International Communications in Heat and Mass Transfer 98 (2018) 143–154

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Transient behaviors of mixed convection in a square enclosure with an inner impulsively rotating circular cylinder

Wei Zhang*, Yikun Wei, Hua-Shu Dou, Zuchao Zhu

Faculty of Mechanical Engineering and Automation, Zhejiang Sci-Tech University, Hangzhou, Zhejiang 310018, China

ARTICLE INFO

Keywords:
Mixed convection
Rotating cylinder
Enclosure
Transient
Shear stress

ABSTRACT

This work numerically investigated the unsteady mixed convective heat transfer between a square enclosure and an inner concentric circular cylinder maintained at different temperatures. The cylinder undergoes a transition of motion whose physics is significant for engineering applications such as cooling of rotating shaft and lubricating of journal bearing. It keeps stationary at first and steady-state natural convective flow is achieved, then impulsively rotates at a constant angular velocity and the flow arrives at a new final steady-state. Although the mixed convective heat transfer for rotating cylinder configurations are analyzed in a number of works, the transient behaviors of the thermal and flow characteristics from the onset of cylinder rotation to the final steady-state are not investigated, which restrains the thorough understandings on the relevant flow physics and mechanisms. The present work performs the first numerical investigation on the targeted physical problem. The objective is to explore the transient characteristics of the mixed convective heat transfer, and to identify the effects of cylinder rotation and rotation speed (tangential velocity at cylinder surface, V_0) on the flow and heat transfer behaviors. The results are presented and analyzed by the temporal variations and spatial distributions of the thermal field, mean heat transfer rate, local heat transfer rate and velocity/temperature in certain regions where the flow is significantly affected. Our numerical results for the first time identify two flow regimes depending on the tangential velocity: *mixed buoyancy-shear stress dominated flow* for $V_0 \leq 1.0$ where the effect of natural convection is pronounced and the flow is driven by both buoyancy and shear stress from the cylinder surface, and *shear stress dominated flow* for $V_0 \geq 2.0$ where the flow is almost entirely driven by rotating cylinder and heat transfer is realized by conduction. The transient behaviors of the thermal and flow quantities are characterized and categorized based on the two regimes.

1. Introduction

Natural convection in a rectangular enclosure is one of the most fundamental and classical problems in fluid mechanics and heat transfer. The physical problem has been studied in a variety of numerical works since de Vahl Davis [1], and is also widely encountered in industrial and civil engineering circumstances. In most situations, the enclosure is heated at certain sidewalls and cooled at the remained ones, while the sidewalls can also be adiabatic. The buoyancy arising from temperature variation drives the fluid to circulate within the enclosure, thus heat transfer occurs between the cold and hot sidewalls. Characterized by the Rayleigh number, the heat transfer is primarily dominated by conduction in the low-*Ra* regime and the isotherms are generally parallel with the enclosure sidewalls, while in the high-*Ra* regime the heat transfer is convection dominated and the isotherms are deviated from each other and become approximately horizontal at the center of enclosure (thermal stratification) [1].

In many engineering applications, the configuration could be more complex with additional entities within the enclosure, and the fluid circulation and heat transfer are highly integrated. Kim et al. [2] studied natural convection in a square enclosure with an inner circular cylinder located at its vertical centerline at $Ra = 10^3$–10^6; the enclosure and cylinder are kept at low and high temperatures, respectively. The flow is symmetric about the vertical centerline for this *Ra*-regime, and two or four vortices may form in the fluid domain depending on the vertical position of the cylinder. The physical problem is different from the differentially heated enclosure configuration [1] in that the flow is confined in a smaller domain, and is characterized by several parameters including the diameter and position of the cylinder, the Rayleigh number and the Prandtl number. Considering its significance in relevant engineering applications, problems with similar geometrical and physical configurations have also been investigated (e.g., [3,4]).

---

* Corresponding author.
E-mail address: zhangwei@zstu.edu.cn (W. Zhang).

https://doi.org/10.1016/j.icheatmasstransfer.2018.08.016

During the past two decades, there were a number of studies, mostly carried out by numerical simulations, focused on the mixed convective heat transfer in a square enclosure with an inner rotating circular cylinder. This model is different from the stationary cylinder case in that the shear stress from the surface of rotating cylinder also contributes to the fluid circulation in addition to the buoyancy. Many practical applications in industrial and civil engineering circumstances can be simplified to this model, such as mixing chambers, rotating-tube heat exchangers, lubricating of journal bearings and cooling of rotating shafts. For example, for the air lubricating journal bearing working at room temperature, the dimensional values of the enclosure diameter, temperature difference and lubricant viscosity are in the order of $O(10^{-1})$ meter, $O(10^{1})$°C and $O(10^{-5})\mathrm{m}^2\text{•}\mathrm{s}^{-1}$, respectively, thus the Rayleigh number is order of $O(10^5–10^7)$ where buoyancy takes significant effect as well as the shear stress depending on the rotating velocity. The understandings of the transient characteristics of the thermal and flow processes could help to optimize the design and operational programs for better performance of the system. Ghaddar & Thiele [5] studied mixed convection in a vertical rectangular enclosure with an inner eccentrically placed rotating circular cylinder. The enclosure walls are isothermal, while the cylinder surface is maintained at a constant heat flux. Compared with the stationary cylinder case, the temperature distribution on the cylinder surface is more uniform. The heat transfer is enhanced and weakened at low and high Rayleigh numbers, respectively. Fu et al. [6] investigated mixed convective flow in a differentially heated enclosure with a heated inner rotating circular cylinder. They found that the rotation direction of the cylinder is crucial for this specific configuration: the heat transfer is enhanced if the tangential speed at the cylinder bottom is aligned with the temperature gradient. The same problem is numerically studied by Costa & Raimundo [7] who were the first to use the heatline visualization technique to analyze the two-way heat transfer interactions between the fluid and solid walls. It was found that the cylinder size affects the heat transfer by reducing the fluid domain. For small cylinders, the mean Nusselt number is the maximum if the cylinder is stationary, while the heat transfer rate increases substantially with rotation speed for large cylinders. For different nanofluids, Roslan et al. [8] found that the strength of fluid circulation is stronger for high nanoparticle concentration, high thermal conductivity and small cylinder diameter. Liao & Lin [9] analyzed mixed convection between a cold enclosure and a hot cylinder using the immersed boundary method, and explored the effects of Prandtl number, Rayleigh number and cylinder diameter. The mean Nusselt number only slightly varies with the Richardson number (Ri) in the conduction dominant regime at $Ra = 10^4$, while it greatly increases with Ri for the convection dominant flow at $Ra = 10^6$, and large Nusselt number is obtained for small cylinder cases. Chatterjee et al. [10] studied the mixed convection of nanofluid in a lid-driven square enclosure with an inner rotating circular cylinder. The flow is affected by Richardson number, cylinder rotation speed and nanoparticle concentration. They found that the heat transfer is more pronounced for high rotation speed, but is not necessarily enhanced by the cylinder rotation since other parameters also affect. In the work of Chatterjee & Halder [11], the hydromagnetic mixed convection in a square enclosure with two inner rotating cylinders is studied. The flow is determined by the Rayleigh number, Hartmann number and rotation speed. The flow is stabilized by thermal and magnetic fields, but becomes more unstable at large Hartmann number and exhibits multicellular structure. In addition to the circular geometry, there are also numerical studies on rotating square [12,13] or triangular [13] cylinder in an enclosure. For a three-dimensional rotating curved rectangular duct, Islam et al. [14] and Mondal et al. [15] investigated the transient thermal and flow processes by using the high accuracy spectral method with consideration of the centrifugal, Coriolis and buoyancy forces. The flow undergoes different variation scenarios with the rotating speed as the duct rotates in the same or opposite directions with the main flow. Multi-cellular flow structure is also observed that the convective heat transfer is enhanced by the secondary flows and in the chaotic flow regime.

The earlier studies, some of which were reviewed above, undoubtedly demonstrate the significant effect of cylinder rotation on the flow patterns and heat transfer characteristics. However, most of the relevant studies only focused on the final steady-state flow adapting to the cylinder rotating at a constant speed, while the transient behaviors of the flow and heat transfer processes from the onset of cylinder rotation to the final steady-state are not considered. This is crucial for the present physical configuration in that the time-dependent flow may approach to the final steady-state through various ways and exhibits entirely different transient patterns, and the flow and temperature characteristics may experience particular global and local variations that cannot be intuitively anticipated from the initial and final solutions nor based on experience. The present work performed the first numerical investigation on the transient mixed convection heat transfer in a square enclosure with a concentric rotating circular cylinder. All the walls of the enclosure are kept at a low temperature, while the cylinder is maintained at a constant high temperature. The cylinder is first stationary and initial steady-state natural convective flow is obtained, and then it impulsively rotates about its center at a constant angular speed, thus the flow and thermal patterns are consequently perturbed and will reach the final steady-state after a certain time period. The objective of our work is to reveal the transient behaviors during the transition from initial to final steady-state and correlate it with the cylinder rotation, and to identify the effects of rotation speed of the cylinder on the transient behaviors of flow and heat transfer characteristics. The effects are demonstrated by the temporal variations or distributions of characteristic quantities regarding the thermal and flow fields, mean and local heat transfer rates, and velocity and temperature distributions in regions where the flow is significantly affected.

2. Numerical setup

2.1. Problem description

The schematic of the physical model is illustrated in Fig. 1. A square enclosure of side length L is horizontally placed with a concentric inner circular cylinder of fixed diameter $D = 0.4\,L$. Both geometries are assumed sufficiently long in the axial direction (z-direction which is perpendicular to the paper) so that the fluid can be considered circulating within an annular channel where the end effects are omitted. The enclosure and cylinder are maintained at constant low temperature θ_c and high temperature θ_h, respectively, the approximation of which is consistent with the realistic condition that the thermal conductivity of solid is orders of magnitude that of fluid. The enclosure is filled with air ($Pr = 0.71$) whose thermophysical properties are assumed constant except for the density variation which follows the Boussinesq approximation. The inner circular cylinder is first stationary and steady-state flow and heat transfer patterns are obtained, then it impulsively rotates at a constant angular speed that the tangential velocity of the cylinder surface is V_0 and the flow becomes steady again, which is termed as final steady-state in the following sections.

All simulations are performed at a typical Rayleigh number $Ra = 10^6$ where the transient flow is regarded two-dimensional. It is well known that in real-life applications, the flow becomes three-dimensional at large Rayleigh numbers that the flow develops in the axial-direction of the annuli which cannot be resolved through two-dimensional simulations. However, it has been confirmed in a number of works in literatures that the two-dimensional simulation is valid at the present Rayleigh number and could reveal the primary flow physics for both stationary [2,3] and rotating [9] cylinder configurations. For a geometrically similar octagonal cylinder within a circular enclosure, Zhang et al. [16] experimentally investigated the circulating flow using the smoke visualization technique for Rayleigh number up to $Ra = 1.5 \times 10^6$ and observed the well two-dimensional flow structures.