International Communications in Heat and Mass Transfer 92 (2018) 1–14

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Forced, natural and mixed convection benchmark studies for indoor thermal environments

Bahadir Erman Yuce[a], Erhan Pulat[b,*]

[a] School of Natural and Applied Sciences, Mechanical Engineering Department, Uludag University, Gorukle Campus, TR-16059 Bursa, Turkey
[b] Faculty of Engineering, Mechanical Engineering Department, Uludag University, Gorukle Campus, TR-16059 Bursa, Turkey

ARTICLE INFO

Keywords:
CFD
Validation
Turbulence models
Indoor environment

ABSTRACT

In this study, some well-known experimental studies related to forced, natural, and mixed convections were used for validation of k-ε and k-ω turbulence models. For this purpose ANSYS-Fluent 16.0 is used. International Energy Agency IEA Annex20 room, a tall differentially heated rectangular cavity, and a mixed convective air flow within a square chamber with a heated bottom wall were considered for forced, natural, and mixed convection respectively. Standard, RNG and Realizable models of k-ε group, and Standard, SST and BSL models of k-ω group with enhanced wall treatment for near wall modeling were tested by comparing the velocity and temperature distributions with available measurement values of employed geometries. In total, the results of Standard and RNG k-ε models are in good agreement with experimental measurements. Although the performance of k-ω group models is well in natural convection, some results of these models do not agree well with test data in forced and mixed convection cases.

1. Introduction

Energy supply and use are related to global warming and well known environmental problems such as air pollution, acid precipitation, ozone depletion, forest destruction and emission of radioactive substances. These problems and issues that are directly and indirectly related them must be taken into consideration for sustainable development [1]. One of the Sustainable Energy Development Strategies that typically involve technological change is energy savings [2].

According to data of United Nations Environmental Programme [3], buildings use about 40% of global energy, 25% of global water, 40% of global resources. So building sector attracts attention from both energy use [4] and greenhouse gas (GHG) emissions point of views [5]. Also, energy consumption in buildings can be reduced by 30 to 80% using available effective technologies. Investment in building energy efficiency is accompanied by significant direct and indirect savings, which help offset incremental costs, providing a short return on investment period. Building ventilation has an important role in energy consumption in buildings. In addition, the productivity and efficiency of occupants are dependent on the conditions of thermal environment [6,7].

The main aim of HVAC is to provide fresh air and to regulate indoor air for comfortable conditions for occupants in buildings [8]. Room ventilation can be achieved in different types of distribution systems and ventilation and air distribution systems in buildings were briefly summarized and discussed by Awbi [9]. Despite of the low speed velocity regions, system air flow is often turbulent because of mechanical blowing system. Ventilation can be classified into natural, mixing and displacement ventilation. Mixed and displacement ventilation types are commonly used types. The CFD simulations and the measurements suggest that displacement ventilation is more energy efficient than a mixing system [10]. Although the traditional mixing systems have poor ventilation efficiency and are less energy efficient, they still occupy a large portion of the market. Relatively new impinging jet ventilation system shows similar tendencies with displacement ventilation but this new system may need further studies by ventilation researchers and designers to fully understand its performance under various conditions experienced in practice [11].

Airflow and velocity characteristics are very important to regulate and control the indoor air parameters such as temperature, humidity and contaminant concentration. These parameters also have significant effect on air quality, thermal comfort, health and energy savings [12].

Pioneering Computational Fluid Dynamics (CFD) study was first introduced in the ventilation industry in the 1970s [13]. Air distribution is all about airflow and transport of heat and airborne pollutants. Air distribution is governed by the conservation principle. It is mathematically described by a set of partial differential equations, known as the Navier–Stokes equations. These can be solved analytically only for simple and ideal conditions. For complex geometry and/or complex

* Corresponding author.
E-mail addresses: bahadiryuce@uludag.edu.tr (B.E. Yuce), pulat@uludag.edu.tr (E. Pulat).

https://doi.org/10.1016/j.icheatmasstransfer.2018.02.003

boundary conditions, numerical methods may be used to solve these equations or solve their modeling versions, given the initial and boundary conditions, i.e. Computational Fluid Dynamics (CFD) [14].

Indoor environment design requires details of air velocity/temperature distributions, relative humidity maps, contaminant concentrations, and turbulence levels. Most indoor airflows are actually rather complicated, and often driven by both pressure gradient and thermal buoyancy forces. There are three typical convection modes; i.e. forced convection like spring free cooling flow near the ceiling, natural convection as winter heating by radiators, and mixed convection as summer cooling using an air-conditioning unit. Despite the challenges in predicting the airflow precisely, both experimental measurements and computational simulations have been used in the past. Most experiments adopt a full-scale test chamber to setup an artificial environment to isolate the space from the external. While this permits the controllable flow and thermal boundary conditions, the cost would be high and turn-around testing period is normally very long [15]. So in recent years, CFD has taken a prominent role in the simulation of indoor environment airflow problems [16,17]. However room ventilation is a complex turbulent system and it requires reliable benchmark studies especially for simulation studies required turbulence model using.

In this study, some well-known experimental studies related to forced, natural and mixed convections were used for validation of k-ε and k-ω turbulence models. For this purpose ANSYS-Fluent 16.0 is used. International Energy Agency IEA Annex20 room, a tall differentially heated rectangular cavity, and a mixed convective air flow within a square chamber with a heated bottom wall were considered for forced, natural and mixed convection respectively.

2. Materials and method

2.1. Turbulence models and numerical procedure

It is well known that above a critical value of the Reynolds (*Re*) number, flow becomes turbulent characterised by the appearance of statistical fluctuations of all the variables around mean values, and it is impossible to describe them in a deterministic manner. However, these fluctuations can be computed numerically in direct simulations of turbulence (DNS) or at a lower level of approximation by the Large Eddy Simulation (LES) approach, whereby only the small-scale turbulent fluctuations are modelled and the larger scale fluctuations are computed directly [18]. The other family, the Reynolds-Averaged Navier-Stokes (RANS) model does not account the turbulent fluctuations and calculates only the turbulent averaged flow. In RANS approach, turbulence models used in computations fall into two general categories as Eddy Viscosity Models (EVM) and stress transport models including Algebraic Stress Models (ASM) and Reynolds Stress Models (RSM). A hybrid combination of LES and RANS is known as Detached Eddy Simulation (DES), and in this approach, the RANS approximation is kept in the regions where the boundary layers are attached to the solid walls [18,19]. In addition to above approaches, recently, microscopic dynamics approaches such as Lattice Boltzmann Method (LBM) have attracted significant attention for turbulent flow simulation [20]. Although DNS and LES can provide invaluable information about the details of the flow field, and are expected to be more accurate than RANS, they are not yet practical as engineering tools due mainly to the extreme computational requirements and serious problems with boundary conditions [21,22]. Similar problem arises for LBM [20] and we need approximate models to represent turbulence for industrial applications [18]. So engineering calculations still rely on the use of the RANS equations with the aid of turbulence modeling [21]. In addition, despite the undisputable progress in the development of advanced models, the rudimentary k-ε (and, to a less extent, k-ω) model serves still as the most frequently used closure [23], and the systematic verification and validation of RANS turbulence models is also still an open issue [19]. So RANS based two groups of two equation turbulence models, k-ε (Std. k-ε, RNG k-ε, Realizable k-ε) and k-ω (Std. k-ω, SST k-ω and BSL k-ω) groups are considered in this study. Klinzing and Sparrow [24] compared five turbulence models commonly offered in commercially available CFD software, including FLUENT, ANSYS CFX, and STAR-CD. They found that the most suitable turbulence model for external flows was the standard k-ε model with enhanced near-wall treatment. For this reason Enhanced wall treatment (Ewt) is used for near wall modeling. Two-dimensional and steady Reynolds Averaged Navier-Stokes (RANS), continuity and energy equations are solved through ANSYS-Fluent 16.0 software. For natural and mixed convection cases, the gravity force is taken into consideration with the Boussinesq approach, and this effect is expressed in terms of Rayleigh (Ra) or Archimedes numbers (Ar). Fluent uses the finite volume method and well-known SIMPLE scheme is used in pressure-velocity coupling. In the recent study of Limane et al. [25] about thermo-ventilation in a cavity with heated floor, buoyant Boussinesq SIMPLE option among available four options were preferred by comparing RNG k-ε model results with experimental measurements and considering some qualitative evaluations. For spatial discretization, Least Squares Cell Based in gradient, Second Order and Second Order Upwind in pressure and momentum respectively, and First Order Upwind in both turbulent kinetic energy and its dissipation rate are used. Convergence criterion is 10^{-6} for all parameters. Standard forms including default coefficients are used for all turbulence models and further information about them can be found in [26]. For all considered cases, geometries, boundary conditions, and mesh independency studies are given in the subsequent sections. In the mesh independency studies, Std. k-ε turbulence model with enhanced wall treatment are used for all cases. Very fine meshes were constructed in the near walls of all considered cases since enhanced wall treatment was used in the near-wall modeling. For example, representative maximum mean $y+$ value is 2.22 for mixed convection case in selected grid size of 160×160.

In this study, International Energy Agency IEA Annex20 room, a tall differentially heated rectangular cavity, and a mixed convective air flow within a square chamber with a heated bottom wall is used for forced, natural, and mixed convection respectively to compare considered turbulence models.

2.2. Forced convection

Forced convection is a type of heat transport in which fluid motion is generated by an external source like pump, fan etc. This case is studied with well-known IEA Annex20 room which is showed in Fig. 1a. Results of this study will be compared with experimental results of Nielsen [27] taken from Laser Doppler Anemometer (LDA) measurements. All walls are assumed as adiabatic. There are no temperature differences between walls so buoyancy effect is neglected. Air enters through the channel on the upper left corner with a velocity of 0.455 m/s to ensure Reynolds number of 5000 based on the inlet height (h) and blow out of the channel on the lower right corner. Dimensions are given in Fig. 1a. Grid structure is one of the most important parameters affecting accurate solution. Element number and quality of the grid structure not only achieves the accurate result, but also affects the duration of the solution progress. A denser grid structure was obtained in the walls, and hexahedral element type was selected. Results were obtained at four different element numbers (4000, 18,150, 28,100, 43,100) to ensure obtaining mesh independency. Almost all velocity profiles coincide each other as seen from Fig. 1b,c,d,e and element number of 28,100 was selected.

2.3. Natural convection

Natural convection is another type of heat transport in which fluid motion is generated by buoyancy force due to density variations of fluid with temperature. A tall cavity which is well known in natural convection studies is used for this case. The airflow because of natural