International Journal of Heat and Mass Transfer 112 (2017) 185–200

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Mixed convection heat transfer of turbulent flow in a three-dimensional lid-driven cavity with a rotating cylinder

Ali Khaleel Kareem [a,b,*], Shian Gao [a]

[a] Department of Engineering, University of Leicester, Leicester LE1 7RH, United Kingdom
[b] Engineering Department, University of Thi-Qar, 64001 Nassiriya, Iraq

ARTICLE INFO

Article history:
Received 3 February 2017
Received in revised form 21 March 2017
Accepted 25 April 2017

Keywords:
Rotating cylinder
Lid-driven cavity
Mixed convection
Turbulent flow
URANS
LES

ABSTRACT

A numerical study has been carried out to investigate the combined forced and natural convection heat transfer in a differentially heated 3D obstructed cavity with a thermally insulated rotating circular cylinder. The cavity has a hot stationary bottom wall and a cold top lid-driven wall, and all the other walls completing the domain are motionless and adiabatic. The simulations are performed for different Reynolds numbers, $Re = 5000$, 10,000, 15,000 and 30,000, and for dimensionless rotational speeds of the cylinder, $0 \leq \Omega \leq 10$. The performance of two turbulence methods, Large Eddy Simulation (LES) and Unsteady Reynolds-Averaged Navier-Stokes (URANS), has been evaluated in this research. The flow and thermal fields are studied through flow vectors, isotherm contours and iso-surfaces temperature, as well as through the average Nusselt number (Nu_{av}) and velocity components. The results demonstrate clearly that the flow patterns and the thermal fields are influenced strongly by increasing either the rotating cylinder speed or the Reynolds number. Furthermore, both LES and URANS solutions can capture the essential feature of the primary eddies in the cavity. But this study has shown convincing evidence that only the LES method can predict the structure details of the secondary eddies that have profound effects on the heat transfer behaviour within the enclosure.

1. Introduction

Over the last few decades, the numerical simulations of turbulent flows and heat transfer have become one of the essential attentions of engineering applications in the industrial and engineering fields. Indeed, for high value of Reynolds number (turbulent flow), the expanded scope of turbulent vortices has gained major interests. Large Eddy Simulation (LES) method is increasingly involved in predicting the detailed turbulent flow scales [1,2], though the Reynolds Averaged Navier-Stokes (RANS) approach is still a useful tool in evaluating the time averaged features of the turbulent flows [3–6]. Even though the LES costs more computational resources than the RANS modelling, this method performs better in terms of data availability and accuracy [7].

The effect of combined natural convection, which emerges as a sequence of buoyancy effects, with forced convection, which occurs as a result of the fluid motion due to shear forces that are offered by external means such as the partial physical motion of the domain, is defined as mixed convection. There have been quite a few studies over the last few years on heat convection problems in the 2D cavities containing either a rotating or stationary cylinder with different dimensionless diameters and boundary conditions. A 2D combined convection heat transfer of heated top lid-driven wall cavity that has an internal central circular cylinder and heater was simulated by Ray and Chatterjee [8]. It was demonstrated that the internal circular objects lead to a considerable increment in the Nusselt number (Nu). A study of mixed and natural convection of stationary and rotating centred cylinder in a 2D square cavity, with different rotating speeds, was carried out by Liao and Lin [9]. It was concluded that by reducing the Richardson number (Ri) the mean value of Nusselt number (Nu_{mean}) decreases. Heat transfer was enhanced by using small aspect ratio between the inner cylinder and the outer cavity which leads to generating a greater Nu. Hydro-magnetic mixed convection heat transfer in a 2D moving wall cavity with central rotating conducting solid cylinder was demonstrated numerically by Chatterjee et al. [10]. It was summarised that an increase in rotation of the conductive cylinder generates the enhancement of the heat transfer within the cavity.

Hussain and Hussein [11] numerically simulated laminar steady state mixed convection of air within differentially heated cavity containing conductive rotating cylinder. It can be pointed out that when the forced convection dominates, major vortices were

* Corresponding author at: Department of Engineering, University of Leicester, Leicester LE1 7RH, United Kingdom.

E-mail addresses: akkak2@le.ac.uk, alikhaleel17@yahoo.com (A.K. Kareem).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.04.118
0017-9310/© 2017 Elsevier Ltd. All rights reserved.

Nomenclature

CFL	Courant–Friedrichs–Lewy number
D	width of the cavity on z-axis (m)
d	cylinder diameter (m)
FVM	finite volume method
Gr	Grashof number ($g\beta_m \Delta T W^3/\nu_m^2$)
h	convective heat transfer coefficient (W/m^2 K)
k	turbulent kinetic energy (m^2/s^2)
L	width of the cavity on x-axis (m)
Nu	Nusselt number
Pr	Prandtl number (ν_m/α_m)
Ra	Rayleigh number (*Gr Pr*)
Re	Reynolds number ($U_{0,m}W/\nu_m$)
Ri	Richardson number (Gr/Re^2)
$\bar{S}_{ij}$	large-scale strain rate tensor for grid-filter
T	temperature of the fluid (K)
t	time
u	velocity component at x-direction (m/s)
U	dimensionless velocity component at x-direction
U_0	lid velocity (m/s)
v	velocity component at y-direction (m/s)
V	dimensionless velocity component at y-direction
W	dimensionless velocity component at z-direction
x	distance along the x-coordinate
X	distance along the non-dimensional x-coordinate (x/L)
Y	distance along the non-dimensional y-coordinate (y/H)
Z	distance along the non-dimensional z-coordinate (z/D)

Greek symbols

α	thermal diffusivity of the fluid (m^2/s)
β	volumetric coefficient of thermal expansion (1/K)
μ	dynamic viscosity of the fluid (Pa/s)
ν	kinematic viscosity of the fluid (m^2/s)
ν_{sgs}	sub-grid scale (SGS) viscosity
ρ	density of the fluid (kg/m^3)
ε	dissipation rate of turbulent kinetic energy (m^2/s^3)
δ_{ij}	Kronecker's delta
$\bar{\Delta}$	grid-filter width
τ_{ij}	subgrid-scale (SGS) stress tensor
ω	rotational speed (rad/s)
Ω	dimensionless rotational speed

Subscripts

av	average value
b	buoyancy
C	value of cold temperature
H	value of hot temperature
rms	root mean square
sgs	sub-grid scale
t	turbulent

founded around the cylinder. No influences were noticed on both the flow and thermal fields when changing the cylinder location at equal domination between the natural and forced convection. A natural convection of a 2D square cavity with cold and hot cylinders was numerically investigated by Park et al. [12]. Different locations of cold and hot cylinders were the main concern. The outcome showed that when the surfaces of the cylinders and the cavity were close to each other the Nusselt number increases. An unsteady natural convection of two heated horizontal rotating cylinders that erected within a 2D closed square cavity was simulated by Karimi et al. [13]. It was observed that at a low Rayleigh number (*Ra*) (less than 10^4) the distance between the cylinders has a clear effect on the averaged-area of the Nusselt number. When the Rayleigh number is higher than 10^4 and no more than 10^7, the influence of spacing between the circular cylinders could be ignored. A study of 2D natural convection in a closed square cavity with two horizontal inner cylinders was numerically carried out by Yoon et al. [14]. The upper cylinder was cooled, while the lower one was heated. The lower and upper half of the cavity was the place of the equidiameter cylinders. It was concluded that an increase in the radius of the cylinders at all values of the Rayleigh number drives the increment of the heat transfer rate and dominates the cold upper circular cylinder on a wider area.

An inner sinusoidally heated circular cylinder placed in a cavity was involved in a study of a 2D numerical unsteady natural convection heat transfer by Roslan et al. [15]. It can be pointed out that the flow field has two inner vortices and a heated cylinder provided a warm-chamber, which impacts on the heat transfer. Although the heat transfer was not changed by changing cylinder radius at the lowest value of the parameters, temporal increasing in the heat transfer was found by increasing the cylinder radius to the maximum value of the dimensionless parameters. In addition, it was observed that oscillating heat source of the cylinder caused augment in the heat transfer rate. A natural convection of circular cylinder within a 2D rhombus enclosure filled by water was numerically observed by Choi et al. [16]. It was noticed that the thermal features of the heat transfer between the cavity and its cylinder stick in accordance with the value of the Rayleigh number and the cylinder location. Increasing Rayleigh number would therefore lead to an increment in the Nusselt number for both the enclosure and its cylinder. In addition, when the cylinder is located on the bottom wall of the rhombus cavity, the Nusselt numbers for both the enclosure and its cylinder reach the maximum values. The lowest value of the Nusselt numbers occurred when the cylinder was nearby the inner top of the cavity. The research of a heated hollow cylinder within the middle of the moving wall enclosure was completed by Billah et al. [17] at different range parameters, including the size diameter of the cylinder, the Richardson number and the thermal conductivity of the fluid. The sequences of installation cylinder on the mixed convection heat transfer coefficient were mainly targeted. It has been noticed that a significant influence of the cylinder on the heat transfer ratio as well as on the cylinder diameter size occurred. Khanafer and Aithal [18] evaluated a laminar combined convection heat transfer and flow patterns of moving wall cavity that has a central cylinder. It was concluded that the heat transfer fields can be controlled by the cylinder body within the cavity. The obstacle size and location can affect the heat transfer and flow characteristics. Laminar mixed convection of a heated square blockage within moving wall enclosure was studied numerically by Islam et al. [19] in order to understand the effects of the central and eccentric locations of the square body at different sizes as well as the constant Reynolds number on the heat transfer and flow patterns. It was observed that at the domination of the forced convection there were no clear differences in the heat transfer when changing either the location or the size of the installed body. An obvious influence was noticed on the Nusselt number when the natural convection was controlling the domain.

A combined convection heat transfer of nanofluid in a 2D moving wall square cavity that contains a rotating cylinder was studied