Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **14**, No. 6, pp. 1567-1586

DOI: 10.4208/aamm.OA-2021-0220
December 2022

Interaction of Radiation and Turbulent Natural Convection: A Pseudo-Direct Numerical Study

Alexander Nee[1,*] and Ali J. Chamkha[2]

[1] *Research and Educational Center of I.N. Butakov, National Research Tomsk Polytechnic University, Tomsk, Russia*
[2] *Faculty of Engineering, Kuwait College of Science and Technology, Doha District, Kuwait*

Received 29 June 2021; Accepted (in revised version) 3 February 2022

Abstract. This paper presents a hybrid lattice Boltzmann solver for turbulent buoyancy-driven flow coupled with surface thermal radiation. The two-relaxation time scheme for the Boltzmann equation combined with the implicit finite difference scheme for the energy equation is implemented to compute the heat transfer and fluid flow characteristics. The accuracy and robustness of the hybrid approach proposed in this study are assessed in terms of the numerical and experimental data of other researchers. Upon performing the simulation, the Rayleigh number is ranged from 108 to 1010 whereas the surface emissivity is changed from zero to unity. During computations, it is found that the overall temperature of the cavity is increased as a result of enhancing the surface radiation. Convective plumes are formed both at the isothermal and the thermally-insulated walls with the $Ra \geq 10^9$ and $\varepsilon \geq 0.6$. In the conditions under study, the overall heat transfer rate is raised by around 5% when taking into account the surface thermal radiation.

AMS subject classifications: 80A20, 76F65

Key words: Pseudo-direct numerical simulation, surface radiation, hybrid lattice Boltzmann scheme, turbulent natural convection.

1 Introduction

Turbulent natural convective mechanism of heat transfer is encountered in many practical applications such as building heating, fuel cells cooling, metal industry, etc. Despite a significant development of computational fluid dynamics tools, an accurate prediction of turbulent flow behavior is still a very complicated task since a conventional direct

*Corresponding author.
Emails: nee_alexander@mail.ru (A. Nee), a.chamkha@kcst.edu.kw (A. Chamkha)

numerical simulation (DNS) approach is computationally inefficient. Moreover, numerical implementation becomes much more difficult when taking into account the radiative heat transfer [1]. Based on the literature review, the works devoted to the combined heat transfer by turbulent natural convection and radiation can be classified by the turbulence model used. Below, the up-to-date works will be discussed.

As could be expected, mathematical modelling predominantly performed in terms of the Reynolds averaged Navier-Stokes (RANS) approach. The popularity of the RANS is due to acceptable time of code execution. However, it should be stressed that this approach contains several empirical constants. Sharma and co-workers considered turbulent buoyancy driven flow in rectangular [2] and inclined square [3] enclosures filled with a transparent medium when taking into account the surface radiation. Turbulent natural convection was computed in terms of the standard kinetic energy and its dissipation rate ($k-\varepsilon$) model. The effect of walls emissivity on turbulent buoyancy driven flow of the humid air in a cavity with cylindrical obstacles was analyzed by Iyi et al. [4]. Heat transfer via turbulent natural convection and surface thermal radiation in an enclosure partially filled with a porous medium was studied in [5]. Dash and Dash [6] considered turbulent natural convection from a hot horizontal cylinder. The radiation transport equation was solved by means of the discrete ordinate method. Parmananda et al. [7] examined the effect of non-Boussinesq formulation on turbulent heat transfer and fluid flow characteristics. A low Mach number model was utilized to perform numerical study. Xaman et al. [8, 9] analyzed the turbulent buoyancy flow coupled with the conjugate heat transfer and surface radiation. These studies were aimed to find out how a vertical glazed wall affected the thermal and flow behavior. Sheremet et al. [10, 11] considered the same heat transfer modes in a cavity bounded by solid walls with a discrete heater. Moreover, the authors implemented the standard $k-\varepsilon$ model with a coordinate transformation in order to perform computations on a non-uniform mesh.

While the studies [2–11] implemented the $k-\varepsilon$ model, Zamora and Kaiser [12–14] performed computations with the standard kinetic energy and its specific dissipation rate ($k-\omega$) model. They considered the effect of temperature-dependent physical properties, cavity shape and the presence of a hot immersed body on local and mean fluid dynamics characteristics. Radiation was described by means of the IMMERSOL model. Hemmer et al. [15] studied thermal and flow fields in a room with convective and radiant heaters. The shear stress transport (SST) $k-\omega$ model of Menter was applied to calculate turbulent fluxes. The same turbulence model was used to analyze the effect of aspect ratio [16] on the flow structure and electronics cooling characteristics [17]. Comparative studies of performance of different $k-\varepsilon$ and $k-\omega$ models in combined heat transfer problems by turbulent natural convection and radiation are discussed in [18–20].

By now, few papers analyze the turbulent buoyancy driven flows coupled with radiation in terms of the LES and DNS techniques. Kogawa et al. [21] built a complex three-dimensional model to analyze how the non-radiation, surface radiation, volumetric radiation and combined radiation affected the heat transfer and turbulent fluid flow patterns. The large eddy simulation coupled with the dynamic Vreman subgrid model