BUILD SIMUL (2014) 7: 35 – 55
DOI 10.1007/s12273-013-0103-1

Detailing the effects of geometry approximation and grid simplification on the capability of a CFD model to address the benchmark test case for flow around a computer simulated person

Giacomo Villi (✉), **Michele De Carli**

Department of Industrial Engineering, Università degli Studi di Padova, Padua, Italy

Research Article

Abstract

This paper details the use of a simplified CFD model to predict the flow patterns around a computer simulated person in a displacement ventilated room. The use of CFD is a valuable tool for indoor airflow analysis and the level of complexity of the model being investigated is often critical to the accuracy of predictions. The closer the computational geometry is to the real geometry of interest, the more accurate the corresponding results are expected to be. High complexity meshes enable elaborated geometries to be resolved. The drawback is, however, their increased computational cost. The Fire Dynamics Simulator (FDS) model (Version 5) enabled to investigate the effects of geometry and computational grid simplification on the accuracy of numerical predictions. The FDS model is based on a three-dimensional Cartesian coordinate system and all solid obstructions are forced to conform to the underlying numerical grid which is a potential limitation when dealing with complex geometries such as those of a human body. Nevertheless, the developed computational model was based exclusively on a three-dimensional rectangular geometry. At the same time, in order to limit the total number of grid cells, a relatively coarser grid than those used for similar simulations was adopted in the investigation. The developed model was then assessed in terms of its capability of reproducing benchmark temperature and air velocity distributions. The extent to which numerical results depend on different simulation settings was detailed and different boundary conditions are discussed in order to provide some guidance on the parameters that resulted to affect the accuracy of the predicted results. The comparison between numerical results and measurements showed that a simplified CFD model can be used to capture the airflow characteristics of the investigated scenario with predictions showing a favourable agreement with experimental data at least in the qualitative features of the flow (the detailed investigation of the local airflow field near the occupant can not be probably conducted apart from considering the real human geometry). Significant influence of simulator geometry and of boundary conditions was found.

Keywords

LES,
indoor airflow,
CFD,
ventilation

Article History

Received: 17 July 2012
Revised: 29 November 2012
Accepted: 18 December 2012

Indoor/Outdoor Airflow and Air Quality

1 Introduction

Air distribution in rooms is the result of the complex interaction between the ventilation system and local disturbances induced by factors such as occupants. A more detailed investigation of the distribution of air properties such as temperature and velocity within a room is beneficial to the design of an energy efficient, comfortable and healthy indoor environment. In order to predict the details of the ventilation mechanism, realistic conditions such as the presence of obstructions and localized heat sources have to be incorporated into the analysis. A person acts as an obstacle when exposed to a flow (Brohus 1997); behind the person a wake is generated whilst in front some air is pushed away. The human body is continually exchanging energy with the surrounding environment and the excess human temperature results in an ascending plume with velocities which can exceed 0.2 m/s and even reach 0.5 m/s locally (Zukowska et al. 2007). A nude

E-mail: giacomo.villi@unipd.it