
Vladimíra MICHALCOVÁ¹, Lenka LAUSOVÁ², Iveta SKOTNICOVÁ³, Stanislav POSPÍŠIL⁴**NUMERICAL AND EXPERIMENTAL MODELS OF THE THERMALLY
STRATIFIED BOUNDARY LAYER****Abstract**

The article describes a change of selected turbulent variables in the surroundings of a flow around thermally loaded object. The problem is solved numerically in the software Ansys Fluent using a Transition SST model that is able to take into account the difference between high and low turbulence at the interface between the wake behind an obstacle and the free stream. The results are verified with experimental measurements in the wind tunnel.

Keywords

Thermal stratification, turbulence, complex terrain, CFD, Transition SST model, climatic wind tunnel.

1 INTRODUCTION

Wind conditions influence the effect of wind on both building constructions and the dispersion of pollutants from different surface or elevated sources. Those are the processes occurring in the atmospheric boundary layer (hereinafter ABL). The physical and thermal properties of the underlying surface in connection with the dynamics and thermodynamics in the lower layers of the atmosphere influence wind velocity distribution in the thermally stratified ABL. The atmospheric turbulences are characterized by a high rate of irregularities, dimensionality, diffusivity, dispersion and a very wide range of motion scales.

From the viewpoint of the effect of wind on building constructions, it is generally assumed that the influence of heat convection can be ignored. However, there is actually almost always a certain vertical heat flow in MVA, which affects the flow field velocity profiles of wind. These thermal effects, especially their effects on building constructions, whether in terms of load and architectural solutions, are an item of interest globally. They are investigated both experimentally [1, 2] and numerically [3, 4].

¹ Ing. Vladimíra Michalcová, Ph.D., Department of Building Structures, Faculty of Civil Engineering, VŠB - Technical University Ostrava, Ludvíka Podéště 1875/17, 708 33 Ostrava - Poruba, Czech Republic, phone: (+420) 597 321 348, e-mail: vladimira.michalcova@vsb.cz.

² Ing. Lenka Lausová, Ph.D., Department of Building Structures, Faculty of Civil Engineering, VŠB - Technical University Ostrava, Ludvíka Podéště 1875/17, 708 33 Ostrava - Poruba, Czech Republic, phone: (+420) 597 321 326, e-mail: lenka.lausova@vsb.cz.

³ doc. Ing. Iveta Skotnicová, Ph.D., Department of Structural Mechanics, Faculty of Civil Engineering, VŠB - Technical University Ostrava, Ludvíka Podéště 1875/17, 708 33 Ostrava - Poruba, Czech Republic, phone: (+420) 597 321 957, e-mail: iveta.skotnicova@vsb.cz.

⁴ doc. Ing. Stanislav Pospíšil, Ph.D., Department of Structural Mechanics, Faculty of Civil Engineering, VŠB - Technical University Ostrava, Ludvíka Podéště 1875/17, 708 33 Ostrava - Poruba, Czech Republic, phone: (+420) 597 321 919, e-mail: stanislav.pospisil@vsb.cz.

2 TASK DESCRIPTION

In cooperation with the department of experimental research CWT CET in Telč [5, 6, 7], a task corresponding to experimental measurements in the climatic wind tunnel CET in Telč is numerically modeled in the presented work [8]. It is the case of flow around a heated model of a hill of a height $h = 200$ mm (Fig. 1) at two different air velocities and temperatures of the object surface. The aim is to define changes in the flow field in the stratified boundary layer [9, 10]. The task is solved using CFD codes in the software Ansys Fluent.

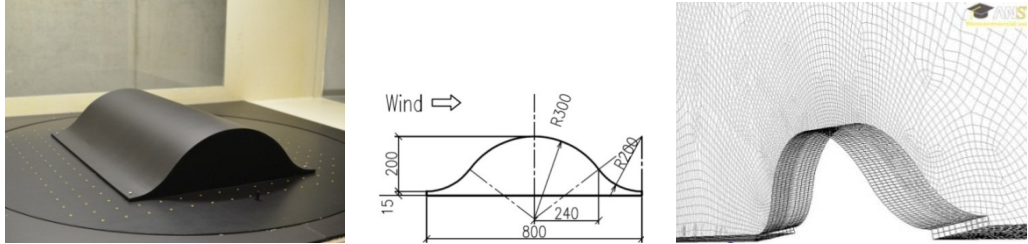


Fig. 1: The model of a hill in the wind tunnel, scheme of the object and in the numerical model

2.1 The description of the numerical model

The solution is performed using Ansys Fluent software. From the viewpoint of numeric modelling, the issue is interesting in terms of flow characteristics. It is flow with a transition from low turbulence at the beginning to fully developed turbulence behind the obstacle which is flown around [11, 12, 13]. The object which is flown around and which is thermally burdened contributes both to the change of momentum and turbulent flow properties [14]. The Transition SST model was chosen for the solution, because it is suitable for showing the significant change of flow field momentum in the transition area of turbulent flow at low Re numbers. The standard k - ω model is an empiric model based on the solution of two transport equations for kinetic energy k (1) and dissipation of this energy ω (2):

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left(\Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \quad (1)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho \omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left(\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \quad (2)$$

where:

k – is kinetic energy [m^2s^{-2}],

ω – is specific dissipation rate [s^{-1}],

G_k – is generation of turbulence kinetic energy due to mean velocity gradients [m^4],

G_ω – is generation of ω [$\text{kg}\cdot\text{m}^{-3}\text{s}^{-2}$],

Γ_k, Γ_ω – is effective diffusivity of k and

Y_k, Y_ω – is dissipation of k and ω due to turbulence [$\text{kg}\cdot\text{m}^{-1}\text{s}^{-3}$], [$\text{kg}\cdot\text{m}^{-3}\text{s}^{-2}$],

S_k, S_ω – user-defined source terms and

D_ω – cross-diffusion term [$\text{kg}\cdot\text{m}^{-3}\text{s}^{-2}$].

Both equations describe the anticipated shear flows well. The Transition SST model, in addition to the original k and ω equations, uses two transport equations: the transport equation for the intermittency γ (3) and the transport equation for the transition momentum thickness Reynolds number Re_{θ^*} (4). By this, the model is able to take into account the difference between high and low turbulence at the interface between the wake behind the barrier and free flow:

$$\frac{\partial(\rho\gamma)}{\partial t} + \frac{\partial(\rho u_j \gamma)}{\partial x_j} = P_{\gamma 1} - E_{\gamma 1} + P_{\gamma 2} - E_{\gamma 2} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\gamma} \right) \frac{\partial \gamma}{\partial x_j} \right] \quad (3)$$

$$\frac{\partial(\rho Re_\alpha)}{\partial t} + \frac{\partial(\rho u_j Re_\alpha)}{\partial x_j} = P_\alpha + \frac{\partial}{\partial x_j} \left[\sigma_\alpha (\mu + \mu_t) \frac{\partial Re_\alpha}{\partial x_j} \right] \quad (4)$$

where:

μ – is dynamic viscosity [$\text{kg} \cdot \text{m}^{-1} \text{s}^{-1}$],

μ_t – is turbulent dynamic viscosity [$\text{kg} \cdot \text{m}^{-1} \text{s}^{-1}$],

$P_{\gamma 1}, P_{\gamma 2}$ —are transition sources, which include: strain rate magnitude and empirical correlation that controls the length of the transition region,

$P_{\gamma 2}, P_{\gamma 2}$ —sources pro the destruction/relaminarization, which include: vorticity magnitude, a critical Re number and also necessary empirical correlation and

P_α – cross-diffusion term [$\text{kg} \cdot \text{m}^{-3} \text{s}^{-2}$].

Near the surface of the thermally burdened object which is flown around, the buoyancy forces play an important role. The Boussinesq model, which describes the transfer of heat by natural convection and buoyancy, is included into the heat transfer equation (see ANSYS Fluent 14.0 Theory Guide). This model adds a source term describing the change in current density into the momentum equation. In such a case, the following applies for the real density of media ρ :

$$(\rho - \rho_0) \cdot g \approx \rho_0 \cdot \beta \cdot (T - T_0) \cdot g \quad (5)$$

where:

ρ_0 – is the constant flow density [$\text{kg} \cdot \text{m}^{-3}$],

g – is gravitational acceleration [$\text{m} \cdot \text{s}^{-2}$],

β – is the thermal expansion coefficient [K^{-1}],

T_0 – is the operating temperature [K] and

T – current temperature [K].

Equation (5) is solved by means of Boussinesq approximation:

$$\rho = \rho_0 \cdot (1 - \beta \cdot \Delta T) \quad (6)$$

If the true density changes are small, as in the described task, this approximate solution can be considered accurate.

2.2 Calculation area

The task is solved in a 3D computation area with the following dimensions $b \times h \times l = 0.1 \times 1.8 \times 8$ m (Fig. 2). The boundary condition (BK) at the input is the velocity inlet, at the output pressure outlet and on the symmetry of the sides because of free passage of the flowing medium. The upper and lower surface of the calculation area is ensured by the BK wall describing the limited space of the wind tunnel. The number of cells is $2 \cdot 10^5$. Flow is considered compressible, specific heat is $c_p = \text{const}$ [$\text{J} \cdot \text{kg}^{-1} \text{K}^{-1}$]. With the current change of temperature the physical properties of air thermal conductivity λ [$\text{W} \cdot \text{m}^{-1} \text{K}^{-1}$] and dynamic viscosity μ [$\text{Pa} \cdot \text{s}$], [$\text{kg} \cdot \text{m}^{-1} \text{s}^{-1}$] are also changed.

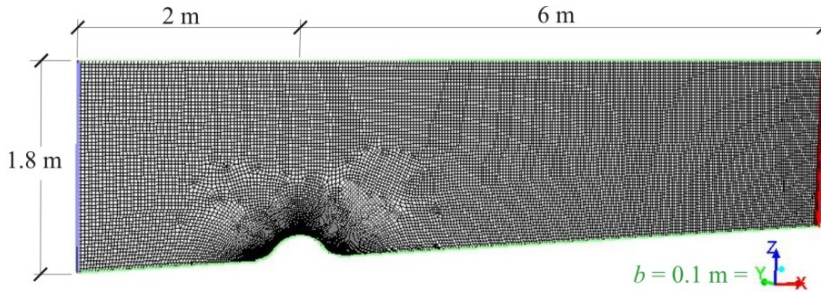


Fig. 2: The calculation area diagram

3 RESULTS AND DISCUSSION

The results for the primary air velocity $v_0 = 0.5 \text{ m}\cdot\text{s}^{-1}$ and $v_0 = 1.5 \text{ m}\cdot\text{s}^{-1}$ are presented. These speeds were chosen because there was an experiment with them in the tunnel and are available measured data. The input air temperature is $T_0 = 20 \text{ }^\circ\text{C}$. In either case, flow around a hill is solved both unheated with a surface temperature of $T = 20 \text{ }^\circ\text{C}$, and heated with a surface temperature of $T = 150 \text{ }^\circ\text{C}$. The courses of the two stated velocities and temperatures are monitored. Fig. 3 and Fig. 4 show the isolines of velocity near the flow around object obtained from the experiment and the numerical calculation.

The four main streams are presented.

- flow with the basic air velocity of $v_0 = 0.5 \text{ m}\cdot\text{s}^{-1}$ at temperatures of the flow around object of $T = 20 \text{ }^\circ\text{C}$ and $T = 150 \text{ }^\circ\text{C}$ (Fig. 3),
- flow with the basic air velocity of $v_0 = 1.5 \text{ m}\cdot\text{s}^{-1}$ at temperatures of the flow around object of $T = 20 \text{ }^\circ\text{C}$ and $T = 150 \text{ }^\circ\text{C}$ (Fig. 4).

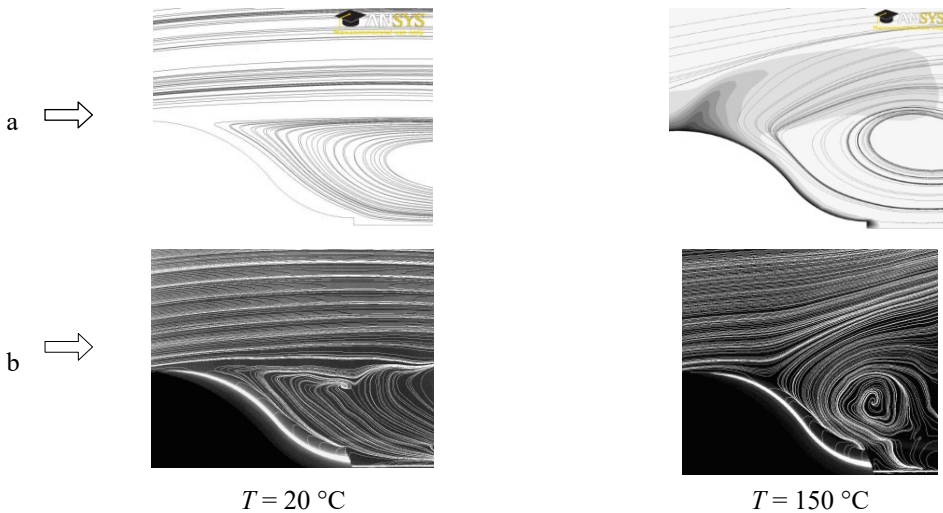


Fig. 3: Velocity isolines near the flow around object for the air velocity of $v_0 = 0.5 \text{ m}\cdot\text{s}^{-1}$
a) numerical calculation; b) experimental data

Figures show that over the axis of the hill, the flow swirl manifests slightly, while in the wake behind the hill, the swirl is significant. In both approaches (experiment and numerical calculation) there are distinct changes in the vortex structures behind the flow around object. The thermal load of the objects results in the lifting of the vortex and it is assumed to be more significant at lower speeds.

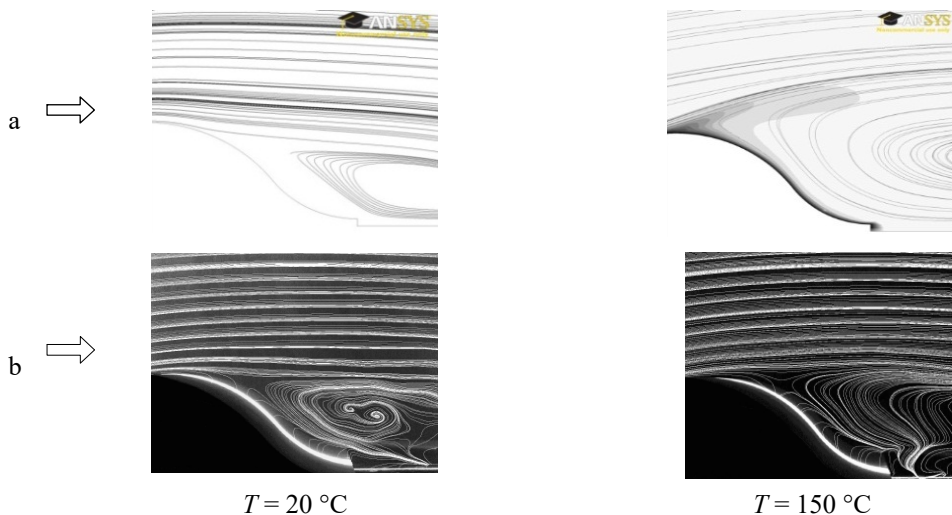


Fig. 4: Velocity isolines near the flow around object for the air velocity of $v_0 = 1.5 \text{ m} \cdot \text{s}^{-1}$
a) numerical calculation ; b) experimental data

Vertical temperature curves for both the basic velocity v_0 in three sections over the hill, where $x = 0 \text{ mm}$ applies for the axis of the hill, are available in Fig. 5. They clearly show a jumbled temperature field near the object up to about 1.5 times the height of the hill.

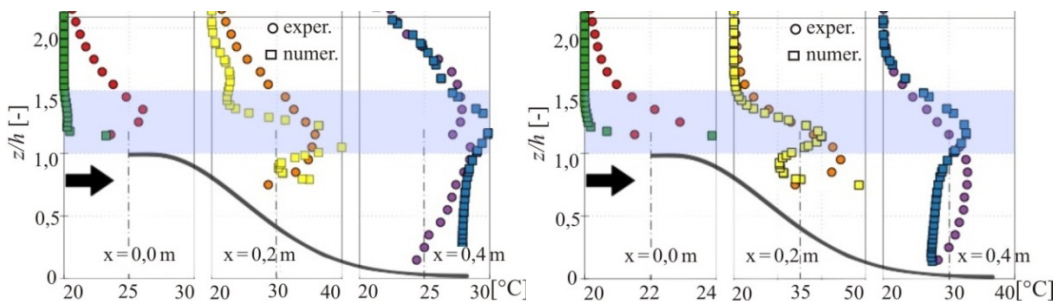


Fig. 5: The vertical temperature profiles; $T = 150 \text{ °C}$; left $v_0 = 0.5 \text{ m} \cdot \text{s}^{-1}$; right $v_0 = 1.5 \text{ m} \cdot \text{s}^{-1}$

4 CONCLUSIONS

This article is focused on describing the changes in selected turbulent variables near a thermally loaded object. The task is solved numerically using the software Ansys Fluent, specifically the Transition SST model. Two cases of a free speed flow were examined with the basic velocity $v_0 = 0.5 \text{ m} \cdot \text{s}^{-1}$ and $v_0 = 1.5 \text{ m} \cdot \text{s}^{-1}$. It is a complex physical task which falls within the area of basic research. The aim was to investigate the possibility of numerical modeling of the problem. The presented results are consistent with the results of experimental measurements with sufficient accuracy. The following works of the authors will focus on testing other numerical models and their evaluation in terms of objectivity of the results and performance solutions.

ACKNOWLEDGMENT

The paper has been supported by the project of “Conceptual development of science and research activities 2016” on the Faculty of Civil Engineering, VŠB-TU Ostrava.

LITERATURE

- [1] KIM H.G., LEE C.M. and LIM H.C. An experimental and numerical study on the flow over two-dimensional hills, *Journal of Wind Engineering and Industrial Aerodynamics*. 1997, LXVI, pp. 17-33. ISSN 0167-6105.
- [2] FERREIRA A.D., LOPES A.M.G., VIEGAS D.X. & SOUSA A.C.M. Experimental and numerical simulation of flow around two-dimensional hills, *Journal of Wind Engineering and Industrial Aerodynamics*. 1995, LIV/LV, pp. 173-181. ISSN 0167-6105.
- [3] MARUYAMA Y., TAMURA T., OKUDA Y. & OHASHI M. LES of turbulent boundary layer for inflow generation using stereo PIV measurement data, *Journal of Wind Engineering and Industrial Aerodynamics*. 2012, CIV-CVI, pp. 379-38. ISSN 0167-6105.
- [4] OHYA Y. & UCHIDA T. Laboratory and numerical studies of the atmospheric stable boundary layers, *Journal of Wind Engineering and Industrial Aerodynamics*. 2008, CIV-CVI, pp. 379-388. ISSN 0167-6105.
- [5] MICHALCOVÁ, V., KUZNĚTSOV, S. & POSPÍŠIL, S. Models of load on buildings from the effects of the flow field. *Transactions of the VŠB - Technical University of Ostrava: Construction Series*. Volume 13, Issue 2, Pages 91-97. ISSN 1804-4824 (Online), DOI: [10.2478/tvsb-2013-0014](https://doi.org/10.2478/tvsb-2013-0014), December 2013.
- [6] MICHALCOVÁ, V., KUZNĚTSOV, S. & POSPÍŠIL, S. Numerical modelling of air flow attributes in a Contractions chamber. *Transactions of the VŠB - Technical University of Ostrava: Construction Series*. Volume 14, Issue 2, Pages. 11-16. ISSN 1804-4824 (Online), DOI: [10.2478/tvsb-2014-0026](https://doi.org/10.2478/tvsb-2014-0026). December 2014.
- [7] MICHALCOVÁ, V., KUZNĚTSOV, S. & POSPÍŠIL, S. Numerical and experimental study of the load of an object due to the effects of a flow field in the atmospheric boundary layer. *Journal of Mathematics and Computers in Simulation*. 2014, VIII, pp. 135-140. ISSN 19980159.
- [8] KUZNĚTSOV, S., POSPÍŠIL, S. at al. Wind-tunnel and computational simulations of the thermally stratified atmospheric boundary layer above a hill. In *Proceedings of the 2nd World Congress on Mechanical, Chemical, and Material Engineering*, MCM'16, Paper No. HTFF 108.
- [9] QU Y., MILLIEZ M., MUSSON-GENON L. & CARISSIMO B. Numerical study of the thermal effects of buildings on low-speed airflow taking into account 3D atmospheric radiation in urban canopy, *Journal of Wind Engineering and Industrial Aerodynamics*. 2012, CIV-CVI, pp. 474-483. ISSN 0167-6105.
- [10] PILLAI S.S. & YOSHIE R. Experimental and numerical studies on convective heat transfer from various urban canopy configurations, *Journal of Wind Engineering and Industrial Aerodynamics*. 2012, CIV-CVI, pp. 447-45. ISSN 0167-6105.
- [11] NAKAYAMA, H., TAKEMI, T. & NAGAI, H. Large-eddy simulation of plume dispersion under various thermally stratified boundary layers, *Adv. Sci. Res.* 2014, XI, pp. 75-81, DOI:10.5194/asr-11-75-2014.
- [12] KOTRASOVÁ, K. & KORMANÍKOVÁ, E. Hydrodynamic analysis of fluid effect in rigid rectangular tank due to harmonic motion, *Key Engineering Materials*. 2015, DCXXXV, pp. 147-150. ISSN: 10139826. DOI: [10.4028/www.scientific.net/KEM.635.147](https://doi.org/10.4028/www.scientific.net/KEM.635.147).
- [13] PIETERSE, J.E. & HARMS, T.M. CFD investigation of the atmospheric boundary layer under different thermal stability conditions, *Journal of Wind Engineering and Industrial Aerodynamics*. 2013, CXXI, pp. 82-97. ISSN 0167-6105.
- [14] OHYA, Y., HASHIMOTO, H. & OZONO, S. A numerical study of a thermally stratified boundary layer under various stable conditions, *Journal of Wind Engineering and Industrial Aerodynamics*. 1997, LXVII-LXVIII, pp. 793-804. ISSN 0167-6105.