NUMERICAL SIMULATION OF A FLOWMETER FOR NATURAL VENTILATION

Sezin Eren Ozcan, Turkey
Supervisors: M.L. Riethmuller & Ph. Planquart

Natural ventilation is a sustainable and energy saving solution for residential buildings, offices, animal stables, greenhouses and even food storage systems. Effective and widespread use of natural ventilation requires better control of air flux through these buildings. Unfortunately, existing measurement techniques are unable to determine total air movement through openings due to bi-directional and non-uniform flow behaviour.

Tracing of air velocities by temperature was already applied at hot wire anemometer at micro-scale, and at some other studies at more higher scales. Temperature measurements are simple, fast, economical, and robust. Therefore, a temperature-based measurement technique was suggested for estimation of total ventilation rate through an opening. This method involves a heat source in the middle of an air inlet where dissipation of heat from that heat source is measured and related to ventilation rate of air through that opening. Preliminary experiments has already been performed at Catholic University of Leuven using infrared camera as a reference technique to obtain 2D temperature maps.

The objective of research project at von Karman Institute involved numerical simulation of experimental results obtained by 2D infrared thermography, and implementation of system parameters and geometry to observe the effect of different conditions on flow field.

Simulation of natural convection was difficult, since many factors affect the final results enormously. Therefore, comprehensive study of simplified and detailed heater geometry was performed at 2D and 3D, partial and full geometry simulations. Partial simulations were performed in a wind tunnel structure with uniform velocity which is equal to total ventilation rate through the opening. Simplified heater was applied at 2D simulations, while detailed heater geometry was tested at 3D partial simulations. At full simulations, simplified and detailed heater was investigated.

Partial simulations of heater in a wind tunnel showed that simplified heater geometry resulted different flow and temperature profiles after the heater. Analysing thermal wake at a certain vertical distance away from the simplified heater, maximum temperature was decreasing with increased velocity; while a reverse trend was observed with detailed heater and experiments. Partial 3D simulations with detailed heater indicated it is possible to obtain similar profiles due to thermal buoyancy. When a uniform velocity profile was introduced, air rises up easier due to lower air velocities around the heater.

3D full scale simulations with simplified heater were useful to compare turbulence models and discretisation schemes at a reduced number of cells. 2nd order, up-wind disretisation scheme and k-e standard turbulence modeling performed better in this condition. Besides, boundary cell adaptation around the heater has a profound effect when compared to experiments.

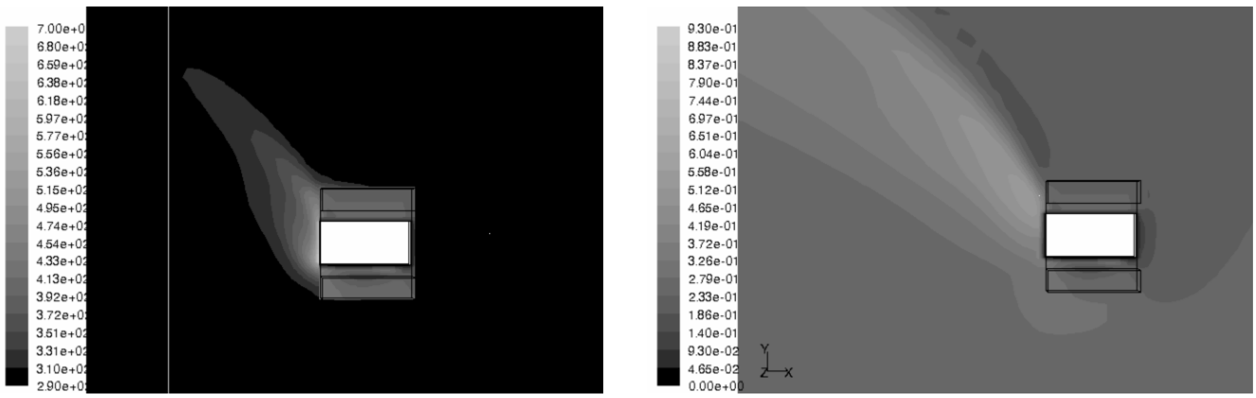

Figure 1: Temperature and velocity contours around heating element