KINEMATIC BEHAVIOR OF SOLID PARTICLES IN ENCLOSED LID-DRIVEN CAVITY USING CIP METHOD

ALI AKBARI SHELDAREH

A project report submitted in partial fulfilment of the Requirements for the award of the degree of Master of Engineering (Mechanical Engineering)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

FEBRUARY 2012
To my beloved family and friends
And my respectful supervisor

Thank you for all kindness and sacrifices you made for me.
ACKNOWLEDGEMENTS

In the name of Allah, the most Gracious, the Most Merciful.

I have faced difficulties during preparation of this thesis, but with the help and support of Dr. Nor Azwadi Che Sidik, I have overcome all of them. I sincerely thank Dr. Nor Azwadi Bin Che Sidik for his guidance, help and motivation. I have learnt a lot from him and his positive contributions and responsibility were a boon to me.

I would like to thank my Supervisor Dr. Nor Azwadi Bin Che Sidik, the Head of Department of Postgraduate studies in the Mechanical Department for his warm assistance in finalizing the research problem and giving useful comments. A great deal of appreciation also goes to the Faculty of Mechanical Engineering (FKM). I am especially grateful to my fellow course mate Arman Safdari, for his sincere assistance, useful insights, and his positive contributions.

Nevertheless, I would like to thank to my family for loving me and being supportive in the duration of completing this thesis. I wish to thank my parents, Dr. Fereydoun Akbari Sheldareh and A. Ghanbari, for their love, support and kindness. They have always been a source of inspiration and encouragement for me. I wish to thank my brothers Arash and Amir Hossein who are always giving me positive energy and kindness, brothers who are the apple of my eyes.
ABSTRACT

The Cubic Interpolated Pseudo-Particle Navier Stokes equation (CIP-NSE) was applied to investigate the two-dimensional laminar square lid driven cavity flow of water at Reynolds number 1000. CIP-NSE scheme was used to solve hyperbolic term of the vorticity transport equation. In the CIP-NSE, the gradient and the value of the vorticity at the nodes is determined and the stream function is then determined using the vorticity equation. It is discovered that the numerical simulation of CIP-NSE provided a very good agreement with the established benchmark results by previous researchers. The Runge-Kutta method has been used to calculate the velocity and position of the particle with the effects of Drag force and Gravitational forces. The hard sphere model has been applied to show the collisions effect on particles in the Lid-Driven cavity. The main result achieved from the investigation is that, as the density of particles increases the number of particles collision in first seconds of the investigation decreases and the number of particles settled on the floor of the cavity increases, so for higher density of particles there have been large number of particles settlement on the floor and the collision at starting of investigation decrease as the particles moves slower, and for the lighter particles and lower density of particles number of collision at starting of investigation in more as the particles are lighter and move faster but the particles settlement on the floor of cavity are less in compare to higher density of particles. All simulation have been done for four different density of particle which are 1000, 1200, 1700, and 2000 (kg/m³).
ABSTRAK

# TABLE OF CONTENTS

<table>
<thead>
<tr>
<th>CHAPTER</th>
<th>TITLE</th>
<th>PAGES</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACKNOWLEDGEMENTS</td>
<td></td>
<td>iv</td>
</tr>
<tr>
<td>ABSTRACT</td>
<td></td>
<td>v</td>
</tr>
<tr>
<td>ABSTRAK</td>
<td></td>
<td>vi</td>
</tr>
<tr>
<td>LIST OF TABLES</td>
<td></td>
<td>xi</td>
</tr>
<tr>
<td>LIST OF FIGURES</td>
<td></td>
<td>xii</td>
</tr>
<tr>
<td>LIST OF ABBREVIATIONS</td>
<td></td>
<td>xiv</td>
</tr>
<tr>
<td>LIST OF SYMBOLS</td>
<td></td>
<td>xv</td>
</tr>
<tr>
<td>CHAPTER 1</td>
<td>INTRODUCTION</td>
<td>1</td>
</tr>
<tr>
<td>1.1</td>
<td>Introduction</td>
<td>1</td>
</tr>
<tr>
<td>1.2</td>
<td>Computational Fluid Dynamic (CFD)</td>
<td>4</td>
</tr>
<tr>
<td>1.2.1</td>
<td>Governing Equation in CFD</td>
<td>6</td>
</tr>
<tr>
<td>1.2.2</td>
<td>The Navier-Stokes Equations</td>
<td>7</td>
</tr>
<tr>
<td>1.3</td>
<td>Problem Statement</td>
<td>9</td>
</tr>
<tr>
<td>1.4</td>
<td>Objectives of the research</td>
<td>9</td>
</tr>
<tr>
<td>1.5</td>
<td>Significance of study</td>
<td>10</td>
</tr>
<tr>
<td>1.6</td>
<td>Scope of the Study</td>
<td>11</td>
</tr>
</tbody>
</table>
## CHAPTER 2

### LITERATURE REVIEW

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Introduction</td>
<td>12</td>
</tr>
<tr>
<td>2.2</td>
<td>Background of Study</td>
<td>13</td>
</tr>
<tr>
<td>2.2.1</td>
<td>The Navier-Stokes Equation</td>
<td>17</td>
</tr>
<tr>
<td>2.2.2</td>
<td>Analytical Solution to Navier-Stokes Application</td>
<td>18</td>
</tr>
<tr>
<td>2.3</td>
<td>Stream Function-Vorticity Navier-Stokes Approach</td>
<td>21</td>
</tr>
<tr>
<td>2.4</td>
<td>Essence of Finite Difference</td>
<td>22</td>
</tr>
<tr>
<td>2.5</td>
<td>Cubic Interpolated Pseudo-Particle (CIP)</td>
<td>23</td>
</tr>
<tr>
<td>2.6</td>
<td>Two-Phase Flows</td>
<td>24</td>
</tr>
<tr>
<td>2.7</td>
<td>Forces Acting on Particle</td>
<td>25</td>
</tr>
<tr>
<td>2.7.1</td>
<td>Gravitational Force</td>
<td>25</td>
</tr>
<tr>
<td>2.7.2</td>
<td>Buoyancy Force</td>
<td>26</td>
</tr>
<tr>
<td>2.7.3</td>
<td>Drag Force</td>
<td>26</td>
</tr>
<tr>
<td>2.8.1</td>
<td>Particle Collisions</td>
<td>27</td>
</tr>
<tr>
<td>2.8.2</td>
<td>Particle-Particle Collision</td>
<td>28</td>
</tr>
<tr>
<td>2.8.2.1</td>
<td>Hard Sphere Model</td>
<td>29</td>
</tr>
<tr>
<td>2.8.2.2</td>
<td>Soft Sphere Model</td>
<td>29</td>
</tr>
</tbody>
</table>

## CHAPTER 3

### RESEARCH METHODOLOGY

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Introduction</td>
<td>30</td>
</tr>
<tr>
<td>3.2</td>
<td>Overview</td>
<td>31</td>
</tr>
<tr>
<td>3.3</td>
<td>Primary Data</td>
<td>31</td>
</tr>
<tr>
<td>3.3</td>
<td>Secondary Data</td>
<td>31</td>
</tr>
<tr>
<td>3.4</td>
<td>Governing Equations in Cavity Flow</td>
<td>32</td>
</tr>
<tr>
<td>3.5</td>
<td>Stream Function-Vorticity Approach</td>
<td>32</td>
</tr>
</tbody>
</table>
CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 Introduction 71

4.2 Simulation of Fluid in Lid-Driven Cavity 72

4.3 Code Validation 73

4.3.1 Code Validation for Fluid 74

4.3.1.1 Code Validation of Fluid Flow Using CIP Method 79

4.3.2 Code Validation for Solid Particle 80

4.3.2.1 Code Validation for Solid Particle 80

4.3.2.2 Particle Flow in Lid-Driven Cavity 80

4.4 Main Results of the Research 82

4.4.1 Comparison Between Particle Density of 1000 and 1200(kg/m³) 83
4.4.2 Comparison Between Particle Density of 1700 and 2000 (kg/m³)  

4.5 Graphs of the Main Simulation  
4.5.1 Particles Settlement Graph  
4.5.2 Particles Collision Graph  

4.6 Summary of Results  

CHAPTER 5  
CONCLUSION AND RECOMMENDATION  

5.1 Conclusion  
5.2 Recommendation  

REFERENCES
### LIST OF TABLES

<table>
<thead>
<tr>
<th>TABLE NO</th>
<th>TITLE</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Table 3.1</td>
<td>Relation Between Pre- and Post-Collisional Velocities</td>
<td>69</td>
</tr>
<tr>
<td>Table 4.1</td>
<td>Comparison of Velocity ((U)) at Vertical Midsection of Re 1000 for Various Grids Including CIP-NSE With Ghia Result.</td>
<td>76</td>
</tr>
<tr>
<td>Table 4.2</td>
<td>Comparison of Velocity ((V)) at Horizontal Midsection of Re 1000 for Various Grids Including CIP-NSE With Ghia Result.</td>
<td>78</td>
</tr>
<tr>
<td>Table 4.3</td>
<td>Comparison Between (\rho_p = 1000) and (\rho_p = 1200), Time 1-10 s</td>
<td>83</td>
</tr>
<tr>
<td>Table 4.4</td>
<td>Comparison Between (\rho_p = 1000) and (\rho_p = 1000), Time 20-40 s</td>
<td>84</td>
</tr>
<tr>
<td>Table 4.5</td>
<td>Comparison Between (\rho_p = 1000) and (\rho_p = 1200), Time 45-55 s</td>
<td>85</td>
</tr>
<tr>
<td>Table 4.6</td>
<td>Comparison Between (\rho_p = 1700) and (\rho_p = 2000), Time 1-10 s</td>
<td>86</td>
</tr>
<tr>
<td>Table 4.7</td>
<td>Comparison Between (\rho_p = 1700 (kg/m^3)) and (\rho_p = 2000 (kg/m^3)), Time 15-25 s</td>
<td>87</td>
</tr>
<tr>
<td>Table 4.8</td>
<td>Comparison Between (\rho_p = 1700) and (\rho_p = 2000), After 30 s</td>
<td>88</td>
</tr>
</tbody>
</table>
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>FIGURE NO</th>
<th>TITLE</th>
<th>PAGE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Figure 1.1</td>
<td>Classification of Fluid Dynamics Solution</td>
<td>3</td>
</tr>
<tr>
<td>Figure 2.1</td>
<td>The Couette Flow at Steady State</td>
<td>19</td>
</tr>
<tr>
<td>Figure 2.2</td>
<td>Numerical and Analytical Graph of Couette Flow</td>
<td>20</td>
</tr>
<tr>
<td>Figure 3.2</td>
<td>The Grid Used in Simulation.</td>
<td>42</td>
</tr>
<tr>
<td>Figure 3.1</td>
<td>Rectangular Meshing of the Cavity</td>
<td>43</td>
</tr>
<tr>
<td>Figure 3.4</td>
<td>Square Meshing of the Cavity</td>
<td>44</td>
</tr>
<tr>
<td>Figure 3.5</td>
<td>Comparison of CIP Method for First Order Wave Equation With Classical Method With CFL 0.2</td>
<td>46</td>
</tr>
<tr>
<td>Figure 3.6</td>
<td>Meshing in Two Dimensional CIP</td>
<td>55</td>
</tr>
<tr>
<td>Figure 3.7</td>
<td>Particle-Particle Collisions</td>
<td>62</td>
</tr>
<tr>
<td>Figure 3.8</td>
<td>Relative Motion of Two Spheres.</td>
<td>64</td>
</tr>
<tr>
<td>Figure 3.9</td>
<td>Particle-Wall Collision Schematic</td>
<td>68</td>
</tr>
<tr>
<td>Figure 3.10</td>
<td>Flow Chart of the Project</td>
<td>70</td>
</tr>
<tr>
<td>Figure 4.1</td>
<td>The Schematic Diagram for a 2D Lid-Driven Cavity</td>
<td>72</td>
</tr>
<tr>
<td>Figure 4.2</td>
<td>The Boundary for a 2D Lid-Driven Cavity</td>
<td>73</td>
</tr>
<tr>
<td>Figure 4.3</td>
<td>Streamline Plot Using CIP-NSE 129x129 Grid and Ghia Result With 129x129 Grid, Re Number 100, 400 and 1000.</td>
<td>74</td>
</tr>
<tr>
<td>Figure 4.4</td>
<td>Comparison of U-Velocity Along Vertical Lines.</td>
<td>75</td>
</tr>
<tr>
<td>Figure 4.5</td>
<td>Comparison of V-Velocity Along Vertical Lines.</td>
<td>77</td>
</tr>
<tr>
<td>Figure 4.6</td>
<td>Comparison of Velocity Profiles of CIP-NSE and Ghia Through the Center of the Cavity, $U$ and $V$ Along the Centerline for 128x128.</td>
<td>79</td>
</tr>
<tr>
<td>Figure 4.7</td>
<td>Trajectory of a Particle in a Driven Cavity.</td>
<td>81</td>
</tr>
<tr>
<td>Figure 4.8</td>
<td>Graph of Particle Settlement</td>
<td>89</td>
</tr>
<tr>
<td>Figure 4.9</td>
<td>Graph of Particle Collisions Number</td>
<td>90</td>
</tr>
</tbody>
</table>
LIST OF ABBREVIATIONS

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CIP</td>
<td>Cubic Interpolated Pseudo-particle</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>PDE</td>
<td>Partial Differential Equation</td>
</tr>
<tr>
<td>NSE</td>
<td>Navier-Stokes Equation</td>
</tr>
<tr>
<td>FDM</td>
<td>Finite Difference Method</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
</tr>
<tr>
<td>FVM</td>
<td>Finite Volume Method</td>
</tr>
<tr>
<td>CIPNSE</td>
<td>Cubic Interpolated Pseudo-particle Navier-Stokes Equation Method</td>
</tr>
</tbody>
</table>
LIST OF SYMBOLS

$\textit{AR}$ - Aspect Ratio

$H$ - Height of cavity

$p$ - Pressure

$\rho$ - Density

$Re$ - Reynolds Number

$t$ - Time

$T$ - Dimensionless time

$u$ - Velocity in x direction

$u_\infty$ - Lid velocity

$U$ - Dimensionless velocity in x direction

$v$ - Velocity in y direction

$V$ - Dimensionless velocity in y direction

$W$ - Width of cavity

$x$ - Axial distance

$X$ - Dimensionless axial distance

$y$ - Vertical distance

$Y$ - Dimensionless vertical distance

$\mu$ - Dynamic viscosity

$\nu$ - Kinematic viscosity

$\omega$ - Vorticity

$\Omega$ - Dimensionless vorticity
\psi - Stream function
\Psi - Dimensionless stream function
W - Gravitational force
F_b - Bouyancy force
F_D - Drag force
\rho_p - Particle density
g - Gravity
C_D - Coefficient of drag
m_j - Mass of particle
F_{pj} - External force
\omega_j - Angular velocity
T_j - Particle torque
u_j - X direction velocity of particle
v_j - Y direction velocity of particle
V_i^{(0)} - Pre-collision velocity
V - Post-collision velocity
G - Collisional relative velocity
e - Restitution coefficient
a - Particle radius
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
\( \infty, e \)  -  Free stream condition
CHAPTER 1

INTRODUCTION

1.1 Introduction

One of the biggest inventions of mankind is the computer. Nowadays, the lack of a computer may cause many problems. The world is changing rapidly, with the computer’s evolution as evidence. Since its invention in the early 20th century, the computer started off as big as a house and was incapable of rapid calculations.

However it was not the end of the story; it was just the beginning of a great invention. After years of struggle and improvements made by companies, they have improved the computer in many aspects such as the size, weight and the performance. People used to need days or months to execute a task on an old version of the computer; the same task can be done in mere minutes on today’s computers.

Researchers and accountants benefit a lot from the improvements of today’s computer performance. They save more time and can perform more tasks in mere minutes, hence they will have more time doing other tasks and research.

Centuries before the invention of the computer, researchers can only count on experimental data and results to comprehend the actions of the flow of fluids and derive many other correlations and relationship. An example of the relationship which is famous and widely used is the Reynolds Number (Re) which was discovered once hundreds of successful experiments and investigations have been done.
The experiments were conducted by Osborn Reynolds in the 1880s constructing the founding of the dimensionless Reynolds number, Re, as the important parameter for the resolve of the flow system in pipes, whether turbulent or not [1] the next amazing and successful experiment ever accomplished in last few decades was the airplane which was designed and developed in 1903. Oliver and Wilbur Wrights were the team successfully lead the world into a new aspect and those conquests were attained so many praiseworthy experience and experiments.

Clearly, the most challenging part of experimentation is to develop an effective data which need a large number of experiments. The outcome from experiment is very hopeful because it is the real thing that is really happening. By some means, it is difficult when showing an experiment since the preparation of the composition and devices is boring if it does not follow the instruction in a correct way.

As the world technologically advances, computers also improved. CFD or Computational Fluid Dynamic is one of the applications which can be presented by a computer. CFD simulates fluid flow, and hence is a great tool to help solve problems in fluid flow. Many simulations were done using the CFD and it has been a great help for engineers and scientists. CFD is an easier and more cost-effective alternative to conducting an experiment, which could be expensive and time-consuming.

Furthermore, the use of CFD is current and will always produce a good result if the formulation, especially the numerical simulation, was correctly selected and evaluated. More researches were carried out, and a large amount of numerical method was applied using a computer.

Generally, solutions for fluid dynamics can be introduced through experiments where many relationships are established, and can be classified into three major categories which have more relationships.
There are three different categories of solution for fluid dynamics problems: the first one is through experiments, where the problem will be investigated in an experimental manner and in a sample mode; the second category is a theoretical solution, which deals with the fact that most problems dealing with fluid dynamics have its own assumptions and mathematical equation that will result in analytical solutions. The final category of solution is the recently generated and recently used method known as CFD, which stands for Computational Fluid Dynamics. This particular classification is shown in Figure 1.1

![Figure 1.1](classification_of_fluid_dynamics_solution.png)
1.2 Computational Fluid Dynamic (CFD)

Computational Fluid Dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. The CFD has become an essential tool in solving problems governing the Navier-Stokes equation and the continuity equation, or any equation which are derived from these equations.

CFD works by showing on a computer how fluid behaves. One method is to divide the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for inviscid, and Navier-Stokes equations for viscous flow). In many instances, other equations are solved simultaneously with the Navier-Stokes equations.

Other equations that may be included are those describing species concentration (mass transfer), chemical reactions and heat transfer, among others. More advanced codes allow the simulation of more complex cases involving multi-phase flows (e.g. liquid/gas, solid/gas, liquid/solid), non-Newtonian fluids (such as blood), or chemically reacting flows (such as combustion).

The basic approach in the use of CFD includes preprocessing, simulation, and post-processing. In preprocessing, the geometry of the problem is defined and the volume occupied by fluid is divided into meshes. During this process, both physical modeling and boundary conditions are defined.

Simulation begins after the process and the equations are solved iteratively. Post-processing is where the postprocessor is used for the analysis and visualization of the result. Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics which solves fluid dynamics problems by using numerical methods. In this method, computers play an important role in computing and calculating the fluid flow problem.
There are many applications of CFD which are useful in the fields of research, education, automotive, design and sports, among others. This thesis focuses on using CFD to solve non-linear partial differential equation (PDE) where the analytical solution typically does not exist. Regardless, some flows with analytical solutions have applied with numerical method for validation purposes.

The base of CFD is the well-known and unsolvable non-linear incompressible full Navier-Stokes equation. There are two types of CFD: the numerical type and another type, which uses computer software to simulate. The second type uses a form of software to simulate or calculate the CFD problem.

Software like FLUENT©, which is very easy to use and can be used to simulate virtually any fluid flow problems, has some disadvantages, such as the user’s lack of knowledge about the equations applied, the assumptions or other criteria. This software is generally used for practical applications and for complicated geometry and complex conditions. In spite of that, FLUENT© software is established when it comes to numerical method but it is not publicized.

The earlier type of simulation is very notable because who create the codes could understand the simulation, the assumption, boundary conditions and other variables very well. This type of simulation is appropriate for information sharing because many papers are published frequently which touts the use of new methods, for example the Lattice Boltzmann method, Bifurcation method and CIP. The better method is determined by carrying out comparison and validation between the aforementioned methods. The simulation requires the user to be well-versed in programming software such as FORTRAN, C++, and MATLAB; example applications are the simulation of flow over cylinder [2] and the experimental [3].
1.2.1 Governing Equation in CFD

There are many variables and parameters in fluid flow which control the characteristic of the flow. Generally, these parameters are related to the physics of the flow, the nature of the fluid or the surrounding system. Some of those variables which are usually arising in fluid flow are listed:

- Temperature $T$
- Pressure $P$
- Velocity $u$
- Fluid density $\rho$
- Fluid viscosity, dynamic ($\mu$), and kinematic ($\nu$)

These are important variables in CFD simulation because they are useful and are generally incorporated in three major governing equations. These governing equations are very important for CFD and also for heat transfer simulation. These equations can also be modified depending on the physics of the fluid flow or based on the assumption which can be made. The equations involved in incompressible fluid flow are:

- The continuity equation (conservation of mass)

$$ \nabla \cdot \mathbf{u} = 0 \quad (1.1) $$

- The Navier-Stokes Equation (conservation of momentum)

$$ \frac{D}{ Dt} \mathbf{u} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} + \mathbf{f} \quad (1.2) $$

- The energy equation (conservation of energy)

$$ E_i + \nabla (Ep) = q - p \nabla u + f \cdot u \quad (1.3) $$
The first two equations play an important role in the formulation which is needed to produce the numerical simulation. These equations will be transferred into a new equation based on the physical model and it is also different from one another if the applied numerical method is different.

### 1.2.2 The Navier-Stokes Equations

The Navier-Stokes equations were named after the French engineer and scientist Claude Louis Henri Navier and the English mathematical physicist George Gabriel Stokes. The equations’ essential form was set forth by Navier in 1822; however, the origin of viscous stress was not properly represented. The latter was addressed by others, in particular by Poisson and Saint-Venant, but independently developed by Stokes in 1845.

Stokes constructed a number of solutions to the equations of viscous flow, which confirmed their ability to describe fluid dynamical phenomena.

The equation which describes the motion of fluid substances, i.e. substances which can flow, 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 which governs the fluid flow is the continuity equation and Navier-Stokes equation given by:

\[ \nabla \cdot \mathbf{u} = 0 \]  

\[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla P + \nu \nabla^2 \mathbf{u} \]

(1.4)  

(1.5)
With velocity $u$, pressure $P$, and kinematic shear viscosity. The Navier–Stokes equations are a set of nonlinear partial differential equations which, unlike algebraic $v$ equations, do not explicitly establish a relation among the variables of interest (e.g. velocity and pressure). Rather, they establish relations among the rates of change.

Navier-Stokes equation is well known in the field of fluid dynamics. The equation is nonlinear and usually the flows that use 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. In the equation lie an unsteady term, a diffusive term, a pressure term, a convective term and the external force which is a complete package for momentum conservation. However, there is no analytical solution to this equation as there are many Partial Difference term in the equation.

During the writing of this thesis, this equation is still not solved but many types of numerical methods were tried out by scientist and engineers and hence produce their own solution of numerical simulation. However, there still are exceptions, because some fluid flows having the analytical solution and this exception will be discussed later in the next chapter.
1.3 Problem Statement

Many classical numerical methods have been applied to investigate the behaviors of particles in a lid-driven fluid cavity by solving the Navier-Stokes equation accompanied with Newton’s second law and the CIP method. Yet, these numerical methods are still insufficient; for higher order of accuracy, more grids are needed to satisfy the methods.

- Many numerical solutions are being applied to solve Navier-stokes equation but they still lack accuracy
- Low Mesh Grid has a higher accuracy.
- How to properly describe the flow of a particle within a lid driven fluid cavity.
- Effect of collisions on particle flow and the trajectory of particle.
- The effects of gravitational forces and drag forces and collisions on particle flow.

1.4 Objectives of the research

The objective of this thesis is to investigate solid particles behaviors in a lid-driven cavity flow while considering the drag force and gravitational force. In addition, the objective is to observe the effects of collision which is divided to two parts and will be further defined in chapters two and three. This research is mainly based on study of the flow in a square two-dimensional cavity with particle moving and particle collision is limited to hard sphere models only. Meanwhile, the CIP method is applied to solve the Navier-Stokes equation to express the result with less grid structure, which will increase the order of accuracy. The Runge-Kutta method is used to calculate the drag force and gravitational force exerted on the particle.
1.5 Significance of study

Simulation allows scientists to virtually construct the experimental conditions so that they can investigate real conditions without actually experiencing that particular phenomenon. In some cases, it would be quite impossible to perform that experiment with the existing facilities and defined conditions. In the field of computational fluid dynamics, the most interesting areas in this field are description of fluid flow and the prediction and profile of the flow.

Moreover, viscous fluids while in rotary motion have diverse industrial and commercial applications. The main focus for researchers has been lid-driven cavity flows, where the fluid is set into motion by part of the containing boundary. These types of flows are tedious for analyzing fundamental aspects of recirculation fluids: in spite of the apparently simple geometry, lid-driven cavity flows may involve a high degree of complexity. This is an interesting problem, which may yield much information about the interaction between fluid and particle and particle-particle and particle-wall collisions in a wide range of practical configurations. This has not been widely studied before this.
1.6 Scope of the Study

The scope for this particular research is bound by two matters and will therefore be adhered to throughout the research, which are:

Solve the advection equation with the application of CIP for NSE by:

- Comparing the results with practical and simulated benchmarks with over other methods.
- Comparing the dynamics of solid particles with the results that have been revealed so far.
- Simulating multi-particle behavior in a lid-driven cavity while considering the effects of collision and gravitational force and drag force.

Implementation of results verification:

- Two-dimensional incompressible, unsteady, lid-driven cavity.
- Two-dimensional incompressible lid-driven flow in square cavity without particle affecting, focusing on the streamline plots and velocity plots.
- Two-dimensional incompressible lid-driven flow in square cavity representing the dynamics of solid particle, focusing on the orbit of the solid particle.
- Two-dimensional incompressible lid-driven cavity flow with multiple particles and collisions effect and gravitational force and drag force.
- Particle collision is limited to Hard-Sphere model only.
- Gravitational force and drag force are solved using Runge-Kutta method.
REFERENCES


