World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:6, No:7, 2012

An Enhance of the Energy Effectiveness of the Convectors Used for Heating or Cooling

K. Fraňa, M. Müller, F. Lemfeld

Abstract**—**The objective of this paper is to present a research study of the convectors that are used for heating or cooling of the living room or industrial halls. The key points are experimental measurement and comprehensive numerical simulation of the flow coming throughout the part of the convector such as heat exchanger, input from the fan etc.. From the obtained results, the components of the convector are optimized in sense to increase thermal power efficiency due to improvement of heat convection or reduction of air drag friction. Both optimized aspects are leading to the more effective service conditions and to energy saving. The significant part of the convector research is a design of the unique measurement laboratory and adopting measure techniques. The new laboratory provides possibility to measure thermal power efficiency and other relevant parameters under specific service conditions of the convectors.

Keywords**—**Heating, cooling, floor convectors, large eddy simulation, measurement techniques.

Open Science Index, Mechanical and Mechatronics Engineering Vol:6, No:7, 2012 publications.waset.org/5154.pdf

I. INTRODUCTION

THE objective of the paper is to introduce techniques and methods including new constructed labs used for measurement of the heating/cooling power of the indoor convectors. Convectors are a facility located directly in the floor or at the vertical walls and widely used for heating or cooling of the air inside the living rooms or halls. The quality of the room ventilation and determination of the appropriate temperature stratification is popular and widely investigated topic in the house architecture. The energy efficiency and thermal comfort living has been a subject of the research for instance in [1]. The basic idea is to find a balance between thermal comfort condition and energy consumption in the living room in respect to the type of the heating system. In case of the convectors, principally, there are two different convector type based on the physical point of view. First one is a convector that is adopting a natural convection. Advantageous of such facilities is relatively simple construction, less materials and no fans required for operation. Disadvantageous are higher demand on the air flow optimization in order to reduce aerodynamics drag and so improve the thermal efficiency. Despite of the huge effort to improve flow conditions and the thermal efficiency, the final thermal power stays relatively low. The second type of the convector uses a fan for air circulation so called force convection. It is well known that this technical solution is much more effective. In generally forced convection provides better parameters for heat convection and reduces the sensitivity on the aerodynamics drag inside of the heat exchanger. On other hand fans are mostly a source of the noise pollution, increase the energy consumption, need the electricity for power and increase the price of the final products. The basic idea of our research is to optimize the heat and mass transfer inside of the convectors of the both type and so increase the heat efficiency or cooling effectiveness.

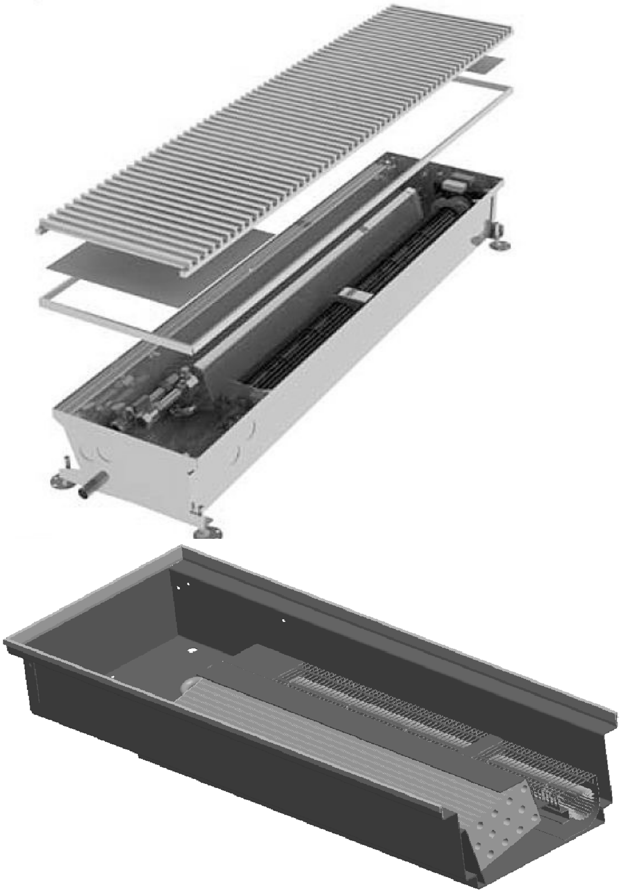

Fig. 1 The convector located above the floor (above) and a view into the convector located mostly directly in the floor (below).

Because of the low temperature differences appeared in the cooling then in the heating regime, the cooling regime is more relevant for optimization of the convectors. Optimized convectors are provided by partner LICON Heat s.r.o. that belongs to the famous company produced such facilities in the Czech Republic.

Fig. 1 shows a measured convector mostly located above the floor and the figure at right side shows floor convector and its components.

In order to optimize an existing convectors or even to design a new concept, the properly measurement of the heating and cooling power efficiency is required.

K.Fraňa is with the Technical university of Liberec, Department of Power Engineering Equipment, 461 17, Liberec, Czech Republic (phone: +420 48 535 3436; fax: +420 48 535 3434; e-mail: karel.frana@seznam.cz).

M.Müller is with the Technical university of Liberec, Department of Power Engineering Equipment, 461 17, Liberec, Czech Republic (phone: +420 48 535 3428; fax: +420 48 535 3434; e-mail: milos.muller@tul.cz).

F.Lemfeld is with the Technical university of Liberec, Department of Power Engineering Equipment, 461 17, Liberec, Czech Republic (phone: +420 48 535 3436; fax: +420 48 535 3434).

Besides that, the velocity profiles in the inlet or outlet of the convectors will be measured, impact of the exhausted air on the flow inside of the controlled room will be simulated and flow structures will be visualized using appropriate visualization techniques. Such results help to identified important processes and to determine their effect on the thermal power efficiency from the global point of view. In previous publications a lot of paper dealt with an effect of the specific conditions (wall, roughness of walls, locations of the heater panel, emissivity of the materials etc.) on the heating effectiveness and intensity of the heating system. This subject has been investigated experimentally and numerically and the appropriate combination of the parameters has been found which leading significantly to increase of the heat transfer and simultaneously heat output from the radiator to the room [2]. In our paper, we followed the same idea how to increase the effectiveness of the heating/cooling convectors and simultaneously to increase the efficiency of the heating or cooling, and therefore, reduce the energy consumption.

The paper is organized as follows: section II reveals lab design for the experimental investigation and measurement techniques preferred for experimental investigations. The section III contains important results of the numerical simulation and studies of the flow behavior inside of the heat exchanger located inside of the convectors. Section VI summarizes significant results and consequences.

II. LABORATORY AND MEASUREMENT TECHNIQUES

The laboratory is composed from the controlled room bounded by the thermal isolated artificial walls. The controlled room is opened so that the top and one side of the room is opened the rest is covered by walls. One wall is cooled representing an effect of the cooled window. Experimental setup used for the measurement is shown in Fig. 2. For the measurement of the heating power, the convector no. 5 is connected to the electrically heated boiler no.2. In order to achieve a stable temperature output the boiler is supplied by the autotransformer no.1. This allows regulating the electric input power of the boiler according to the output heating power of the convector. In cooling regime the convector no. 5 is connected to the cool water reservoir no.4, which is supplied by chillier no.3. As the chillier has only switch of/on regulation, the cool water reservoir has a separate circuit which is heated by separate electric heater regulated via autotransformer. The temperature difference on the inlet and the outlet is measured via three thermocouples (each). Two thermocouples in each group are used for the measurement. The taken temperature is the closer value to the third thermocouple value. The mass flow is measured using induction flow meter no.6 with error under 1.25% in range from 0.1 to 10 l/min. The thermocouples and the flow meter are connected to the computer via CompactRio data acquisition system and monitored in real time by LabView software.

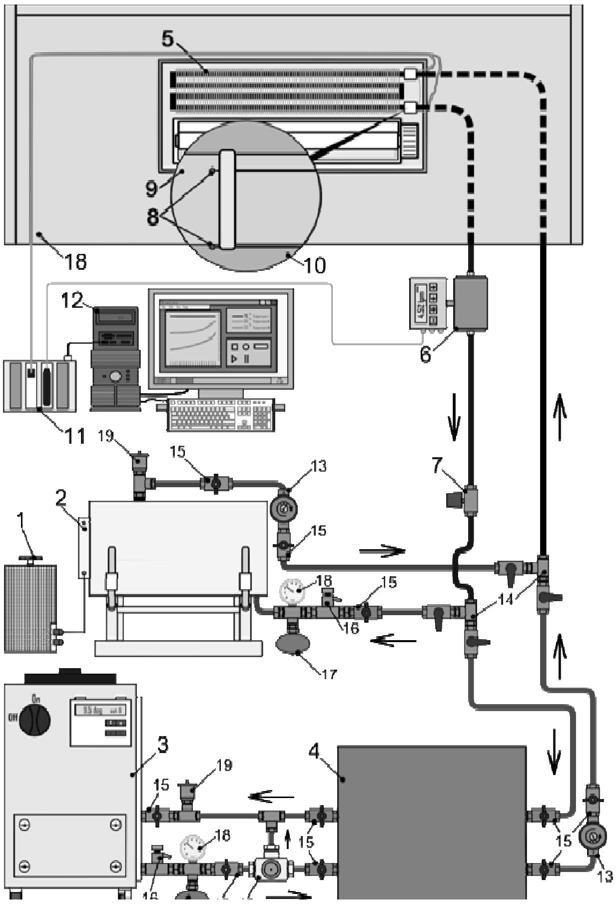

Fig. 2 Sketch of the laboratory

Fig. 3 shows a cooling wall that has been installed because of a simulation of the effect of the cooled window on the temperature stratification above the convector. Logically, it can be assumed that the temperature stratification in the some part of the controlled space can affect a power heat efficiency of the convectors so this effect has to be taken into account for experiment as well. From the measurement of the temperature at inflow and outflow and flow rate at the side of the supplied water, the heat power intensity of the convector can be determined calculating an equation expressed as follows

$$\dot{Q} = \dot{m} C_p \, \Delta t \tag{1}$$

where $\dot{m}$ is a flow rate in kg/s, C_p is a heat capacity of the water for a constant pressure and Δt is a temperature difference.

Fig. 3 Cooling wall and the measured convectors situated in the floor

Values such as flow rate or temperatures must be measured with an appropriate accuracy in order to reach realistic results. Notes, during the measurement, no water condensation is assumed.

In some particular case it is helpful to look at a snapshot of flow and vortex structures created at air inflow or air outflow of the convectors.

Fig. 4 Visualization of the flow structures at in- and outflow

For the visualization of the flow inside and outside the convector a smoke generator is used. The investigated area is illuminated by the continuous-wave laser with the wavelength of 532 nm. The circular laser beam is transformed through a cylindrical lens to a sheet of light. The images are taken by common CCD camera or high speed CCD camera (Redlake motion Pro X3) allowing double frame mode.

The results from the visualization can be then elaborated using PIV algorithms to obtain the velocity field. The large scale vortex structures are forming at the inflow of the convectors, however, at the outflow, the air stream is composed by a huge number of the small structures. The flow at output is more intensified containing developed more or less homogeneous turbulent flow. The flow visualization can indicate as well, that no air bypass between incoming and leaving air stream exist. In many constructions, this matter must be checked in order to prevent the loss of the energy.

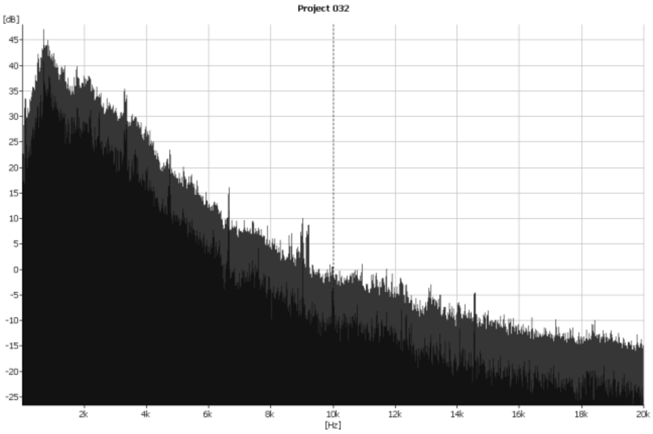

Fig. 5 Noise power spectra measured close to the fan

Fig. 5 depicts a noise measurement carried out using Briel and Kjaer sound level meter. At a first glance, no dominate frequency can be identified and produced noise level is laying under desirable averaged value that is prescribed about 40dB. From the analyses of the noise generation spectrum aeroacustic sound (loss of the air stream flow energy) and other noise type can be identified. The convectors are serviced mostly in the living room or even room reserved for sleeping or relaxing so existence of the permanent noise disturbance is improper.

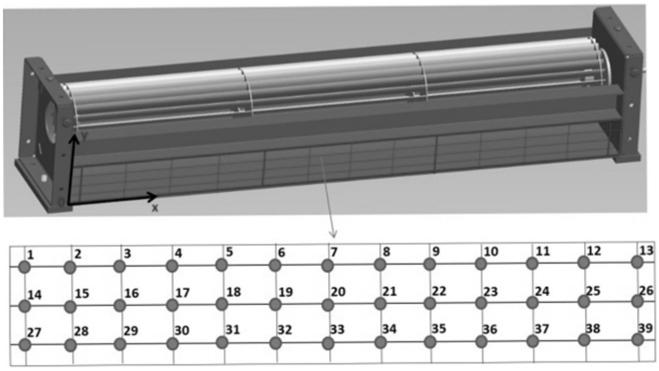

Fig. 6 Sketch of the fan with measured grid points

For a purpose of the numerical simulation, the correct boundary conditions must be set up especially for the inlet. To avoid a big complexity of the numerical simulation in case of the fan simulation, the flow velocity profile from the fan is measured and results are used directly as an inlet boundary condition for flow simulation.

In the real system the flow in the suction of the ventilator influences conditions on the ventilator inlet, which is not considered in this case.

Fig. 6 shows a fan used for force air ventilation and a grid with points at which the normal time-averaged velocities have been measured.

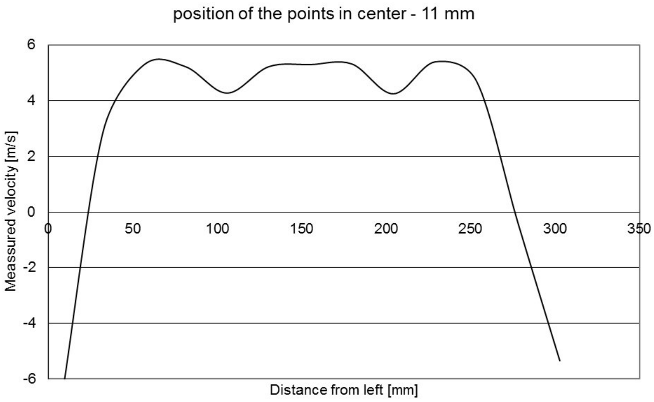

Fig. 7 Example of the measured velocity profile used later as a definition of the boundary condition

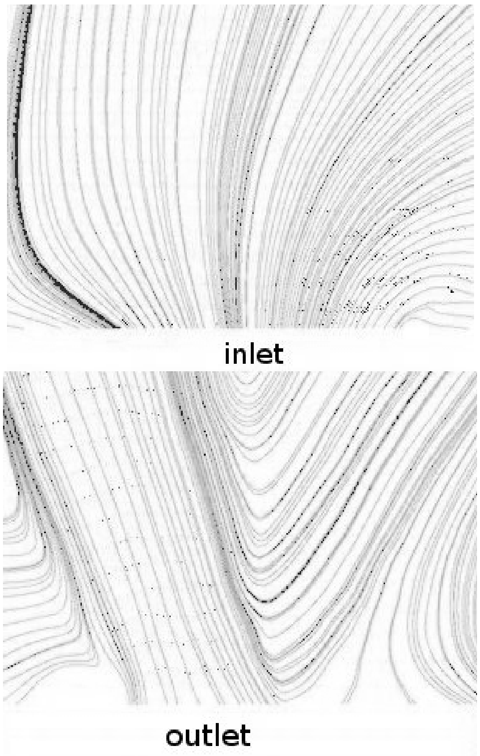

Fig. 8 The flow pattern at outlet and inlet

Fig. 7 illustrates a velocity profile representing by points starting from 14 up to 26. The velocity profile was measured using thermo anemometric probe connected to a data logger. The probe was traversed along the grid shown in the Fig. 6.

Fig. 8 presents results of the input and output flow above a floor convector measured using PIV technique. The flow pattern on the inlet (above) is influenced by the suction into the inlet flow (below).

III. MATH NUMERICAL RESULTS

Numerical simulation has been carried out using commercial computational code ANSYS. The simulation has been considered as a three-dimensional unsteady turbulent flow. The computational grid has been composed by hexahedral type elements with grid refinement at the wall of the tubes. Fig. 9 illustrates a global computational grid and detailed view on the grid resolution inside of the heat exchanger.

The computational grid and its resolution have been studied in order to avoid a dependency between grid resolution and numerical results.

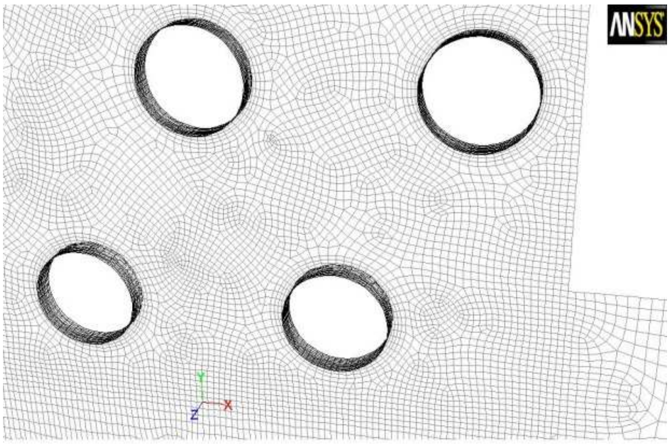

Fig. 9 The global computational grid and local zoom of the grid close to the walls.

The inlet conditions have been set up based on the experimentally measured velocity profiles created behind the fan. Temperature distribution at the tube walls has to be fixed at the constant value given by real temperature of the cooling or heating supplied water, particularly 9°C. The turbulent flow is captured by Large Eddy Simulation with subgrid-scale model based on the Smagorinsky approach [3]. This turbulent approach has been successfully used in the past for similar applications for instance in [4]. The total thermal power is calculated from (1), where the temperature difference between input and output at the air side is taken from the numerical simulation of the time-averaged temperature field. The flow rate is calculated directly by commercial code ANSYS and it is simultaneously used as a checking of the balance between input and output air mass. As in experimental Studies, no water condensation or other phase changes are not assumed.

Fig. 10 shows a time-averaged velocity field depicted using vector fields and a time-averaged temperature distribution in the whole convector. The first result interpretation can reveal qualitatively the intensity of the flow past tubes. In practice, the aim of the simulation is to ensure that all tubes will be confronted at maximal surface size with the air flow in order to reach the best heat convection parameter.

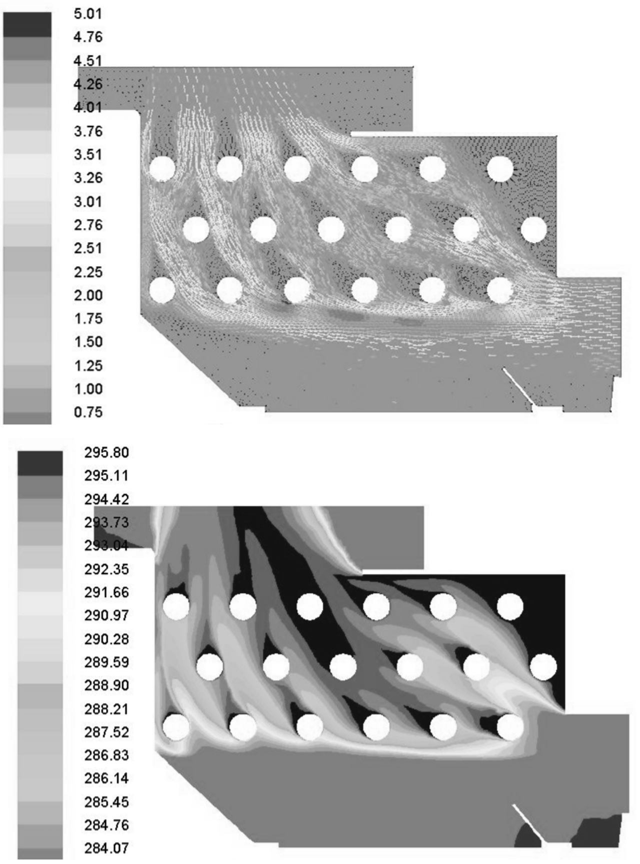

Fig. 10 The time-averaged velocity field and temperature distribution inside of the heat convectors

Furthermore, the temperature distribution can reveal information about the intensity of the heat exchange in other words, the effectiveness of the heat transport from the cooled or heated tubes to the cooled or heated air. This fact can be particularly observed evidently by color intensity at the air outflow. Large size of the colors associated to the lower temperature reveals cooler air temperature and so more intensified heat transfer.

In case of the velocity filed, the particular attention is focused on the space below the heat exchanger in which the strong vortices are created affecting the redistribution of the air flow entering into the heat exchanger.

In order to control such flow, the spoiler is used to control partially the intensity and direction of the incoming air from the fan.

IV. CONCLUSION

This paper presented methods and approaches used for investigation of the heating or cooling convectors. Obtained results such as total thermal power and effectiveness, noise level measurement, vortex structure visualization help to optimize or even design new concept of the convector. Measured results are used for validation of the numerical results. Furthermore, the validated numerical results provide details about flow behavior inside of the container and can contain other information such as turbulent flow properties, existence of the big vortex structures that influencing the flow redistribution inside space of the convectors.

Currently, obtained results have been leading to the practical changes of the convector construction; the thermal effectiveness could be increased approximately about 20 percent based on type and service conditions of the convector. In perspective we improve the numerical model for more complex flow simulation taking into account the phase changes of the air and different temperature distribution at the walls of the tubes in the heat exchanger.

ACKNOWLEDGMENT

This work was financially supported from the project of the Technology Agency of the Czech Republic under no. TA01020231.

REFERENCES

[1] A. Hamza, G.M. Morsy, Energy efficiency and indoor thermal perception: a comparative study between radian panel and portable convective heater, *Energy Efficiency*, 3, (2010), pp. 283-301

[2] A.K.A. Shati, S.G. Blakey and S.M.B. Beck, The effect of surface roughness and emissivity on radiator output, *Energy and Buildings* 43, (2011), 400-406

[3] P. Sagaut., Large Eddy Simulation for Incompressible Flows, *Second Edition Springer-Verlag Berlin Heidelberg New York* (2002)

[4] C. Liang, G. Papadakis, Large Eddy Simulation of cross-flow through a staggered tube bundle at subcritical Reynolds number, *Journal of Fluid and structures 23* (2007), pp. 1215-1230

K. Fraňa (M'99–SM'05–F'07) is an associated professor at the Technical university of Liberec at the Department of Power Engineering Equipment. He is interested in numerical simulation of turbulent flows; design a computational code for turbulence modeling and in numerical methods. He is a manager of the many European projects and domestic research projects relating to the energy saving technologies. He is a supervisor of the several Ph.D. students and his team is focused on the research in the material science, new production technologies in respect to the energy saving. In past, he was a scientific worker at the Technical University in Dresden, Germany. He was working on the development and testing the finite-element computational code used for magnetohydrodynamics flow simulations. He is an author of many paper contribution such as: K. Fraňa, J. Stiller, A numerical study of flows driven by a rotating magnetic field in a square container, European Journal of Mechanics / Fluids B (2008), doi:10.1016/j.euromechflu.2007.10.001, ISSN 0997-7546, Stiller, J., Fraňa, K. and Cramer, A., Transitional and weakly turbulent flow in a rotating magnetic field, Physics of Fluids, 18 (2006), Fraňa, K., Honzejk, V., Horakova K., A numerical simulation of the magnetically driven flows in a square container using the Delayed Detached Eddy Simulation, Computational Fluid Dynamics 2010: Proceedings of the Sixth International Conference on Computational Fluid Dynamics, ICCFD6, St Petersburg, Russia, on July 12-16, 2010 by Alexander Kuzmin Springer; 1st Edition (May 25, 2011)