THE COMPARISON OF NUMERICAL AND EXPERIMENTAL INVESTIGATION OF FLOW INSIDE REVERSING CHAMBER.

Robert Kłosowiak[1], Jarosław Bartoszewicz[1]
[1]*Chair of Thermal Engineering , Poznan University of Technology, Poznan, Poland*

Abstract The papers presents the results of numerical and experimental studies of jet flowing inside axisymmetric reverse chamber. The test results will be present in the form of distributions of components of axial and radial velocities and intensity of turbulence. The investigated cases of the phenomena include the flow with the interaction of the walls, and therefore have characteristics of free-and confined flows [5]. The numerical results will be compared with the experimental results. The numerical and experimental results will be present in the form of normalized radial distributions for different velocity of the outlet of the inner tube against the bottom of the reverse chamber. Research is intended to demonstrate the similarities and differences between free and confined jets.

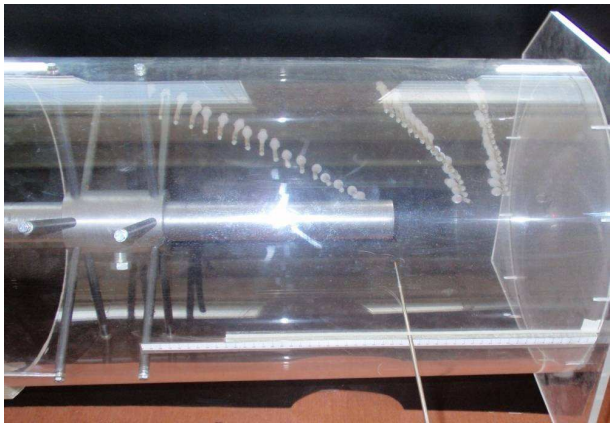

Figure 1. Schema of reversing chamber

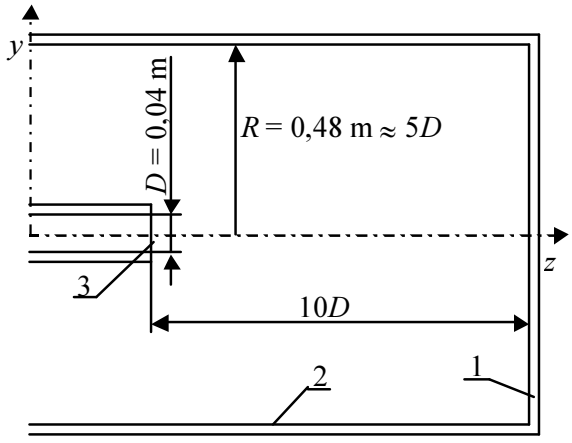

Figure 2. Schematic model reversing chamber 1 - impinging wall, 2 – runoff wall, 3 - outlet pipe

NUMERICAL ANALYSIS

For numerical simulation of turbulent flow in cylindrical vessel the Ansys program will be used. For simulation of turbulent flow CFX will use the standard high-Reynolds-number form of the k- model or Reynolds Stresses Model.

EQUIPMENT

The measurement of velocity and its fluctuation will be carried out by the CTA anemometer. The standard X probe TSI-1241 will use to measure the two components of velocity. The position of the jet axis will indicate by the laser beam. Probes will be connect to the TSI-1050 constant temperature anemometer bridge. The auto trigger option will be select. Subsequently, the recorded signal will be processes and analyses by means of the same program.

RESULTS

The results of numerical simulations are presented for z/D=(0-9) measurement cross sections, so that it can be read characteristic values for the velocity distributions. The results of numerical simulations are very close to the experimental results. Only in some cases there are clear discrepancies in the presented graphs. This proves the correctness of the performed experimental studies and numerical simulations.

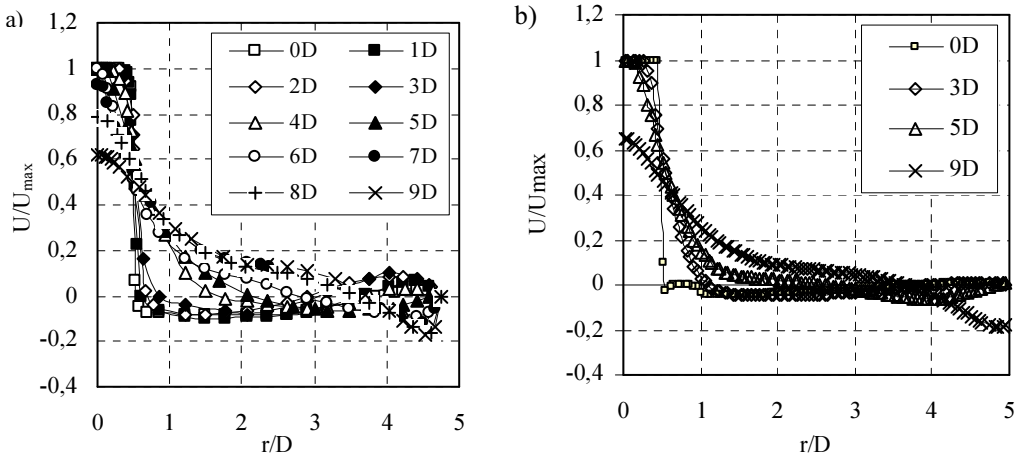

Figure 3. The axial distribution of velocity for 10[m/s], a) experimental results, b) numerical calculation

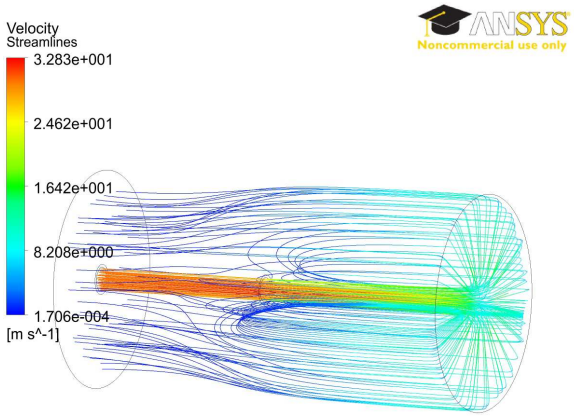

Figure 4. Results of numerical analysis flow inside reversing chamber for 30 m/s

References

[1] Cho Y. et al. Theoretical and experimental investigation of wall confluent jets ventilation and comparison with wall displacement ventilation, Building and Environment, 43, 1091–1100, 2008.

[2] Koseoglu M.F., Baskaya S., Experimental and numerical investigation of natural convection effects on confined impinging jet heat transfer, International Journal of Heat and Mass Transfer, 52 1326–1336, 2009.

[3] Launder, B. E., and Spalding, D.B., 1972, "Mathematical models of turbulence", Academic Press.

[4] Launder, B. E., and Spalding, D.B., 1974, "The numerical computation of turbulent flows", Comp. Meth. in Appl. Mech. & Eng., Vol. 3, pp. 269.

[5] Bartoszewicz J., Kłosowiak R., Bogusławski L., The analysis of the flow structure in a jet at variable geometry of the reverse chamber, International Journal of Heat and Mass Transfer Volume 55, Issues 11–12, Pages 3239–3245, May 2012