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):114-119(December 2011)

NUMERICAL SIMULATION OF INCOMPRESSIBLE FLOWS IN ONE-SIDED LID-DRIVEN SQUARE CAVITY BY FINITE ELEMENT METHOD

M. RASHID, A.B.M.A. KAISH, M.M. ISLAM AND M.T. 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):114-119(December 2011)

NUMERICAL SIMULATION OF INCOMPRESSIBLE FLOWS IN ONE-SIDED LID-DRIVEN SQUARE CAVITY BY FINITE ELEMENT METHOD

M. RASHID[1], A.B.M.A. KAISH[2], M.M. ISLAM[1] AND M.T. 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, Islam MT (2011) Numerical simulation of incompressible flows in one-sided lid-driven square cavity by finite element method. *J. Innov. Dev. Strategy.* 5(3), 114-119.

The paper computes the two-dimensional incompressible flow through one-sided lid driven square cavity based on finite element method (FEM). Unlike the existing finite element method for incompressible flow simulation, which was based on primitive-variables Navier–Stokes equations, the target macroscopic equations of the present model are vorticity-stream function equations. In this investigation, we have tried to present one-sided lid driven square cavity by finite element method with a clear and simple statement using ANSYS 11.0 finite element software. Numerical simulations of one-sided lid-driven square cavity are performed for a range of Reynolds number of 100 to 2000. The results obtained from ANSYS finite element simulation confirm that the present model is acceptable, efficient, stable and simple for two-dimensional incompressible flow. The ANSYS finite element program is useful in this study as well as varieties of enclosed fluid flows.

***Key words**: incompressible flow, one-sided lid-driven square cavity, finite element method*

INTRODUCTION

The flows observed in nature usually differ from the stream line flow or the laminar flow of a viscous fluid. Numerical simulation of fluid flow has been a major topic of research not only for scientist but also for engineers for the past few decades (Chen 2009; Shankar and Deshpande, 2000). 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. The Navier-Stokes equations do not posses analytical solutions, only has to resort to numerical methods (Perumal and Dass, 2010). 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 one 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).

It is increasingly recognized that the lid-driven square cavity flow problem is not technically important but also of great scientific interest, because it displays almost all fluid mechanical phenomena in the simplest geometrical settings (Bruneau and Sadd, 2006). Despite its simple geometry, the lid-driven flow retains a rich fluid flow physical manifested by multiple counter rotating re-circulating regions appear at the corners of the cavity depending on the Reynolds numbers (Erturk 2009; Chen 2009). Flow in a square one-sided lid-driven square cavity is an interesting research problem in the field of computational fluid dynamics because many important incompressible flow phenomena such as corner vortices, longitudinal vortices, Taylor–Görtler vortices, transition and turbulence all occur in the same closed geometry (Chen 2009). The simplicity of the geometry of the cavity flow makes the problem easy to code and apply boundary conditions. Furthermore, this type of flow is encountered in numerous practical applications such as photographic films, short dwell coaters and flexible coater used for production high-grade papers, and in liquid-film drying devices.

A number of experimental and numerical studies have been conducted to investigate the flow flied of a lid-driven cavity flow from more than one hundred years ago (Cheng and Hung, 2006; Ghia *et al.* 1982; Chen 2009). Numerous researchers have explored the different applications of large eddy simulation with finite element method. Popiolek *et al.* (2006) adopts the finite element analysis of laminar and turbulent flows using large eddy simulation the tow and three dimensional flows in a lid-driven cavity over a backward facing step. 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. A finite element method (FEM) is frequently used in CFD. FEM consist in essentially setting up a uniform grid in the problem domain, discretizing the governing equations with respect to the grid by replacing the derivatives with their finite element and solving the resulting algebraic equations numerically.

The study presents very fine grid steady solutions of the driven cavity flow at different Reynolds numbers.

MATERIALS AND METHODS

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

Governing equations

The governing equation for the incompressible Navier-Stokes equations for two dimensional applications in vorticity-stream function formulation is given by

The vorticity transport equation

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

Stream function equation

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

Where ψ and ω are the stream function and vorticity respectively. Components x and y are Cartesian coordinates and *Re* is the Reynolds number.

The velocities u and v are obtained from:

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

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

It is intended to obtain the steady state solution from the discretized equations in a time marching fashion. It is well known that in an explicit technique that uses a forward time stepping of first order accuracy and a central spatial discretization, the stability limit restricts the time-step to a very small value and the convergence to the steady state is painfully slow. This however, limits accuracy of the results and is no solution to the problem encountered. The governing equation given by (1) and (2) can be used for either steady or unsteady. Note that, time appears explicitly in the vorticity transport equation indicate as a parabolic equation. Thus any scheme for the solution of the parabolic equation can be utilized to solve equation (1). Even time does not appear in equation (2), it is still for unsteady flow. Therefore, for unsteady flow computation, the stream function is solved at each time step by any scheme previously introduce for the solution of elliptic equations.

Finite Element Analysis

Conventional numerical methods solve the Navier-Stokes equations to obtain the macroscopic variables of interest, such as velocity and pressure. An incompressible viscous flow in one-sided lid-driven square cavity whose top direction with a uniform velocity is the problem investigated in the present work. A schematic diagram of the enclosure with coordinate system and boundary conditions is shown in Fig. 1. Fine 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). A grid across flow domain needs to be defined (fig 1).

A finite element is used in which variation of variables within elements is approximated by a function and residual or error term is minimized. This technique is used for the solution of the governing equations. The cavity domain consists of a square. A uniform mesh of 100 by 100 cells has been used initially.

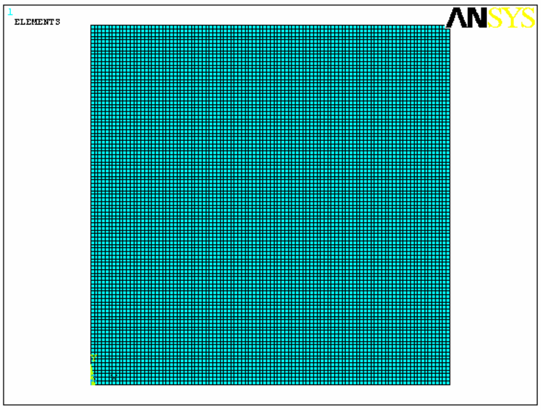

Fig. 1. Refined grid 100× 100

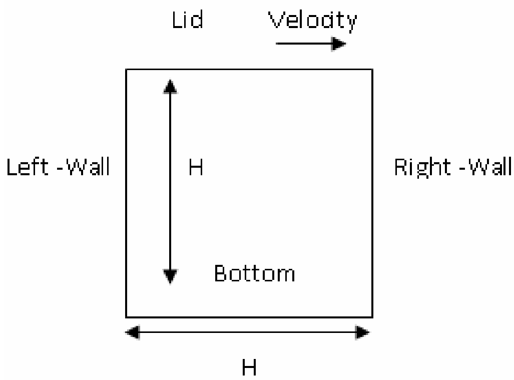

Fig. 2. Schematic diagram of the physical model

Finite element analysis of 2-dimensional incompressible laminar flow in one-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. 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 the following fig. 1.

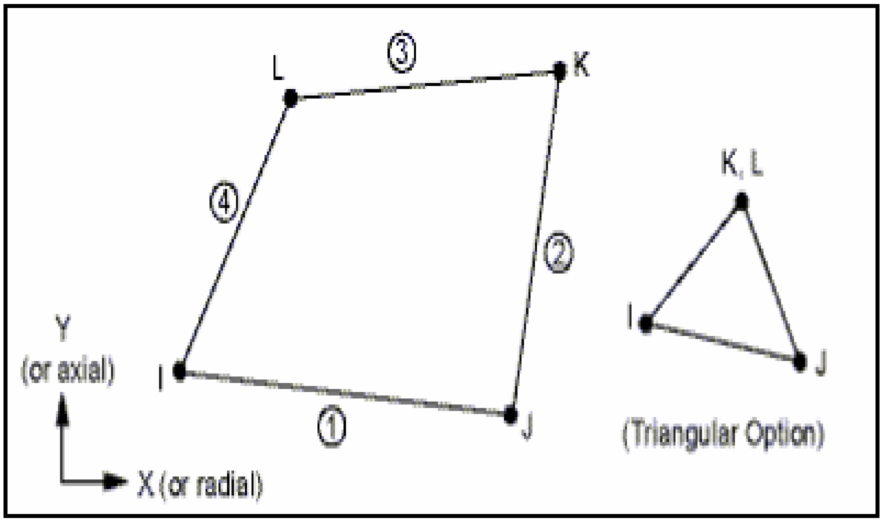

Fig. 3. FLUID141 geometry

Table 1. Air properties at 70^0C

Kinematics viscosity (in^2/s)	Reynolds number (*Rn*)	Velocity (in/s)
0.03086	100	0.3086
	400	1.2344
	1000	3.086
	2000	6.172

A laminar flow analysis requires specifying density and viscosity. In this case, predefined density and viscosity of air are used.

NUMERICAL RESULTS & DISCUSSION

The finite element method are applied to this one-sided lid-driven cavity flow, whose bottom, left and right walls are fixed and the top with a uniform velocity along X-axis is computed through finite element methods. Only uniform grid (100×100) was applied for the finite element computation. First, the case of a one-sided lid-driven flow in a square cavity with aspect ratio of 1.0 is considered. The developed finite difference code is used to compute the one-sided lid-driven square cavity flow for a range of Reynolds numbers of 100 to 2000 using 100×100 grid size is investigated here. As Figs. 4, 6, 8 and 10 illustrate, when Re$\leq$ 10000, the flow is laminar and steady. For low Re $\leq$ 1000, only three vortices appear in the cavity, a primary one near the center and a pair of secondary ones in the lower corners of the cavity. Fig. 4 shows the streamline patterns for *Re*=100 with top wall generate primary vortex moving from the left to right. Fig. 5 shows the velocity component along a vertical line passing through the center of the cavity at Reynolds number 100. At *Re* = 400, Figure 6 shows secondary vortices are symmetrically placed about the horizontal centerline near the centre of the right wall. Fig. 7 show the velocity component along a vertical line passing through the center of the cavity at Reynolds number 400.

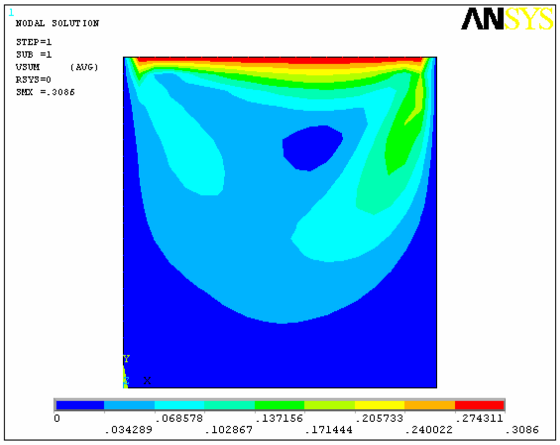

Fig. 4. Velocity distribution in one-sided lid-driven cavity at *Rn* =100

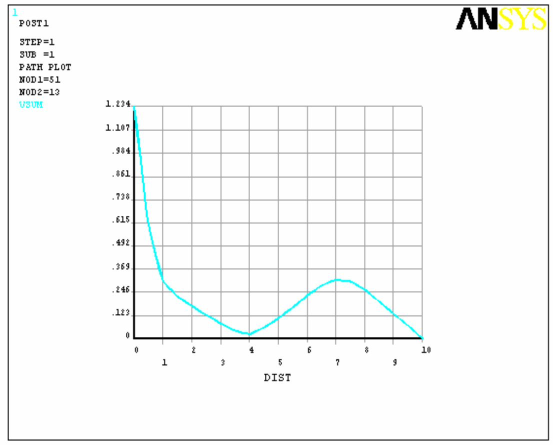

Fig. 7. Flow velocity along vertical mid-section (*Rn* =400)

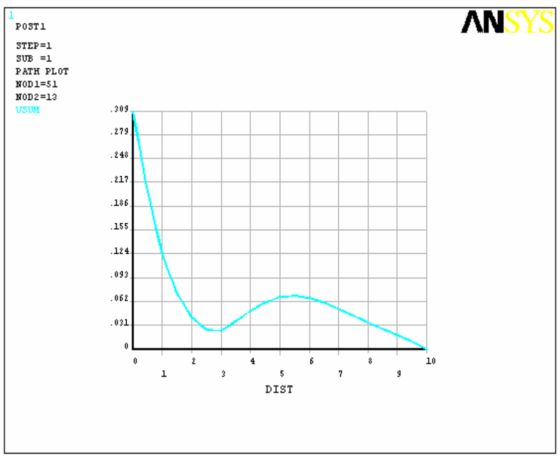

Fig. 5. Flow velocity along vertical mid-section (*Rn* =100)

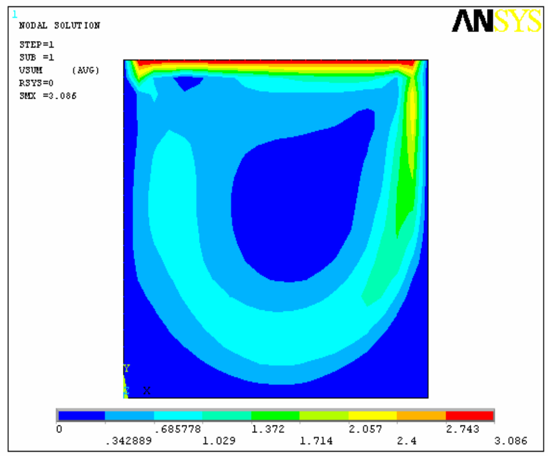

Fig. 8. Velocity distribution in one-sided lid-driven cavity at *Rn* =1000

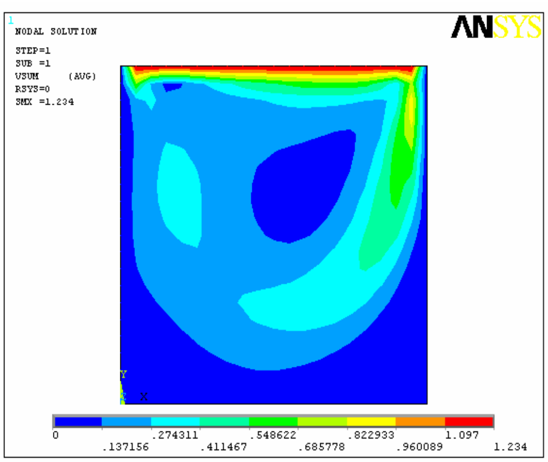

Fig. 6. Velocity distribution in one-sided lid-driven cavity at *Rn* = 400

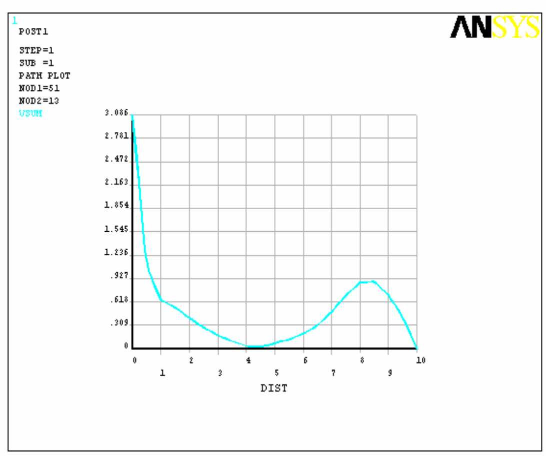

Fig. 9. Flow velocity along vertical mid-section (*Rn* =1000)

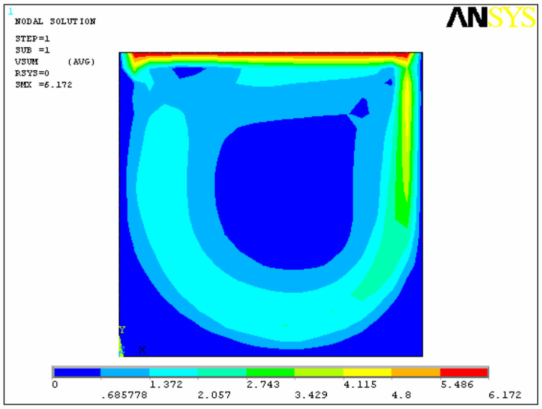

Fig. 10. Velocity distribution in one-sided lid-driven cavity at *Rn* =2000

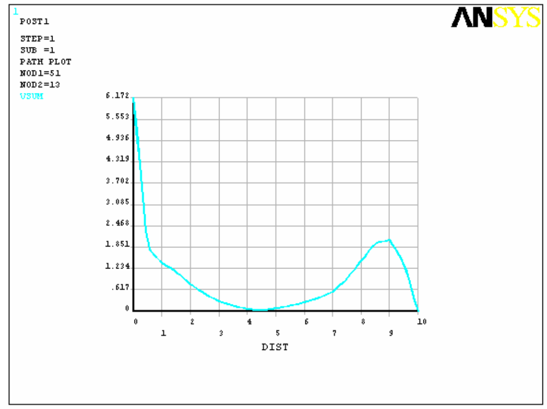

Fig. 11. Flow velocity along vertical mid-section (*Rn* =2000)

At *Re* =1000, Fig. 8 shows the streamline patterns with top wall generate secondary vortex. Fig. 9 shows the velocity component along a vertical line passing through the center of the cavity at Reynolds number 1000. As the Reynolds number increased to 2000, Fig. 10 shows the secondary vortices are seen to grow in size and the appearance of tow tertiary vortices. Fig. 11 shows the velocity component along a vertical line passing through the center of the cavity at Reynolds number 2000. As the Reynolds number is increased the location of the primary vortex moves, while the smaller corner vortices grow and undergo some interesting dynamics. Increasing the aspect ratio of the cavity has a similar effect; however, the bottom corner vortices will continue to grow until they form a single secondary vortex at the bottom of the cavity.

CONCLUSION

In this study we analyzed a simple one-sided lid-driven square cavity based on finite element methods using ANSYS. Unlike the existing primitive variables one-sided lid driven square cavity based on FEM, the target macroscopic equations of the present model are vorticity-stream function equations. Therefore, the present model can be employed straightforwardly for other types of incompressible fluid simulation. As the Reynolds number is increases the location of the primary vortex moves, while the smaller corner vortices grow and undergo some interesting fluid dynamic phenomena. The bottom corner vortices will continue to grow until they form a single secondary vortex at the bottom of the cavity with the increasing Reynolds numbers. 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.

Bruneau CH, Saad M (2006) The 2D lid-driven cavity problem revisited. Computers & Fluids, 35, 326-348.

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

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

Erturk E (2009) Discussions on driven cavity flow. *Intl J. Numer. Meth. Fluids*, 60, 275-294.

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.

Ghia U, Ghia KN, Shin CT (1982) High-Re solutions for incompressible flow using Navier- Stokes equations and a multigrid method. Journal of Computational Physics, 43, 387-411.

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.

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.

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.

Shankar PN, Deshpande MD (2000) Fluid mechanics in the driven cavity. Annual Review of Fluid Mechanics, 32, 93-136.