Numerical Optimization of Flow Stabilizer in a Channel with Rapid Change of Flow Direction

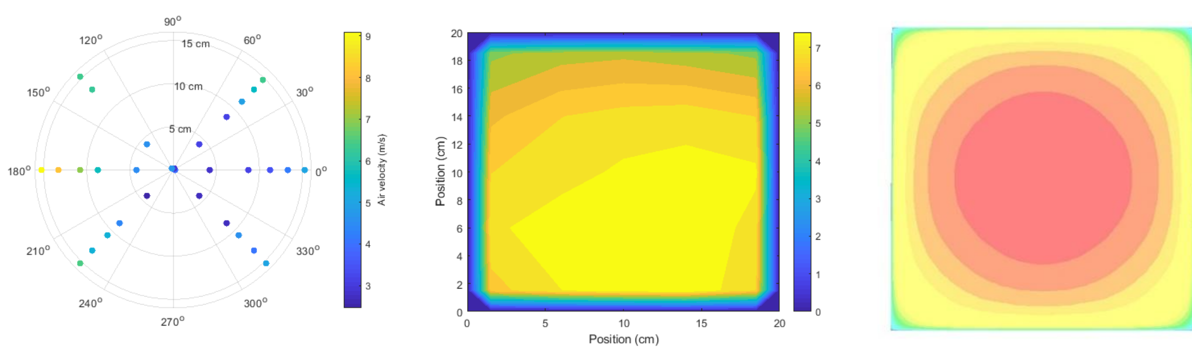
Michał Wasik[1, a)] Łukasz Cieślikiewicz[1, b)] Piotr Łapka[1, c)] and Michał Kubiś[1, d)]

[1]Institute of Heat Engineering, Faculty of Power and Aeronautical Engineering, Warsaw University of Technology, Nowowiejska St. 21/25, 00-665 Warsaw, Poland.

[a)] Corresponding author: michal.wasik@itc.pw.edu.pl
[b)] lukasz.cieslikiewicz@itc.pw.edu.pl
[c)] plapka@itc.pw.edu.pl
[d)] michal.kubis@itc.pw.edu.pl

Abstract. The stability and alignment of velocity profile in experimental stands is very important during validation of numerical models. In case of distorted flow profile reconstruction of its variation is difficult and might result in large error in numerical calculations. Therefore, uniform and developed flow profiles are required. In real flows in channels stability of velocity profile may be disturbed by elements of the experimental system like fans and throttles as well as rapid changes of flow direction. In the last case homogeneity of flow profile may be preserved by applying flow guides. However, applicability of this method is limited to ducts where inlet profile is aligned. In case of some perturbated inlet velocity profiles their disruption may be transferred to the outlet through flow guides.

In this paper analyzed flow precluded use of flow guides because inlet flow profile had annular character which is presented in Fig. 1a and which was caused by axial fan. Additionally, the duct cross-section changed from round to square one – see Fig. 2. These resulted in outlet velocity profile with high irregularity as presented in Fig. 1b which is far from required velocity profile shown in Fig. 1c.



(a) (b) (c)

FIGURE 1. Velocity profile: (a) measured in inlet round duct, (b) measured in outlet square duct and (c) required during validation of numerical models.

To obtain velocity profile in the experimental stand more similar to the required one shown in Fig. 1c several modifications of the stand were proposed. At first, to make velocity profile more symmetrical semi-cylindrical turbulizator of 90 mm diameter was used. This action raised maximum velocity to the center of the duct. Additionally, a grid of small ducts with cross-section dimensions of 25×30 mm and length of 300 mm were used as flow stabilizers. Diameter of the turbulizator as well as position of each element and length of stabilizers were optimized applying the ANSYS CFD environment. Developed geometry and mesh used in numerical calculations are shown in Fig. 2.

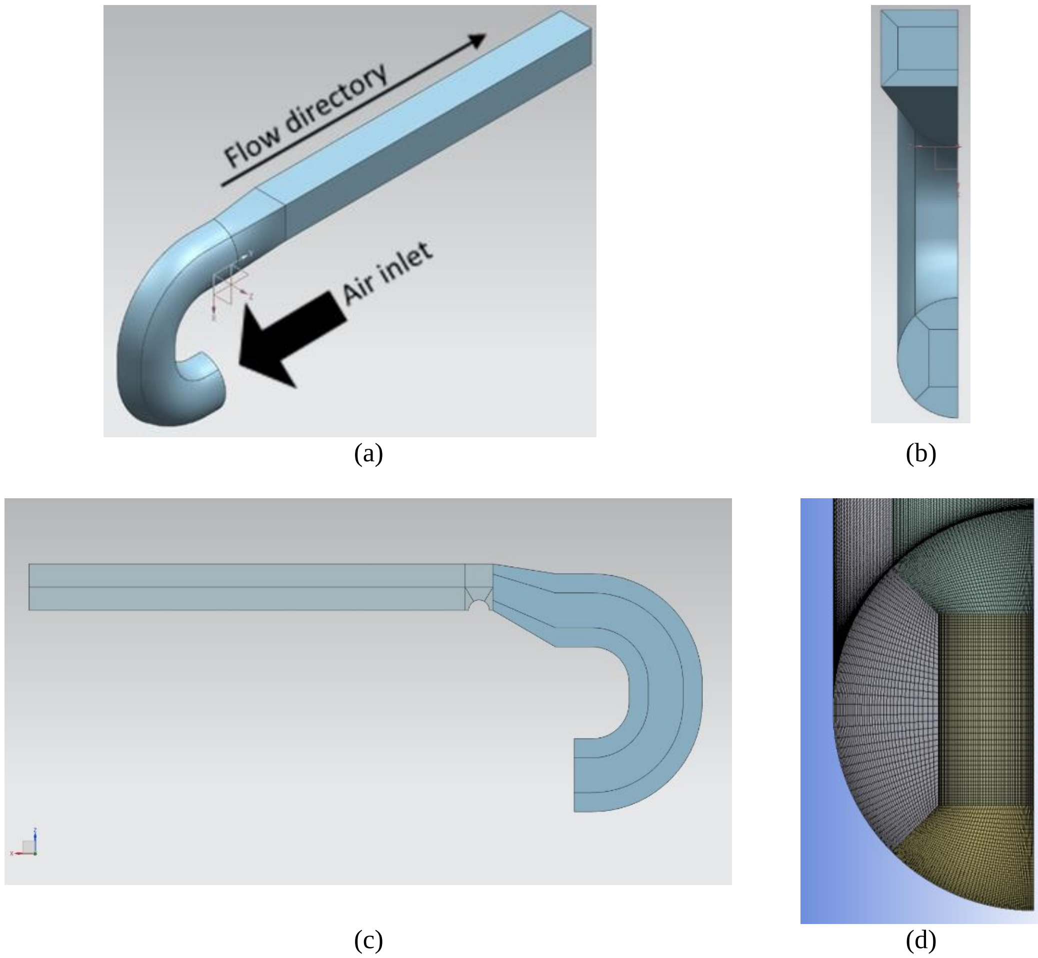


FIGURE 2. Half of considered geometry: (a) isometric view, (b) front view, (c) side view with turbulizer and (d) generated structural mesh.

Numerical simulations were conducted on geometry prepared in the Simens NX and by applying orthogonal mesh generated in the ANSYS Meshing tool. The channel was symmetrical hence only its half was taken into consideration. Due to complex computational domain generated mesh had over 7 million of elements. Despite complicated shape structural mesh was obtained by splitting the duct into four zones and sweeping the mesh from air inlet to outlet – see Fig. 2. The grid was refined closed to walls of the channel. Maximum aspect ratio, minimum orthogonal quality and maximum skewness were equal to 32, 0.43 and 0.75, respectively. The simulations were carried out in the ANSYS Fluent software. The turbulent flow of incompressible air was assumed in the channel and the k-ω SST turbulence model was applied in the numerical model.

The simulations were split up into three cases, i.e., references one (without stabilization), with semi-cylindrical turbulizator and with flow stabilizer. In the second case influence of diameter of tubulizator was investigated while in the third one the effect of length of the grid of small ducts was determined. The goal of optimization was to reach the most homogenous velocity profile simultaneously with the pressure drop in the duct as small as possible. High value of flow resistance yields low energy efficiency of experimental stand and decreases velocity range obtained in the stand.

ACKNOWLEDGMENTS

This work was supported by the statutory funds of Faculty of Power and Aeronautical Engineering of Warsaw University of Technology.