EXPERIMENTAL AND NUMERICAL PREDICTION OF FLOW FIELD AROUND A PANEL RADIATOR

Tamer Calisir[1,*], Senol Baskaya[1], Hakan Onur Yazar[2], Sinan Yucedag[2]
[1]Dept. of Mechanical Engineering, Gazi University, Ankara, Turkey
[2]DemirDöküm A.S., R&D Dept., Bilecik, Turkey
*Corresponding author: tamercalisir@gazi.edu.tr

Panel radiators are one of the most used heating devices in domestic, business and industrial environments all over the world. Due to rising energy prices and a wish to create environmentally friendly heating systems, there is a demand for a reduction in the total energy consumption. Accordingly, there is an increasing demand for more efficient heating systems. So it is important to increase the efficiency of heating devices such as panel radiators. In the general design of radiators, the heating water is circulated in the hollow radiator and heat is transferred to the cooler surrounding air through the panels and convectors. Most of the heat transferred by the radiators is by natural convection. Hence, it is important to understand the natural convection flow around and inside the radiator, in order to design a more efficient panel radiator.

The main goal of this study is to observe the flow field around a panel radiator presently manufactured using numerical methods and particle image velocimetry (PIV) measurement technique. The aim was to predict the velocity and flow field of the heated air over a PCCP (panel-convector-convector-panel) radiator, and also the boundary layer which forms on the front panel of the radiator.

Experiments were performed for a panel radiator with the dimensions of 600x1000 mm under controlled laboratory conditions with a room temperature of 20°C. Same conditions were implemented in the FloEFD CFD code. Experiments and simulations were performed for a water inlet and outlet temperature of 75°C and 65°C, respectively. These temperatures were selected according to the EN 442-2 standard. The radiator performance test rig consists of a radiator, circulating constant temperature water bath, two immersion type thermocouples to measure water temperatures at the inlet and outlet of the radiator, a Coriolis mass flowmeter, a computer connected data logger for tracking the temperatures of the room and radiator inlet and outlet ports, and necessary piping. The mass flow rate was adjusted to achieve the mentioned outlet temperature.

PIV measurements were performed at different sections along the radiator, and the flow field was observed at the upper sections of the radiator. Hence, the air velocity which flows through the convectors and leaves at the upper section of the radiator was obtained.

Numerical results were verified with the PIV measurement results, and a good agreement was achieved. It was observed that the flow is mainly vertical and in the upward direction. However, after approximately 0.15 m above the radiator the flow begins to change direction towards the wall, where the radiator is mounted. At approximately 0.25 m above the radiator the flow sticks to the wall, and flows at almost constant velocity. It was also observed that the velocity values change with the measured location, which is due to the non-uniform temperature distribution on the panel radiator.

The findings of this study are important and will serve in the study to design a new panel radiator with a higher heat output. Hereafter, it will be possible to use lower circulation water temperatures in daily use, reducing fossil fuel consumption.