Advanced Powder Technology 23 (2012) 736–743

Contents lists available at SciVerse ScienceDirect

Advanced Powder Technology

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

Original Research Paper

Numerical investigation on deposition of solid particles in a lid-driven square cavity with inner heated obstacles

V. Golkarfard [a,*], S.A. Gandjalikhan Nasab [a], A.B. Ansari [a], G.H. Bagheri [b]

[a] Department of Mechanical Engineering, Shahid Bahonar University of Kerman, P.O. Box 76169-133, Kerman, Iran
[b] Section des sciences de la Terre et de l'environnement, Université de Genève, Switzerland

ARTICLE INFO

Article history:
Received 4 February 2011
Received in revised form 28 August 2011
Accepted 28 September 2011
Available online 12 October 2011

Keywords:
Cavity
Laminar mixed convection flow
Particle deposition
Thermophoresis effect
Obstacle

ABSTRACT

The deposition of aerosol particles in laminar mixed-convection flow in a lid-driven cavity with two heated obstacles is investigated numerically by an Eulerian–Lagrangian method. Lagrangian particle transport calculations are carried out to track 2000 particles that initially exerted with random distribution in flow regime and also assumed that the effect of particles on the fluid is neglected. All the affecting forces on particle equation of motion, such as Brownian, thermophoresis, drag, lift and gravity are considered. The main goal is to study the effective parameter on deposition of particles such as free convection, distance and size variation of obstacles. Numerical results showed that free convection is an effective parameter that affected the deposition. As a main result, it is observed that deposition decreases with increasing in the Richardson number. Results showed that by increasing obstacles distance, deposition increases. Finally, it is revealed that the size of obstacles has a great effect on particle deposition, such that by increasing the obstacles size, the deposition increases.

1. Introduction

Many researches have been carried out on the fluid flow and heat transfer in enclosed environments with a rigid obstacle inside. These types of situations occur in many engineering applications especially in heat exchangers such as cooling systems in reactors, cooling of the electrical components and heating, ventilation and air conditioning (HVAC) systems [1–4]. Increasing heat transfer from a heated body inside an enclosure is in the most interest in such applications, either this body should be cooled, e.g. electrical circuits, or should heat the environment, e.g. radiators inside buildings.

Particle deposition on heating or cooling systems can significantly increase energy consumption and decrease heating or cooling performance [1,5] and in some cases may results in the failure of the system [6–8]. For example, repair cost of failed electronic devices caused by indoor air pollutants, only in U.S. telephone offices, were around \$200 million annually at the beginning of 1990s [9]. Considering this value is only for a part of U.S offices and also since then using electronic devices increased dramatically, this value should be much greater all over the world nowadays.

Fluid flow and heat transfer inside a lid-driven cavity combined with its simple geometry, makes it appropriate for fundamental studies on the particle deposition on heated obstacles. However cavity's geometry appears simple, but its flow can involve many interesting phenomena that can possibly occur in laminar incompressible flows such as circulations, secondary flows and chaotic particle motion [10]. From the past, lid-driven cavity has been used for many applications like representing the new numerical scheme or validating. de Vahl Davis [11] represented the most significant review of numerical methods in rectangular cavities and many researchers use his results as a benchmark. Another interesting and important research fields in lid-driven cavity flow is temperature effects on fluid flow and it have been studied extensively in the literature. Oztop and Dagtekin [12] studied mixed convection in a lid-driven cavity with differentially heated walls in the range of $0.01 < Ri < 100$ and $Pr = 0.7$. They found that both direction of moving walls and Richardson number has an important role on the fluid flow and heat transfer in the cavity. Recently Hussain and Hussain [13] investigated on the numerical simulation of mixed convection in a square cavity with a rotating conductive circular cylinder inside the cavity as an obstacle. Their results showed that parameters such as Richardson number, Reynolds number and obstacle location has an important effect on the fluid flow and heat transfer inside the cavity.

There are some researches on the deposition of particles inside a lid-driven cavity without any heat transfer mechanism. Tsorng et al. [14] investigated an experimental research on the behavior of solid particles in an isothermal lid-driven cavity flow. They used

* Corresponding author. Tel.: +98 917 988 3458; fax: +98 341 212 0964.
E-mail address: vahid.golkarfard@gmail.com (V. Golkarfard).

0921-8831/\$ - see front matter © 2011 The Society of Powder Technology Japan. Published by Elsevier B.V. and The Society of Powder Technology Japan. All rights reserved.
doi:10.1016/j.apt.2011.09.006