HEAT TRANSFER PHENOMENON OF HEATED CYLINDER AT VARIOUS LOCATIONS IN A SQUARE CAVITY

MOHD RUZAINI BIN ZAKARIA

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

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

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: MOHAMAD MAZWAN BIN MAHAT

Position: LECTURER

Date: 24 NOVEMBER 2008

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 concurrently submitted for award of other degree.

Signature:

Name: MOHD RUZAINI BIN ZAKARIA

ID Number: MA06051

Date: 20 NOVEMBER 2009

ACKNOWLEDGEMENTS

I am grateful and would like to express my sincere gratitude to my supervisor Mr Mohamad Mazwan Bin Mahat 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, and his belief that this is only a start of a life-long learning experience. I appreciate his consistent support from the first day I applied to graduate program 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. I also would like to express very special thanks supervisor for the suggestions and co-operation throughout the study. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

I acknowledge my sincere indebtedness and gratitude to my parents and for their understanding. My sincere thanks go to all my friend and lecturer, who helped me in many ways and made my stay at UMP pleasant and unforgettable. Many special thanks go to all of people that involve in this study for their excellent co-operation, inspirations and supports during this study. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals.

ABSTRACT

In this thesis, the theory of lattice Boltzmann method is been described in first chapter. The lattice Boltzmann equation method has been found to be useful in many application involving interfacial dynamics and complex boundaries. First, the introduction of this report is described the objective of the project. This project objective is to study the plume behavior of heated cylinder at various locations in square cavity. Next, the problem statement is explained in further detailed. The problems solve using the lattice Boltzmann method theory and some flow simulation. The background of the project is relating to the lattice Boltzmann method equation that is involving the Navier-Stoke equation, the governing equation and Bhatnagar-Gross-Krook (BGK) approximation. Then, the literature will explain and described further detail about the lattice Boltzmann method. The methodology is the simulation of the isothermal and thermal of the lattice Boltzmann. The isothermal include the Poiseuille and Coutte flow. The thermal include the Porous Coutte flow. The isothermal and thermal of lattice Boltzmann equation have been derived from the Boltzmann equation by discretization in both time and phase space. The result of heated cylinder at various locations in square cavity at different Rayleigh number that has been done compute that when the Rayleigh number is increase the flow will become distorted and the plume will emerge in the enclosure. This is because of the buoyancy induced and convection become more predominant than conduction. The isotherms move upward and larger plumes exist on the top of the inner square, which gives rise to the stronger thermal gradient on the top of the enclosure. Therefore, the flow strongly imposes on the above of the enclosure, which also cause the form of a thinner thermal boundary layer in this area and develops the heat transfer.

ABSTRAK

Tesis ini menerangkan teori kaedah kekisi Boltzmann dalam bahagian satu. Persamaan kekisi Boltzmann telah ditemui amat berguna kerana membabitkan dinamik antara muka dan sempadan kompleks. Pertama, pendahuluan menerangkan tentang objektif projek ini.. Objektif projek ini adalah untuk mengkaji sifat pemanasan silinder dalam ruang segi empat. Seterusnya, permasalahan projek ini diterangkan dengan lebih terperinci. Masalah The problem is diselesaikan dengan kaedah kekisi Boltzmann dan simulasi aliran. Latar belakang projek ini berkaitan dengan kaedah kekisi Boltzmann yang membabitkan persamaan Navier-Stoke dan penghampiran Bhatnagar-Gross-Krook (BGK). Kemudian, penulisan akan menerangkan lebih lanjut tentang kaedah kekisi Boltzmann. Dalam simulasi Isothermal dan pemanasan kekisi Boltzmann. Dalam Isothermal terdapat aliran Poiseuille dan Couette. Dalam pemanasan terdapat aliran Porous Couette. Persamaan Thermal dan Isothermal Boltzmann diterbitkan daripada persamaan kekisi Boltzmann melalui diskit masa dan fasa.Hasil pemanasan silinder dalam ruang segi empat pada nombor Rayleigh yang berlainan menunjukkan apabila nombor Rayleigh meningkat, aliran akan menjadi bengkok dan bentuk seperti pelepah akan terbentuk dalam ruang tersebut. Ini adalah kerana apungan berlaku dan konveksi akan menjadi lebih dominan daripanan konduksi. Isotherm akan bergerak ke atas dan pelepah yang lebih besar akan terbentuk di atas ruang segi empat, yang mana akan meningkatkan kecerunan suhu di atas ruang segi empat tersebut. Oleh itu, aliran yang kuat terjadi di atas ruang segu empat tersebut, seterusnya menjadikan lapisan sempadan suhu menjadi nipis di kawasan ini dan menyebabkan pemindahan haba berlaku.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

f	Density distribution function
c	Microscopic velocity
Ω	Collision integral
f^{eq}	Equilibrium distribution function
f_i	Initial density distribution function
τ	Relaxation time
u	Velocity
P	Pressure
υ	Kinematic shear viscosity
χ	Thermal diffusivity

LIST OF ABBREVIATIONS

BGK	Bhatnagar-Gross-Krook
CFD	Computational fluid dynamic
ELB	Entropic lattice Boltzmann
LBE	Lattice Boltzmann equation
LBGK	Lattice Bhatnagar-Gross-Krook
LBM	Lattice Boltzmann method
LGA	Lattice gas automata

CHAPTER 1

INTRODUCTION

1.1 OBJECTIVE

This project objective is to study the plume behavior of heated cylinder at various locations in square cavity. This project also attempts to deal with the analysis of an investigation of the natural convection of heat transfer in a square enclosure containing solid cylinder. The effects of the cylinder position on the heat transfer and the flow structures inside the cavity are to be studied and highlighted.

1.2 PROBLEM STATEMENT

This project is to study the heat transfer phenomenon of heated cylinder at various locations in square cavity. The project scope is to analysis heat transfer limit to natural convection only. The problem will be tested at $Ra = 10^5$and 10^6. This study will include the natural convection interactions in a heated cavity with an inner body. A specifically developed numerical model, based on the lattice Boltzmann method (LBM), is used for the solutions of the governing equations. Natural convection in heated enclosures, housing inner bodies has received significant attention because of its interest and importance in industrial applications. Some applications are solar collectors, fire research, electronic cooling, aeronautics, chemical apparatus, building constructions and nuclear engineering. This will contributes to the development of the LBM. Effects of the cylinder position on the heat transfer and the flow structures inside the cavity are to be studied and highlighted.

1.3 PROJECT BACKGROUND

The Lattice Boltzmann Method (LBM) is another approach to finite difference, finite element and finite volume method for solving Navier-Stoke Equation. Lattice Boltzmann approach has found current achievement in the host of fluid dynamical study including in porous media, magnetohydrodynamics, immiscible fluid and turbulence. The numerical and experimental study of natural convection of heat transfer in the partitioned enclosure has received significant interest in the recent year due to the useful engineering application. The application that related to this project is the solar collectors, thermal insulation, cooling of the electronic component and designing building. In nearly all of the earlier studies on natural convection in a square cavity containing partitions or solid bodies, with or without heat generation, the influence of radiation is ignored. There have been not many studies on the both heat-transfer problem involving convection and radiation. On the other hand, it is well recognized that when natural convection in air is involved, the heat transfers by convection and radiation are usually of the equal order of magnitude. In this project the objective is to study the plume behavior of heated cylinder in the square cavity at the various locations using the Lattice Boltzmann Method. The analysis of the heat transfer will be limited to natural convection only. A complete parametric study is made for different Rayleigh numbers. The problem will be tested at Ra = $10^5$and $10^{6.}$

The mathematical relationship governing fluid flow is the famous continuity equation. The Navier-Stoke Equation is given by:

$$\nabla \cdot \mathbf{u} = 0 \tag{1.1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla P + \upsilon \nabla^2 \mathbf{u} \tag{1.2}$$

u = velocity

P = pressure

υ = kinematic shear viscosity

Source: J Ryong Lee, Man Yeong Ha and S. Balachandar (2007)

As well known, the Navier-Stoke equation is nonlinear partial differential equations. It is too difficult and there is no analytical answer to them except for a small amount of particular cases. The information about physical process of fluid dynamics is often given by real dimension. The study analysis involving full scale tools can be used to guess how indistinguishable copies of the tools would act upon under the same state. On the other hand, in nearly all cases the investigations are costly and frequently unattainable.

At the present time, the fresh development in the computing power of microprocessor, numerical and solution of flow problems can be brought to the desktop. The employ of computer is necessary to come to a decision of the fluid motion of a few problems. The Computational Fluid Dynamic (CFD) has developed to turn out to be significant tool in solving the Navier-Stoke equation, continuity equation or the equations that are achieve from them. CFD is the science for determination the numerical answer to the governing equation during space or time to attain numerical details of the entire flow field of consideration. To accurately replicate fluid flow on the computer, it is essential to work out the Navier-Stoke equation with infinite exactness.

Lattice Boltzmann method is an additional technique to finite difference, finite element, and finite volume process for solving the Navier-Stoke equations. Lattice Boltzmann develops since the expansion of the lattice gas automata and takes over a few appearances from its pioneer, the lattice gas technique. The significant development to improve the computational competence has been made to Lattice Boltzmann method. The continuous Boltzmann equation is express as in Eq. (1.3)

$$f(x+c\Delta t, t+\Delta t) - f(x,t) = \Omega(f) \tag{1.3}$$

f = density distribution function

c = microscopic velocity

Ω(f) = collision integral

Source: Nor Azwadi C.S. (2007)

The development is the completion of Bhatnagar-Gross-Krook (BGK) estimate that is the single relaxation approximation. The primary pace in the lattice Boltzmann method is to follow the progression of single particle distribution. This will involve the probable quantity of molecules in an assured amount at assured moment complete since huge number of particles in a structure that travel liberally with no collisions for extended space judge against to their sizes. Following the distribution functions are achieved, the hydrodynamic equation can be attained. The most important purpose of LBM advance is to construct a connection or relation involving the microscopic and macroscopic dynamics, slightly than to deal with macroscopic dynamic straightforwardly. The goal is to attain macroscopic equation since microscopic dynamics by signify of statistic.

The collision integral equation is express as in Eq. (1.4)

$$\Omega(f) = \frac{1}{\tau}\left(f^{eq} - f\right) \tag{1.4}$$

τ = relaxation parameter

f^{eq} = equilibrium distribution function

Source: Nor Azwadi C.S. (2007)

The combination of the continuous Boltzmann equation and collision integral equation will give the Lattice Boltzmann BGK equation. The Lattice Boltzmann BGK is express as in Eq. (1.5)

$$f(x + c\Delta t, t + \Delta t) - f(x,t) = -\frac{f - f^{eq}}{\tau} \tag{1.5}$$

f_i = density distribution function

τ = relaxation parameter

f^{eq} = equilibrium distribution function

Source: Junya Onishi, Yu Chen and Hirotada Ohashi (2001)

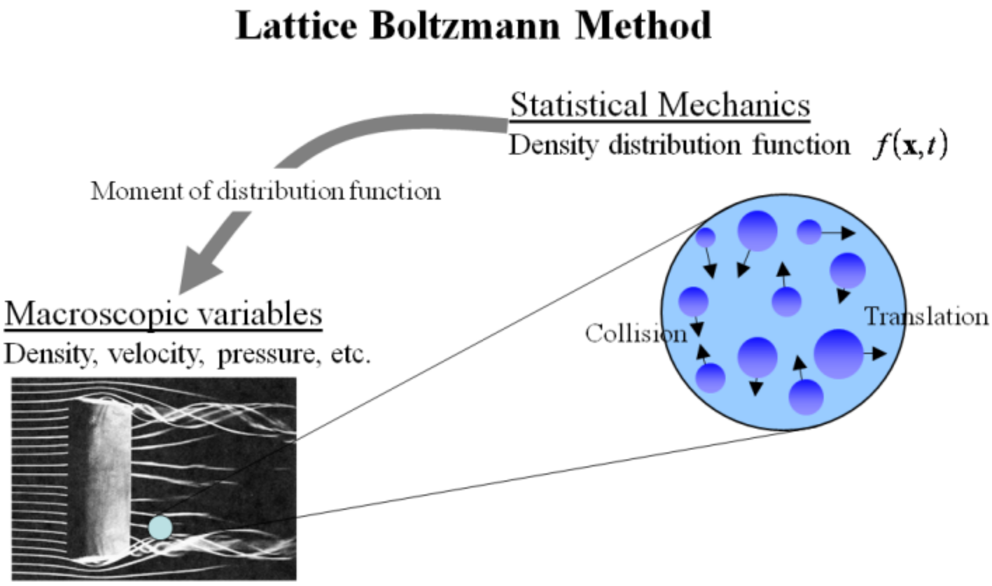

Figure 1.1: Lattice Boltzmann Method

Source: Nor Azwadi C.S. (2007)

The newest thermal lattice Boltzmann method goes to three sorts: the passive scalar approach, multispeed approach and thermal energy distribution model. The figure 1.1 shows the lattice Boltzmann method and the relationship between the macroscopic and microscopic variables. The multispeed technique employ the equivalent purpose in determine the macroscopic velocity, pressure and temperature. To preserve the kinetic energy in the collision on every one lattice point, this model necessary extra dissimilarity of velocity than isothermal form. The equilibrium distribution function commonly include elevated order velocity structure but this form on the other hand has severe numerical instability and not competent. The passive scalar model has enhanced numerical constancy than the multispeed form. The flow ground and temperature of the inactive scalar model distinguish by two distribution functions. Macroscopic function is advection by flow speed but does not manipulate the flow ground. The isothermal and thermal lattice Boltzmann equation (LBE) is resulting from the Boltzmann equation by discretization in together time and stage space. The origins straightforwardly link the LBE to the Boltzmann equation. Consequently, the LBE can be constructing on well-known origin of the Boltzmann equation and the effect of Boltzmann equation can be prolonged to the LBE. To verify the newest developed lattice arrangement, the numerical simulations of the porous plate coutte flow complexity and the natural convection in a square or cubic cavity have to be figure.

The macroscopic equation for isothermal equation is express as in Eq. (1.6)

$$\nabla \bullet u = 0$$

$$\frac{\partial u}{\partial t} + u\nabla \bullet u = -\nabla P + \left(\frac{2\tau - 1}{6}\right)\nabla^2 u \qquad (1.6)$$

$$\upsilon = \frac{2\tau - 1}{6} \qquad (1.7)$$

u = velocity

P = pressure

υ = kinematic shear viscosity

τ = relaxation time

Source: Nor Azwadi C.S. (2007)

The numerical answer of the porous coutte flow problem for a great variety of the Rayleigh numbers is representing that the form is suitable and numerically steady for the computational of elevated Rayleigh. The computations of natural convection in a cavity predictable the flow element for dissimilar Rayleigh number. The models utilize shorter imitation time and can be relate successfully in engineering function.

The macroscopic equation for thermal express as in Eq. (1.8) and (1.9)

$$\nabla \bullet u = 0$$

$$\frac{\partial u}{\partial t} + u\nabla \bullet u = -\nabla P + \left(\frac{2\tau_f - 1}{6}\right)\nabla^2 u \qquad (1.8)$$

$$\frac{\partial T}{\partial t} + \nabla \cdot (uT) = \left(\tau_g - \frac{1}{2}\right)\nabla^2 T \tag{1.9}$$

$$\tau_f = 3\upsilon + \frac{1}{2} \tag{1.10}$$

$$\tau_g = \chi + \frac{1}{2} \tag{1.11}$$

u = velocity

P = pressure

υ = kinematic shear viscosity

τ = relaxation time

Source: H.N. Dixit and V. Babu (2006)

1.4 PROJECT FLOW CHART

The figure 1.2 shows the project flow chart which is basically referred to the theory of lattice Boltzmann method. The theory of lattice Boltzmann contained the governing equation, basic principle of lattice Boltzmann, Collide Function of BGK, Equilibrium Distribution Function, Time Relaxation, Discretization of Microscopic Velocity and the Derivation of Navier Stoke Equation. After the theory of lattice Boltzmann has been studied, the isothermal fluid flow is simulated. The isothermal fluid flows have two basic flows which is the flow in pipe or the Poiseuille flow and the Couette flow. The extension of lattice Boltzmann model is the thermal lattice Boltzmann theory and the Porous Couette flow is simulated. The final part of the flow chart is to do the main project that is to study the Heated Cylinder Geometry and Boundary Condition Analysis

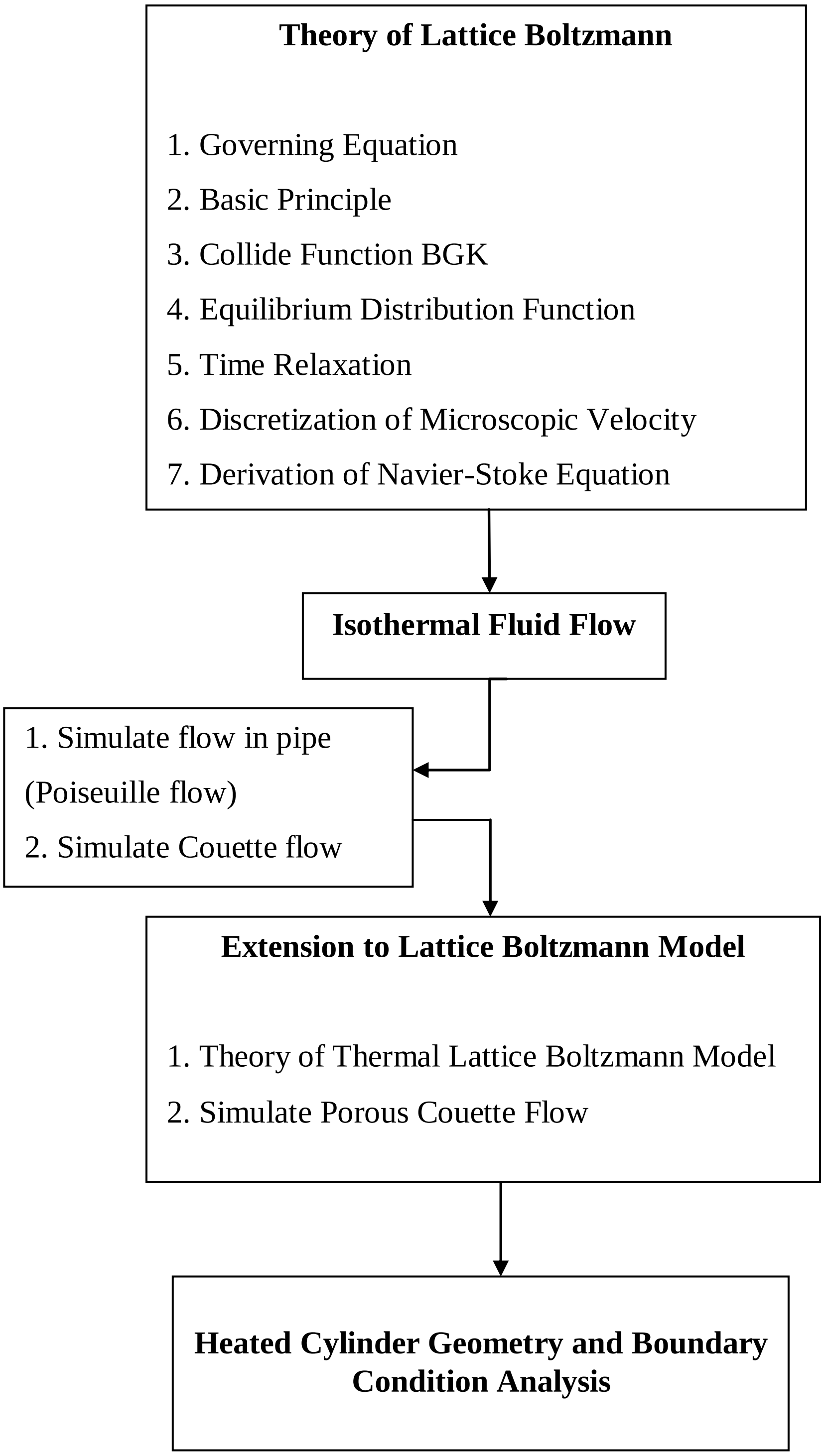

Figure 1.2: Project flow chart

CHAPTER 2

LITERATURE REVIEW

2.1 NAVIER STOKE EQUATION

In the last few years we have seen a quick growth of latest numerical methods for the result of partial differential equations, especially Navier-Stokes equations. The history of the Navier–Stokes equations is it named after Claude-Louis Navier and George Gabriel Stokes. Navier-Stokes equations explain the motion of fluid material that is material which can flow. These equations obtained from relate Newton's second law to fluid movement and collectively with the statement that the fluid pressure is the sum of a spread viscous expression, plus a pressure expression (Batchelor, G.K., 1967). They are one of the mainly practical sets of equations because they explain the physics of a large number of phenomena of academic and economic attention. The application is the weather, ocean currents, water stream in a pipe, flow about an airfoil and movement of stars within a galaxy. These equations in together complete and shorten outline are employed in the design of airplane and vehicle, the learning of blood stream, the devise of power post and the investigation of the effect of pollution.

In a purely mathematical sense, the Navier–Stokes equations are in the great attention. On the other hand, mathematicians have not yet confirm that in three dimensions answers always subsist or that if they do subsist they do not include any infinities, singularities or discontinuities (Batchelor, G.K., 1967). These are known the Navier–Stokes continuation and smoothness troubles. The Clay Mathematics Institute has known this one of the seven mainly significant open questions in mathematics. The Navier–Stokes equations are differential equations which do not explicitly create a relation between the variables of concern example like velocity and pressure. They establish relations among the rates of change. The Navier–Stokes equations for simple case of an ideal fluid can affirm that acceleration is proportional to the gradient of pressure. It also states not position but rather velocity (Frisch, U., Hasslacher, B. and Pomeau, Y. 1986). A result of the Navier–Stokes equations is called a velocity field or flow field, which is a description of the velocity of the fluid at a given point in space and time. Once the velocity field is resolved for, other amount of concern such as flow rate or drag force may be establish. This is dissimilar from what one normally sees in classical mechanics, where answers are typically trajectories of position of a particle or deflection of a continuum. Studying velocity as an alternative of position makes more sense for a fluid but for visualization reasons one can compute a variety of trajectories.

The Navier-Stoke equation is nonlinear partial differential equations. The information about physical process of fluid dynamics is frequently given by genuine measurement (Nor Azwadi C.S, 2007). The experimental study involving full scale equipment can be used to expect how identical copies of the equipment would perform under the same state. Yet, in nearly all cases the tests are expensive and always impossible.

$$\nabla \cdot \mathbf{u} = 0 \tag{2.1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla P + \upsilon \nabla^2 \mathbf{u} \tag{2.2}$$

u = velocity

P = pressure

υ = kinematic shear viscosity

Source: J Ryong Lee, Man Yeong Ha and S. Balachandar (2007)

2.2 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational fluid dynamics (CFD) is one of the undergrowth of fluid mechanics that uses numerical scheme and algorithms to work out and study problems that engage fluid flows. Computers are used to carry out the millions of calculations needed to simulate the relations of fluids and gases with the not easy surfaces used in engineering. Still with high-speed supercomputers barely inexact solutions can be attain in many cases. Continuing study, on the other hand, may give way software that give betters accuracy and speed of difficult simulation situations such as transonic or turbulent flows (Acheson, D. J., 1990). Software is frequently carried out by a wind tunnel with the final justification coming in flight analysis. The basic of CFD problem is the Navier-Stokes equations, which describe whichever single-phase fluid flow. Navier-Stokes equations can be simplified by eliminate terms explaining viscosity to give in the Euler equations (Batchelor, G.K., 1967). Advance simplification, by eliminate terms explaining vorticity give in the complete possible equations. Finally, these equations can be linearized to give in the linearized possible equations.

In these days, CFD has developed from a mathematical attention to become significant instrument in solving Navier-Stoke equation and the continuity equation. It is the science of determining numerical answer of the governing equation of fluid flow during proceeds the solution through space or time to achieve a numerical explanation of the whole flow field of attention (Acheson, D. J., 1990). For the fact, numerical researcher must select a method to discretise the difficulty. The settings up of the numerical simulation initiate with built a computational grid. The flow variables are calculated at the node point of this grid in some approach and at in-between points. The spacing between grid points has to be very well sufficient to achieve a high enough degree of precision. There are some benefits but to remain the number of grid point small since of additional grid point indicate more computer memory needed and a greater time is desired to carry out each iteration of the calculation (Nor Azwadi C.S, 2007). The uncomplicated computational grid rectangular lattice by unchanging spacing between node points in every dimension. There are series of way that use unstructured grids where the density of the node point is not constant and is higher in the area where the precision is wanted. Unstructured meshes often end up being connected in a triangular or tetrahedral style since these form fill space well and they needed least number of vertices. Several way employ adaptive meshes where the node point are generated and devastated as flow featured shift though the computational domain. This will remain number of nodes to a least but still providing the wanted dimension for the certain flow elements.