European Journal of Mechanics B/Fluids 44 (2014) 69–81

Contents lists available at ScienceDirect

European Journal of Mechanics B/Fluids

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

On the steady flow in a rectangular cavity at large Reynolds numbers: A numerical and analytical study

E.M. Wahba *

Mechanical Engineering Department, American University of Sharjah, United Arab Emirates
Mechanical Engineering Department, Faculty of Engineering, Alexandria University, Egypt[1]

HIGHLIGHTS

- Simulations for rectangular cavity flows are reported up to Reynolds number 20000.
- The eddy structure in shallow and deep cavities is analyzed.
- An analytical model is developed for a large Reynolds number.
- Vorticity in the inviscid core is found to be a function of the cavity aspect ratio.

ARTICLE INFO

Article history:
Received 4 February 2013
Received in revised form
1 October 2013
Accepted 29 October 2013
Available online 6 November 2013

Keywords:
Cavity flow
High Reynolds number
Core vorticity

ABSTRACT

Steady flow in a rectangular cavity at high Reynolds numbers is numerically and analytically investigated. Numerical simulations are reported up to a maximum Reynolds number, Re, value of 15000 for deep cavities and 20000 for shallow cavities using a compact fourth-order accurate central difference scheme and a stream function–vorticity formulation. At high Reynolds numbers, the eddy structure in shallow cavities consists of counter-rotating primary eddies, with each eddy behaving as an inviscid core with uniform vorticity. For deep cavities, the increase in Reynolds number results in the growth and eventually merger of the corner eddies into new primary eddies. Two merger patterns are identified, a symmetric pattern and an asymmetric pattern depending on a local Reynolds number based on the properties of the bottom primary eddy. A cavity with effectively infinite depth, $D = 10$, is also numerically investigated up to a maximum Re value of 10000. Numerical results indicate that for an infinitely deep cavity and at a large Reynolds number, inertia effects would dominate near the upper moving wall, while Stokes flow behavior would dominate away from the moving wall. An overlap region would exist, in which both inertia and viscous effects are of comparable magnitude. Finally, an analytical solution is developed for the steady flow in a rectangular cavity at large Reynolds numbers. Results from the analytical model are compared to numerical solutions obtained from the full Navier–Stokes equations for both one-sided and four-sided driven cavity configurations.

© 2013 Elsevier Masson SAS. All rights reserved.

1. Introduction

Driven cavity flow has received considerable attention over the past fifty years, starting with the early work of Burggraf [1] for a square cavity. The flow configuration is relevant to a number of industrial applications such as solar collectors [2] and short-dwell coaters [3]. Computational fluid dynamics researchers use driven cavity flow as a classical benchmark problem for the assessment and validation of newly developed Navier–Stokes solvers. Flow inside a cavity is also used as a model problem to study flow stability, bifurcation and transition to turbulence through direct numerical simulations [4,5].

A schematic of the driven rectangular cavity is provided in Fig. 1. The velocity of the top side is defined as V_t, while the velocities of the bottom, left and right sides are defined as V_b, V_l and V_r, respectively. Flow dynamics in the cavity depend on two non-dimensional parameters, namely the Reynolds number, Re, and the cavity aspect ratio, D. Flow in a square cavity, $D = 1$, has been the subject of numerous studies. The first numerical results for the driven square cavity flow were reported by Burrgraf [1], spanning the creeping flow limit, $\mathrm{Re} = 0$, up to Reynolds number 400. The main characteristics of the flow in this Reynolds number range consist of a single primary eddy and two secondary eddies located at the bottom left and right corners of the cavity. Moreover, and for large Reynolds number, Burrgraf [1] coupled an inviscid rotational vortex to thin boundary layers at the cavity walls and evaluated

* Correspondence to: Mechanical Engineering Department, American University of Sharjah, United Arab Emirates. Tel.: +971 509410575.
E-mail addresses: emwahba@yahoo.com, ewahba@aus.edu.

1 On leave.

0997-7546/$ – see front matter © 2013 Elsevier Masson SAS. All rights reserved.
http://dx.doi.org/10.1016/j.euromechflu.2013.10.005

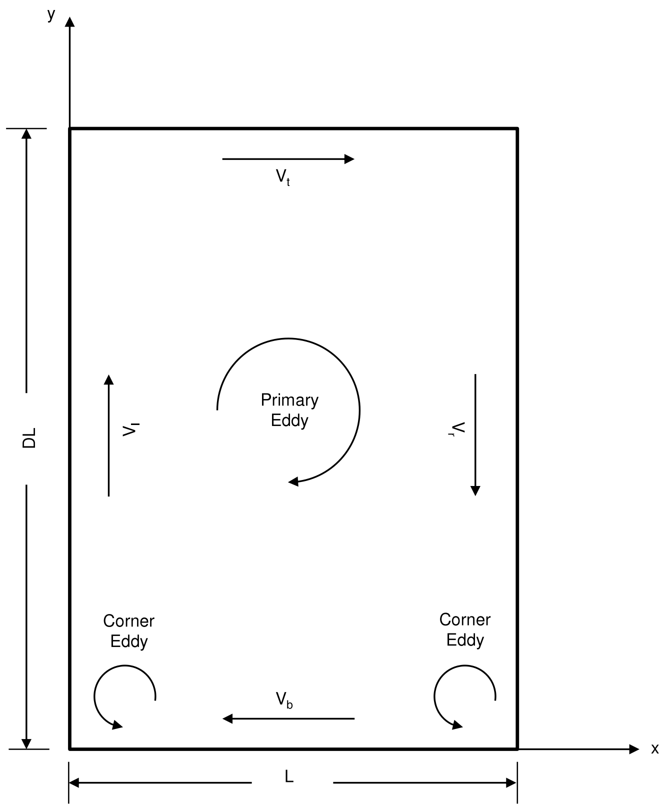

Fig. 1. Schematic of the driven rectangular cavity.

the core vorticity in the cavity to be 1.886. Numerical simulations at higher Reynolds numbers followed, showing that new tertiary eddies are formed at the bottom corners. Ghia et al. [6], Benjamin and Denny [7], Liao and Zhu [8], and Schreiber and Keller [9] presented simulations up to Re = 10000, while simulations up to Re = 20000 followed by Erturk et al. [10], Hachem et al. [11] and Botti and Di Pietro [12]. Recently, Wahba [13] reported steady flow simulations inside a square cavity up to Re = 35000.

On the contrary, and compared to the large number of papers devoted to the square cavity, a limited number of studies were carried out for rectangular cavity flows. Pan and Acrivos [14] carried out numerical investigations for Stokes flow in cavities with aspect ratios ranging from 0.25 to 5 by a relaxation method. In addition, experimental investigations were conducted for a Reynolds number range from 20 to 4000 for cavities of finite depth, as well as for cavities of effectively infinite depth, $D = 10$. Based on the experimental results, they stated that for cavities of infinite depth, viscous and inertia forces should remain of comparable magnitude throughout the whole cavity, even at a high Reynolds number. Shankar [15] developed a calculation procedure, based on an eigenfunction expansion, to study the eddy structure of Stokes flow in a rectangular cavity. He showed how the second primary eddy is formed from the merger of the two primary corner eddies. The secondary corner eddies then assume the role of the primary ones. As a result, and for an infinitely deep cavity, the structure of Stokes flow would be comprised of an infinite sequence of counter-rotating primary eddies of diminishing strength.

As pointed out in an excellent review paper by Shankar and Deshpande [16], very little work has been done on deep cavities at a high Reynolds number. They attributed the limited numerical work in this area to computational difficulties due to the slow penetration of the flow field into the cavity depth and the large number of grid points required, even for the creeping flow case. As the Reynolds number is increased, the nonlinear convection terms are introduced and more numerical difficulties arise. Recently, some studies tried to investigate this topic. Cheng and Hung [17] provided steady flow simulations up to a Reynolds number of 5000 and an aspect ratio of 7, while Patil et al. [18] presented numerical results up to a Reynolds number of 3200 and an aspect ratio of 4. Recently, steady flow simulations were reported by Lin et al. [19] using a multi-relaxation time lattice Boltzmann method up to Reynolds number 5000 and aspect ratio 4.

As can be seen from the above review, and up the author's best knowledge, numerical investigations for steady rectangular cavity flow in the literature are limited to a maximum Reynolds number value of 5000 and a maximum aspect ratio of 7. The objective of the present study is to numerically investigate how the eddy structure inside the cavity transforms beyond these limits. Specifically speaking, numerical simulations are performed for deep and shallow cavities up to Re = 15000 and 20000, respectively. Moreover, numerical simulations are carried out for a cavity of effectively infinite depth, $D = 10$, up to Reynolds number 10,000. The present study also addresses another intriguing question regarding the core vorticity value for rectangular cavity flows at a high Reynolds number. As mentioned above, Burrgraf [1] evaluated the theoretical core vorticity for a square cavity to be 1.886. The present study aims to provide similar evaluations for rectangular cavities of varying aspect ratios and multiple moving walls. This is done through the development of an analytical model for rectangular cavity flows at large Reynolds numbers.

2. Numerical model for rectangular cavity flow

In the present section, details of the governing equations and numerical methods are provided. Moreover, verification and validation procedures for the developed numerical model are also presented.

2.1. Governing equations and numerical methods

We consider steady two-dimensional flow of a viscous incompressible Newtonian fluid inside a rectangular cavity, in which the motion is generated by one or more of the cavity walls. The governing Navier–Stokes equations can be conveniently expressed in terms of a non-dimensional stream function–vorticity formulation as follows:

$$\frac{\partial^2\psi}{\partial x^2}+\frac{\partial^2\psi}{\partial y^2}=-\omega \tag{1}$$

$$u\frac{\partial\omega}{\partial x}+v\frac{\partial\omega}{\partial y}=\frac{1}{\mathrm{Re}}\left(\frac{\partial^2\omega}{\partial x^2}+\frac{\partial^2\omega}{\partial y^2}\right). \tag{2}$$

The normalization of the governing equations is carried out using the cavity length, L, as the length scale and the top wall velocity, V_t, as the velocity scale. The velocity components, u and v, are related to the stream function, ψ, through the following definitions:

$$u=\frac{\partial\psi}{\partial y},\qquad v=-\frac{\partial\psi}{\partial x}. \tag{3}$$

The compact fourth-order-accurate central difference scheme of Gupta et al. [20] is used to discretize the governing equations (1)–(2). This higher order compact (HOC) scheme is known to provide stable flow computations at high Reynolds numbers, as demonstrated recently by Wahba [13] who used the HOC scheme to compute steady flows inside a driven square cavity up to Re = 35000. The velocity components are evaluated using a compact fourth-order discretization [21] of Eq. (3). The no-slip boundary condition is applied at the cavity walls by enforcing a zero value for the stream function, and evaluating the wall vorticity using Jensen's formula [22].

Download English Version:

https://daneshyari.com/en/article/650350

Download Persian Version:

https://daneshyari.com/article/650350

Daneshyari.com