Inclination Angle Effects On Heat Transfer In A Semi-Circular Enclosure

Falah A. Abood[1], Sana M. Shrama[1], Amani J. Majeed[2], and Ahmed M. Al-Mukhtar[3,4*]

[1] *Mechanical Engineering Department, College of Engineering, University of Basrah, Basrah, Iraq*
[2] *Petroleum Engineering department, College of Engineering, University of Basrah, Basrah, Iraq*
[3] *Al-Hussain University College, Iraq*
[4] *Institute of Structural Mechanics, Bauhaus-Universitat Weimar, Germany*
[*] *Corresponding author. E-mail: almukhtar@hotmail.de*

Received: Oct. 27, 2022; Accepted: Jan. 17, 2023

The investigation of the effects of rotary cylinders on the transfer of heat in a semicircular cavity occupied with air is presented in this paper. A constant and 2D laminar flow was assumed. Moreover, a Boussinesq approximation is applied to calculate density variations with temperature. Using a flexpde software program to solve the governing equations of energy, momentum, and continuity. The studied parameters are the rotational speed ($\Omega = 0 - 2000$), the angle of inclination ($\gamma = 0 - 90°$), Rayleigh number (10^4-10^7), and Prandtl number (0.7). The findings indicate that the average Nusselt number decreases with the increase in inclination angle, as the greatest value of the Nusselt number was found for an angle of inclination close to 0°. Furthermore, for the range of inclination angles between $0° - 45°$, the average Nusselt number increases gradually with increasing angular velocity values. The results have been benchmarked and a good agreement has been obtained. The objective of this study is to scrutinize the effect of the angle of inclination, and the rotational position of the cylinder on heat transfer. The largest and lowest values of the average Nusselt number take place at a location 0.1 and 0.2 of the cylinder center.

Keywords: Nusselt number; combined convection; rotating cylinder; inclined semi-circular cavity.

http://dx.doi.org/10.6180/jase.202311_26(11).0015

1. Introduction

The mixed convection heat transfer throughout the cavities has a variety of engineering and industrial applications. For instance its use by electronic cooling and in the manufacture of solar collectors and building insulation [1–3]. The effects of convective flows in cavities, and ducts have been studied using the experimental procedures and numerical simulations showing the effect of a rotating body inside the cavity; see Refs [4–6]. By rotating the cylinder, the fluid and heat exchange naturally occur both through conductive and convective, without any thermal conditions being imposed at its surface.

The numerical studies for natural and mixed convection with rotating and stationary cylinder are carried out; see Refs [7, 8]. The range of Ra ($10^4 - 10^6$), Pr (0.07–7.0) and the cylinder-to-enclosure aspect ratio have been used [8]. The surface roughness has effects on turbulent flow through a rotating cylinder [9, 10]. Researchers found that the drag coefficients reduced as the Reynolds number increased because of the turbulence layer that moved the separation bubble toward the end of the body.

The mixed convection heat transfer properties inside a trapezoidal cavity for rotating heated cylinders are studied in Ref. [11]. Both the inertia of the rotating cylinder as well as the sidewall angles influence the properties of heat transfer more than a motionless heated cylinder in a square cavity [11, 12]. However, numerical studies have been conducted on Nusselt number (Nu) in rotating circular cylinders [13, 14]. Nu for the fixed rotation rate increases as the Reynolds number or as the Richardson number increases.