Computers & Fluids 109 (2015) 27–37

Contents lists available at ScienceDirect

Computers & Fluids

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

Numerical investigation and reduced order model of mixed convection at a backward facing step with a rotating cylinder subjected to nanofluid

Fatih Selimefendigil [a,*], Hakan F. Öztop [b]

[a] Department of Mechanical Engineering, Celal Bayar University, Manisa, Turkey
[b] Department of Mechanical Engineering, Technology Faculty, Fırat University, Elazığ, Turkey

ARTICLE INFO

Article history:
Received 13 January 2014
Received in revised form 20 November 2014
Accepted 4 December 2014
Available online 16 December 2014

Keywords:
Rotating cylinder
Nanofluid
Backward facing step
POD

ABSTRACT

In the present study, numerical investigation of mixed convection at a backward facing step with a rotating cylinder subjected to nanofluid is conducted and a reduced order model of the system is obtained by using the proper orthogonal decomposition method. The governing equations are solved with a finite element based commercial solver. The effects of various pertinent parameters, Reynolds number, cylinder angular velocity and nanofluid volume fraction on the fluid flow and heat transfer characteristics are numerically studied. It is observed that flow field and thermal patterns change for different parameters and heat transfer enhancement is obtained for some combinations of parameters. Length averaged Nusselt number plots indicate that there is almost a linear increase in the heat transfer enhancement with increasing Reynolds number and nanoparticle volume fraction. Heat transfer enhancement is obtained for cylinder angular velocities of $\Omega = -4.5$ and $\Omega = 1.5$. A reduced order model of the system with proper orthogonal decomposition method is obtained and it provides accurate results when compared to high fidelity CFD model of the system.

© 2014 Elsevier Ltd. All rights reserved.

1. Introduction

Flow separation and its subsequent reattachment is of vital importance for many engineering applications such as flow around airfoils, buildings and collectors of power systems. The flow over a backward facing or forward facing step is a problem where both flow separation and reattachment occur. A comprehensive review is presented in Abu-Mulaweh [1] for laminar mixed convection over vertical, horizontal and inclined backward- and forward-facing steps. Numerical and experimental studies for heat transfer and fluid flow characteristics related to the backward or forward facing step have been investigated by many researchers in laminar or turbulent flow regimes, for 2D or 3D configurations [2–7].

Heat transfer and fluid flow characteristics over a backward or forward facing step in a channel with the insertion of obstacles has received less attention in the literature [8–10]. Yilmaz and Oztop [11] have numerically investigated the turbulent forced convection over double forward facing step by using adiabatic steps with different lengths and heights.They reported that the second step can be used to control the fluid flow and heat transfer characteristics. Oztop et al. [12] have numerically studied the turbulent forced convection of double forward facing step with obstacles. They used rectangular obstacles before each step to control the fluid flow and heat transfer characteristics. Kumar and Dhiman [13] have numerically studied the heat transfer enhancement in laminar forced convection flow over a backward facing step with the insertion of an adiabatic circular cylinder. They considered different cross-stream positions of the circular cylinder for the Reynolds number between 1 and 200. They obtained heat transfer enhancement up to 155% compared to no-cylinder case. Several studies have been conducted to investigate the mixed convection in enclosures with rotating or stationary cylinders. Costa and Raimundo [14] have numerically studied the mixed convection in a differentially heated square enclosure with an active rotating circular cylinder. The effects of the radius, cylinder angular velocity and thermal conductivity and thermal capacity of the cylinder on the mixed convection problem is studied. Hussain and Hussein [15] have numerically investigated the mixed convection in an enclosure with a rotating cylinder using finite volume method. Their result showed that cylinder locations have an important influence in enhancing the convection heat transfer in the square enclosure. Paramane and Sharma [16] have studied the forced convection heat transfer across a rotating circular Yan and Zu cylinder with a constant non-dimensional rotation rate for Reynolds number between 20 and 160. They reported that rotation can be used as a drag reduction and heat transfer suppression technique. Yan

---

* Corresponding author.
E-mail addresses: fatih.selimefendigil@cbu.edu.tr (F. Selimefendigil), hfoztop1@gmail.com (H.F. Öztop).

http://dx.doi.org/10.1016/j.compfluid.2014.12.007
0045-7930/© 2014 Elsevier Ltd. All rights reserved.

Nomenclature

g	gravitational acceleration (m/s^2)
Gr	Grashof number, $g\beta\Delta TH^3/\nu^2$
h	local heat transfer coefficient (W/m^2 K)
k	thermal conductivity (W/m K)
H	step size (m)
n	unit normal vector
Nu	local Nusselt number, hH/k
p	pressure (Pa)
Pr	Prandtl number, $\frac{\nu}{\alpha}$
Re	Reynolds number, $\frac{u_0 H}{\nu}$
T	temperature (K)
u, v	x–y velocity components (m/s)
u_0	average velocity (m/s)
x, y	Cartesian coordinates (m)

Greek characters

α	thermal diffusivity (m^2/s)
β	thermal expansion coefficient (1/K)
θ	non-dimensional temperature, $\frac{T-T_c}{T_h-T_c}$
ν	kinematic viscosity (m^2/s)
ρ	density of the fluid (kg/m^3)
Ω	nondimensional angular velocity of cylinder, $\frac{\omega H}{2u_0}$
ϕ	nanoparticle volume fraction

Subscripts

c	cold
h	hot

and Zu [17] have studied the heat transfer and fluid flow past a rotating isothermal cylinder using LBM method. The effects of the peripheral-to-translating-speed ratio, Reynolds number and Prandtl number on flow and thermal fields are discussed. Rehimi et al. [18] studied experimentally the flow past a circular cylinder located between parallel walls for Reynolds number between 30 and 277. They reported differences between the unconfined cylinder case. They observed that von-Karman instability is shifted to a higher Reynolds number for the confined cylinder case. Jiang and Lin [19] have studied the flow past two tandem cylinders of different diameters in a centrally placed channel using LBM method for Reynolds number between 20 and 120. Sahin and Owens [20] have used finite volume method to investigate the flow field around a circular cylinder placed in a channel. The effect of the lateral wall proximity on the stability, wake structure behind the cylinder is investigated for a range of blockage ratio at Reynolds up to 280. They observed different bifurcations and transition region from symmetric vortex shedding to asymmetric vortex shedding with increasing blockage ratio. Singha and Sinhamahapatra [21] have numerically investigated the flow about a circular cylinder placed in a channel using a finite volume based method. They performed simulations up to Re = 250 and the blockage ratio is varied using the channel height. They observed that due to the interaction between the cylinder wake and channel wall, transition to vortex shedding regime is delayed.

In heat transfer applications, nano-sized particles (average particle size less than 100 nm) are added in the base fluid such as water or ethylene glycol to obtain better thermal properties compared to base flow. Nanofluids have improved heat transfer characteristics with little pressure drop as compared to base fluids [22–24]. Roslan et al. [25] have theoretically studied the convective heat transfer in a differentially heated square enclosure with an inner rotating cylinder. The free space between the cylinder and the enclosure walls is filled with different types of nanofluids. Their results showed that the maximum heat transfer values are obtained at high nanoparticle concentration. Kamyar et al. [26] have revised the literature related to computational fluid dynamics application of nanofluids.

To the best of authors' knowledge a study of laminar mixed convection over a backward facing step in a channel with a rotating cylinder subjected to nanofluid has never been studied in the literature. In the present study, our aim is to investigate the effects of cylinder angular velocity, nanoparticle volume fraction and Reynolds number on the fluid flow and heat transfer characteristics. Moreover, a reduced order or surrogate model of the high fidelity CFD computations for the range of the parameters will be obtained by using the proper orthogonal decomposition.

2. Numerical simulation

2.1. Problem description

A schematic description of the physical problem considered in this study is shown in Fig. 1. A channel with a backward facing step is considered. The step size of backward facing step is H and channel height is $2H$. At the inlet of the channel, a parabolic velocity (U) and a uniform temperature ($T = 300$ K) are imposed. The downstream length starting from the edge of the step to the exit of the channel is $35H$ to ensure that the recirculation length downstream of the step is independent of the computational domain. The downstream bottom surface of the backward facing step is maintained at $T = 310$ K, while the other walls of the channel are assumed to be adiabatic. An adiabatic rotating cylinder with diameter ($D = H$) is mounted at the location $(x_c, y_c) = (H, H)$ where the coordinate system is positioned at the step on the bottom wall of the channel. In the study by Singha and Sinhamahapatra [21], the effect of the channel height (h) to cylinder diameter (D) ratio on the fluid flow is investigated for Reynolds number between 45 and 200. The h/D value is between 2 and 8. In their study, it is shown that at Re = 100, when $h/D \leqslant 3$, then steady flow conditions are observed. They also showed that at Re $\geqslant$ 150, the critical value for the h/D is below 2. In our case, h/D ratio is kept at 2. We confirm the steady nature of the flows for our parametric ranges by comparing the results of unsteady and steady simulations. In the study by Kumar and Dhiman [13], the vertical position of the cylinder at backward facing step is also investigated. In our case, we can also vary the cylinder position, but our aim in this study is to investigate the effect of other parameters (cylinder angular velocity, nano-particle volume fraction and Reynolds number) on the flow and heat transfer characteristic. The cylinder location is chosen such that the top of the cylinder is located above the top surface of the step and the fluid flow over and under the cylinder in the vicinity of the step will be affected to some extent. With these choice of the cylinder location parameters, the flow field downstream of the step is influenced by the presence of the cylinder. For an unconfined case, the wake of the cylinder for Re = 250 may have three dimensionality and the flow may be turbulent [21]. But, the existence of the wall in the vicinity of the cylinder for a confined case enhances the stability properties of the wake

Download English Version:

https://daneshyari.com/en/article/761487

Download Persian Version:

https://daneshyari.com/article/761487

Daneshyari.com