International Communications in Heat and Mass Transfer 38 (2011) 625–632

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Finite element simulation of hydromagnetic convective flow in an obstructed cavity ☆

Rehena Nasrin

Department of Mathematics, Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh

ARTICLE INFO

Available online 24 February 2011

Keywords:
Cavity
Hydromagnetic
Obstacle
Free convection
Joule heating and finite element simulation

ABSTRACT

The hydromagnetic natural convective flow and heat transfer characteristics in a square cavity with a solid circular heated obstacle located at the center have been investigated numerically. The left vertical surface of the cavity is uniformly heated of temperature T_c and other three surfaces are adiabatic. The obstacle consists of constant heat T_h. Under all circumstances the condition $T_h > T_c$ is maintained. The physical problem is represented mathematically by sets of governing equations and the developed mathematical model is solved by employing Galerkin weighted residual finite element simulation. The behavior of the fluid in the ranges of Prandtl number (0.073–2.73), Hartmann number (0–50) and Joule heating parameter (1–7) is explained in details. It is found that the flow and temperature fields are strongly dependent on the above stated parameters for the ranges considered. The variation of the average Nusselt number (*Nu*) for various Prandtl number (*Pr*) is also presented.

1. Introduction

Free convection flow of an electrically conducting fluid within a cavity in the presence of magnetic field and Joule heating is of extraordinary technical importance because of its frequent occurrence in many industrial applications such as geothermal reservoirs, thermal insulations and petroleum reservoirs. When current flows in a wire, the resistance of the wire causes a voltage drop along the wire, as a result electrical energy is lost. This lost electrical energy is converted into thermal energy called Joule heating. Several numerical and experimental methods have been developed to explore flow characteristics inside cavities with and without obstacle. These geometries have realistic engineering and industrial purposes, for example in the design of solar collectors, thermal design of building, air conditioning, cooling of electronic devices, furnaces, lubrication technologies, chemical processing equipment, drying technologies etc. Many authors have recently analyzed heat transfer in enclosures with partitions, fins and block which manipulates the convective flow phenomenon. A vital application is the use of MHD (magnetohydrodynamics) acceleration to shoot plasma into fusion devices or to produce high energy wind tunnels for simulating hypersonic flight. These types of problems also arise in electronic packages, micro electronic devices during their actions.

House et al. [1] analyzed the effect of a centered, square, heat conducting body on natural convection in a vertical enclosure. They showed that heat transfer across the cavity enhanced or reduced by a body with a thermal conductivity ratio less or greater than unity. The buoyancy driven convection in a rectangular enclosure with a transverse magnetic field was studied by Garandet et al. [2]. The geometry considered in the numerical study of Oh et al. [3] was that the conducting body generated heat within the cavity. Under these situations, the flow was driven by a temperature difference across the cavity and a temperature difference caused by the heat-generating source. Roychowdhury et al. [4] rigorously investigated the natural convective flow and heat transfer features for a heated cylinder placed in a square enclosure with different thermal boundary conditions. Natural convection in a horizontal layer of fluid with a periodic array of square cylinder in the interior was conducted by Ha et al. [5], in which the transition of the flow from quasi-steady up to unsteady convection depended on the presence of bodies and aspect ratio effect of the cell.

Tsay et al. [6] analyzed the thermal and hydrodynamic interactions among the surface-mounted heated blocks and baffles in a duct. They focused particularly on the effects of the height of baffle, distance between the heated blocks, baffle and number of baffles on the flow structure and heat transfer characteristics for the system at various *Re* and Gr/Re^2. The problem of natural convection in a horizontal layer of fluid with a periodic array of square cylinders in the interior was considered by Lee et al. [7]. Later, Shokouhmand and Sayehvand [8] carried out the numerical study of flow and heat transfer in a square driven cavity. They exposed that at the higher values of Reynolds number, an inviscid core region was developed, but secondary eddies were present in the bottom corners of the square at all Reynolds numbers. Roy and Basak [9] explored finite element method of natural

☆ Communicated by W.J. Minkowycz.
E-mail address: rehena@math.buet.ac.bd.

0735-1933/$ – see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.icheatmasstransfer.2011.02.005