10.2478/tvsb-2014-0026

Transactions of the VŠB – Technical University of Ostrava

No. 2, 2014, Vol. 14, Civil Engineering Series

paper #26

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

NUMERICAL MODELLING OF AIR FLOW ATTRIBUTES IN A CONTRACTIONS CHAMBER

Abstract

The article describes air flow turbulent attributes in the enclosed chamber of a rectangular cross-section contraction for the purpose of confirming its optimal shape. The task is solved numerically using Ansys Fluent software. Right models were selected based on the evaluated results at a contraction's outlet which were compared to the physics experiment.

Keywords

Aerodynamics, wind tunnel contraction, CFD.

1 INTRODUCTION

The crucial characteristics of a wind tunnel are the flow quality inside the test chamber and the overall performance. Three main criteria that are commonly used to define them are: maximal achievable velocity, flow uniformity and the minimum turbulence level. Therefore, in general, the aim of the contraction design is to get a controlled flow in the test chamber, achieving necessary flow performance and quality parameters [1, 2, 3 and 4].

The aim of this paper is the characterization of a flow field inside the contraction chamber in order to meet required criteria and the confirmation of the optimal contraction shape. The task has been solved at FAST VŠB – TU using Ansys Fluent software in collaboration with the experimental research CET in Telč. Input parameters values are set in accordance with the requirements of the aerodynamic tunnel department under the earlier mutual collaboration [5].

2 PHYSICAL EXPERIMENT

The physical modelling was carried out in the climatic wind tunnel laboratory in the “Centrum Excelence Telč” (CET) of the Institute of Theoretical and Applied Mechanics of the Academy of Science of the Czech Republic. The main feature of the planned contraction of the Wind Tunnel CET will be a variable open-jet test section for a full-scale and model-small-scale testing with a cross-sectional area according to Fig. 1. Respective scan points are 20 mm away from one another both in horizontal and vertical direction. The variation in size is achieved by a rectangular contraction with two interchangeable outlet sections resulting in overall contraction ratios of 2.02 and test section velocities from 3 to 55 m/s. The main design criteria for the contraction shapes were the avoidance of re-laminarization and boundary layer separation as well as the velocity nonuniformity in the cross-sectional outlet area lower than 0.3%. A short contraction length is just as important a requirement.

[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, e-mail: vladimira.michalcova@vsb.cz.

[2] Prof. Ing. Sergej Kuznetsov, Ph.D., Institute of Theoretical and Applied Mechanics of the Czech Academy of Sciences, Centre of Excelence Telč, Batelovská 485, 588 56 Telč, Czech Republic, e-mail: kuznetsov@itam.cas.cz.

[3] Doc. Ing. Stanislav Pospíšil, Ph.D., Institute of Theoretical and Applied Mechanics of the Czech Academy of Sciences, Prosecká 809/76, 190 00 Prague 9, Czech Republic, e-mail: pospisil@itam.cas.cz.

The results were evaluated at a distance of 0.36 m downwind of the contraction outlet. Velocity distribution of the within monitored cross-section is defined by a non-dimensional speed coefficient given by the ratio of the actual velocity to reference velocity.

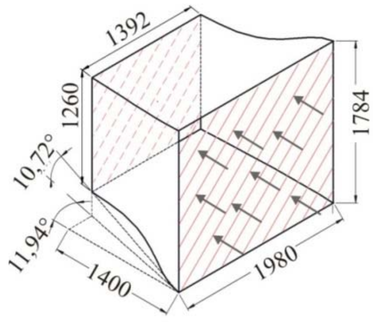

Fig. 1: Contraction diagram [mm]

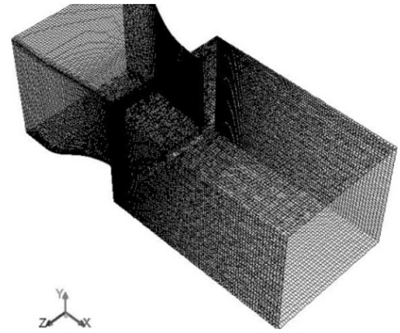

Fig. 2: Grid for numeric computation

3 NUMERICAL MODELLING

Suitable numerical models to characterize the flow field in the enclosed contraction chamber were chosen on the basis of experimental results at the contraction outlet from the turbulence velocity and intensity distribution's point of view [6]. These were examined, evaluated and compared with the physical experiment. Namely it is RSM model, which solves turbulences using anisotropically direct computation/calculation of Reynolds stress. In terms of RANS models based on modelling isotropic turbulence, it's Realizable k-ε a SST k-ω. models that demonstrated a satisfying result. The tested numerical models can be subsequently used also for other task types [7; 8; 9].

The computational dimension is 4.9 m long. This encompasses an inlet chamber (0.5 m), a nozzle (1.4 m), and a 3 meter long contraction downwind area (test section), as shown in Fig. 2. The computational grid uses $9 \cdot 10^5$ hex cells. There is a distinct concentration in the vicinity of the contact of both contraction outlet areas for better data transmission at the boundary line. The steady rise of cell sizes that are farther away from the boundary of the contraction for the purpose of the correct computation at the wall is an important influence.

Consistent types of boundary conditions were set for all computations at the inlet - it is a so-called inlet, and at the outlet velocity - it is a pressure-outlet condition. The boundary conditions on all the sides and lower and top areas of the computational area were defined by wall so, that they correspond to the circumscribed space [10].

The models used make it possible to solve the task in a stationary way, but due to the wrong convergence, the computations had to be done in a non-stationary way. For the RSM model the time step was set to $1 \cdot 10^{-3}$ seconds and $5.1 \cdot 10^3$ iterations took place.

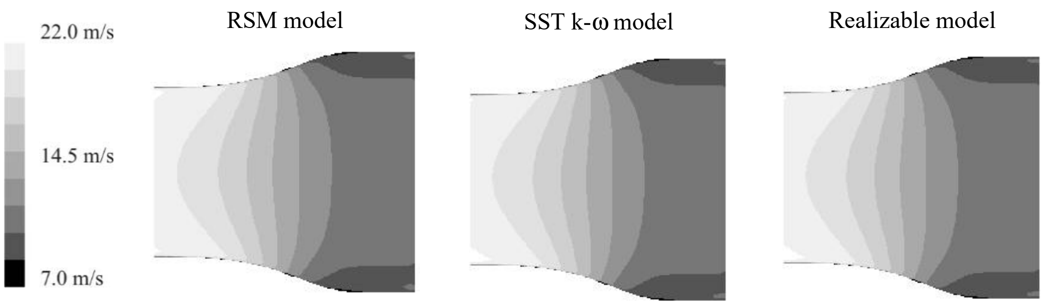

Fig. 3: Velocity field in a vertical longitudinal section in the contraction

4 RESULTS

The velocity field of the outlet flow rate are evident in Fig. 3. Distributions velocity fields of the downwind area of a nozzle (distance is 0.36 m), which are obtained by wind tunnel tests, and numerical computations are shown in Fig. 4.

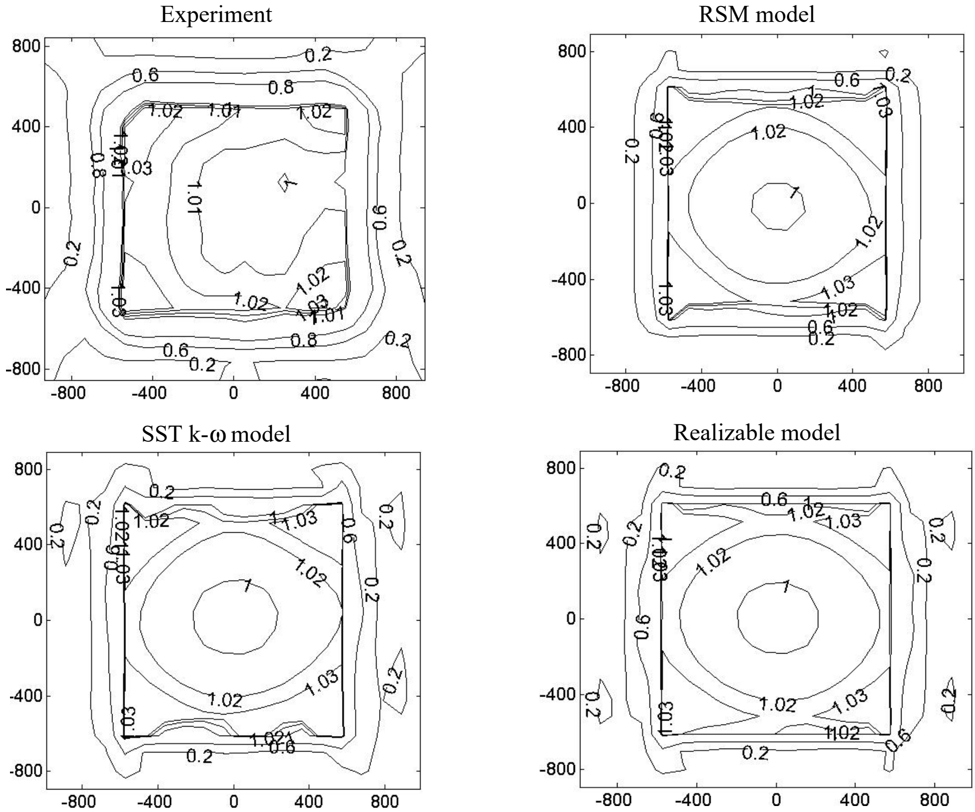

Fig. 4: Velocity coefficient isoline 0.36 m after contraction

Graphs in Fig. 5 and Fig. 6 describe the development of the velocity and intensity alongside the longitudinal axis of the observed area.

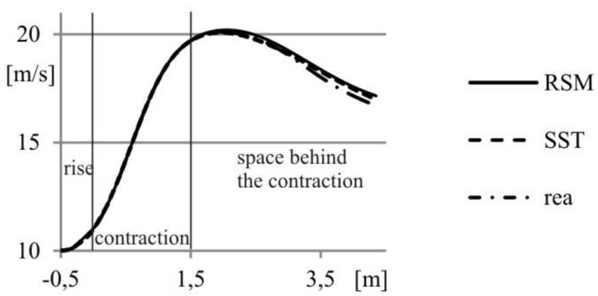

Fig. 5: The velocity development alongside the longitudinal axis of the observed area

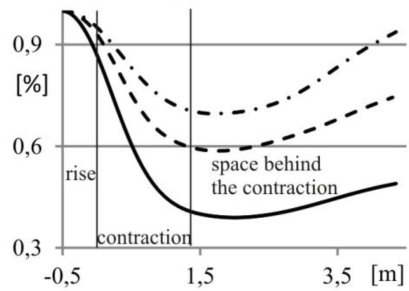

Fig. 6: The intensity development alongside the longitudinal axis of the observed area

The velocity profile in the three indicated cross sections is shown in Fig. 7. The turbulence intensity distribution of the contraction flow rate is evident in Fig. 8.

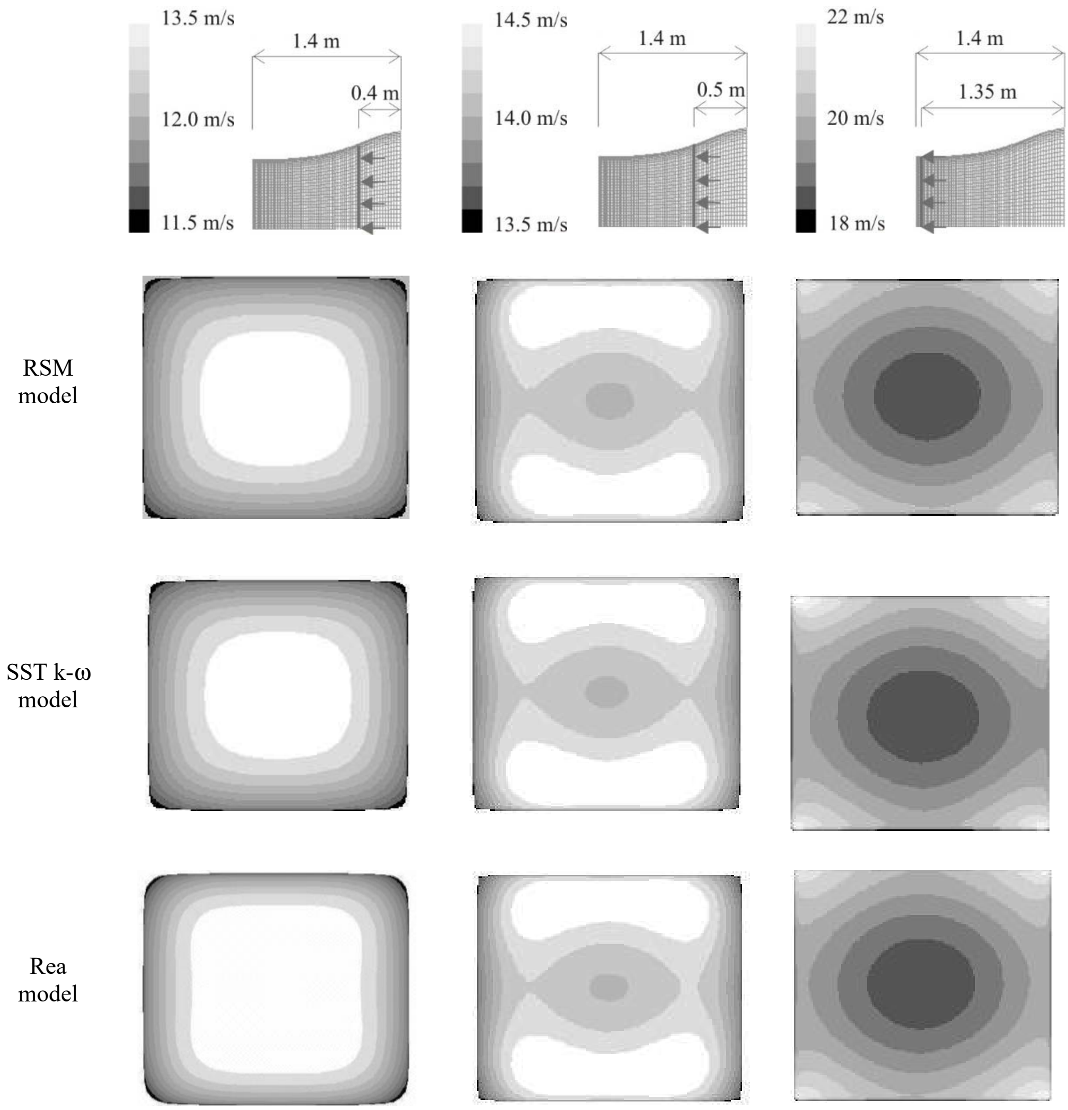

Fig. 7: Cross-section velocity fields in the contraction

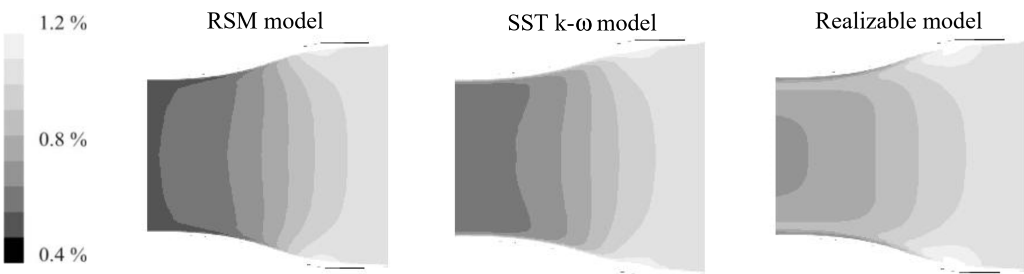

Fig. 8: Turbulence intensity in a vertical longitudinal section in the contraction

In Fig. 9 there are turbulence intensity isolines in cross sections at the distance of 0.36 m after the contraction outlet determined by the physical experiment and numerical modelling.

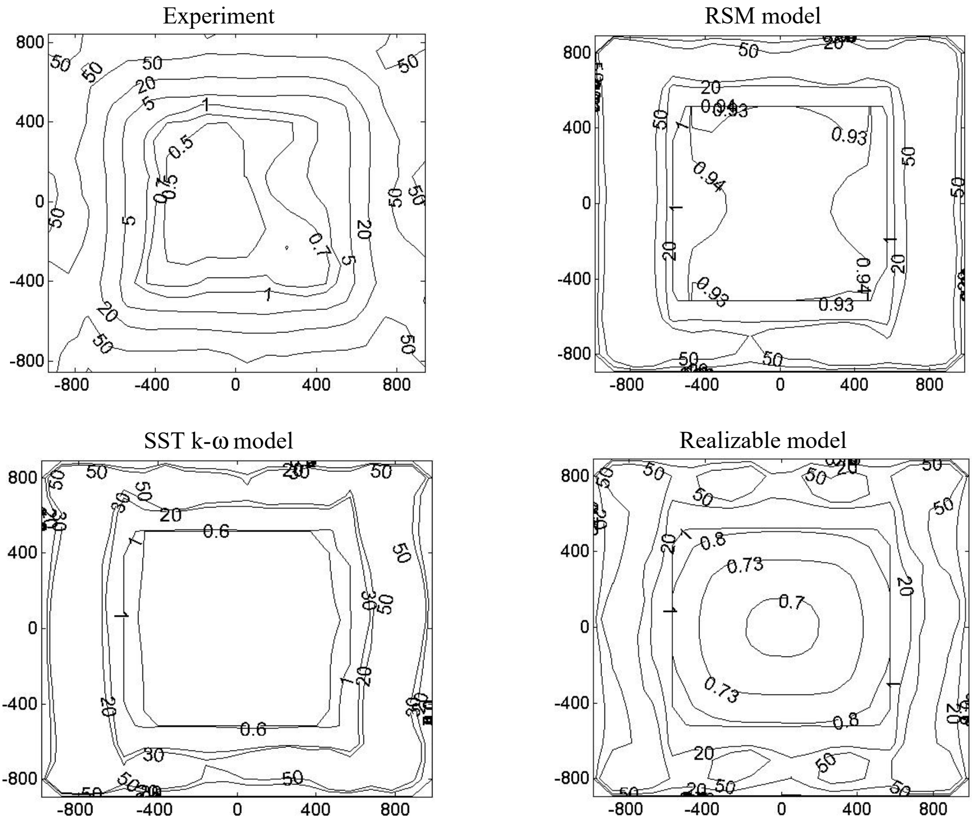

Fig. 9: Turbulence intensity isoline 0.36 m after contraction from numerical modelling [%]

4 CONCLUSIONS

The desired acceleration of a flow field was confirmed by both the mentioned-above calculations and experiment. The numerical models describe a field velocity profile with very satisfactory results. There are almost identical flow field profiles of all the models in the observed area of the contraction and almost twice as high outlet velocity compared with input parameters is evident. The results have also shown a slight increase of velocity near the peripheral walls of the contraction (Fig. 3 and Fig .4). The flow field pattern is intact or unaltered within the distance of 0.4 m from the area entry; the first acceleration near the walls became evident in 0.5 m. Near the contraction outlet (1.36 m), the acceleration of velocity by the walls is evident, approx. 18 %. The velocity values of the whole computation area are identical regardless of the computing method used; the slight distinction becomes apparent 1.5 m after the contraction outlet (Fig. 5). The demand of the turbulence intensity reduction was confirmed during its problematic description (Fig. 8 and Fig. 9). The differences of turbulence values were, nevertheless, found at the contraction outlet and especially alongside the longitudinal axis of the observed area (Fig. 6). More thorough investigations of the turbulence intensity along with a planned physical measurement of a flow field inside the contraction in yet a greater distance behind the nozzle outlet are going to be a topic of the follow-up research, on the basis of which it will be possible to determine an optimal model which is to serve to a more detailed characterization of the modelled process and an optimal contraction shape in terms of as homogeneous a velocity field as possible as well as the acquisition of a minimal outlet turbulence, especially in an anticipated spot of the model.

ACKNOWLEDGMENT

This contribution is the result of the research supported by the conceptual development of science, research and innovation for 2014 allocated to the VŠB-Technical University of Ostrava by the Ministry of Education, Youth and Sports of the Czech Republic and was supported by the Grant Agency of the Czech Republic under the project GAČR 14-12892S of the Centre Excellence Telč of the Institute of Theoretical and Applied Mechanics.

REFERENCES

[1] Sargison, J. E., Walker, G. J. & Rosi, R. Design and calibration of a wind tunnel with a two dimensional contraction. In: *Proceeding of the 15th Australian Fluid Mechanic Conference*, Sydney, Australia, 2004, pp. 143-146. ISBN: 1-864-87695-6.

[2] Fang, F. M., Chen, J. C. & Hong, Y. T. Experimental and analytical evaluation of flow in a square-to square wind tunnel contraction. *Journal of Wind Engineering and Industrial Aerodynamics*. 2001, LXXXIX. Nr. 1, pp. 247-262. ISSN 0167-6105.

[3] Fang, F. M. A design method for contractions with square end sections. *Journal of Fluids Engineering*. 1997, CXIX. Nr. 2, pp. 454-458. ISSN 1555-1415.

[4] Wolf, T. Design of a variable contraction for a full-scale automotive wind tunnel. *Journal of Wind Engineering and Industrial Aerodynamics*. 1995, LVI. Nr. 1, pp. 1-21. ISSN 0167-6105. DOI: 10.1016/0167-6105(94)00010-B.

[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* [online]. Warsaw, Poland: Versita, 2013, Vol. 13, Issue 2, pp. 91-97 (7 pp). ISSN 1804-4824 (Online); ISSN 1213-1962 (Print). DOI: 10.2478/tvsb-2013-0014.

[6] Michalcová, V., Kuznětsov, S., Pospíšil, S. & Brožovský, J. Numerical and Experimental Investigations of Air Flow Turbulence Characteristics in the Wind Tunnel. *Applied Mechanics and Materials*, 2014, Vol. 617, pp. 275-279. DOI:10.4028/www.scientific.net/AMM.617.275.

[7] Lausova, L. & Skotnicova, I. Analysis of experimental measurements and numerical simulations of a heat field in the light weight building structure. *Advaced Materials Research*, 2014, Vol. 969, pp. 33-38. DOI:10.4028/www.scientific.net/AMR.969.33.

[8] Kormaniková, E. & Kotrasová, K. Design of a variable Strength optimal design of the fiber reinforced composite on microscopic level, In *Proceeding of the 12th International Multidisciplinary Scientific GeoConference*, Varna, Bulgaria, 2012, pp. 499-506. ISSN 1314-2704. DOI: 10.5593/sgem2012/s12.v3009.

[9] Kaveh, G., Mohammad, R. S. & Manshadi, M.D. Experimental and analytical evaluation of flow in a square-to square wind tunnel contraction. *Aerospace Science and Technology*. 2011, XV. Nr. 2, pp. 137-147. ISSN 1270-9638. DOI: 10.1016/j.ast.2010.06.009.

[10] Kocich, R., Bojko, M., Macháčková, A. & Klečková, Z. Numerical analysis of the tubular heat exchanger designed for co-generating units on the basis of microturbines. *Journal of Heat and Mass Transfer*. 2012, LV. Nr. 19/20, pp. 5336-5342. ISSN 00179310.

Reviewers:

Doc. Ing. Oľga Hubová, PhD., Department of Structural Mechanics, Faculty of Civil Engineering, Slovak University of Technology in Bratislava, Slovakia.

Doc. Ing. Jiří Kala, Ph.D., Institute of Structural Mechanics, Faculty of Civil Engineering, Brno University of Technology, Czech Republic.