SIMULATION OF CENTER LOCATION OF FLUID FLOW IN SHEAR DRIVEN CAVITY USING LATTICE BOLTZMANN METHOD

WAN MOHD HANIF BIN OTHMAN

BACHELOR OF MECHANICAL ENGINEERING
UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS

JUDUL: SIMULATION OF CENTER LOCATION OF FLUID FLOW IN SHEAR DRIVEN CAVITY USING LATTICE BOLTZMANN METHOD

SESI PENGAJIAN: 2011/2012

Saya, **MOHD FAHMI BIN JASNI (890629-04-5351)**

mengaku membenarkan tesis (Sarjana Muda / ~~Sarjana / Doktor Falsafah~~)* ini disimpan di perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis ini adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi / badan di mana penyelidikan dijalankan)
√	**TIDAK TERHAD**	

Disahkan oleh:

___________________________ (TANDATANGAN PENULIS)

___________________________ (TANDATANGAN PENYELIA)

Alamat Tetap:

PT5798,
Tmn Desa Barakah,
17500, Tanah Merah,
Kelantan.

MUHAMMAD AMMAR BIN NIK MU'TASIM
(Nama Penyelia)

Tarikh: **15 JUNE 2012**

Tarikh: **15 JUNE 2012**

CATATAN: * Potong yang tidak berkenaan.

** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.

◆ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara Penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

SIMULATION OF CENTER LOCATION OF FLUID FLOW IN SHEAR DRIVEN CAVITY USING LATTICE BOLTZMANN METHOD

WAN MOHD HANIF BIN OTHMAN

Thesis submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled " *Simulation Of Center Location Of Fluid Flow In Shear Driven Cavity Using Lattice Boltzmann Method* " is written by Wan Mohd Hanif Bin Othman I have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. I herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering.

(MR. NASRUL AZUAN ALANG)

Examiner Signature

SUPERVISOR’S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering.

Signature
Name of Supervisor: MR. MUHAMMAD AMMAR BIN NIK MU'TASIM
Position:
Date:

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurently submitted for award of other degree.

Signature
Name: WAN MOHD HANIF BIN OTHMAN
ID Number: MA08141
Date: 12 JUNE 2012

Dedicated to my parents,

Othman bin Hj Wan Su and Ramizan binti Mat Isa

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor En Mr. Muhammad Ammar Nik Mu'tasim for his germinal ideas, invaluable guidance, continuous encouragement and constant support in making this research possible. He has always impressed me with his outstanding professional conduct, his strong conviction for science, I appreciate his consistent support from the first day I disscuss about my topic and to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career..

My sincere thanks go to all my labmates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. I acknowledge the sincerity of my parents-in-law, who consistently encouraged me to carry on my higher studies in Malaysia. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study.

ABSTRACT

The study of center location of lid driven cavity using lattice Boltzmann method is all about the of the fluid dynamic base on the simulation and prediction the flow. The study is base on steady flow and transient flow using Lattice Boltzmann method to understant the capability of Lattice Boltzmann. The simulation of Lattice Boltzmann is using FOTRAN software. The result is compare with Ghia et al to validate the stream function is in good arrangement and also can determine the capability of LBM. The result have been compare with simulation lid driven cavity using ANSYS(FLUENT) using navier stoke solver at the transient flow in Reynolds number 100 to 10000. From the simulation, the lattice Boltzmann method is capable at Reynolds number below than 7500.

ABSTRAK

Kajian simulasi tentang pusaran utama didalam rongga segiempat sama yang tudungnya digerakkan utk mengkaji pergerakan bendalir menggunakan kaedah kekisi Boltzmann. Dalam kajian bendalir secara simulasi, ramalan pergerakan berdasarkan kaedah berangka digunakan. Simulasi ini adalah untuk mengkaji kebolehupayaan kaedah kekisi Boltzmann berbanding kaedah lain. hasil dari simulasi ini telah dibandingkan menggunakan hasil simulasi Ghia et al untuk perbandingan struktur aliran fungsi berada dalam keadaan baik. Simulasi juga dijalankan menggunakan perisian ANSYS (FLUENT) iaitu berasaskan navier stoke dalam keadaan bergerak untuk mengkaji pergerakan pusat utama. Daripada simulasi ini, kekisi lattice Boltzmann berupaya menunjukkan hasil baik dari simulasi semasa bergerak, atau simulasi semasa akhir pergerakan pada nombor Reynolds sebelum 7500.

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURES

LIST OF SYMBOLS

u	Velocity vector
t	Time
$F(x,c,t)$	Density distribution function
f_i	Discretised Density Distribution Function
f_i^{eq}	Discretised Equilibrium Density Distribution Function
x	Space vector
C	Micro Velocity Vector
a	Acceleration
T	Temperature
k_b	Boltzmann Constant
g	Particle's relatice velocity
	Gravity Force
R	Gas Constant
D	Dimension
W_k	Weight coefficient
U	Top wall horizontal velocity
u	Horizontal velocity
v	Vertical velocity
n	Mole number
V	Volume
υ	Kinematic shear viscosity
ρ	Density
Ω	Collision operator
ψ	Stream function
τ	Average time between excessive collision
Re	Reynolds number
Nu	Nusselt number
Pr	Prandtl Number
Ra	Rayleigh number

LIST OF ABBREVIATIONS

BGK	Bhatnagar – Gross - Krook
CFD	Computional fluid dynamic
D2Q9	Two dimentions nine velocity
FD	Finite Difference
LB	Lattice Boltzmann
NSE	Navier stoke equation
FEM	Finite Element Method
LBM	Lattice Boltzmann method
LBE	Lattice Boltzmann Equation
LGA	Lattice gas Approsh

CHAPTER 1

INTRODUCTION

1.1 LID DRIVEN CAVITY FLOW (LDCF)

Understanding of fluid dynamic is very important in most branch of engineering especially in mechanical engineering. The fluid dynamic touches in many aspect of daily life such as air conditioning system to comfort people in the room and the simple thing such make a tea in a cup. To understand the fluid dynamic, we need to visualize the movement or flow of the fluid. The fluid mechanics need to visualize with time and space for more understanding and we can realize with the problem on it. The visual of fluid mechanic can be produce from the experiment that is high cost and the simulation by software which is almost accurate and low cost.

The shear driven cavity or also called lid-driven cavity flow is not only technically important to solve fluid flow problem but also the great scientific interest because it displays almost all fluid mechanical phenomena in the simplest of geometrical settings (Ghia et al 1982; D.A. Parumal, A.K Dass, 2009). Lid-driven cavity flow problem also has received considerable attention mainly because of its geometric simplicity, physical abundance, and its close relevance to some fundamental engineering (T. P. Chiang, W. H. Sheu 1997). The simplicity of the geometry of the cavity flow makes the problem easy to code and apply boundary conditions. Even though the problem looks simple in many ways, the flow in a cavity retains all the flow physics with counter rotating vortices appear at the corners of the cavity. (E. Erturk 2009).

The study about 2-D driven cavity flow problem is discussed in details in terms of physical and mathematical and also numerical aspects (E. Erturk 2009). The lid-driven-cavity problem is one of the most important benchmarks for numerical Navier–Stokes solvers. It can be subject grouped into three categories, in the first category of studies, steady solution of the driven cavity is sought. In these types of studies the numerical solution of steady incompressible Navier-Stokes equations are presented at various Reynolds numbers such as the results from Ghia et al 1982 and Erturk et al 2005 . In the second category of studies, the bifurcation which is the place where something divides into two branches, of the flow in a driven cavity from a steady regime to an unsteady regime is studied. In these studies a hydrodynamic stability analysis is done and the Reynolds numbers at which a Hopf bifurcation occurs in the flow are presented. The results from second category are from Fortin et al., Gervais et al., Sahin and Owens. In the third category of studies, the transition from steady to unsteady flow is studied through a Direct Numerical Simulation (DNS) and the transition Reynolds number is presented. The paper in the third category is such as Che Sidek, N.A and Nik Mu'tasim, M.A (2009). There are also the study in three dimensions (3D) of lid driven cavity that is from S. Albensoeder and H.C. Kuhlmann.

Ghia et al.1982 were among the first to publish benchmark data on the lid driven cavity flow. These classical papers are frequently referenced even today (S. Albensoeder, H.C. Kuhlmann). This paper was using Navier- stokes equation and a multigrid method in 2-D simulation. The Reynold number that been used for this paper were from 100 to10000 with meshes consisting of as many as 257 × 257 grid points. This paper was group into first category according to E. Erturk 2009.

A lid-driven cavity consists in a cavity bounded by solid walls. One of these walls is allowed to translate along itself, dragging the fluid which adheres to it (S. Nguyen et al 2006). When the lid was moving in u velocity in figure 1, it will cause on the flow of fluid. Basically, the governing equations for 2D deep cavity flow are developed from Navier-Stokes equation and continuity equation. (Mat Sahat, M.I , Che Sidek, N.A , 2010)

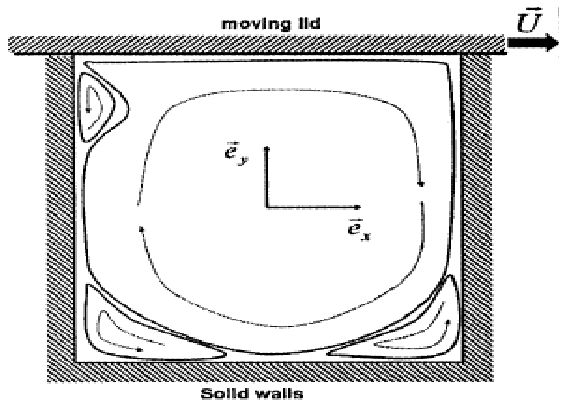


Figure1.0: Lid-driven cavity configuration

Source: S. Nguyen et al 2006

1.2 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics (CFD) that was developed over 40 years ago by engineers and mathematicians. The development of CFD because they want to solve heat and mass transfer problems in aeronautics, vehicle aerodynamics, chemical engineering, nuclear design and safety, ventilation and industrial design. The development of this technology in the 1950s and 1960s made such research possible, and CFD was one of the first areas to take advantage of the newly emergent field of scientific computing. In the process, it was soon realized that CFD could be an alternative to physical modeling in many areas of fluid dynamics, with its advantages of lower cost and greater flexibility.

Computational fluid dynamics is therefore an area of science made possible by and fundamentally linked to, computing. Its growth has paralleled that of computer power and availability, and as we move into an age of cheap, powerful desktop computing it is now possible, with a little knowledge, to run large and complex 3D simulations on an average personal computer.

However, most research advances in CFD continue to originate in the aeronautics and industrial design communities as a result of the significant investment levels available in these areas. In such cases it may be possible to characterize the complete set of process mechanisms that exist and also obtain good experimental data for model validation. Major research questions, therefore, concern improvements to the quality of the numerical solution, the scales of flow resolved by the model for fixed computational costs and the representation of sub-grid-scale processes such as turbulence.

The applications of CFD are used as tools for research, design, education, Automotive, Sports and many other fields. In this thesis, the focus is based on usefulness of CFD base on comparing Navier stoke Equation and Latice Boltzmann Method.

There are two type of CFD simulation which is the numerical and the other one is ready to use software. The software ready to use such as FLUENT© is very easy to use and infinite type of flow problem with many variables can be easily solved but there are disadvantages such as the user probably doesn’t know to the depth about the formulations that has been applied, the assumptions and a lot more. This software normally used for the practical application which the complicated geometry and conditions and used navier stoke as a solver. Despite that, this software is based on the numerical method but it is not being revealed. It purposes is solely to reduce the tough part and to make it user friendly (Mat Sahat, M.I 2010). The CFD simulation which is used numerical method software such as FORTRAN, C++, Matlab need the user create the codes and understand very well the formulation, the assumption, boundary conditions and others. This style of simulation usually applicable for knowledge sharing as many publications spawn everyday with new type of method for example the Lattice Boltzmann method that been used for this paper, Bifurcation method and more, claiming the method is among the best through various comparison and validation with the earlier or the classical method like Ghia et al. 1982. The simulation requires the creator to be well-verse in programming software that been used.

1.3 PROBLEM STATEMENT

The lid driven cavity problem is one of the most important benchmarks for numerical Navier–Stokes solvers. From this statement we can conclude that the result from the Navier-Strokes from lid driven cavity problem can be compare with other analysis base on lid driven cavity flow. This result will compare with lattice Boltzmann method. The critical for this analysis may be from the translation flow because in this regime the flow will change from laminar to turbulent. This study will provide information or subject to use of LBM and its extent toward solving fluid flow analytically.

1.4 OBJECTIVE

The objective for this analysis is to simulate the flow in the lid driven cavity by using lattice Boltzmann method. The analysis will use FORTRAN. The result from lattice Boltzmann method will be compare with Navier Stoke equation that is from Ghia et al 1982 and ANSYS (FLUENT©).

1.5 SCOPE

i. To simulate the flow in the lid driven cavity by using lattice Boltzmann method.
ii. The result from lattice Boltzmann method will be compare with Navier Stoke equation using ANSYS (FLUENT©)
iii. To understand the capability of LBM method in fluid flow based on qualitative study.

1.6 SCOPE OF ANALYSIS

i. In this analysis, the simulation will apply Lattice Boltzmann using FORTRAN language. The simulation is used to analyze the stream function with different Reynolds number that is from 100 to 10000.

ii. The boundary condition is no slip condition. The mashes size of cavity is 301×301.

iii. Validation is done using LBM, LDC in steady state condition.

iv. The result of this analysis will be compare with the result from experiment from Ghia et. al. (1982).

v. The result for LBM will compare to FLUENT software using Navier Stroke equation in transient flow condition in LDC.

CHAPTER 2

LITERATURE REVIEW

2.1 NAVIER STOKES EQUATION

The Navier–Stoke equation is well known equation for fluid dynamic describes the motion of fluid substances. These equations arise from applying Newton's second law to fluid motion, together with the assumption that the fluid stress is the sum of a diffusing viscous term (proportional to the gradient of velocity), plus a pressure term.

The mathematical relationship governing fluid flow is the continuity equation, from general differential equation from conservation of mass,

$$\frac{\delta\rho}{\delta t} + \vec{\nabla}.\left(\rho\vec{\mathrm{V}}\right) = 0 \tag{2.1}$$

The mathematical relationship governing fluid flow is the famous continuity equation

$$\nabla . u = 0 \tag{2.2}$$

And the Navier-strokes equation

$$\frac{\delta u}{\delta t} + u.\nabla u = -P + v\nabla^2 u \tag{2.3}$$

With velocity u, pressure P, kinematic shear viscosity v.

After the implementation of vorticity equation, two main equation which are literally derived from Navier-Stokes equation and continuity equation are as followed (Mat Sahat, M.I , Che Sidek, N.A , 2010)

$$\frac{\delta\Omega}{\delta T} + U\frac{\partial\Omega}{\delta X} + v\frac{\delta\Omega}{\delta \mathrm{Y}} = \frac{1}{Re}\left(\frac{\delta^2\Omega}{\delta X^2} + \frac{\delta^2\Omega}{\delta Y^2}\right) \tag{2.4}$$

$$\frac{\delta^2\Psi}{\delta X^2} + \frac{\delta^2\Psi}{\delta Y^2} = -\Omega \tag{2.5}$$

The Ω represent the vorticity and ψ is stream function. Re is the Reynolds numbers of the flow in the cavity and T is the time.

2.2 LATTICE BOLTZMANN

In the last one and a half decade or so Lattice Boltzmann Method (LBM) has emerged as a new and effective approach of computational fluid dynamics (CFD) and it has achieved considerable success in simulating fluid flows and heat transfer (D.A. Parumal, A.K Dass, 2009).

The lattice Boltzmann method allowed particles to move on a discrete lattice and local collisions conserved mass and momentum. Unlike than continuum field approach, kinetic theory assumes that a fluid is made of a huge number of molecular constituents, whose motion obeys Newtonian mechanics. Directly solving the system with a large number of degree of freedom, which is in the order of the Avogadro's number (10^{23}), is impossible. The movement of every individual molecular is not in the current purpose to be discussed, but one molecular is interested in the collective behavior of such system. The statistical description of the system will become predictable. The statistical approach provide a bridge between the macroscopic realm of hydrodynamics and the microscopic realm of atoms and molecular.

Because of the continuity and Navier Stokes equations are only continuous forms of the mass and momentum conservation statements and method that locally conserves mass and momentum will follow some kind of continuity and Navier Stokes equations and it was shown that the lattice gas methods could be used to simulate (rather noisy) hydrodynamics.

However, the lattice gas methods had several drawbacks consisting mainly of their noisy nature and the appearance of some additional terms in the Navier Stokes level equations that limited their success. It was then discovered that instead of discrete particles a density distribution could be adevecting which eliminated the noisiness of the method and allowed for a more general collision operator. This is the lattice Boltzmann method which has been extraordinarily successful for many applications including turbulence, multi-component and multi-phase flows as well as additional applications.

2.2.1 Classical Boltzmann Equation

A statically description of a system can be in term of the distribution function $f(x,c,t)$ where $f(x,c,t)$ is define such as $f(x,c,t)dxdc$ is the number of partical whose position and velocities are essentially dx and dc at time t. If there were no collision, then a short time Δt later each particle would move from x to $x + c\Delta t$ and each particle velocity would change from c to $c + a\Delta t$, where a is the acceleration due to external forces on a particle at x with a velocity c. The number of molecules $f(x,c,t)dxdc$ when there is no collision is equal to the number of molecules $f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc$, therefore

$$f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc - f(x,c,t)dxdc = 0 \tag{2.6}$$

However the collisions do occur between the molecules there will be a net difference between the molecules $f(x,c,t)dxdc$ and the number of molecules $f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc$. This can be expressed by

$$f(x + c\Delta t, c + a\Delta t, t + \Delta t)dxdc - f(x, c, t)dxdc = \Omega(f)dxdcdt \quad (2.7)$$

Which is $\Omega(f)dxdcdt$ is the collision operator. On dividing by $dxdcdt$, and letting dt tends to zero gives the Boltzmann equation for f

$$\frac{\delta f}{\delta t} + c_a \frac{\delta f}{\delta x_a} + a \frac{\delta f}{\delta c_a} = \Omega(f) \quad (2.8)$$

2.2.2 Boltzmann Collision Function

Any solution of the Boltzmann equation, $\frac{\delta f}{\delta t} + c_a \frac{\delta f}{\delta x_a} + a \frac{\delta f}{\delta c_a} = \Omega(f)$ required that an expression for the collision operator $\Omega(f)$. If the collision is to conserve mass, momentum and energy it is required that

$$\int \begin{bmatrix} 1 \\ c \\ c^2 \end{bmatrix} \Omega(f)dc = 0 \quad (2.9)$$

Collision can change the distribution function $f(x, c, t)$ in two ways;

i. Some particles originally having velocities c will have some different velocity after collision. This causes in $f(x, c, t)$.
ii. Some particles have other velocities may have the velocity c after a collision, increasing $f(x, c, t)$.

The form of the collision function can be found by assuming that

i. Only binary collisions need to be considered (dilute gas)
ii. The influence of container walls may be neglected
iii. The influence of the external force (if any) on the rate of collision is negligible
iv. Velocities and position of a molecule are uncorrelated (assumption of molecular chaos)

Suppose two particle with initial velocities c and c_1 have velocities $c^{'}$ and $c_1^{'}$ after a collision. Since all particles have same mass, conservation of momentum and energy required that

$$c + c_1 = c^{'} + c_1^{'} \tag{2.9}$$

$$\frac{1}{2}|c|^2 + \frac{1}{2}|c_1|^2 = \frac{1}{2}\left|c^{'}\right|^2 \frac{1}{2}\left|c_1^{'}\right|^2 \tag{2.10}$$

For an elastic collision, the magnitude of the relative velocity is a coalitional invariant

$$|c - c_1| = \left|c^{'} - c_1^{'}\right| \tag{2.11}$$

Under all these assumptions, the Boltzmann equation takes on following form:

$$\Omega(f) = \iint (f f_1 - f^{'} f_1^{'}) g\sigma d\Omega dc^{'} \tag{2.12}$$

Where$f = f(x, c, t)$, $f_1 = f(x, c_1, t)$, $f^{'} = f(x, c^{'}, t)$, $f_1^{'} f\left(x, c_1^{'}, t\right)$, g is the particles relative velocities before the Collision and σ is the scattering cross section.

2.2.3 Bhatnagar-Gross-Krook(BGK) Collision Model

The Boltzmann equation without the external force where

$$\frac{\delta f}{\delta t} + c_a \frac{\delta f}{\delta x_a} + a \frac{\delta f}{\delta c_a}\bigg| \text{ (Collision)} \tag{2.13}$$

which is,

$$\frac{\delta f}{\delta t} = \Omega(f) \tag{2.14}$$

represent the change in distribution function per unit time due to collision. The particular interest is in the change in distributing function f in time of order τ_f, the average time between excessive collision. Assuming that at near equilibrium, the system is closed to local Maxwell-Boltzmann state. Moreover, the post-collision distribution function, $f^{''}s$, should be closer to equilibrium than the pre-collision $f^{'}s$, because of H- theorem. The distribution function f can be related to the equilibrium distribution function f^{eq} via Taylor's series expansion

$$f^{eq}(x,c,t)f \approx f(x,c,t) + \frac{\delta f}{\delta t}\bigg|(collision)\ (\delta t) + o(\delta t)^2 \tag{2.15}$$

$$\frac{\delta f}{\delta t}\bigg|(collision) = \frac{f^{eq}(x,c,t)f - f(x,c,t)}{\delta t} = \frac{f^{eq}(x,c,t)f - f(x,c,t)}{\tau_f} \tag{2.16}$$

Where the small time interval δt have ben replaced by the characteristic time between collisions τ_f. This model is frequently called collision model after Bhatnagar, Gross and Krook who first introduced.

2.2.4 The Lattice Boltzmann Equation

The Boltzmann equation with BGK collision model can be expressed as

$$\frac{\delta f}{\delta t} + c_a \frac{\delta f}{\delta x_a} = \frac{f - f^{eq}}{\tau_f} \tag{2.17}$$

That is well known as the BGK Boltzmann equation. The Maxwell-Boltzmann equilibrium distribution function is define as

$$f^{eq} = \rho \left(\frac{1}{2\pi RT}\right)^{d/2} exp\left\{-\frac{(c-u)^2}{2RT}\right\} \tag{2.18}$$

The BGK lattice Boltzmann equation can be derived by further discretise using an Euler time step in time step in conjunction with an upwind spatial discretization and then setting the grid spacing divided by the time step equal to the velocity

$$\frac{f(x,t+\Delta t)-f(x,t)}{\Delta t}+c\,\frac{f(x+\Delta x,t+\Delta t)-f(x,t+\Delta t)}{\Delta x}=\frac{f-f^{eq}}{\tau_f} \quad (2.19)$$

$$\frac{f(x,t+\Delta t)-f(x,t)}{\Delta t}+c\,\frac{f(x+\Delta x,t+\Delta t)-f(x,t+\Delta t)}{\mathrm{c}\Delta x}=\frac{f-f^{eq}}{\tau_f} \quad (2.20)$$

As a result

$$f(x+c\Delta t,t+\Delta t)-f(x,t)=-\Delta \mathrm{t}(\frac{f-f^{eq}}{\tau_f}) \quad (2.21)$$

The LBGK model with single relaxation time, which is a commonly used lattice Boltzmann method, is given by (D.A. Parumal, A.K Dass, 2009)

$$f_i(x+c_i\Delta t,t+\Delta t)-f_i(x,t)=-\frac{1}{\tau}[f_i(x,t)-f_1^0(x,t)] \quad (2.22)$$

Where f_i is the particle distribution function, $f_1^0(x,t)$ is the equilibrium distribution function at x, t, c_i is the particle velocity along the ith direction and τ is the time relaxation parameter. The D2Q9 square lattice used here has nine discrete velocities. A square lattice is used, each node of which has eight neighbors connected by eight links as shown in Fig. Particles residing on a node move to their nearest neighbors along these links in unit time step. The particle velocities are defined as

$$C_i=o \qquad\qquad i=0 \quad (2.23)$$

$$C_i=\left(\cos\left(\frac{\pi}{4(i-1)},\sin\left(\frac{\pi}{4(i-1)}\right)\right)\right),i=1,2,3,4 \quad (2.24)$$

$$C_i=(\cos\left(\frac{\pi}{4(i-1)}\right)\sin(\frac{\pi}{4(i-1)}))\,,i=5,6,7,8 \quad (2.25)$$

The macroscopic quantities such as density ρ and momentum density ρu are obtained as velocity moments of the distribution function f_i as follows:

$$\rho = \sum_{i=0}^{N} f_i \tag{2.26}$$

$$\rho u = \sum_{i=0}^{N} f_i C_i \tag{2.27}$$

Where N = 8. In the D2Q9 square lattice, a suitable equilibrium distribution function that has been proposed is

$$f_i^{(0)} = w_i \rho \left[1 - \frac{3}{2}u^2\right], \text{ i=0} \tag{2.28}$$

$$f_i^{(0)} = w_i \rho [1 + 3(c_i u) + 4.5(c_i u)^2 - 1.5u^2], \text{ i=1, 2, 3, 4} \tag{2.29}$$

$$f_i^{(0)} = w_i \rho [1 + 3(c_i u) + 4.5(c_i u)^2 - 1.5u^2], \text{ i=5, 6, 7, 8} \tag{2.30}$$

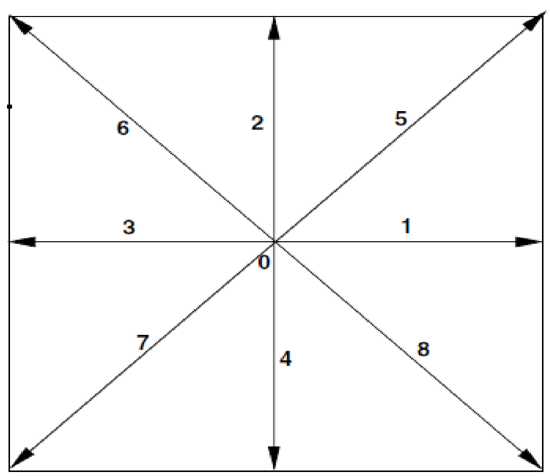


Figure 2: D2Q9 lattice and velocities

Source: D.A. Parumal, A.K Dass, 2009

Where the lattice weights are given by w_0 = 4/9, w_{1-4} = 1/9 and w_{5-8}= 1/36. The relaxation time which fixes the rate of approach to equilibrium is related to the viscosity by (S.Hou et al, 1995)

$$\tau = \frac{6v+1}{2} \tag{2.31}$$

The units for v are lu^2ts^{-1}. Note that τ >1/2 for positive (psychical) velocity. Numerical difficulties can arise as τ approaches ½. A value of τ = 1 is safest and lead to $v = \frac{1}{6}lu^2ts^{-1}$.(M.C. Sukop et al, 2005)

2.3 DISCRETISATION OF LBM

Discretisation is the process of dividing into a finite number of elements a continuum object. The lattice Boltzmann method start from the following Boltzmann equation for discrete velocity distribution in two and three dimensions

$$\frac{\delta f_i}{\delta t} + c_i.\nabla f_i = \Omega(f_i) \tag{2.32}$$

A commonly used LBM is the so-called lattice BGK model where the collision $\Omega(f_i)$ is replaced by the BGK collision model

$$\frac{\partial f_i}{\partial t} + c_i.\nabla f_i = -\frac{1}{\tau_f}(f_i - f_i^{eq}) \tag{2.33}$$

Where the collision is assumed to lead the particle distribution function relaxes to its equilibrium state at a constant rate. If the time derivative is replace by a first order time difference, first order upwind space discretization is used for the convective term $c_i.\nabla f_i$ and $\Delta x = \Delta t = 1$ is set, the discretised version of lattice Boltzmann equation is obtained

$$f_i(x + c_i, t_1) - f_i(x, t) = -\frac{1}{\tau_f}(f_i - f_i^{eq}) \tag{2.34}$$

Although the lattice Boltzmann equation (LBE) has demonstrated to be an effective computational tool for a broad variety of complex physical systems, however, the LBM suffer several limitations. One of these is that the LBM is constructed on a special class of uniform and regular lattices. The limitation of using uniform lattices is particularly severe in many practice application where the complex geometry of boundaries cannot be well fitted by regular lattices.

It should be pointed out, however, that this particular descretization and the condition $\Delta t = \Delta x$ are not necessary. The used of Eq. (2.34) reflects the historical fact that LBM came about as refinement or a by-product of the lattice bolgas automaton method (LGA). Boolean particles reside in a discrete lattice, subject to the automaton dynamic of streaming and collision. One of the main ideas driving the initial LGA efforts was to produce the simplest microdynamics that would yield hydrodynamic behaviors. Recovering rotational invariant macroscopic equation from a discrete finite velocity microscopic dynamic impose constraint on the symmetry of the lattice used, unlike the continuum Boltzmann equation with infinite velocities, for which rotational invariance is automatically recovered. For LBM this is obtained from physical symmetry. By the physical symmetry we mean the symmetry attached to the velocity space in equilibrium distribution function and a sufficient number of moving velocity direction N. the lattice symmetry requirements are those numbers of lattice direction (in x space) and the number of lattice links are the same as those for particle distribution function.

2.4 FINITE DIFFERENCE LATTICE BOLTZMANN METHOD

The chronological discretized is obtained by using second order Rungge-Kutta (modified) Euler method. The time evolution of particle distribution is then derived by

$$f_i^{n+\frac{1}{2}} = f_i^n + \frac{\Delta t}{2}\left(-c_i.\nabla f_i^n - \frac{f_i^n - f_i^{eq,n}}{\tau_f}\right) \tag{2.35}$$

$$f_i^{n+1} = f_i^n + \Delta t\left(-c_i.\nabla f_i^{n+\frac{1}{2}} - \frac{f_i^{n+\frac{1}{2}} - f_i^{eq,n+\frac{1}{2}}}{\tau_f}\right) \tag{2.36}$$

The second order Eqs. (3.6~3.9) or third order Eqs. (3.10~3.13) upwind scheme can be applied to calculate the spatial gradient in Eq. (3.2)

$$c_{ix}\delta_x f_i = c_{ix}\frac{3f_i(x,y)-4f_i(x-\Delta x,y)+f_i(x-2\Delta x,y)}{2\Delta x}, c_{ix} > 0 \tag{2.37}$$

$$c_{ix}\delta_x f_i = -c_{ix}\frac{3f_i(x,y)-4f_i(x-\Delta x,y)+f_i(x-2\Delta x,y)}{2\Delta x}, c_{ix} < 0 \tag{2.38}$$

$$c_{iy}\delta_y f_i = c_{iy}\frac{3f_i(x,y)-4f_i(x,y-\Delta y)+f_i(x,y-2\Delta x,y)}{2\Delta y}, c_{iy} > 0 \tag{2.39}$$

$$c_{ix}\delta_y f_i = -c_{iy}\frac{3f_i(x,y)-4f_i(x,y-\Delta y)+f_i(x,y-2\Delta y)}{2\Delta y}, c_{iy} < 0 \tag{2.40}$$

$$c_{ix}\delta_x f_i = c_{ix}\frac{f_i(x+2\Delta x,y)-2f_i(x-\Delta x,y)+9f_i(x,y)}{6\Delta x} + c_{ix}\frac{-10f_i(x+\Delta x,y)-2f_i(x-2\Delta x,y)}{6\Delta x}, c_{ix} > 0 \tag{2.41}$$

$$c_{ix}\delta_x f_i = c_{ix}\frac{-f_i(x-2\Delta x,y)+2f_i(x-\Delta x,y)-9f_i(x,y)}{6\Delta x} + c_{ix}\frac{10f_i(x+\Delta x,y)-2f_i(x-2\Delta x,y)}{6\Delta x}, c_{ix} < 0 \tag{2.42}$$

$$c_{iy}\delta_y f_i = c_{iy}\frac{f_i(x,y+2\Delta y)-2f_i(x,y-\Delta y)+9f_i(x,y)}{6\Delta y} + c_{iy}\frac{-10f_i(x,y+\Delta y)-2f_i(x,y-2\Delta y)}{6\Delta y}, c_{iy} > 0 \tag{2.43}$$

$$c_{iy}\delta_y f_i = c_{iy}\frac{-f_i(x,y-2\Delta y)+2f_i(x,y-\Delta y)-9f_i(x,y)}{6\Delta y} + c_{iy}\frac{10f_i(x,y+\Delta y)-2f_i(x,y+2\Delta y)}{6\Delta y}, c_{iy} \tag{2.44}$$

The combinations of this specifics space and time discretization result in second or third order space and second order in time.

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

The aim for this analysis is to simulate the flow in the lid driven cavity by using lattice Boltzmann method. The analysis will use language program FORTRAN. In the direction of to conduct numerical solution using LBM, discretization in velocity and momentum in space is needed.

For simulation that uses LBM for simple geometry, flow for lid driven square cavity was chosen. The lid driven cavity is a good and simple first test case since it is easy to obtain a converge solution. Furthermore, there are abundant literature reviews available for comparison. The result from lattice Boltzmann method will be compare with Navier Stroke equation using ANSYS.

3.2 FLOW CHART

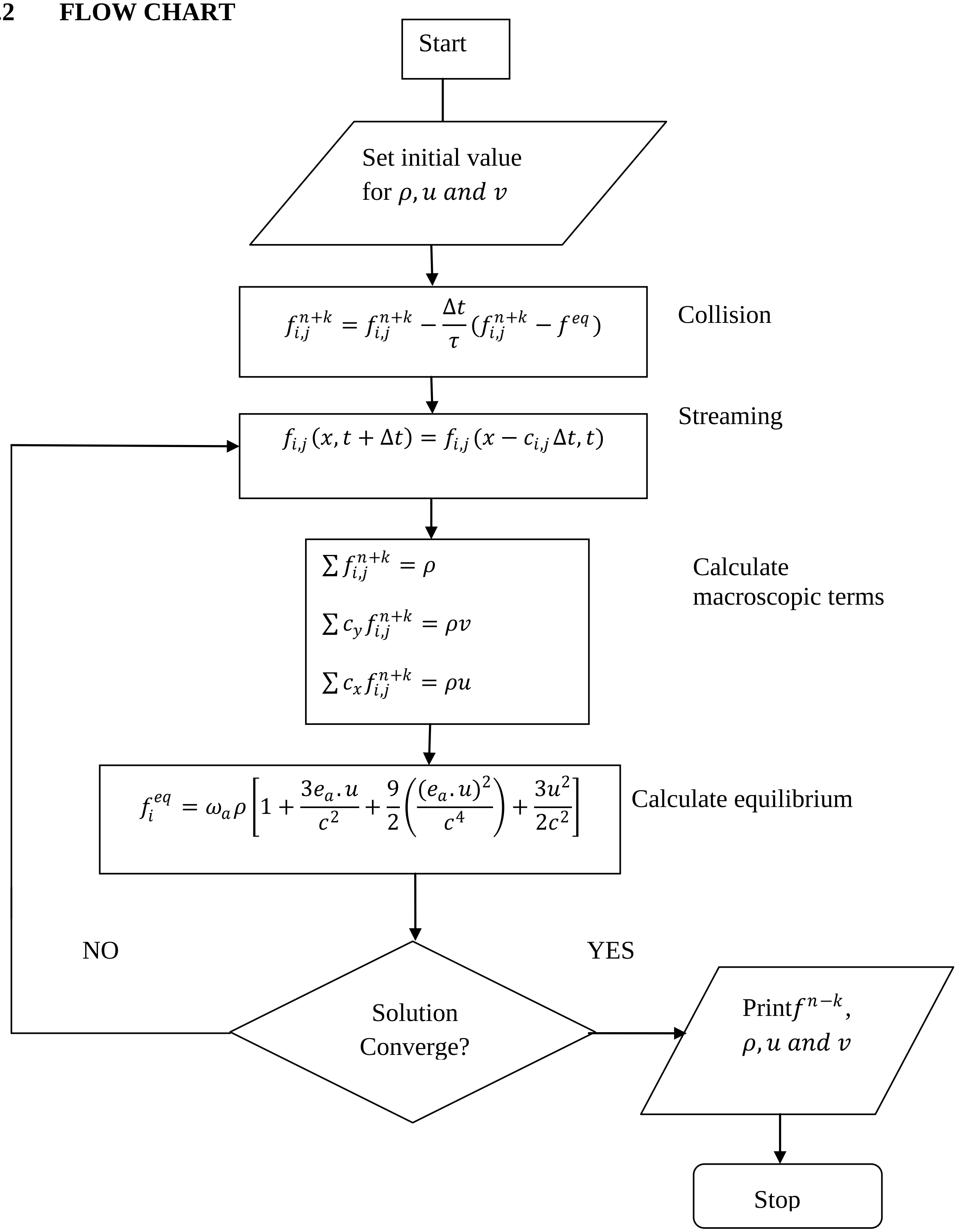


Figure 3.1: Flow chart for Lattice Boltzmann Method (LBM)

3.3 FLOW CHART PROJECT

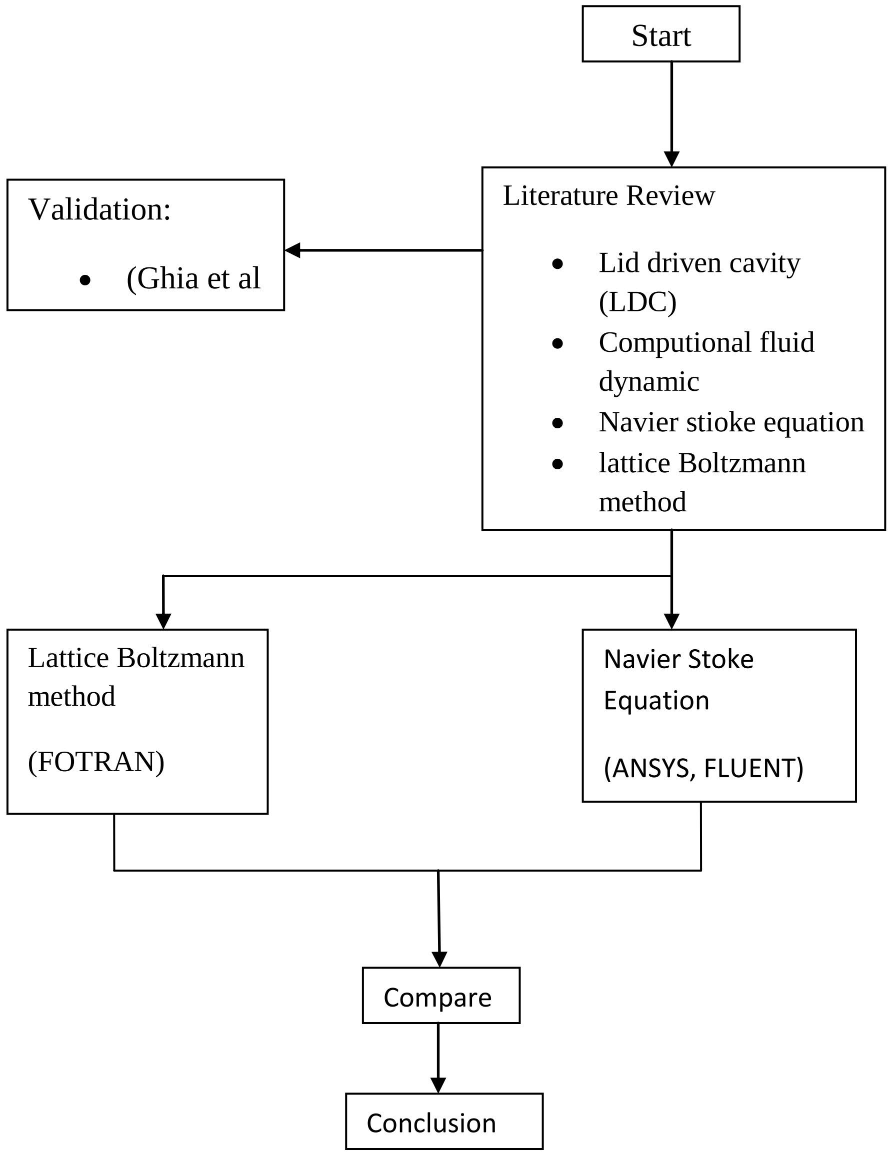


Figure 3.2: Flow chart project

3.4 BENCHMARK OF LID DRIVEN CAVITY

Lid driven cavity has been used as a benchmark problem for many numerical methods due to its simple geometry and complicated flow behavior. The lid-driven-cavity problem also is one of the most important benchmarks for numerical Navier–Stokes solvers. It is usually very difficult to capture the flow phenomena near the singular points at the corner of the cavity. Consequently it is desirable to refine the mesh near this corner. The benchmark of lid driven cavity will divide into two method that is lattice Boltzmann method and Navier Stoke equation. But all method will compare with Ghia et al as mention at chapter 1.

The lid driven cavity flow is a flow inside a cavity where the top wall slides to the right at a constant speed of U while the other three walls are made stationary. This type of flow has been used as a benchmark problem for many numerical methods due to its simple geometry but complicated flow behaviors.

LBM is applied to this cavity flow of height L. the Reynolds number (Re) was varied from 100 to 10000. The table shows the grid size use for corresponding Reynolds number.

Table 3.1: The grid size for LBM

Reynolds number	Grid size
100	301×301
400	301×301
1000	301×301
3200	301×301
5000	301×301
7500	301×301

Using this grid size enable us to obtain steady solution for these various Reynolds numbers (we need to keep a sufficiently large τ within the stability range)

In the simulation, the Reynolds number $Re = LU/v$ the relation between the time relaxation and dynamic viscosity is given by

$$\tau_{f=3v} \tag{3.1}$$

For Navier Stoke equation, the uniform and meshes will used for both Reynolds number simulation. The size of the mesh is 301×301 and $\Delta t = 0.1$ is set for all cases of Reynolds number simulations. The size of the cavity will change that is 1m×1m for all cases. All of this parameter will analyze using FLUENT software for the transient flow.

The result from both solvers will compare with classical paper Ghia et al(1982). The streamline pattern will be plotted and comparison will be made against results found in literature.

3.5 VALIDATION WITH GHIA ET AL

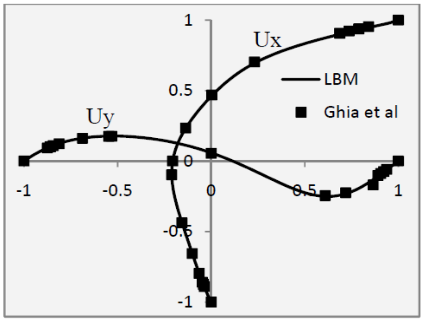
The validation with Ghia et al is to make sure the structure of the stream function in the good condition. The validation with this classical paper is done at the steady flow.

(a) (b)

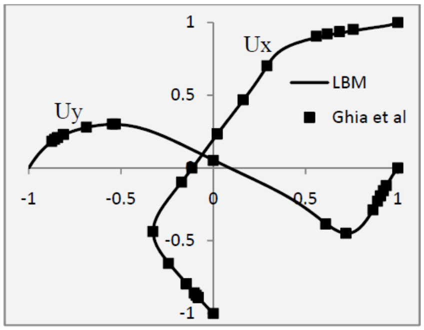
Figure 3.3: The stream function of LBM (a) and Ghia et al at (b) at Re = 400.

The stream function of this figure shows that at Re=400, the stream function of LBM is in the good condition compare with Ghia et al. Besides the stream function figure comparison, the graph of velocity profile at mid also been used to shows the comparison in details. At this comparison, we use the velocity at X axis and Y axis to compare the result.

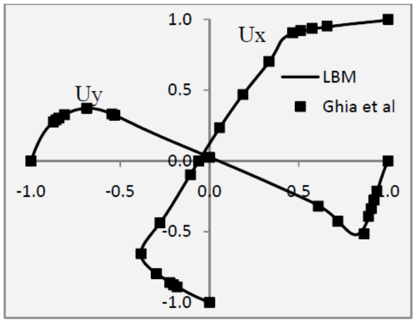
Re = 100



Re = 400



Re = 1000



Re = 3200

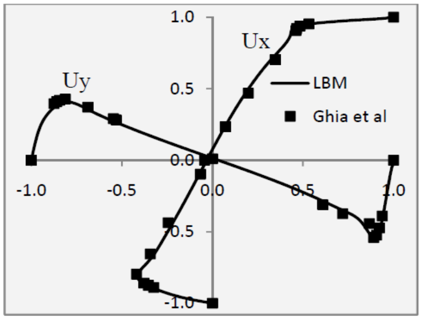
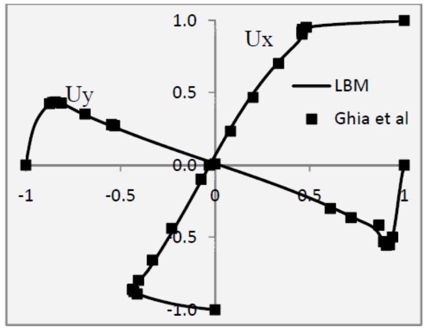
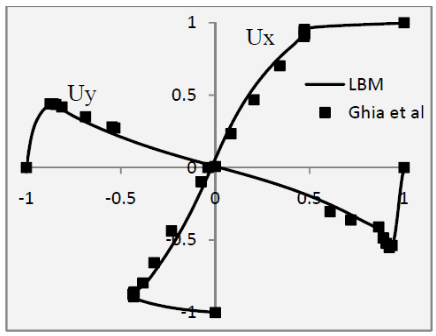


Figure 3.4 : continued

Re = 5000



Re = 7500



Re = 10000

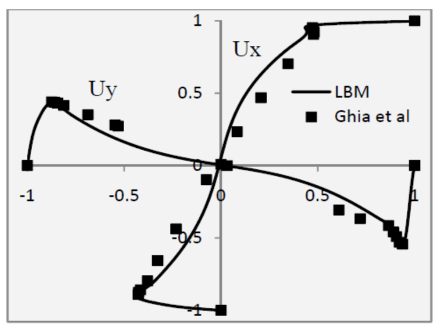


Figure 3.4: Velocity profile at mid-height (x-velocity & y-velocity) of lid driven square cavity using conventional LBM

CHAPTER 4

RESULTS AND DISCUSSTION

4.1 INTRODUCTION

The result from lattice Boltzmann is written using Fotran language. In this chapter, the result from lattice Boltzmann was validate using result from Ghia et al. the result using lattice Boltzmann method is from Reynolds number 100, 400 and 1000.

The result from lattice Boltzmann had been compare with result from latest CFD solver, Ansys Fluent. The ANSYS (FLUENT) is base on Navier Stoke equation. The result using Navier Stoke is in the transient flow, pressure base and in two dimensional (2D).

4.2 VALIDATIONS LATTICE BOLTZMANN WITH GHIA ET AL. 1982

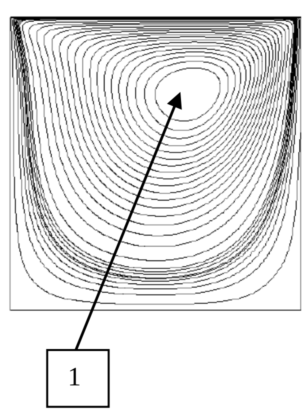
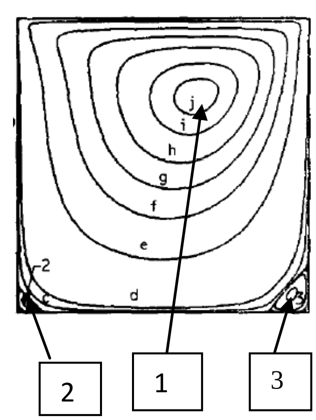
In order to study the center location of lid driven cavity in stream function, the result of LBM was been compare with Ghia et al is in a steady flow condition. This comparison is important because it shows the flow structure is in good agreement with the previous work of Ghia et al 1982. Base on table 4.2 below, LBM have excellent result at Reynolds number 100 to 5000 base on the primary vortex for every Reynolds number that were calculated. We also can see that LBM can produce an excellent agreement with the result predicted by conventional numerical methods. The apparent of LBM flow structure are good agreement compare with the results published in the literature by previous researcher.

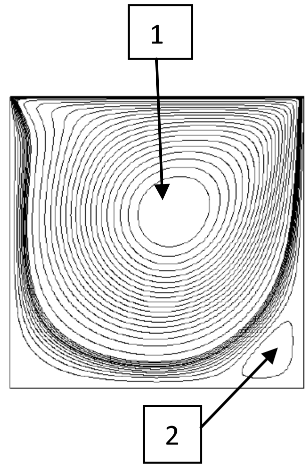
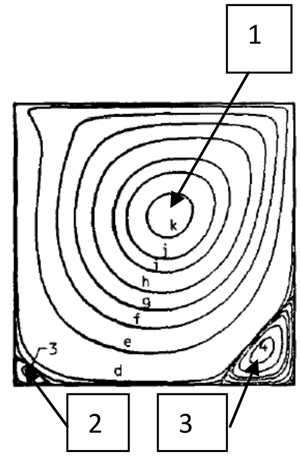
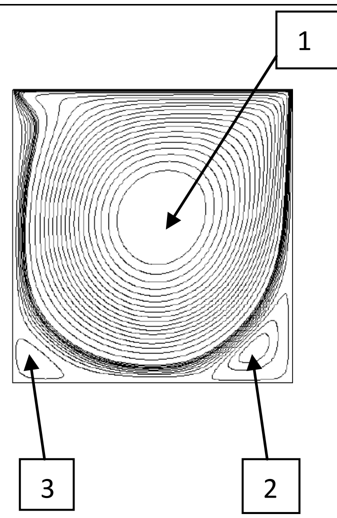
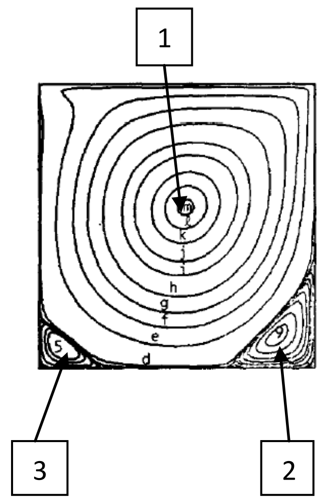
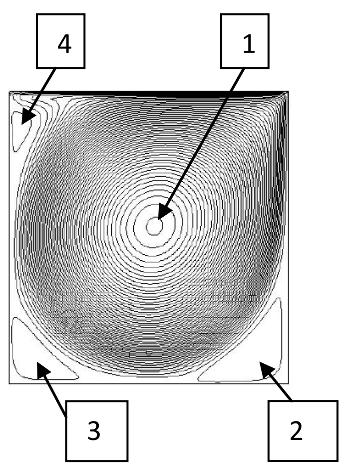
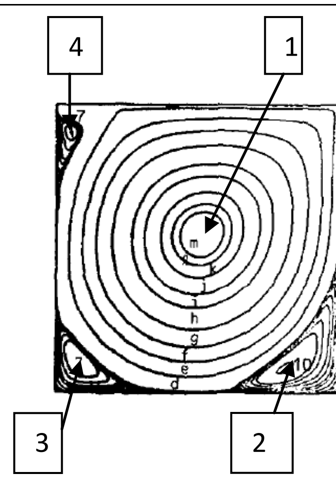
At Re = 100, the result from lattice Boltzmann shows there are only primary eddies compare to result from Ghia et al, there are also have secondary eddies at the bottom left and right. At Re = 400, results from LBM shows the size of primary eddies became bigger until Re = 1000, and the secondary eddies appear at bottom right side at Re = 400, and appear at bottom side left and right at Re = 1000. When the Reynolds number increase to 3200, the size of primary eddies became smaller but the secondary eddies appear at right and left bottom of cavity, and also at the left at the lid. The size of secondary eddies became smaller at Re = 5000 then Re = 3200 but the location of secondary eddies are same.

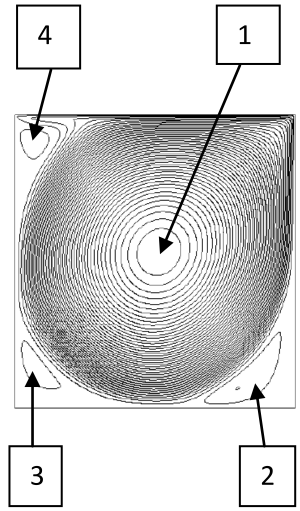
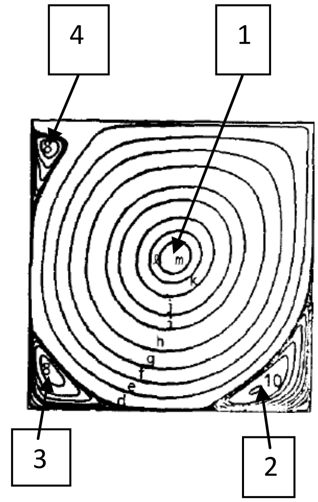
Table 4.1: Location of eddies

Location	Eddies
1	Primary eddies
2	Secondary eddies bottom left
3	Secondary eddies bottom right
4	Secondary eddies lid left

Table 4.2: Comparison between streamlines Lattice Boltzmann Method and Ghia et al from Reynolds number 100 to 5000

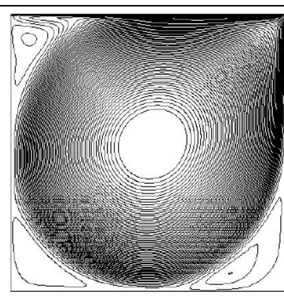
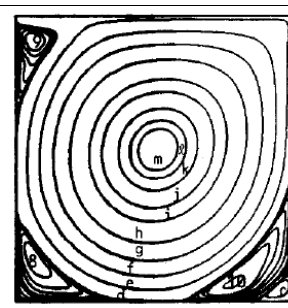
Reynolds number	Lattice Boltzmann method	Ghia et al 1982
100		

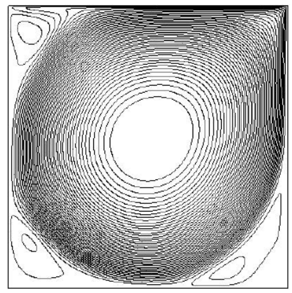
400		
1000		
3200		

5000		

At Re = 7500 and Re = 10000, the results as shows at table 4.2 at steady flow cannot be produce due to errors occurred during simulation. The error was due to the number of grid not sufficient enough to generate stable results. From this case, the results at Reynolds number higher than 7500 cannot be achieve using LBM.

Table 4.3: Comparisons between streamlines lattice Boltzmann method Ghia et al. at Re = 7500 and 10000

Reynolds number	Lattice Boltzmann method	Ghia et al 1982
7500		

10000		

4.3 LATTICE BOLTZMANN STEADY FLOW

The steady flow result below shows the characteristic of stream function at their final movement. The center location of this square moves as when is a change in Reynolds number. Base on figure 4.1 the structure and location of primary vortex or the center also changes in the different Reynolds number. At Re = 100, the location of primary vortex or center location is ¼ from the lid and more to the left, but the secondary eddies didn’t exist in the LBM that show the flow is in laminar flow. As the Reynolds number increase to 400, the center location move to the center and the size became bigger when it increase to 1000. The secondary eddies appear at the right side of the bottom cavity at Re = 400 and when it increase to 1000, they appear at left and right at the bottom of the lid. When the Reynolds number increase to 5000, the center location still at the half of the cavity but the size became smaller and the secondary eddies appear at left and right at the bottom and left side at the lid. The result for the Re = 7500 and 10000 is not valid as we mention before.

Lattice Boltzman Steady Flow

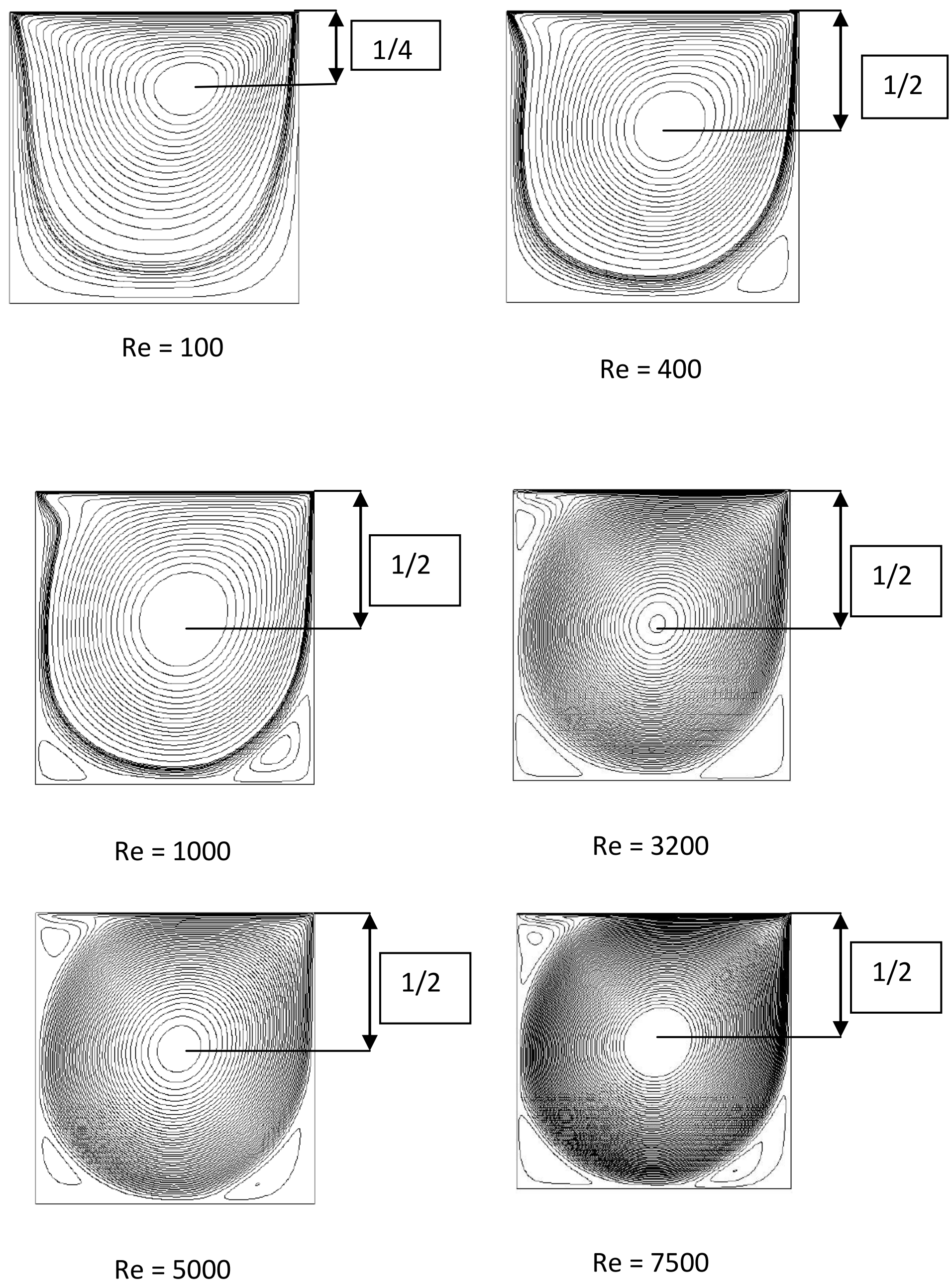
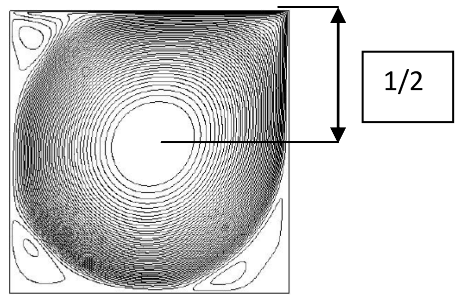


Figure 4.1: Continued



Re = 10000

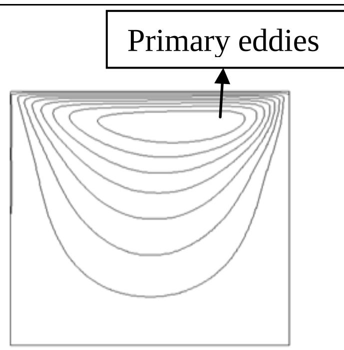
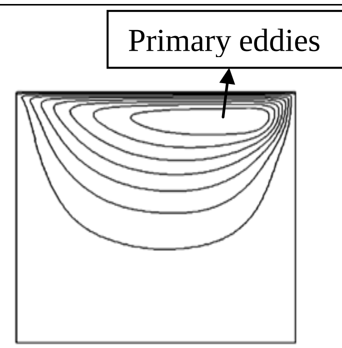
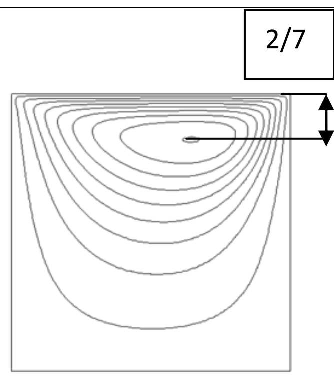
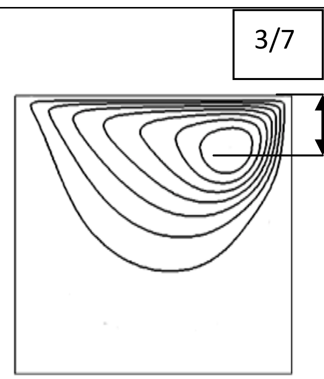
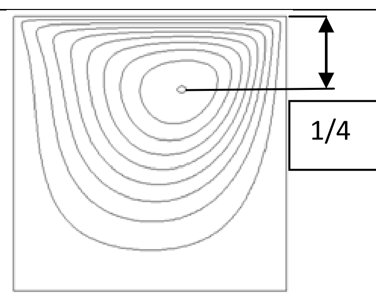
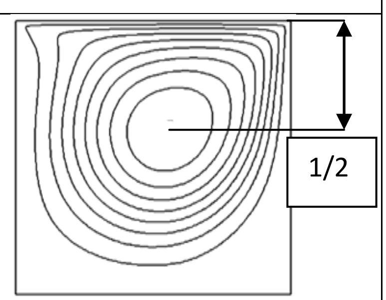
Figure 4.1: Lattice Boltzmann At Steady Flow

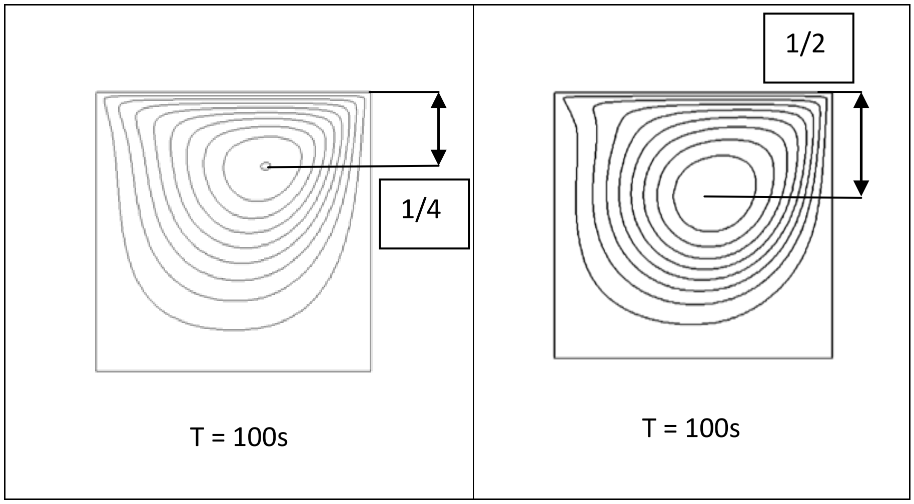
4.4 LATTICE BOLTZMANN (LBM) TRANSIENT FLOW

In order to study the center location of the lid driven cavity, we simulate the movement of the center location in transient flow. At Re = 100, the transient flow is in laminar condition, mean that the stream function look smooth due to time before it became to steady state. Base on table 4.5, at T = 1second, the location of primary eddies is at the lid and wide. When the time increase to 30 second, the primary eddies move 2/7 from the lid and size of the primary eddies became bigger. At 70 second, the primary eddies located almost at ¼ from the lid that is the primary eddies almost at steady flow. The flow of simulation of lid driven cavity is steady at 100 seconds.

At Re = 400, the center location of primary eddies is at the lid at one second, and the size almost same at Re = 100 but the lid more to the right side. When time increase to 30 second, the primary eddies is located 3/7 from the lid and at the right side. The shapes almost circle. At 70 second, the location of primary eddies is located ½ from the lid. The flow of this simulation is steady at 100 second.

Table 4.4: LBM transient flow at Re = 100 and Re = 400

Re = 100	Re = 400
 T = 1s	 T = 1s
 T = 30s	 T = 30s
 T = 70s	 T = 70s



4.5 LATTICE BOLTZMANN CENTER LOCATION

The movements of the center location of lid driven cavity are different at each Reynolds number that had been simulates. When we study the center location of driven cavity, the primary vortex that been produce was been tracked and plot in the graph below.

Figure 4.2 shows the graph of location the movement of the center location at different Reynolds number. The final of this trajectory is when the center location is at the steady flow as we can refer Figure 1. In order to study the center location, the movements of the primary vortex in the stream function need to capture. In higher Reynolds number, the trajectory is more too right side because the lid moves more to the right.

The movement of the center location in lid driven cavity using LBM from Re = 100 until Re = 3200, the result stop when it in the steady flow. When the Reynolds number at 7500, the movement of center location only circle round and it not goes to steady flow. It shows that the flow in the turbulence flow.

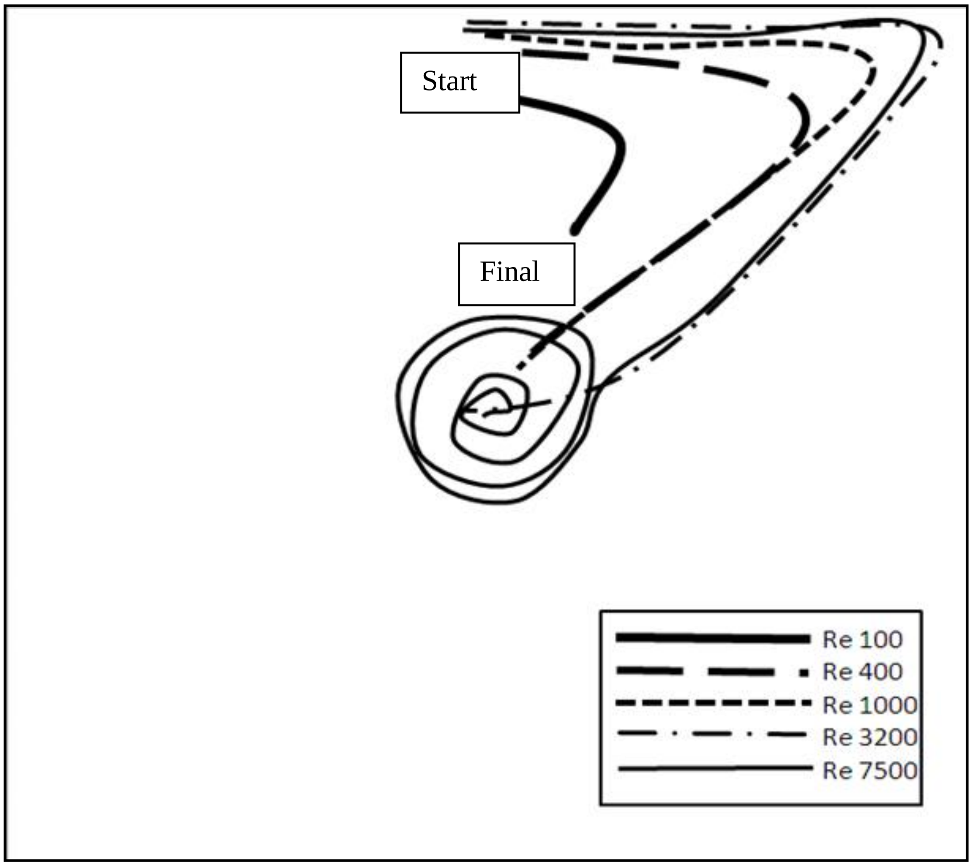


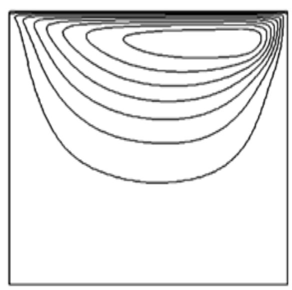
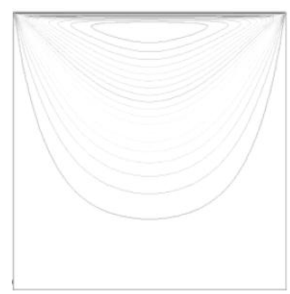
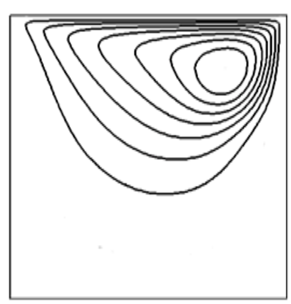
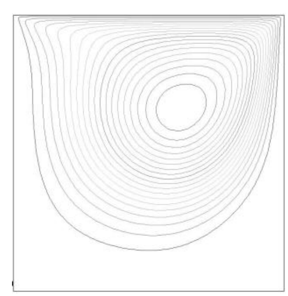
Figure 4.2: The movement of center location of driven cavity using LBM

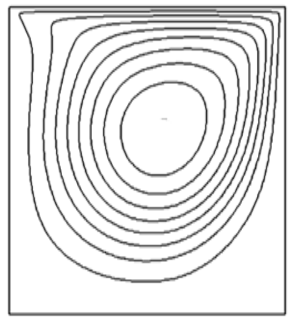
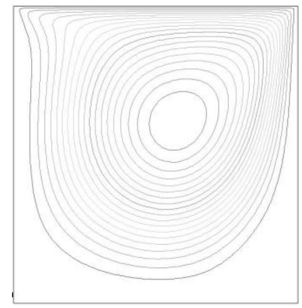
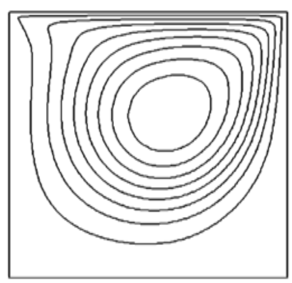
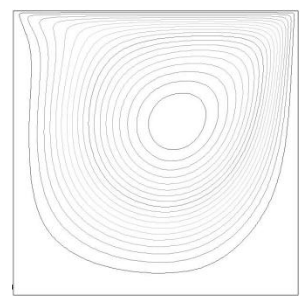
4.6 COMPARISONS WITH ANSYS, FLUENT

The result of lattice Boltzmann in transient flow is being compared with result from Navier Stoke using ANSYS FLUENT software. The comparison was made for test the performance of LBM. In order to study the center location of lid driven cavity, transient flow can capture the movement of the center location as time and as the dependent parameter in the same Reynolds number (Re = 400). From the result at table 4.5, it shows that the result from lattice Boltzmann method in laminar flow is more accurate than Navier Stoke from ANSYS FLUENT.

Using LBM method, the movement of center location of lid driven cavity is wider at the time zero and the ANSYS (FLUENT) result is at the center. At 10 second, the LBM center location move to the right side of cavity and same as ANSYS (FLUENT), but the different can be seen at the stream function. The result for both methods is same at 70 second and 100 second that is when it is in steady flow.

Table 4.5: Comparison between transient flow LBM and navier stoke from ANSYS (FLUENT) at Re = 400

Time (s)	LBM	ANSYS (FLUENT)
1		
30		

70		
100		

4.7 DISCUSSIONS

From the result of steady flow, 4.1 we shows the validation with Ghia et al to shows stream function in the good arrangement. From the result also shows that after Re = 7500, the result from LBM is not valid because of error. LBM is only good at Reynolds number below than 7500.

In the transient flow, the results of movement center location are the same as in the steady flow. When Re = 7500, as we can see at figure 4.1, the movement of stream function only in circular rotation and didn't achieve steady condition. But by using ANSYS (FLUENT), that is base on Navier Stoke solver, the result until Re = 20000, that is in the high Reynolds number can be produce and also can show the location of turbulence (usually secondary's eddies).

From the results, we can conclude that LBM is very good at Reynolds number below than 7500 and show the structure of stream function very well. But after Re = 7500 and above, the result is not sufficient due error.

CHAPTER 5

CONCLUSION AND RECOMENDATIONS

In this paper, the focus is to study the center location of fluid flow in lid driven cavity using Lattice Boltzmann method. This paper shows the steady flow and transient flow of lid driven cavity in two dimensional. The result of LBM is been compare to result using navier stoke that is from Ghia et al and result from (ANSYS) FLUENT. Base on result from LBM at steady flow and transient flow, the LBM is only good at Reynolds number below than 7500, that is we can conclude that LBM good at laminar flow. When the Reynolds number more than 7500, the prediction of the fluid flow cannot be obtain because of the error. Navier stoke still best solver if we want to predict at high Re number and at turbulence flow.

RECOMMENDATION

Even the lattice Boltzmann is not good at high Reynolds numbers, the flow is still good at Reynolds number below 7500. So that, the application of lattice Boltzmann in order to generate accurate simulations especially in transient flow can be use. The application maybe can be use to simulate the fluid flow at the pipe or simulate the wind tunnel using Lattice Boltzmann method.

REFERENCES

C. Sukop, Daniel T. Thone, Jr.,2005 Lattice Boltzmann Modeling, Michael

D. Arumuga Perumal, Anoop K. Dass, Multiplicity of steady solutions in two-dimensional lid-driven cavity flows by Lattice Boltzmann Method

Ercan Erturk, 2009, Discussions On Driven Cavity Flow 2009

Erturk and Gokcol, 2005, Numerical solutions of 2-D steady incompressible driven cavity flow at high Reynolds numbers

Fortin A, Jardak M, Gervais JJ, Pierre R., 1997, Localization of Hopf bifurcations in fluid flow problems. *International Journal for Numerical Methods in Fluids*

Gervais JJ, Lemelin D, Pierre R., 1997, Some experiments with stability analysis of discrete incompressible flows in the lid-driven cavity. *International Journal for*

P.D. Bates, S.N. Lane and R.I. Ferguson, , 2005 Computational Fluid Dynamics modelling for environmental hydraulics *Computational Fluid Dynamics: Applications in Environmental Hydraulics.* Edited by P.D. Bates, S.N. Lane and R.I. Ferguson, John Wiley & Sons, Ltd.

S. Albensoeder and H.C. Kuhlmann, (2004), Accurate three-dimensional lid-driven cavity flow

S. Nguyen, C. Delcarte, G. Kasperski and G. Labrosse (2006), Problems and recent developments in modeling of free surface flows

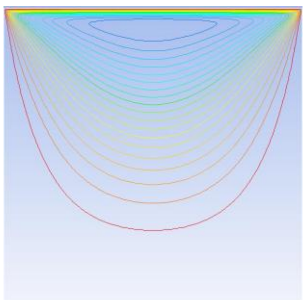
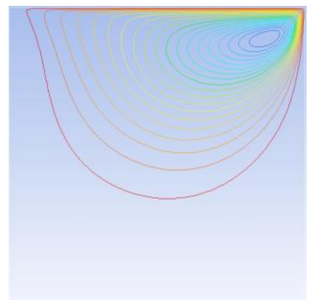
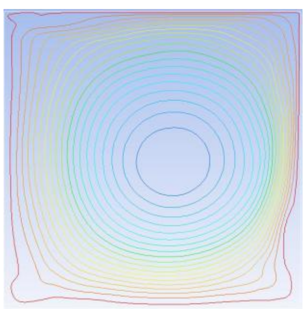
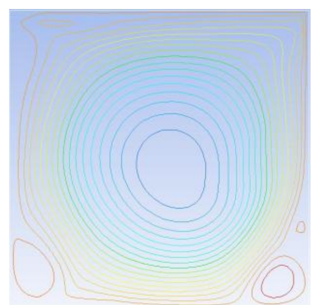
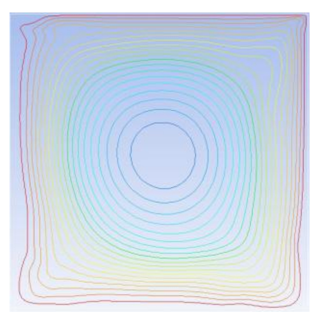
S. Nguyen, C. Delcarte, G. Kasperski and G. Labrosse, 2006, Problems and recent developments in modeling of free surface flows Sahin M, Owens RG. A novel fully-implicit finite volume method applied to the lid-driven cavity flow problem.

U. Ghia, K.N. Ghia, C.T. Shin, (1982) High-Re solutions for incompressible flow using the Navier–Stokes equations and a multigrid method, Journal of Computational Physics 48 387–411

APPENDIX A

APPENDIX A

Table: transient flow at Re= 20000 using ANSYS (FLUENT)

 T=1 second	 T=2 second
 T=20 second	 T= 30 second
 T= 80 second	 T= 100 second