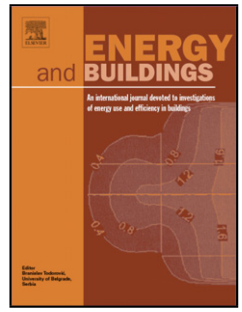

Title: Numerical Simulation of Turbulent Airflow in a Ventilated Room: Inlet Turbulence Parameters and Solution Multiplicity

Author: Erhan Pulat Hıfzı Arda Ersan

PII: S0378-7788(15)00094-8
DOI: http://dx.doi.org/doi:10.1016/j.enbuild.2015.01.067
Reference: ENB 5673

To appear in: *ENB*

Received date: 12-8-2014
Revised date: 12-1-2015
Accepted date: 30-1-2015

Numerical Simulation of Turbulent Airflow in a Ventilated Room: Inlet Turbulence Parameters and Solution Multiplicity

Erhan PULAT[1,*] and Hıfzı Arda ERSAN[2]
[1]Uludag University, Faculty of Engineering, Mechanical Engineering Department, Gorukle Campus, TR-16059 Bursa, Turkey
[2]Uludag University, School of Natural and Applied Sciences, Mechanical Engineering Department, Gorukle Campus, TR-16059 Bursa, Turkey
[*]Corresponding Author: Tel.: +90 224 4428176, Fax: +90 224 4428021, E-Mail: pulat@uludag.edu.tr

Abstract

In this study, airflow and temperature distributions in the well-known International Energy Agency (IEA) Annex 20 room are predicted numerically to investigate the effects of the inlet turbulence intensity and the length scale on the flow characteristics, while considering the possibility of solution multiplicity related to chaos theory. The flow is considered to be turbulent, steady, incompressible, and two-dimensional. Computations are performed using the standard k-ε, RNG k-ε, standard k-ω, and SST k-ω turbulence models with scalable and automatic wall functions, and the results are compared with numerical and experimental results from the literature. The validated turbulence model is then used to investigate the effects of the turbulence intensity and the length scale. At a low inlet turbulence intensity value (Tu=0.01), the length scale variation has no influence on the flow pattern. However, the length scale affects the flow pattern at a high inlet turbulence intensity value (Tu=0.4). At constant low and medium length scale values, an increase in the inlet turbulence intensity from 0.01 to 0.4 affects the flow pattern. However, an increase in the turbulence intensity has no influence on the flow pattern at a constant high length scale value.

Keywords: Room Ventilation, Eddy Viscosity Turbulence Models, Turbulence Intensity and Length Scale, Solution Multiplicity, CFD

1. Introduction

On average, people spend 90% of their life indoors [1]. Good airflow is important in ventilated spaces to provide good air quality and thermal comfort for the occupants, to maintain specified thermal and flow conditions for industrial processes and to ensure the efficient use of energy. The importance of ventilation related energy use is increasing and may represent up to 50% of the total energy use of a building, particularly for certain typologies such as office buildings [1]. Computational fluid dynamics (CFD) is one of the primary methods used to assess indoor airflow [2, 3, 4], among other methods such as the Building Energy Simulation (BES) approach and zonal models. In recent years, CFD has taken a prominent role in the simulation of indoor environment airflow problems [5, 6]. However, indoor airflow is a complex system, and new modeling approaches such as time series modeling can be found in the literature [7].

Pioneering work using the CFD technique to simulate room air motion was performed by Nielsen [8]. The flow inside a building environment is generally turbulent [2], and the IEA Annex 20 room, described in Nielsen [9], is one of the experimental setups for studying the suitability of room turbulent air distribution models [10, 11]. Aside from these studies, which were mainly focused on the comparison of turbulence models, and although the effects of turbulent boundary conditions in ventilated enclosures were identified in Reynolds-averaged