Renewable and Sustainable Energy Reviews 82 (2018) 40–59

Contents lists available at ScienceDirect

Renewable and Sustainable Energy Reviews

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

Turbulent natural convection heat transfer in rectangular enclosures using experimental and numerical approaches: A review

I.V. Miroshnichenko*, M.A. Sheremet

Laboratory on Convective Heat and Mass Transfer, Tomsk State University, 634050 Tomsk, Russia

ARTICLE INFO

Keywords:
Turbulent natural convection
Rectangular cavities
Surface radiation
Heat source
Numerical and experimental results

ABSTRACT

Natural convection is one of the most important modes of fluid flow and heat transfer. This paper presents a review of the research on turbulent natural convection in rectangular cavities using numerical and experimental techniques. In this review we have attempted to summarize the published papers on this topic with some interesting and important results. Numerous configurations of the enclosures with different initial and boundary conditions, heat source locations and radiative properties of medium and walls have been considered under the effects of various parameters such as the Rayleigh and Prandtl numbers, surface emissivity, cavity inclination angle, thermal properties, etc. Finally, some suggestions have been provided for future studies in the considered area.

1. Introduction

Natural convection heat transfer in rectangular enclosures is a field of intense research due to its wide range of engineering applications, which include building insulation, cooling devices for electronic instruments, solar energy collectors, nuclear reactor design etc. The process of heat and mass transfer in the area under consideration has been investigated and analyzed analytically, numerically, and experimentally by many researchers in the past fifty years. Each published paper examines a particular aspect of the problem depending on the aim pursued.

More than 50 years later Batchelor [1] studied two-dimensional convective motion generated by buoyancy forces on the fluid in a long rectangle, of which the two long sides are held at different temperatures. Few years later, Poots [2] numerically investigated laminar free convection in enclosed horizontal, oblique, and vertical gas layers. One main result of this paper is a method developed for obtaining solutions of the two-dimensional governing equations in the form of approximate double expansions involving orthogonal polynomials. Wilkes and Churchill [3] and De Vahl Davis [4] numerically studied the problem with different aspect ratio cavities and, as in previous studies, under isothermal boundary conditions. Since then, the number of studies of natural convection in cavities has increased dramatically. Many authors have investigated free convection in the most common cases of cylindrical, rectangular or other regular geometries. A detailed review of natural convection in enclosures of various (non-square) shapes was carried out by Das et al. [5]. This paper summarized the studies on natural convection heat transfer in triangular, trapezoidal, parallelogrammic enclosures and enclosures with curved and wavy walls filled with fluid or porous media. In addition, they studied also the effects of the various parameters such as the Rayleigh numbers, Prandtl numbers, Darcy numbers, Darcy–Rayleigh number, irreversibility distribution ratios and volume fraction of the nanoparticles. Bairi et al. [6] presented a review of some results of research on natural convection in the parallelogrammic diode cavity. They investigated a great variety of configurations of the enclosures with different shapes and inclinations, initial conditions, thermal boundary conditions, nature of the fluid and radiative properties, heat source distributions. A detailed review of works on the entropy generation analysis during natural convection in various enclosures and processes involving different practical applications was conducted by Biswal and Basak [7]. Vanaki et al. [8], Mahdi et al. [9], Haddad et al. [10] performed reviews of convective heat transfer in nanofluids.

It should be mentioned that most flows that occur in our environment are induced by buoyancy. In many practical situations, the Rayleigh number (which is the ratio of buoyancy forces and the viscous forces) often exceeds the critical values and the fluid flow becomes turbulent [11–13].

There are three most important ways for simulation of turbulence: Direct Numerical Simulation (DNS), Large Eddy Simulation (LES) and Reynolds Averaged Navier Stokes (RANS) models [14–16]. RANS models have been more widely used in turbulent fluid flow simulations and demanded reasonable computational resources. In LES, large eddies with the most energy have being simulated, whereas smaller eddies

* Corresponding author.
E-mail address: miroshnichenko@mail.tsu.ru (I.V. Miroshnichenko).

http://dx.doi.org/10.1016/j.rser.2017.09.005
Received 29 December 2016; Received in revised form 10 July 2017; Accepted 3 September 2017

that cannot be described by the computational grid have being modeled. DNS method is computationally very expensive. It is currently being used mainly to simulate benchmark cases. The present review includes all the above mentioned methods. The effects of some geometrical parameters on the convective heat transfer in the cavity are also considered. Among the most important ones is enclosure inclination angle. This parameter plays an essential role in turbulent heat transfer. Mainly, heat transfer decreases with inclination angle and the lowest heat transfer is formed for the inclination angle $\pi/2$.

The contribution of thermal radiation is an important topic during heat transfer in enclosures [17–19]. Radiation heat transfer depends on several parameters such as the cavity geometry, walls temperature, surface emissivity and thermophysical properties of internal medium. In studies focused on turbulent natural convection, the radiative mode of heat transfer is occasionally neglected because of the overwhelming number of computational resources it demands. However, as practice shows radiation heat transfer has a significant effect on the system and cannot be neglected.

The objective of this review is to present a summary of past and recent studies on turbulent natural convection in rectangular enclosures obtained using numerical and experimental techniques. In the case of numerical technique we have classified papers taking into account different numerical approaches, namely, finite volume and finite element methods, finite difference method and lattice Boltzmann method. The results of a large number of different parameters affected the flow pattern and heat transfer characteristics are summarized. The main attention is paid to explore the causes of the enhancement of heat transfer, which would be useful for understanding the occurring processes in considered area, and accurately predict turbulent natural convection using different numerical methods. Turbulent natural convection within enclosures of different thermal boundary conditions has wide ranges of applications in cooling processes [20–22], building construction [23–25], nuclear reactor design [26,27], solar energy [28,29], etc. The present review can be useful for the future researchers for choosing the suitable process parameters for different convective systems.

2. Simulation of fluid flow and heat transfer

Several numerical methods have been applied successfully in order to simulate fluid flow and heat transfer [31–96]. These approaches are presented below. The numerical approach consists of solving to a complex system of differential equations with corresponding initial and boundary conditions. These governing equations consist of continuity equation, momentum balance and energy balance equations.

2.1. Finite volume method and finite element method

One of the most employed methods is finite volume method. The first time this method has been developed by Patankar [30]. In the finite volume method, volume integrals in a partial differential equation are converted to surface integrals, using the divergence theorem. Advantage of the finite volume method is that it is easily formulated to take into account unstructured meshes. There are many computational fluid dynamics packages that used considered method (ANSYS, FlowVision, Star-CD).

The benchmark problem of laminar and turbulent natural convection flow in a differentially-heated square cavity has been investigated by Barakos et al. [31]. They have showed that the turbulent solution starts deviating from the laminar approximation at $Ra \approx 10^8$. The average Nusselt number along the hot wall shows a sudden increase when the turbulent solution is reached. Henkes and Hoogendoorn [32] have compared the numerical results obtained by 10 groups for turbulent natural convection flow in enclosures. The comparison of computations has led to a numerical reference solution for turbulent natural convection in considered area. They have found that the reference solution overpredicts the averaged wall heat transfer, as compared to experiments, but the velocity in the vertical boundary layers agrees well. Henkes et al. [33] have studied laminar and turbulent natural convection flow in a two-dimensional square cavity by using three different turbulence models such as the standard k–ε model with logarithmic wall functions, low-Reynolds-number model of Chien [34] and low-Reynolds-number model of Jones and Launder [35]. They have shown that differences between the turbulence models are largest for quantities that are determined in the inner layer of the vertical boundary layer, for example, the wall-heat transfer and the wall-shear stress. Numerical simulations of the natural convection flow of air in an inclined square cavity have been performed by Kuyper et al. [36]. They have found the results for the calculation of the average Nusselt number as a function of the cavity inclination angle. Markatos and Pericleous [37] have investigated the buoyancy-driven laminar and turbulent flow and heat transfer in a square cavity with differentially heated side walls. The speculative use of the k–ε model in this work has indicated that, despite its well-known deficiencies in terms of physical realism, it may still lead to a reasonable prediction of the overall flow structure. Sharif and Liu [38] have studied turbulent modes of natural convection in a two-dimensional side-heated square cavity at various angles of inclination. They have found that the average Nusselt number at the hot wall, which is a measure of heat transfer rate, decreases with increasing in inclination angle, while the convection strength, which is a measure of the vigorousness of the flow in the cavity, increases with inclination. A computational study of turbulent natural convection in a side-heated near-cubic enclosure at a high Rayleigh number has been carried out by Dol and Hanjalic [39]. They have shown that the second-moment closure is better in capturing thermal three-dimensionality effects and strong streamline curvature in the corners, at the same time k–ε model still provides reasonable predictions of the first moments away from the corners.

Shati et al. [40] studied numerically turbulent natural convection with thermal radiation in square and rectangular enclosures. They have calculated the average Nusselt number as a function of Grashof and Prandtl numbers for the turbulent natural convection in a square enclosure without radiation interaction. Coupling of turbulent natural convection with radiation in an air-filled differentially-heated cavity has been investigated by Ibrahim et al. [41]. They have revealed that in the considered configuration, gas radiation has little influence on the flow structure, at least when considered alone (without wall radiation). Sharma et al. [42] have examined the interaction of turbulent natural convection and surface thermal radiation in inclined square enclosures. They have ascertained that turbulent viscosity is maximum at the center of the enclosure for $\varphi = 0$, where φ is an inclination angle, and as φ increases; it reduces at the center and increases around the wall zones. It should be noted that the radiative Nusselt number increases linearly with growth of the emissivity. Fusegi and Farouk [43] have performed a study to investigate turbulent and laminar natural convection-radiation interaction in a square enclosure. They have reported that gas radiation slightly reduces heat transfer to the walls, as a result of the attenuation by the participating medium. Moreover, radiation causes hydrodynamic and thermal boundary layers to thicken. The numerical study Mesyngier and Farouk [44] is devoted to the turbulent natural convection-radiation interaction in a square cavity either filled with a single participating gas (H_2O or CO_2) or a homogeneous mixture of two participating gases along with a diluent (N_2). The authors have reported that the results of the simulations show an enhancement of heat transfer in the compositions where H_2O was present, even in small quantities. Ben-Nakhi and Mahmoud [45] have studied conjugate turbulent natural convection in the roof enclosure of a heavy construction building during winter. They have found that as Rayleigh number increases, the heat loss from the room through the enclosure progressively rises and that as the aspect ratio increases the enclosure becomes more thermally efficient. The effect of grid and boundary conditions on turbulent heat and mass transfer in enclosures has been simulated using the Shear