Reprint **ISSN 1997-2571 (Web Version)**

Journal of Innovation & Development Strategy (JIDS)

(J. Innov. Dev. Strategy)

Volume: 5 **Issue: 3** **December 2011**

J. Innov. Dev. Strategy 5(3):120-124(December 2011)

NUMERICAL SIMULATION OF TURBULENT FLOWS IN TWO-SIDED LID-DRIVEN SQUARE CAVITY BY FINITE ELEMENT METHOD

M. RASHID, A.B.M.A. KAISH AND M.M. ISLAM

GGF
Nature is Power

An International Scientific Research Publisher
Green Global Foundation©
Publication and Bibliography Division
100 Leeward Glenway
Apartment # 1601
M3c2z1, Toronto, Canada
E-mails: publication@ggfagro.com, editor@ggfagro.com
http://ggfagro.com/ejournals/current issues

JIDS** issn 1997-2571, HQ:19-10 central place, saskatoon, saskatchewan, s7n 2s2, Canada

ISSN-1997-2571 (Online)
J. Innov. Dev. Strategy 5(3):120-124(December 2011)

NUMERICAL SIMULATION OF TURBULENT FLOWS IN TWO-SIDED LID-DRIVEN SQUARE CAVITY BY FINITE ELEMENT METHOD

M. RASHID[1], A.B.M.A. KAISH[2] AND M.M. ISLAM[1]

[1]Deparment of Mathematics and Physics, Faculty of Computer Science and Technology, Hajee Mohammad Danesh Science and Technology University, Dinajpur, Bangladesh; [2]Graduate Student, Dept. of Civil & Structural Engineering, National University of Malaysia.

Corresponding author & address: Mamunur Rashid, E-mail: mamun_hstu@yahoo.com

Accepted for publication on 2 December 2011

ABSTRACT

Rashid M, Kaish ABMA, Islam MM (2011) Numerical simulation of turbulent flows in two-sided lid-driven square cavity by finite element method. *J. Innov. Dev. Strategy.* 5(3), 120-124.

The paper focuses on simulation of turbulent flow in a two-sided lid-driven square cavity containing an incompressible fluid using a finite element method (FEM). The flow is driven by the parallel motion to two facing walls. The top and bottom horizontal walls of the cavity are moving parallel to each one on its own plane at a constant while the left and right vertical walls are at rest. The governing equations are solved using finite element approach with a clear and simple statement using ANSYS 11.0 finite element software. A fine uniform grid mesh of 100X100 was used to simulate the flows. The numerical simulation is performed for a range of Reynolds number of 3500 to 6500 with an aspect ratio 1. Using ANSYS 11.0 finite element software, it is found that there appears a pair of counter-rotating secondary vortices of equal size near the centre of a wall. The ANSYS finite element program is useful in this study as well as varieties of enclosed fluid flows.

Key words: *turbulent flow, two-sided lid-driven square cavity, finite element method*

INTRODUCTION

The description of turbulence is a benchmark problem in flow engineering and theoretical research in fluid mechanics. It is still a big challenge not only for scientists but also for engineers though numerous efforts have been made on it from more than one hundred years ago (Chen 2009). The flow is turbulent when Reynolds number goes beyond 3000, with the transition to turbulence occurs within the Reynolds number range of 2000-3000. The realistic turbulence always is three-dimensional, which usually too expensive for available computer capability to simulate. The importance of two-dimensional turbulence are idealizes geophysical phenomena in the atmosphere, oceans and magnetosphere and provides a starting point for modeling these phenomena (Elkaim *et al.* 1992; Milane 2004; Chai *et al.* 2006).

Computational fluid dynamics (CFD) involves describing the fluid flow in terms of mathematical models that consist of governing equations in the form of ordinary or partial differential equations (Perumal and Dass, 2010). A finite element method (FEM) consists with setting up a uniform grid which is frequently used in CFD. In conventional numerical methods the macroscopic variables of interest, such as velocity and pressure are usually obtained by solving the Navier-Stokes equation. Such numerical methods for two dimensional steady incompressible Navier-Stokes equations are often tested for code validation.

The two-sided lid-driven square cavity flow has been used as a benchmark problem for many numerical methods due to its simple geometry and complicated flow behaviors. It is usually very difficult to capture the flow phenomena near the singular points at the corners of the cavity (Chen 2009). In two-dimensional analytical and numerical study of a lid-driven square cavity, Burggraf (1966) found the basic vortex to develop from a viscous eddy to an in-viscid rotational core at high Reynolds numbers. In addition to the primary vortex, secondary viscous eddies (Kuhlmann *et al.* 1997) were found to exist in the rigid corners.

Nomenclature

L	Length, cm	x, y	Spatial coordinates
Re	$\rho \times U \times L/\mu$ Reynolds number	μ	dynamic viscosity, gm/cms
T	Time, s	L_{wall}	Left wall
ρ	Density, gm/cm^3	R_{wall}	Right wall
u, v	Velocity of the fluid in x- and y-direction respectively	L_h	Length of the horizontal wall
V	Wall velocity, cm/s	L_v	Length of the vertically wall
υ	Kinemetic velocity, cm/s	ψ	Stream function
w	Wall		

A number of experimental and numerical studies have been conducted to investigate the flow flied of a lid-driven square cavity flow from more than one hundred years ago (Cheng and Hung, 2006; Chen 2009; Freitas *et* *al.* 1985; Parasad and Koseff, 1989). Numerous researchers have explored the different applications of large eddy simulation with finite element method. Popiolek *et al.* (2006) used the finite element analysis for the simulation of laminar and turbulent flows. Jiang and Kawahara (1993) developed a three-step finite element formulation for the solution of an unsteady incompressible viscous flow based on the Taylor-Galerkin scheme. They found that the basic two-dimensional flow is not always unique.

In this paper we consider the interaction of vortices in a closed cavity with square cross-section. The flow is driven by the parallel motion to two facing walls. The top and bottom horizontal walls of the cavity are moving on its own plane at a constant while the left and right vertical walls are at rest. The governing equations are solved using finite element approach and a fine uniform grid mesh of 100X100 using ANSYS 11.0 finite element software.

MATERIALS AND METHODS

The present work is a theoretical study where the finite element method has been applied using ANSYS element.

Governing equations

The vorticity-stream function-based governing equations for two-dimensional turbulent flow (Majda and Bertozzi, 2001; Peyret 2002; Chen and Krafzyk, 2009):

The vorticity transport equation

$$\frac{\partial\omega}{\partial t}+u\frac{\partial\omega}{\partial x}+v\frac{\partial\omega}{\partial y}=\frac{1}{Re}\left(\frac{\partial^2\omega}{\partial x^2}+\frac{\partial^2\omega}{\partial y^2}\right) \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(1)}$$

Stream function equation

$$\frac{\partial^2\psi}{\partial x^2}+\frac{\partial^2\psi}{\partial y^2}=-\omega \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(2)}$$

Where, *Re* is the Reynolds number, and x and y are the Cartesian coordinates. In addition, Reynolds number can be used for indicating the occurrence of turbulent flow.

The velocities u and v are obtained from:

$$u=\frac{\partial\psi}{\partial y} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(3)}$$

$$v=-\frac{\partial\psi}{\partial x} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(4)}$$

Finite Element Analysis

We consider tow-dimensional, two-sided lid-driven square cavity flow with uniform velocity of the horizontal parallel walls as shown in Fig. 2. Two moving walls are referred as the top and the bottom wall for convenience, while left and right wall are stationary. The velocity boundary conditions are employed on the two horizontal walls in same direction in the schematic diagram (Fig. 2).

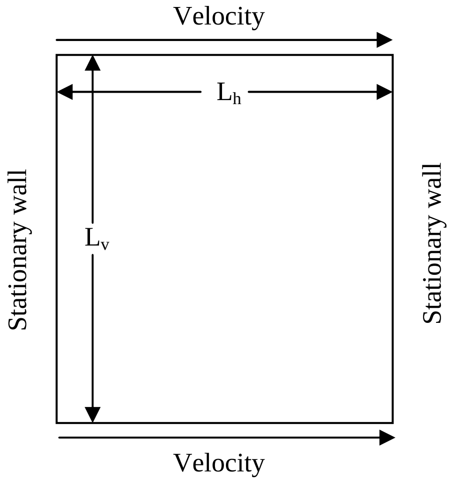

Fig. 2. Schematic diagram of the two-sided lid-driven square cavity for parallel motion

The numerical solution of the governing equations is achieved using finite element method. The technique is applied for the solution of the governing equations by using of square cavity domain, uniform grid mesh as well 2-D Fluid-Thermal (FLUID141) element (Shah *et al.* 2007). Uniform grid mesh is necessary in order to obtain a steady solution and also resolve the vortices appear at the corners of the cavity, as the Reynolds number increases (Erturk *et al.* 2005). Numerical experiments were performed in order to check the grid independence of the solutions. A uniform mesh of 100 by 100 cells has been used in the entire study. A turbulent flow analysis requires specifying density and viscosity. In this case, predefined density and viscosity of air are used.

Table 1. Air properties at 70^0C

Kinematics viscosity (in^2/s)	Reynolds number (*Rn*)	Velocity (in/s)
0.03086	3500	10.801
	5000	15.43
	6500	20.06

The turbulent flow in two-sided lid-driven square cavity is performed using ANSYS FLOTRAN environment. FLOTRAN environment can solve 2-D and 3-D flow, pressure, and temperature distributions in a single phase viscous fluid. 2-D Fluid-Thermal (FLUID141) element is used in this study.

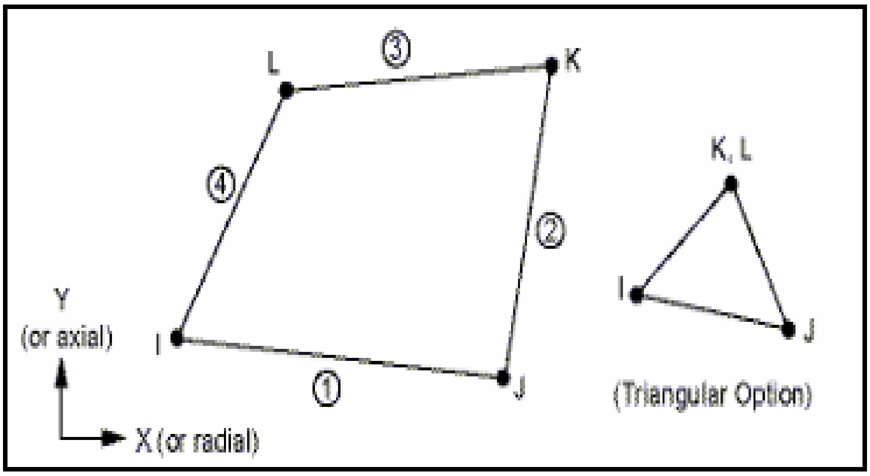

Fig. 3. FLUID141 geometry

FLUID141 is a four nodded quadrilateral or three nodded triangular element to model transient or steady state fluid/thermal systems that involve fluid and/or non-fluid regions. Each node of this element has seven degrees of freedom. The degrees of freedom are velocities and pressure in three directions and temperature. Element geometry is shown in Fig. 3.

RESULTS & DISCUSSION

An incompressible viscous flow in a two-sided square cavity, whose left and right walls are fixed and top and bottom walls are move in the same (parallel motion) direction with a uniform velocity. In the case of parallel wall motion, a free shear layer exists midway between the top and bottom walls apart from the wall bounded shear layers. Figs. 4-9 show the stream patterns for various Reynolds number for a range of 3500 to 6500 on a uniform grid 100×100 structure. The streamlines are found to be symmetrical with respect to the horizontal centerline (Figs 5, 7, 9). Figure 4 shows the streamline pattern for *Re*=3500. It shows that one primary and secondary almost same size vortex the right-hand top and the right-hand bottom corners respectively. Fig. 5 is the flow velocity vectors and clearly shows that one vortex bisect the right-hand wall along the horizontal wall. Figures 6 show the streamline patterns for *Re* = 5000.

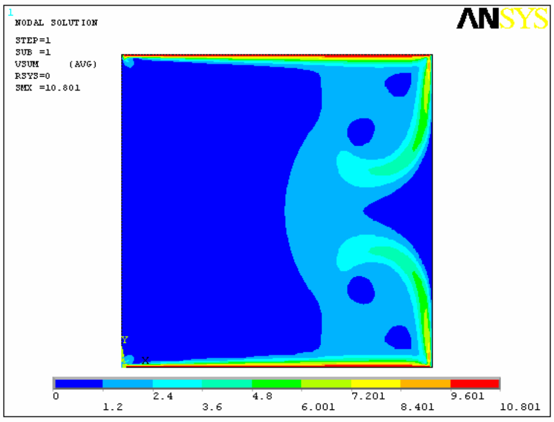

Fig. 4. Velocity distribution in two-sided lid-driven square cavity at *Rn* =3500

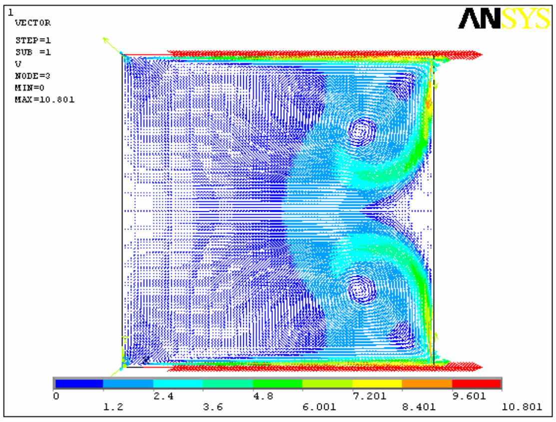

Fig. 5. Flow velocity vector in two-sided lid-driven square cavity at *Rn* = 3500

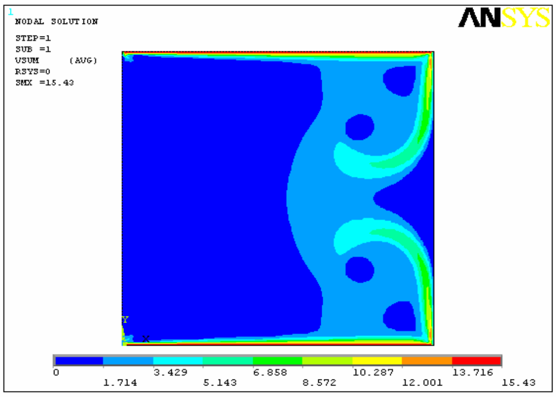

Fig. 6. Velocity distribution in two-sided lid-driven square cavity at *Rn* =5000

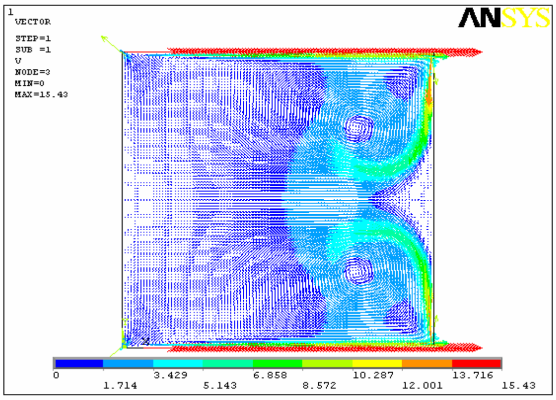

Fig. 7. Flow velocity vector in two-sided lid-driven square cavity at *Rn* = 5000

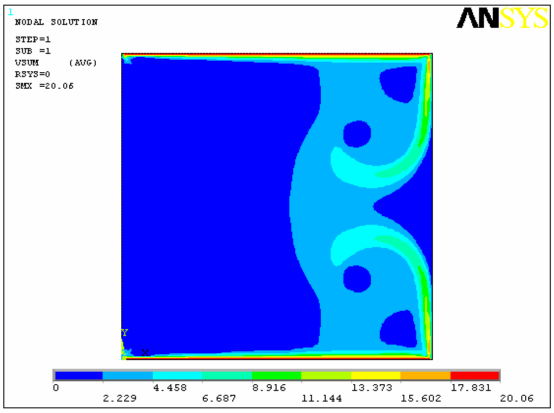

Fig. 8. Velocity distribution in two-sided lid-driven square cavity at *Rn* =6500

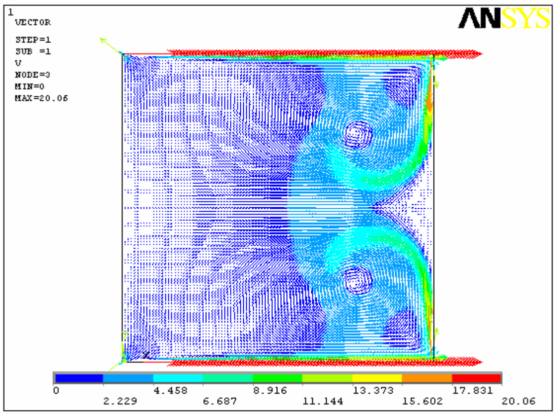

Fig. 9. Flow velocity vector in two-sided lid-driven square cavity at *Rn* = 6500

Fig. 6 show that two vortex also right-hand top and right-hand bottom of the corners and the primary vortex is relatively smaller than the secondary vortex. Fig. 8 show the streamline patterns for *Re* = 6500. With increasing in Remolds number the secondary vortex pair grows in size. At all the Reynolds numbers the counter-rotating pairs of primary and secondary vortices maintain their symmetry about the horizontal centerline. As the Reynolds number is increased the size of the secondary vortex increases at the right corner (at both top and bottom) of the right wall.

CONCLUSION

This paper focuses on simulation of turbulent flow in two-sided lid-driven square cavity is computed based on finite element method using ANSYS. The flow is investigated for parallel wall motion where primary and secondary vertices are well captured by these simulations. In the case of parallel wall motion, besides two primary vortices, there also appears a pair of counter-rotating secondary vortices symmetrically placed about the centerline parallel to the motion of the walls. As the Reynolds number is increases the locations of the primary vortex moves, while the smaller corner vortices are tend to grow. However, the main vortex adjacent to the moving wall grows in diameter as the Reynolds number is increased. Though the present model is designed for tow-dimensional flows, its extension to three-dimensional problems will be considered in future studies.

REFERENCES

ANSYS 11.0 (2007) ANSYS Inc., South Point, Canonsburg, USA.

Burggraf OR (1966) Analytical and numerical studies of the structure of steady separated flows. *J. Fluid Mech.* 24, 113-151.

Chai Z, Shi B, Zheng L (2006) Simulating high Reynolds number flow in two-dimensional lid-driven cavity by multi-relaxation-time lattice Boltzmann method, Chin. Phys. 15, 1855–1863.

Chen S (2009) A large-eddy-based lattice Boltzmann model for turbulent flow simulation. Applied Mathematics and Computation, 215, 591–595.

Chen S, Krafczyk M (2009) Entropy generation in turbulent natural convection due to internal heat generation, *Int. J. Thermal Sci.,* 48(10), 1978-1987.

Cheng M, Hung KC (2006) Vortex structure of steady flow in a rectangular cavity. Computers & Fluids, 35, 1046-1062.

Elkaim D, Reggio M, Camarero R (1992) Camarero, Simulating two-dimensional turbulent flow by using the k—e model and the vorticity-streamfunction formulation. *Int. J. Numer. Meth. Fluids*, 14, 961-980.

Erturk E, Corke TC, Gokcol C (2005) Numerical Solutions of 2-D Steady Incompressible Driven Cavity Flow at High Reynolds Numbers. *Intl J. Numer. Meth. Fluids*, 48, 747–774.

Freitas CJ, Street RL, Findikakis AN, Koseff RJ (1985) Numerical simulation of three-dimensional flow in a cavity. *Intl J. Numer. Meth. Fluids* **5**, 561.

Jiang CB, Kawahara M (1993) The analysis of unsteady incompressible flows by a three-step finite element method. *Intl J. Numer. Meth. Fluids*, 16(9), 793–811, 1993.

Kuhlmann HC, Wanschura M, Rath HJ (1997) Flow in two-sided lid-driven cavities: Non-uniqueness, instabilities, and cellular structures. *J. Fluid Mech.* 336, 267-299.

Majda AJ, Bertozzi A (2001) Vorticity and Incompressible Flow, Cambridge University Press, Cambridge.

Milane RE (2004) Large eddy simulation (2D) using diffusion-velocity method and vortex-in-cell, *Int. J. Numer. Meth. Fluids,* 44, 837–860.

Perumal DA, Dass AK (2010) Simulation of incompressible flows in two-sided lid-driven square cavities. Part1-FDM. Computational Fluid Dynamics, 2(1), 13-24.

Peyret R (2002) Spectral Methods for Incompressible Viscous Flow, Springer-Verlag, New York.

Popiolek TL, Awruch AM, Teixeira PRF (2006) Finite element analysis of laminar and turbulent flows using LES and subgrid-scale models Aplied Mathematical Modelling, 0(2), 177–199.

Prasad AK, Koseff JR (1989) Reynolds number and end-wall effects on a lid-driven cavity flow. Phys. Fluids A 1, 208-218.

Shah P, Rovagnati B, Mashayek F, Jacobs GB (2007) Subsonic compressible flow in two-sided lid-driven cavity. Part II: Unequal walls temperatures. *International Journal of Heat and Mass Transfer* 50, 4219–4228.