Environmental Engineering and Management Journal *August 2014, Vol.13, No. 8, 1957-1964*
http://omicron.ch.tuiasi.ro/EEMJ/

"Gheorghe Asachi" Technical University of Iasi, Romania

COMPUTATIONAL FLUID DYNAMICS BASED MODELING OF A LINEAR HEAT SOURCE

Raluca Teodosiu*, Lidia Niculiţă, Cătălin Teodosiu

Faculty of Building Services and Equipment, Technical University of Civil Engineering
Lacul Tei Bvd., no. 122-124, Bucharest 020396 Romania

Abstract

In this paper, a study is conducted to assess the ability and the accuracy of a CFD approach concerning the behavior of a linear heat source positioned in a test room. Firstly, the main characteristics of our numerical model (mesh generation, turbulence model, near wall treatment, radiation model and boundary conditions) are presented after a briefly description of the test room taken into account. In addition, we show also the significance of these issues within our CFD approach. Furthermore, comparing the results obtained with those from experiments tests, the numerical model precision is evaluated. As a result, we notice a correctly plume description which allow us to achieve a proper general airflow prediction in the heated room (including thermal stratification and pollutant diffusion). On the contrary, we detect relatively important experimental – numerical differences regarding the heat transfer at walls. Therefore, the boundary layer description as well as the radiation model has to be improved. This must be correlated with the measurements precision in order to be able to achieve more complete and accurate experimental data.

Key words: buoyancy driven cavity, CFD - Computational Fluid Dynamics modeling, heat source, thermal plume

Received: February, 2014; Revised final: August, 2014; Accepted: August, 2014

* Author to whom all correspondence should be addressed: e-mail: ralucahohota@yahoo.com