TWO DIMENSIONAL PARTICLE FLOW INSIDE DRIVEN TRAPEZOID APPLYING EULARIAN-LAGRANGIAN APPROACH

NOR HAFRIZ BINTI ABD HALIL

A report submitted in partial fulfillment of
The requirements for the award of the degree of
Bachelor of Mechanical Engineering

Faculty of Mechanical Engineering

UNIVERSITI MALAYSIA PAHANG

JUNE 2013

ABSTRACT

In this report, Navier Stokes Equation (NSE) method is use to simulate lid-driven flow in a two dimensional isosceles trapezoidal cavity. For the first part in this report, the simulation of the NSE are compared with the existing research (Lattice Boltmann Method of lid-driven flow in trapezoidal cavities) and for the second part is to studied about the number of the vortexes based of Reynolds numbers. For these numerical simulations, the effect of Reynolds numbers, various lid movement and number of vortices in the isosceles trapezoidal cavities are studied. Re is varied from 100 to 7500. Numerical result show that, as the Re increases, the phenomena in the cavity becomes more and more complex, and the number of the vortexes increases. Furthermore, as Re is increased, the flow in the cavity undergoes a complex transition (from steady to the periodic flow and finally to the chaotic flow). For the various lid movements, the maximum number of the vortexes is six when upper and the bottom lid move in the same direction.

ABSTRAK

Dalam laporan ini, kaedah Persamaan Navier Stokes (NSE) adalah digunakan untuk mensimulasikan aliran didorong tudung dalam dimensi rongga sama kaki trapezoid dua. Untuk bahagian pertama di dalam laporan ini, simulasi NSE dibandingkan dengan penyelidikan yang sedia ada (Lattice Boltmann Kaedah aliran didorong tudung dalam rongga trapezoid) dan bahagian kedua adalah untuk mengkaji tentang bilangan putaran berdasarkan nombor Reynolds . Untuk simulasi berangka ini, kesan nombor Reynolds, pelbagai pergerakan tudung dan bilangan vorteks dalam rongga trapezoid sama kaki dikaji. Re diubah 100-7500. Menunjukkan hasil berangka bahawa, sebagai Re bertambah, fenomena dalam rongga menjadi lebih dan lebih kompleks, dan bilangan putaran bertambah. Tambahan pula, sebagai Re meningkat, aliran dalam rongga menjalani peralihan kompleks (daripada stabil kepada aliran berkala dan akhirnya aliran huru-hara). Untuk pelbagai pergerakan tudung, bilangan maksimum putaran adalah enam apabila atas dan tudung langkah bawah dalam arah yang sama.

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURES

LIST OF ABBREVIATIONS

E-L	Eularian- Lagrangian
NSE	Navier-Stokes Equation
LBM	Lattice Boltzmann Method
CFD	Computational Fluid Dynamics
FDM	Finite Element Method
FVM	Finite Volume Method
CIP	Cubic Interpolated Pseudo-Particle

LIST OF SYMBOLS

AR	Aspect Ratio
H	Height of cavity
P	Pressure
ρ	Density
Re	Reynolds Number
T	Time
T	Dimensionless time
U	Velocity in x direction
u_∞	Lid velocity
U	Dimensionless velocity in x direction
V	Velocity in y direction
V	Dimensionless velocity in y direction
ν	Kinematic Viscosity
W	Width of the cavity
X	Axial distance
X	Dimensionless axial distance
Y	Vertical distance
Y	Dimensionless vertical distance
M	Dynamics viscosity
v	Kinematic viscosity
Ω	Vorticity
Ω	Dimensionless vortocity
Ψ	Stream function
Ψ	Dimensionless stream function
L	Length

Superscript

N	Current Value
n+1	Next step value
*	Non advection phase value

Subscript

I	*x* direction node
J	*y* direction node
max i	*x* direction maximum node
Max j	*y* direction maximum node
∞,*e*	Free stream condition

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

The main idea of this project is to expose student with many aspect of engineering work; run simulation using MATLAB, edit existing coding, about the particle flow and the effect of Reynolds Number and particle position. The task follows a common product development activity, where student need to apply all their engineering knowledge and skill to complete this project. In random, student must know some basic knowledge which is definition of particle flow, software and about the experiment. In the other hand, student must master in conceptual of modifying code and run the coding using MATLAB. Not only that, student must know how to solve problem in edit the coding.

1.2 PROBLEM STATEMENT

Pattern of the particle flow will change if Reynolds number and particle position not fixed. What will happen to the pattern of the particle flow when it is at different Reynolds number and various lid movements.

1.3 OBJECTIVES

Objectives of this project are:

- To identify flow inside driven trapezoid cavity for different Reynolds number.
- To study the vortex structures of flow for various lid movements.

1.4 SCOPE OF THE STUDY

Scopes for this particular research are bounded by these three matters and therefore will be followed throughout the research :

- Applying Eularian – Langrangian approach
- Study on the vortex structures of flow with various lid movements. ($U_{TOP} = 1, U_{TOP} = -1, U_{Bottom} = 1, U_{Bottom} = -1$).
- Focus on different Reynolds numbers. (100, 1000, 3200, 5000, 7500)
- Do simulation on the vortex structures of flow with different Reynolds number and various lid movements using Matlab.

1.5 INTRODUCTION TO NAVIER STOKES EQUATION

Navier-Stokes equation is well known in the field of fluid dynamic. The equation is nonlinear and usually the flows that apply this equation are considered incompressible. Many fluid flows are governed by this equation because in describing the conservation of momentum, the equation is almost perfect. The equation consists the unsteady term, the diffusive term, pressure term, convective term and the external force. In spite, there is no analytical solution to this equation as there are many Partial differential terms in the equation.

1.6 STRUCTURE OF REPORT

This report consists of five chapters which are Chapter 1 (introduction), chapter 2 (literature review), chapter 3 (methodology), chapter 4 (result and discussion) and chapter 5 (conclusion). Chapter 1 is focused on the basic information about the study which is background of study, problem statement, objectives of study, scope and the structure of this project.

While chapter 2 was focused on the review of past studies that are relate to this study. This chapter includes about particle flow, driven cavity, Lagrangian-Eularian, Reynolds Numbers and some equations that related to this topic. A review of other relevant research studies also provided. The review is detailed so that the present research effort can be properly tailored to add to the present body of literature as well as to justly the scope and direction of the present research effort.

In chapter 3, a review of the methodology that has been suggested in conducting the study was provided. It is start with the designing of the study, what the activities in performing this study has been reviewed. This chapter reviews the planning that has been suggested in conducting the study. A review of data analysis and software that will be used also discussed in general.

Chapter 4 was focused on collecting data from the simulation. The data focus on pattern of particle flow for different Reynolds number and various lid movements that had been used. The data will be used to make analysis. The data has been analyzed using graphs and table. This chapter also provided a discussion for data and analysis of the results.

Finally, chapter 5 was conclusion of this project and discusses some recommendations for futurer work.

CHAPTER 2

LITERATURE REVIEW

2.0 INTRODUCTION

Chapter 2 explains about literature review of the project. All the theories, concepts, and other related standards that related to the study were reviewed. It includes particle flow, Reynolds number, and method that use. A review of other relevant research studies also provided.

2.1 FLOW BEHAVIOUR IN LID DRIVEN CAVITY

In many industrial processes, including the transport of solids in liquids and removal of particulates from gas streams for pollution control, particle flow is most important things. The main problems of flow in industrial design involve the flow of liquids or gas with suspended solids.

Stokes number is a key parameter in fluid-particle flows, which is the ratio of the response time of a particle to a time characteristic of a flow system. The time that a particle takes to respond to a change in carrier flow velocity is known as a particle response time. The Stokes number is small when it is less than 0.1, then the particles have sufficient time to respond to the change in fluid velocity, so the particle velocity approaches the fluid velocity. However, if the Stokes number is large where is greater than 10, the particles have short time to respond to the varying fluid velocity and the particle velocity shows little change.

The relative concentration of the particles in the fluid is referred as the loading. There are several ways to define the loading, such as the ratio of particle mass flow to fluid mass flow. Highly loaded particle flows were involved in many industrial applications.

When the particle loading is small, the fluid will affect the particle properties, such as velocity and temperature and the fluid properties will not be influenced by the particle. This is referred to as one-way coupling. If the condition is that there is a mutual interaction between the particles and fluid, the flow is two-way-coupled. When the Stokes number increasing, the two-way coupling effect will reduce because the particles undergo acceleration.

The flow is termed dilute if the particles motion is controlled by the action of the fluid on the particle. The particles will collide with each other and their motion will be depending on particle-particle collisions, when the particle concentration is sufficiently high and the flow is then regarded as dense.

2.2 DRIVEN CAVITY

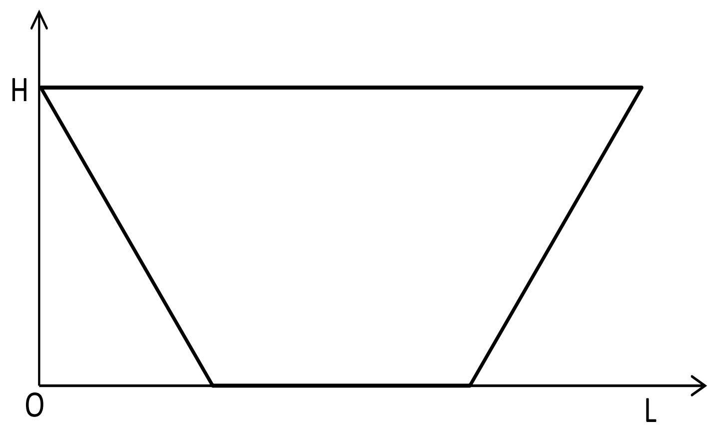

Figure 2.1 Coordinate of the isosceles trapezoidal cavity

In this section, the trapezoidal cavity model is first introduced. L is the length of the top wall and H is the height of the cavity. In all simulations presented here. L and H are set to be 1 and 0.577.

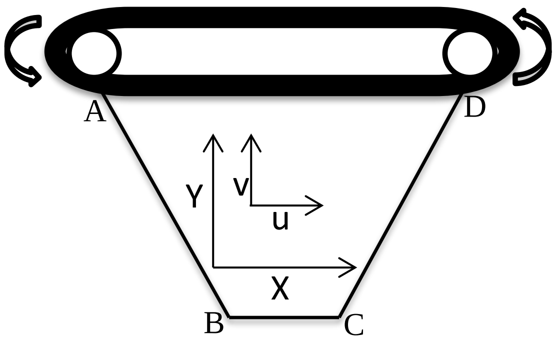

Figure 2.2 The schematic diagram for a 2D lid driven cavity

The flow in the driven cavity will be divided into two groups it the development of new numerical schemes (where the driven cavity is used as a benchmark) and is the analysis of the physics of the flow it is stated in most of the paper. For these objectives, there are a lot of works devoted to the issue of transition to turbulence and regarding to the location of the so-called first Hopf bifurcation.

A few work has been done on rectangular rather than square cavities. Some of researchers used a lattice boltzmann method for the analysis of the 2D vortex structure in a rectangular lid driven cavity for the various Reynolds numbers and for different depth to width ratios. There are some works devoted to the issue of motion of two walls in the cavity rather than one wall. This is an example of the work that been done by some researchers, where the linear stability of two counter-rotating vortices driven by the parallel motion of the two walls is investigated by a FVM. The driven cavity will be effect by the flow temperature, where it also been the subject of some interests, a temperature gradient is introduced and this leads to buoyancy driven flow. The field becomes coupled by temperature distributions and velocity.

The particles are treated as points, where motion is the result of the influence of the fluid phase and the flow of the particles were modelled by the Eularian-Lagrangian (E-L) approach. The biggest drawback of this method is the fact that in the real industrial applications the number of the particles is too large to consider the behaviour all of them.

In order to overcome this problem, a number of particles are often “pooled” to form “virtual particles” with the same parameters, so that the number of the particles to be considered can be reduced. The E–L type of model can be used accurately without introducing virtual particles for small geometries with limited number of particles. The E–L approach makes it possible to investigate the fundamental processes. (Elsevier B.V.2008)

The lid driven cavity flow is most probably one of the most studied fluid problems in computational fluid dynamics field. The easiness of the geometry of the cavity flow makes the problem easy to code and apply boundary conditions. Sometimes the problem looks like simple in many ways, but the flow in a cavity keeps all the flow physics with counter rotating vortices appear at the corners of the cavity. Driven cavity flow is the benchmark of the problem for the numerical methods in term of accuracy and numerical efficiency. In basically the driven cavity flow can be grouped into three categories.

For the first category studies that present steady solution is at the high Reynolds numbers, at the presented solutions of steady 2-D incompressible flow in a driven cavity for Re ≤ 10000. Solutions for Re=12,500 also have presented among these researchers. Moreover, other researchers also have presented steady solutions up to Re=20,000. In the second category, the following two dimensional Direct Numerical Simulation studies on driven cavity flow. For the third category studies, Fortin, Gervais(2000), Sahin(1998) and Abouhamza(2002) are examples of two dimensional hydrodynamic stability studies on driven cavity flow.

Even though that driven cavity flow is studied at this scope in numerical studies, the nature of the flow at high Reynolds number is still not agreed upon. For example many studies come from the first category, where present steady solutions at very high Reynolds numbers showing that there exists a solution for steady 2-D Navier-Stokes equations for the flow inside the driven cavity.

On the other hand, after a two dimensional hydrodynamic stability analysis or Direct Numerical Simulation, the studies from the second and third categories will claim beyond a moderate Reynolds number. The flow in a 2-D driven cavity is unsteady, therefore a steady solution does not exist hence a steady solution at high Reynolds numbers is not computable.

The main purpose of this study is then to discuss the incompressible flow in a 2-D driven cavity in terms of physical, numerical aspects and mathematical, together with a very brief literature survey on numerical studies and experimental. We will also present very fine grid steady solutions for the driven cavity flow at very high Reynolds numbers. We will discuss on the driven cavity flow in an attempt to address the important points mentioned as above. (Ercan Erturk,2009)

2.3 EULERIAN-LAGRANGIAN

Normally Eulerian-Lagrangian approach is widely used to simulate dispersed two-phase of the flows. In this approach the dispersed phase is represented by discrete particle in a Lagrangian frame, while the carrier phase is represented by continuous field in an Eulerian frame reference. The mean interphase momentum transfer term cannot be neglected, and two-way coupling effects must be accounted for when it occur in two-phase flows with non-negligible mass loading. The mean interphase momentum transfer term, it is the average force of exerted by the particles on the fluid and accounts for the presence of the dispersed phase on the fluid phase.

In numerical implementations of the Eulerian-Lagrangian method, the numerical estimate of the mean interphase momentum transfer term (or any other mean quantity) at Eulerian grid nodes is obtained using a finite number of particles, leading to statistical and bias errors (Garg et al,2007).

Statistical error can be reduced either by increasing the number of particles, or by averaging over multiple independent realizations. Bias error is insensitive to the number of independent realizations and becomes zero only in the limit of infinite particles or also called the dense data limit.

In addition to these errors, a finite time is a main step to the usual spatial and a finite number of grid cells and temporal discretization errors that are encountered in numerical simulations of single-phase flow. The scaling of each of these error contributions was bias, statistical, and discretization error.

It will divided with variation of numerical parameters determines the numerical convergence characteristics of any Lagrangian-Eulerian numerical implementation. Although Lagrangian-Eulerian simulations are frequently used to simulate multiphase of flows, their numerical convergence and accuracy properties have not been critically examined until recently.(R.Garg et al, 2008)

A fluid flow (both liquid and air) may be described in two different ways it is the Lagrangian approach (named after the French mathematician Joseph Louis Lagrange), and the Eulerian approach (named after Leonhard Euler, a famous Swiss mathematician). In the Lagrangian approach, one particle is chosen and is followed as it moves through space with time. The line traced out by that one particle is called a particle pathline. An example is a transmitting ocean buoy that observes a set path over regular intervals over a period of time. The path observed is the particle pathline.

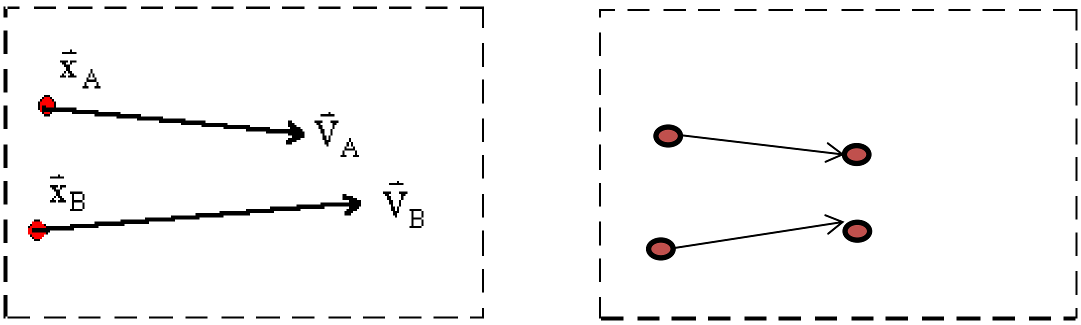

Figure 2.3 Lagrangian description of Particle Flow

In the Lagrangian description of fluid flow, individual fluid particles are "marked," and their positions, velocities are described as a function of time. In the example shown, particles A and B have been identified. Position vectors and velocity vectors are shown at one instant of time for each of these marked particles.

As the particles move in the flow field, their positions and velocities change with time, as seen in the diagram. The physical laws, such as Newton's laws and conservation of mass and energy, apply directly to each particle.

If there were only a few particles to consider, as in a high school physics experiment with billiard balls, the Lagrangian description would be desirable. However, fluid flow is a continuum phenomenon, at least down to the molecular level. It is not possible to track each "particle" in a complex flow field. Thus, the Lagrangian description is rarely used in fluid mechanics. A Eulerian approach is used to obtain a clearer idea of the airflow at one particular instant. One can look at a “photograph” of the flow of, for instance, surface ocean currents at a particular fixed time. The entire flow field is easily visualized. The lines comprising this flow field are called streamlines.

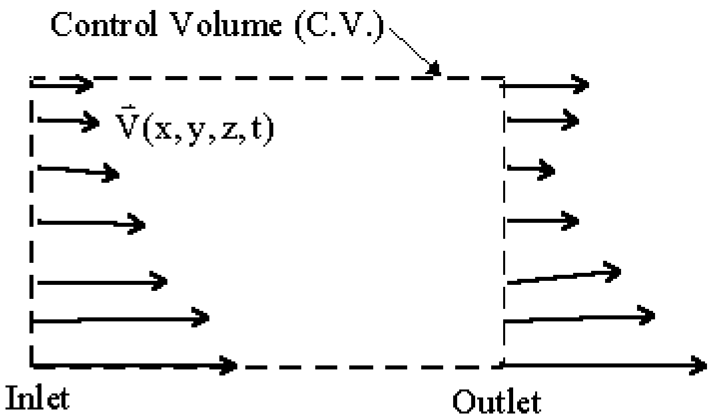

Figure 2.4 Eulerian Description of Fluid Flow

In the Eulerian description of fluid flow, individual fluid particles are not identified. Instead, a control volume is defined, as shown in the diagram. Pressure, velocity, acceleration, and all other flow properties are described as fields within the control volume. In other words, each property is expressed as a function of space and time, as shown for the velocity field in the diagram.

In the Eulerian description of fluid flow, one is not concerned about the location or velocity of any particular particle, but rather about the velocity, acceleration, etc. of whatever particle happens to be at a particular location of interest at a particular time.

Since fluid flow is a continuum phenomenon, at least down to the molecular level, the Eulerian description is usually preferred in fluid mechanics. Note that, the physical laws such as Newton's laws and the laws of conservation of mass and energy apply directly to particles in a Lagrangian description. Hence, some translation or reformulation of these laws is required for use with an Eulerian description.

2.4 REYNOLDS NUMBERS

Reynolds number can be defined for a number of different situations where a fluid is in relative motion to a surface. These definitions generally include the fluid properties of density and viscosity, plus a velocity and a characteristic length or characteristic dimension. This dimension is a matter of convention – for example a radius or diameters are equally valid for spheres or circles, but one is chosen by convention. For aircraft or ships, the length or width can be used. For flow in a pipe or a sphere moving in a fluid the internal diameter is generally used today.

Other shapes such as rectangular pipes or non-spherical objects have an equivalent diameter defined. For fluids of variable density such as compressible gases or fluids of variable viscosity such as non-Newtonian fluids, special rules apply. The velocity may also be a matter of convention in some circumstances, notably stirred vessels. The inertial forces, which characterize how much a particular fluid resists any change in motion are not to be confused with inertial forces defined in the classical way.

$$Re = \frac{\rho v L}{\mu} = \frac{VL}{v} \tag{2.1}$$

The Reynolds number (Re) is a dimensionless number that gives a measure of the ratio of inertial forces to viscous forces and consequently quantifies the relative importance of these two types of forces for given flow conditions.