Abstract:

The use of CFD (Computational Fluid Dynamics) simulation method is an interesting way to get improved or new results concerning thermal conditions in ventilated, heated or air-conditioned rooms. In this work temperature response, of 36 sensor positions at different locations in the room were estimated using a CFD simulation for a large-scale ventilated installation with inlet, outlet and heating elements. A constant air supply rate and a step increase on the incoming air temperature were used as process input. The accuracy of the CFD model was assessed with real time data extracted from the corresponding experiment. CFD model boundary conditions (inlet and wall boundaries) were taken from the experimental readings. Relative error and mean relative error terms were defined and used as criteria for model validation.