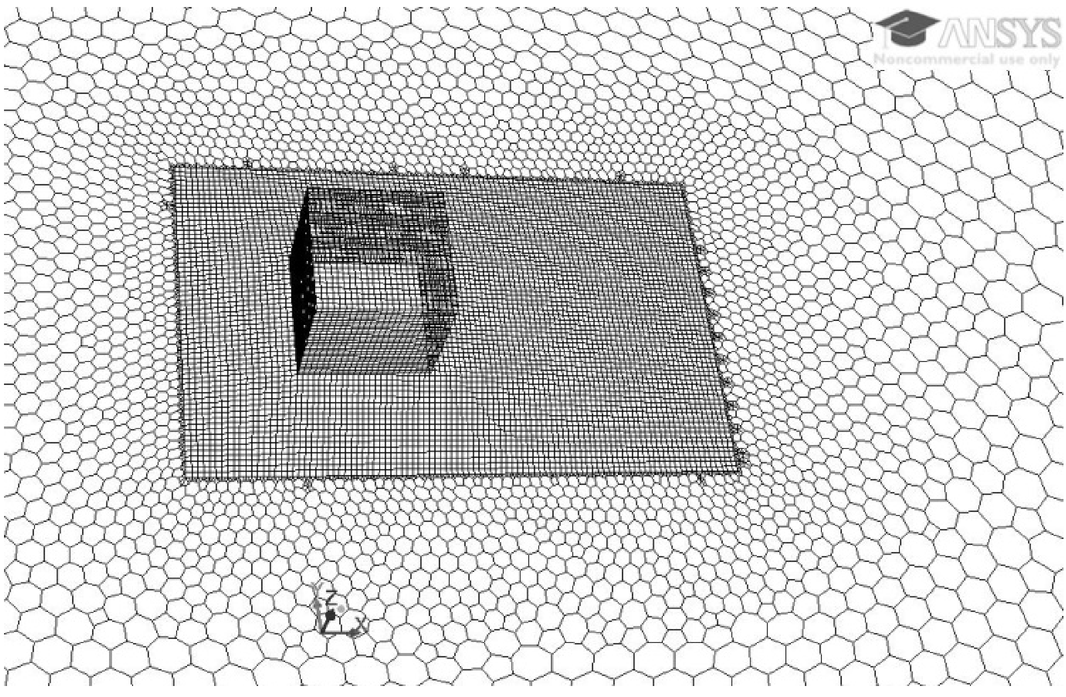

Fig. 4: Combined grid for the ELES model

The SAS model defines the interface on the basis of the linear scale of vortices and behaves as the SST k-ω model in close proximity to the wall and switches automatically to the LES calculation at a larger distance. Its limits do not need to be entered. The calculation area was the same as that for the RANS models for the calculation (Sec. 3.1).

3.3 Boundary conditions

The same types of boundary conditions were set for all calculations. These are the velocity at the inlet and the pressure-outlet condition at the outlet of the calculation domain. The bottom surface is presented using the wall condition, which is the same as in the case of modelling of the open space (atmosphere). The boundary conditions on both sides and on the upper surface of the calculated domain were defined using the wall to correspond to the bounded space of the tunnel, which requires additional requirements on the shape of the grid, the aforementioned uniform distribution of a certain density in the surroundings of the walls (Sec. 3.1).

4 RESULTS AND EVALUATION

As mentioned in Section, the external pressure coefficient on the cube was measured, calculated, and evaluated in two vertical sections perpendicular to each other (Fig. 5 and 6) and in a horizontal section (Fig. 7). Although the concern is the flow with the low turbulence intensity (1 % at the inlet), the RANS models are not suitable for this type of problem because strong anisotropic turbulence arises especially on the sides and on the leeward side of the object in its close vicinity.

On the front side (Fig. 5 to 7), where no strong vortex occurs, all calculations agree with the wind tunnel experimental measurement. There are marked differences in the results of the pressure load along the object, i.e. on the sides and on the upper wall. Figures 5 to 7, especially the course of the load in the transversal vertical section (Fig. 6), prove the unsuitability of RANS models for describing a flow field. For the leeward side of the object, calculations using the Spalart Allmaras and Standard k-ε models approximated the most to the measured values.

The non-stationary calculations using the ELES and SAS models achieved satisfactory results. Moreover, the ELES model gave the results on the front, the upper and the both sides almost identical to those measured in the wind tunnel. The results on the leeward side differ slightly. The SAS model correctly copies the shape of the load curve along all three monitored perimeters of the cube being flowed around. A small displacement could be avoided by using a new denser grid and a longer simulated time of a flow, which will be the object of further examinations.

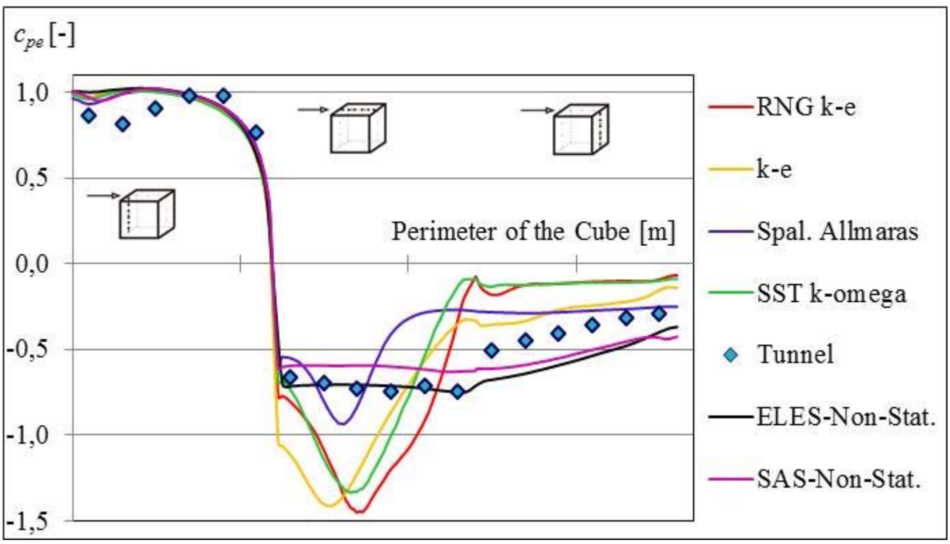

Fig. 5: Vertical longitudinal section

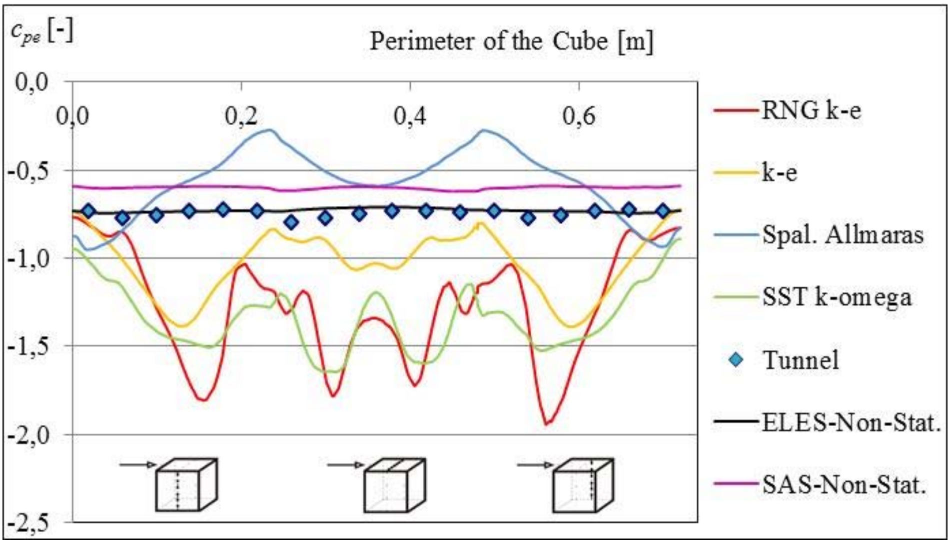

Fig. 6: Vertical cross section

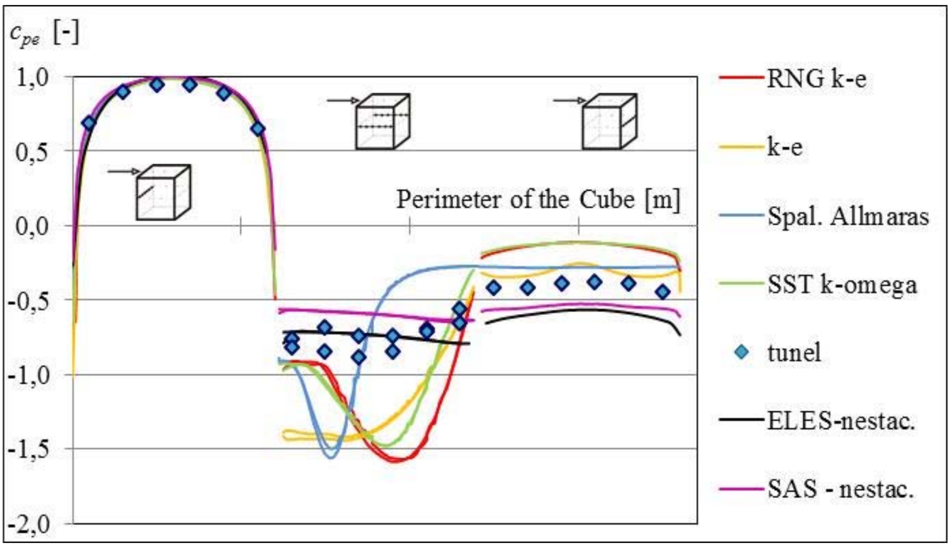

Fig. 7: Horizontal section

A disadvantage of the non-stationary calculations is their demands from the viewpoint of the preparation of calculation (the shape and density of the calculation network and the correct definition of a time step) as well as its time-consuming nature. For both non-stationary problems, a time step of 0.001 second was selected and the calculation simulated a flow for 4 seconds, with the averaging of the variables carried out after 1 second of the simulated action. At that time it was possible to regard the flow field as steady. The resulting time of the flow with the time-averaging of the variables represented approximately eight times the air exchange in the calculation area. There were approx. $4 \cdot 10^4$ iterations within one calculation.

5 CONCLUSION

The solution of this problem proved that a flow around an object of non-aerodynamic shape is a complex action for modelling, whether experimental or mathematical one, and that mutual cooperation between the two approaches is necessary. The results of the numerical simulations using the ELES and SAS models for non-stationary problems showed a very satisfactory similarity with the experiment, which is promising for the further modelling with the atmospheric turbulence.

ACKNOWLEDGMENT

The work is supported from funds of the conceptual development of science, research and innovation for 2012 allocated to the VŠB-Technical University of Ostrava by the Ministry of Education, Youth and Sports of the Czech Republic.

This paper was prepared with the support of the project CZ.1.05/1.1.00/02.0060 that is co-funded from the European Regional Development Fund.

REFERENCES

[1] RICHARDS, P.J., R.P. HOXEY, B.D. CONNELL a D.P. LANDER. Wind-tunnel modelling of the Silsoe Cube. *Journal of Wind Engineering and Industrial Aerodynamics: The Fourth European and African Conference on Wind Engineering*. Vol. 95, 9-11, pp. 1384-1399. ISSN 0167-6105.

[2] TAMURA, T., A. OKUNO a, Y. SUGIO. LES analysis of turbulent boundary layer over steep hill covered with vegetation. *Journal of Wind Engineering and Industrial Aerodynamics: The Fourth European and African Conference on Wind Engineering*. Vol. 95, 9-11, pp. 1384-1399. ISSN 0167-6105.

[3] ZHANG, Y.Q., A.H. HUBER, S.P.S. ARYA, W.H. SNYDER. Numerical simulation to determine the effects of incident wind shear and turbulence level on the flow around a building. *Journal of Wind Engineering and Industrial Aerodynamics: Proceedings of the 1st International Conference on Computational Wind Engineering*. Vol. 93, 46-47, pp. 129-134.

[4] ZHANG, A., M. GU. Wind tunnel test and numerical simulations of wind pressures on buildings in staggered arrangement. *Journal of Wind Engineering and Industrial Aerodynamics: 4th International Symposium on Computational Wind Engineering* (CWE2006)., Vol.96, iss. 10-11, pp. 2067-2079.

Reviewers:

Prof. RNDr. Milada Kozubková, CSc., Department of Hydrodynamics and Hydraulic Equipment, Faculty of Mechanical Engineering, VSB-Technical University of Ostrava.

Prof. Ing. Jozef Melcer, DrSc., Department of Structural Mechanics, Faculty of Civil Engineering, University of Zilina.