UNIVERSITI TEKNOLOGI PETRONAS

Computational Fluid Dynamics (CFD) Study of Flow Development in an Eccentric Annulus

by

Rahimah Bt. Mohamed Nawai

Dissertation submitted in partial fulfillment
of the requirement for the
B. Eng. (Hons.) Mechanical Engineering

SEPTEMBER 2011

Universiti Teknologi PETRONAS
Bandar Seri Iskandar
31750 Tronoh
Perak Darul Ridzuan

CERTIFICATION OF APPROVAL

Computational Fluid Dynamics (CFD) Study of Flow Development in an Eccentric Annulus

by

Rahimah Bt. Mohamed Nawai

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfillment of the requirement for the
BACHELOR OF ENGINEERING (HONS.)
MECHANICAL ENGINEERING

Approved by,

(Professor Dr. Vijay R. Raghavan)
UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
January 2012

CERTIFICATION OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.

RAHIMAH BT. MOHAMED NAWAI

This page is intentionally left blank

To my beloved father, mother and sponsor

To the person who inspires me much with his science thoughts,

Prof. Dr. Vijay R. Raghavan

In loving memory of my grandparents

Acknowledgement

First and foremost, I would like to express my greatest gratitude to Allah S.W.T. Without His blessing, I would not be able to complete my Final Year Project (FYP). I would like to thank my supervisor, Prof. Dr. Vijay R. Raghavan for his willingness to be my project supervisor. He has given valuable guidance, critics and advices along my FYP period. His willingness to educate people contributed tremendously to my project I would like to thank him for sharing his time, experiences, and brilliant ideas until the completion of Computational Fluids Dynamics (CFD) Study of Flow Development in an Eccentric Annulus. It was indeed a great pleasure and exposure for me in understanding this project and applied the knowledge into practice.

Besides that, I would like to thank the authority of Mechanical Engineering Department for providing us with a good environment and facilities to complete this project. Thousands of thanks to the final Year Research Project Coordinator, Mr Faizairi for providing the initial information required to begin the project.

Finally, an honorable mention goes to all individuals that have helped me in any way, but whose name is not mentioned here, Thank you so much for your continuous supports.

This page is intentionally left blank

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

ABSTRACT

Numerous publications have addressed the problems inherent to calculating wellbore hydraulics in eccentric annulus. CFD modeling provides an alternative approach of investigating fluid flow in such complex geometries. This branch of fluid flow analysis complements experimental and theoretical work, providing economically interesting alternatives through the simulation of real flows and allowing an alternative form for theoretical advances under conditions unavailable experimentally.

In this study, results from a series of numerical simulations for the fully developed laminar flow of Newtonian fluids in pipe, concentric and eccentric annular geometries, conducted using the computational fluid dynamics (CFD) code FLUENT, are used to investigate the effect of eccentricity, and friction pressure losses. Unlike the uniform velocity profile applicable for every sector in a concentric annulus, the axial velocity profile for an eccentric annulus is altered, with the peak velocities varying with location. A virtual inspection of the velocity profiles in an eccentric annulus shows that the zone of highest shear exists across the narrowing sector of an eccentric annulus.

The project starts with running simulations with pipe to have the basic understanding of how flow in circular geometry will be look like. The simulation results are expected to have flow development with a higher friction factor value at the entrance. As the flow developing, friction factor starts to decrease and become constant showing that it has already in fully developed flow. For concentric annulus, the fluid flow developing to the inner side of the annulus deviating from the walls as towards inside, the resistance is much lesser as compared to near the wall.

Next, we did simulations with same properties of fluid flow with eccentric annulus. We found that at very high eccentricities, data from the CFD model yields lower friction pressure. The reason being is due to the velocity is higher to the wider gap as compared to the narrow side. The phenomenon resulted due to the same factor of resistance to flow when the gap between the two geometries is lesser.

CHAPTER 1 : INTRODUCTION

1.1 Background of Study

Fluid flow study is very important in oil industry as engineers frequently deals with both Newtonian and Non-Newtonian fluid in annulus in drilling operations. The determination of the annular flow performance is significant in planning and designing of the hydraulics program of a well. Major pressure losses can develop in the constrained space. The knowledge of eccentric and concentric flow behavior help to understand and predict the flow development and pressure losses at normal operation.

Accurate estimation of the frictional pressure losses of drilling fluids in an annulus is a major challenge in developing a hydraulic program design. The most important parameters affecting the frictional pressure loss of drilling fluids are fluid properties, fluid velocity, fluid density, fluid viscosity, and fluid rheology, flow regime, pipe rotation, and pipe eccentricity. Since pipe eccentricity drastically decreases the frictional pressure loss, if it is not considered, frictional pressure loss can be overestimated.

Friction pressure calculations are useful in determining horsepower requirements, bottom hole treating pressure, and maximum wellhead pressure. Optimizing the friction pressure loss calculations through the well bore annulus allows making an appropriate evaluation of the wellbore hydraulics, which is essential to reduce the problems and avoid high costs of operation.

Eccentric of an annulus is defined as the degree of displacement of the geometric centre of a rotating/non-rotating part from the true centre while concentric is when both geometrics, inner and outer is centered to one centre point. Both have been use widely in many applications such as in drilling, heat transfer of fluid and others.

This project has a purpose to give contribution in oil and gas field, by the knowledge for the improvement and its subsequent application on the characterization of concentric and eccentric used in actual industry. Several designs study will be developed to study the flow behavior via Computational Fluid Dynamics (CFD). This simulation will provide us a better knowledge and understanding on flow performance resulted.

1.2 Problem Statements

Behavior and characteristic of a flow is very important to be analyzed in order to design equipment or devices. The basic understanding of how fluid flow in the equipment may raise a few constraints towards the development of the equipment. The constraint may be led to the modification or further improvement to the equipment itself.

In mechanical and petroleum engineering, the knowledge which relates to fluid flow studies is vastly used and applied in the industry. From the simple pipe or tubes to the drilling tubing, the principles of fluid dynamics are the one of the fundamental to be considered. Flow in concentric annulus is not exactly the same as flow in normal tubes. Approximation to the behavior of the flow especially to the laminar flow regime resulted error in calculation to determine the rate of diffusivity of momentum and heat. Hence, many studies done can't really give a general solution as the analysis focuses on specific geometry. One correlation used to analyze and relate the diffusivity rate with the different type of geometry is the hydraulic diameter concept and is successfully applied to relate to Re and Nu analysis.

However, the rising needs of eccentric annulus in the industry led to this project. Eccentric annulus differs to concentric annulus as it has offset to the side between core and the annulus itself. Thus, the similar behavior of concentric annulus cannot be expected from an eccentric annulus. Moises and Shah (2000) reported that annular pressure losses in a fully eccentric annulus could be as low as 40% of the value in concentric annulus. Hence, by taking into account the eccentricity of the tubing, accuracy of mathematical models can be improved (Haciislamoglu, 1989).

This project will discover the flow development in the concentric annulus and its friction as a function of Re and diameter ratio and comparing them with the characteristic owned by eccentric annulus. The friction of eccentric annulus will be in the function of Re, eccentricity and diameter ratio. The frictional behavior are developed using state-of-the art technique in fluid flow analysis; Computational Fluid Dynamics (CFD) modelling and simulation. From this comparison, we will further correlate the result to obtain a hydraulic diameter like correlation.

1.3 Objectives & Scope of Study

The objectives of this project are as follow:

a) Validate and study friction function for a circular passage of laminar flow using computational fluid dynamics software
b) Validate and study friction function for an eccentric annular passage of laminar flow using computational fluid dynamics software

In order to achieve the objectives, the scopes have been narrowed down. The project will focus on understanding the behavior of flow in pipes and how to design such construction in CFD is learnt. The next part, the understanding of flow in concentric annulus and this flow will be compared with without rotation results obtained by Vieira Neto et al (2011).

This will be expanded more to in order to capture the first objectives in this project. Next, the development of understanding in eccentric annulus flow behavior will be then drawn to achieve the second objective pursued. Since eccentricity will give the major impact on the calculations and results, a new correlation based on hydraulic diameter concept is hoped to be developed.

CFD software numerically solves the equations of continuity, momentum, and mass based on finite element methods. In this study, CFD and the proposed model have been used to find out the frictional pressure losses and tangential velocity of Newtonian fluid in both concentric and eccentric annuli in laminar regimes.

1.4 KEYWORDS:

Keyword for this project was summarized as Computational Fluid Dynamics (CFD); FLUENT; Concentric and Eccentric Annular Flow, Laminar.

CHAPTER 2 : LITERATURE REVIEW

2.1 Darcy – Weisbach Equation

The head loss due to friction of a pipe is determined by using the Darcy-Weisbach equation

$$h = \frac{fLv^2}{D2g}$$

Where: h = head loss

f = friction factor

L = length of pipe

v = velocity of fluid trough pipe

D = Diameter of pipe

g = acceleration due to gravity

The Moody diagram gives the friction factor of a pipe. The factor can be determined by its Reynolds number and the Relative roughness of the Pipe.

The rougher the pipe the more turbulent the flow is through that pipe. The relative roughness of a pipe is given by:

$$\frac{e}{D}$$

Where: e = absolute roughness

D = diameter of pipe

2.2 Reynolds Number

The Reynolds number equation was determined by passing dye through a fluid. At low velocities the dye passed in layers and at high velocities the dye diffused into the fluid. This shows that at high velocities the flow is more turbulent than at low velocities. The Reynolds equation is:

$$R = \frac{Dv}{\zeta}$$

Where: R = Reynolds number

D = diameter

v = velocity

ζ = kinematic viscosity of fluid

2.3 Moody Chart

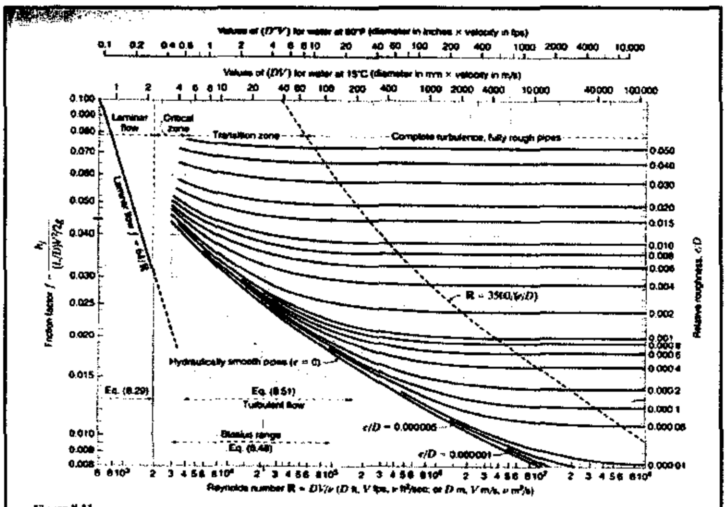


Figure X.11
Moody chart for pipe friction factor (Stanton diagram).

Figure 1 : Moody Chart

By looking at the Moody diagram it shows that the right top corner is completely turbulent and the left top is laminar (smooth flow).

To determine the frictional factor, find the relative roughness value for the pipe on the right. Then locate the pipes Reynolds number on the bottom. Follow the relative roughness curve to where it crosses the determined Reynolds number. Now at that point project a straight line to the left, the number determined on the left is the frictional factor.

2.4 Related Works

2.4.1 Studies of Fluid Flow Behavior in Circular Geometry

A number of studies have been carried out in order to estimate pressure loss in concentric and eccentric annuli for light drilling fluids. Nevertheless, there are a few experimental data on pressure drop and velocity profiles for light drilling fluids flowing through concentric and eccentric annuli (1-3). Hansen (1999) and Mc-Cann (1993) are among the very few investigators to provide experimental data for Newtonian flow in concentric and eccentric annuli.

Singhal (2005) developed new correlations to make accurate predictions of friction pressure losses in a turbulent and laminar regime for Newtonian and non-Newtonian fluids in concentric annuli. Azouz (1993) reported numerically simulated fully developed turbulent flow in concentric and eccentric annuli. They concluded that mixing-length and *k*-models perform equally well for concentric annuli, and the *k*-model carries out slightly better than the mixing-length model for the eccentric annulus.

2.4.2 Studies of Fluid Flow Behavior in Concentric & Eccentric Annulus

One criterion often used in determining an equivalent flow area in annuli is the equivalent diameter that is the difference between inner and outer pipe diameter. Jones (1976) demonstrated that the equivalent diameter was insufficient to accurately correlate geometric effects for rectangular ducts in turbulent flow. Instead a laminar equivalent diameter, dL, which is the equivalent diameter multiplied for a shape factor given by the radius ratio of inner and outer pipe, was determined from theory and used in a modified Reynolds number yielding good results for a broad range of geometries.

Later, Jones and Leung (1981) presented data for smooth concentric annuli and demonstrated that the theoretically determined laminar equivalent diameter that provides similarity in laminar flow for round tubes and concentric annuli also provides similarity in turbulent flow. They proposed that generalized Reynolds number as a function of the laminar equivalent diameter is the only parameter needed to determine the friction factor and recommended the well known standard of comparison. Experimental studies on the turbulent flow of water through an eccentric annulus were first reported by Dogde (1963). His research was focused on the measurement of corresponding flow rates and pressure gradients for various diameter ratios, with eccentricities ranging from 0 to 1. He reported friction factor data for the flow of water in eccentric annuli for diameter ratios of 0.875, 0.750 and 0.688 at Reynolds numbers ranging from 20, 000 to 100,000. Dogde (1963) claimed that diameter ratio of the eccentric annulus will not affect pressure loses in the geometry. This may be due to small difference in diameter ratio used by the experiment.

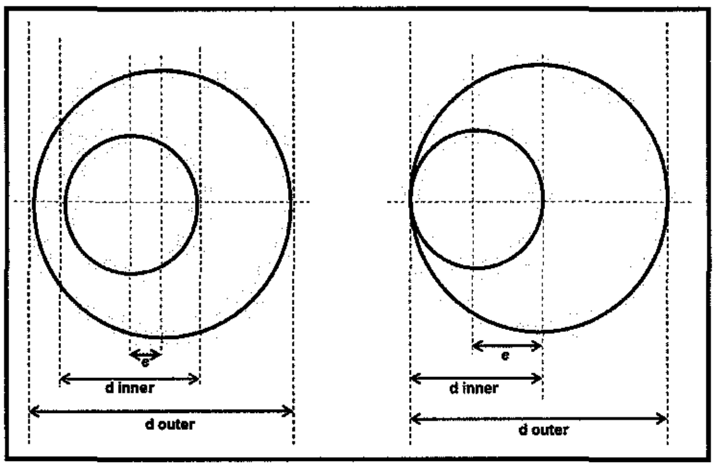


Figure 2 : Eccentric annulus geometries (Left : 0.5 eccentricity, Right : 1.0 eccentricity)

CHAPTER 3 : METHODOLOGY

3.1 Introduction

One method is to discretize 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. In all of these approaches the same basic procedure is followed:

- **Pre-Processing**
  - GAMBIT is used to model the geometry and generate the mesh.
  - The physical bounds of the problem are defined
  - Boundary conditions are defined. This involves specifying the fluid behaviour and properties at the boundaries of the problem.

- **Solver Execution**
  - FLUENT is responsible to solve and do the simulations. The solutions are computed and iterated as laminar, steady state flow, 2DDP and 3 DDP based on the geometry created in GAMBIT.
  - Operation and boundary conditions are defined in the solutions to have more accurate results.
  - Contour displays for the solutions are displayed to be analysed.

- **Post Processing**
  - Finally, post processing method is used for the analysis and visualization of the resulting solution. The different value of eccentricity will be analysed in this process to achieve the objective of this project.

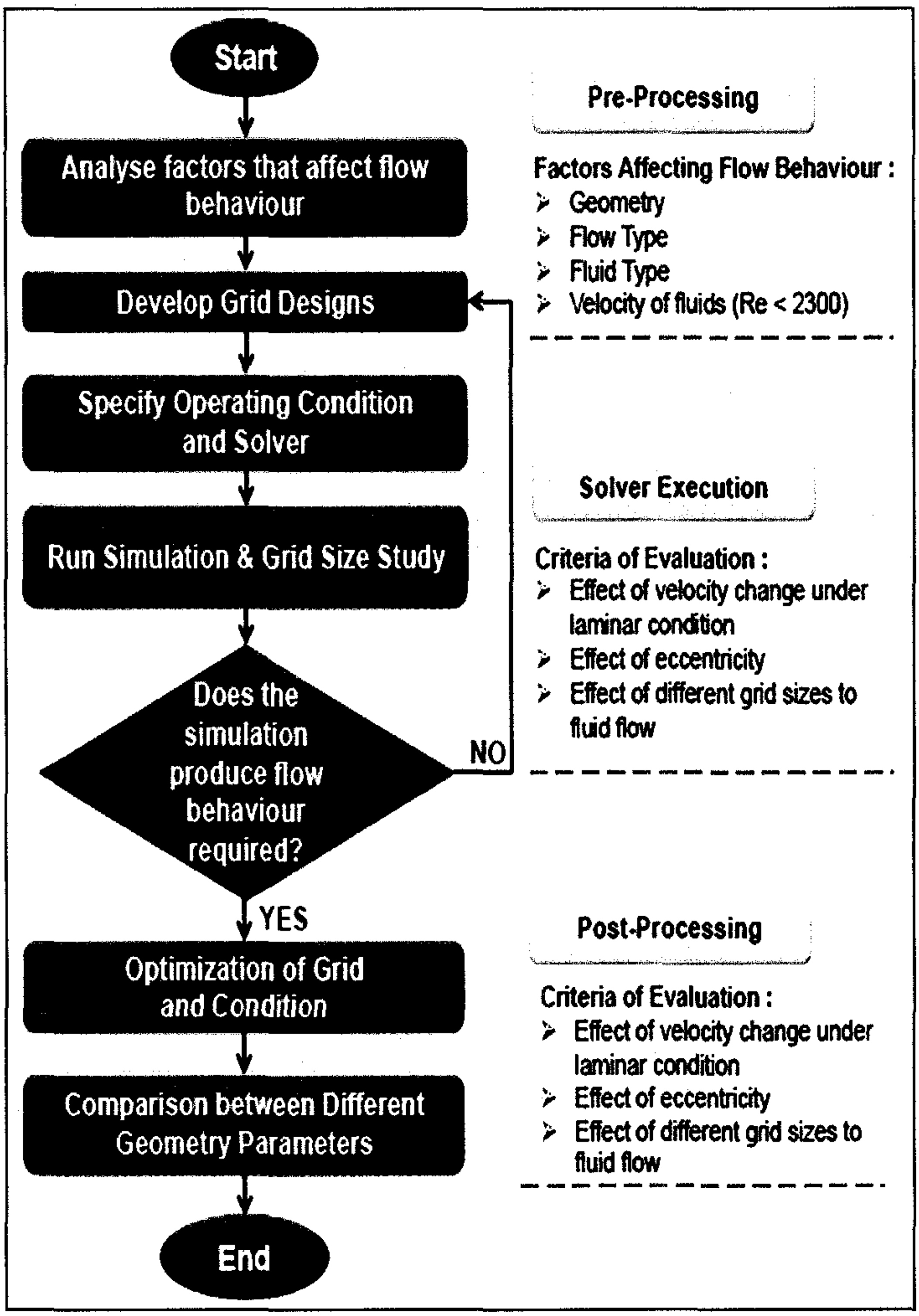


Figure 3 : The flow chart of the project

3.2 Project Work

The methodology of this project as follows (FLUENT Software Training, 2001):

- **Problem identification and Pre-Processing**
  a) Modeling goals:
     - To design different types of pipe, concentric and eccentric annulus
     - To study the fluid flow behaviour in each designs
  b) Domain of the model: 2D axisymmetric pipe flow, 3D concentric annulus, eccentric annulus flow with eccentricity of 0.5, 0.75 and 1.00.
  c) Grid design: created using GAMBIT
     - The edges meshes were created based on First Last Ratio.
     - The faces meshes were using triangle mesh to have finer mesh developed.
     - The volume meshes were using tetrahedral mesh.
  d) Assumption : Newtonian fluid, laminar flow ($Re < 2300$), no heat generation, no slip condition and steady state, 3D flow for annulus and 2D flow the pipe.

- **Solver Execution**
  a) Setting up the numerical method: Single phase model will be used (default)
  b) Conduct mesh size study and convergence check
  c) Computing and monitoring the solution

- **Post processing**
  a) Examining the results and comparison
  b) Considering on optimizing the model

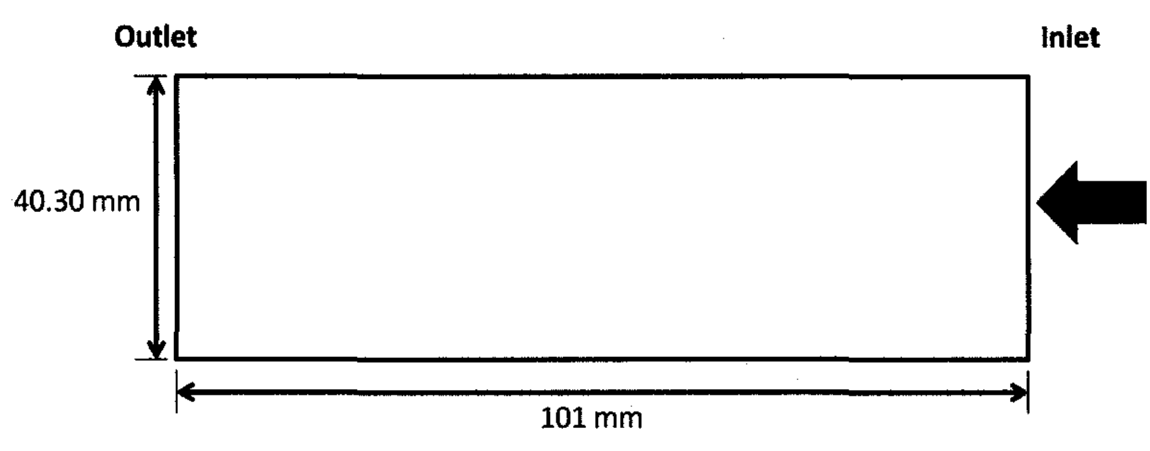
3.2.1 Preliminary Modeling Identification and Pre-processing

3.2.1.1 Pre-processing

The modeling goal is to study the fluid flow behaviour in different types of pipes, concentric and eccentric annulus. For the early stage of this project, the understanding of flow behavior in pipe is essential in order to further understand the behaviour of non-circular duct (i.e. annulus). The analysis uses the same velocity and geometry parameter (i.e. radius) in order to see the development of flow in the respective model.

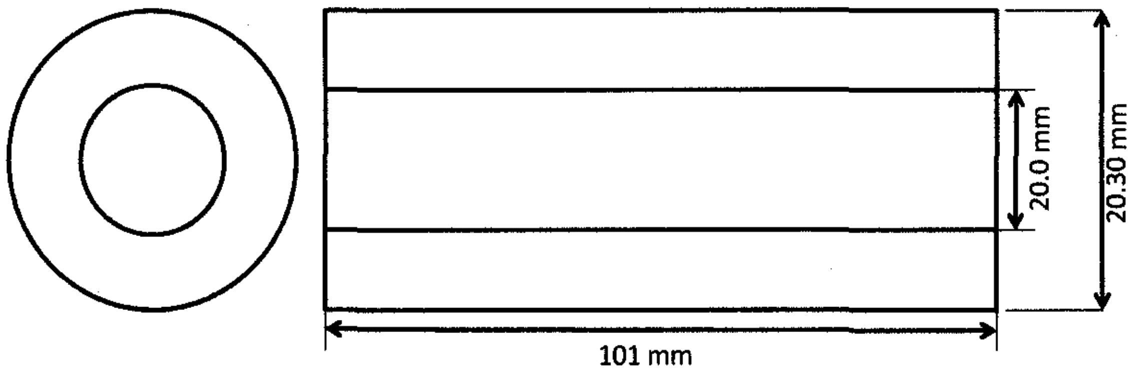
The fluid used in this project is Water (liquid) as water is a Newtonian fluid and most industrial fluids contain water as a part of the mixtures. The basic requirements stated in the early chapters are set to meet objectives. A constant length of pipes is set for us to be able to see the variation of the results obtained. Dimensions for all geometries are based on the geometry used in J.M. Nouri, H.Umur, and J.H. Whitelaw (1992) and the computational length is based on J. L. Vieira Neto, A. L. Martins, A. Silveira Neto, C. H. Ata'ıdel and M. A. S. Barrozo (2010).

Geometry Dimensions - Pipe

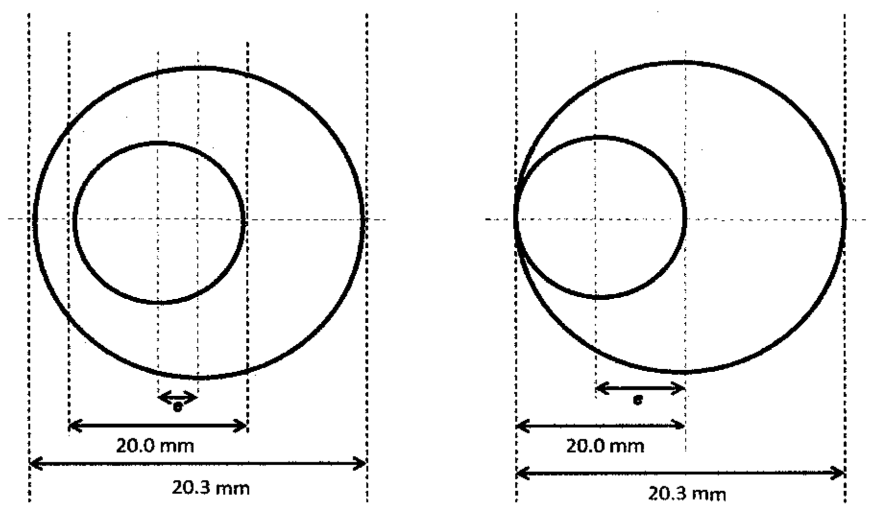


For the non-circular duct analysis, the variation of data is based on velocity profile and the effect on pressure losses of the outlet. For both concentric and eccentric annulus, the aim is to see the effect of centricity of the annulus. Since we are dealing with laminar flow, it is impossible to see any development of flow on rectangular cross sectional area of the annulus. Hence, the analysis will cover the inlet and the outlet of the annulus only.

Geometry Dimensions – Concentric Annulus



Geometry Dimensions – Eccentric Annulus



All of the designs were constructed using GAMBIT software since it has the advantages of flexibility in changing grid size and it is user friendly as well.(FLUENT, 2010). All of design was made in three dimensional (3D), to reduce the computational cost and it is an easy method of constructing and predicting the initial mixing behavior under any condition. (Dr. R Weinekötter, 2011), (Hu, 2010). The simulations also is aim to see any reverse or backflow of the fluid in the geometry. As such event encountered, it will be recorded as observation and finding which will be discussed in later chapter.

3.2.2 Grid Designs

Generally, the design and construction of quality grid is very crucial. Good quality grid ensures the success in doing CFD analysis and help to get more accurate results. An appropriate choice of grid type depends on the geometric complexity, flow filed and the cell and element types supported by solver. (GAMBIT Software Tutorial Guide, 2001)

The quality of the grid is extremely important and can strongly influence the solution. It is used to determine whether or not a valid or a converged solution can be achieved at all. The most important qualification about a computational meshing is that it must define enough points to capture everything of interest that is happening in the computational domain without becoming so extensive and that unreasonable computation times are required.

For this project, we used two basic types of geometry. For pipes, we are using rectangular shape as it is a 2D analysis whereas for the annulus, a cylindrical shape is used to develop the third dimension of the geometry which is the volume.

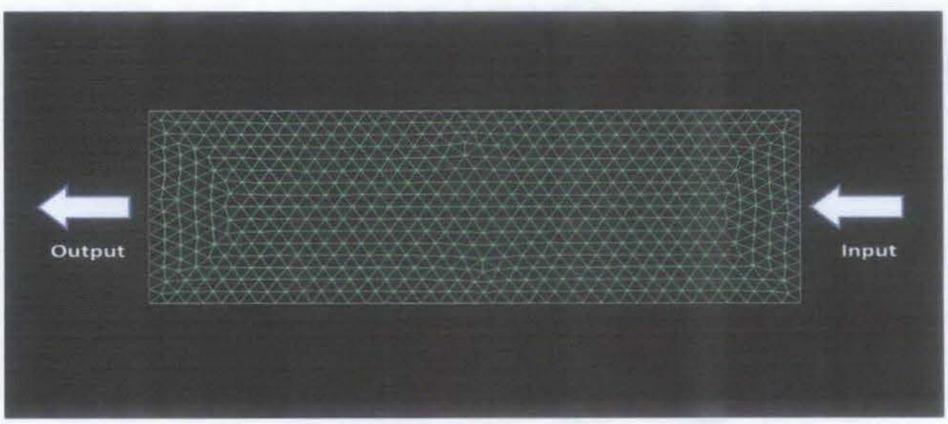


Figure 4 : 2D grid done for pipe using GAMBIT

GAMBIT provides several types of elements to be used to mesh the geometry. It is vital to use a good element and interval of nodes to create a good mesh with reliable accuracy. Although for the rectangular shape, we can use Quadratic element which resulted square meshes to the faces, but in order to sustain the consistency, we are using the same parameters for all geometry; First

Last Ratio type to mesh the edges, triangular shaped mesh for the faces and for the volume, the element is Tetrahedral/Hybrid type. To add the accuracy, we refined the face meshes using L-W Laplacian smoothen

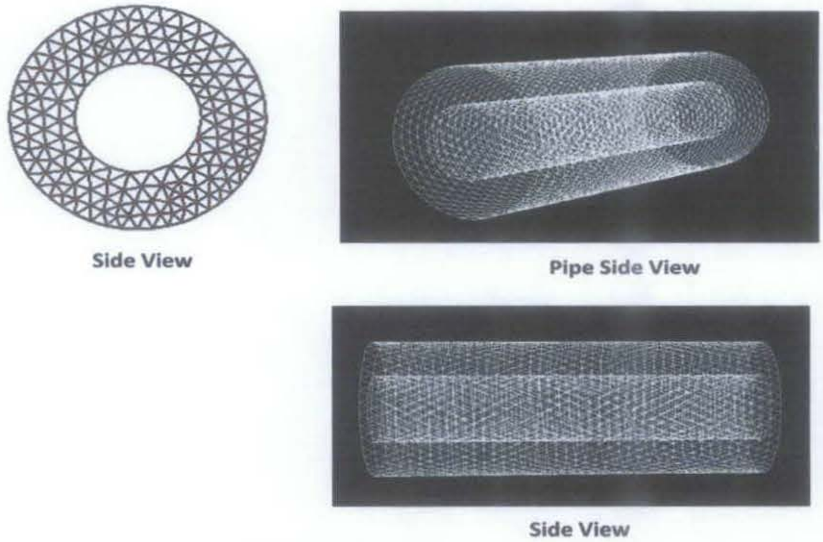


Figure 5 : Concentric computational grid: (a) Face with plans 1–4 and (b) periodic section

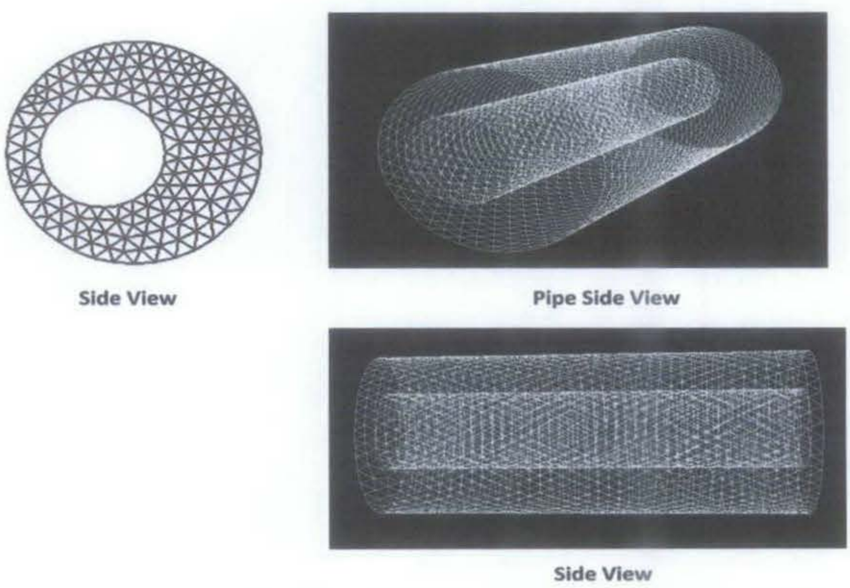


Figure 6 : Eccentric computational grid: (a) Face with plans 1–4 and (b) periodic section

3.2.3 Solver Executions - CFD Simulation Parameters

The simulation is conducted using default model with single phase flow (liquid). We assume that the flow inside the annulus and pipe is laminar, no energy generation, no slip condition and operates at steady state condition. A smooth wall with no-slip condition is imposed on both pipes wall. Water is used as the operating fluids and other operating conditions are similar based on the literature review in chapter two.

3.3 Governing Equations

In this work, the liquid is assumed to be incompressible and miscible. When the impact of gravity was neglected and no other source item existed, the steady flow of an incompressible Newtonian liquid generally described by Navier–Stokes equation and continuity equation as follows. The distribution of species concentration is governed by the diffusion convective equations and none of slip boundary conditions are adopted. The continuity equation for the mixture is:

$$\frac{\partial}{\partial t}(\rho_m) + \nabla \cdot (\rho_m \vec{v}_m) = \dot{m}$$

Where V_m is the mass-averaged velocity:

$$\vec{v}_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_k}{\rho_m}$$

ρ_k is the volume fraction of phase k, *m* represents mass transfer due to cavitations. The momentum equation for the mixture can be obtained by summing the individual momentum equations for all phases. It can be expressed as:

$$\frac{\partial}{\partial t}(\rho_m \vec{v}_m) + \nabla \cdot (\rho_m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla \cdot \left[\mu_m \left(\nabla \vec{v}_m + \nabla \vec{v}_m^{\mathrm{T}}\right)\right] +$$
$$\rho_m \vec{g} + \vec{F} + \nabla \cdot \left(\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_{\mathrm{dr},k} \vec{v}_{\mathrm{dr},k}\right)$$

The equation of mass conservation for incompressible fluids

$$\nabla_i u_i = 0$$

and a convection-diffusion equation for the concentration field

$$\frac{\partial c}{\partial t} + (u_i \cdot \nabla_i)c = D\nabla^2 c$$

Where n is the number of phases, ~F is a body force, and μ is the viscosity of the mixture:

$$\mu_m = \sum_{k=1}^{n} \alpha_k \mu_k$$

$\sim V_{dr;k}$ is the drift velocity for secondary phase k:

$$\vec{v}_{dr,k} = \vec{v}_k - \vec{v}_m$$

The underlying mathematical framework allowing the computation of the time evolution of concentration profiles is the diffusion equation, given as follows:

$$\frac{\partial c}{\partial t} = D\frac{\partial^2 c}{\partial x^2}$$

Where c denotes as the concentration of fluids.

CHAPTER 4 : RESULTS & DISCUSSION

We have set some parameters in order to study the behavior of the flow. Since we are dealing with laminar and Newtonian fluid, we are using **water (liquid)** as the fluid.

Table 1 : Properties of fluid used

No.	Property	Units	Value
1	Density (ρ)	kg/m^3	998.2
2	Viscosity (μ)	kg/m-s	0.001003
3	Specific Heat (Cp)	j/kg-k	4182
4	Thermal Conductivity	w/m-k	0.6

Table 2 : Velocity & Reynolds Number used

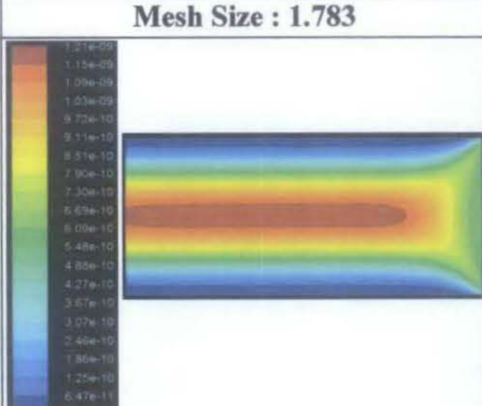
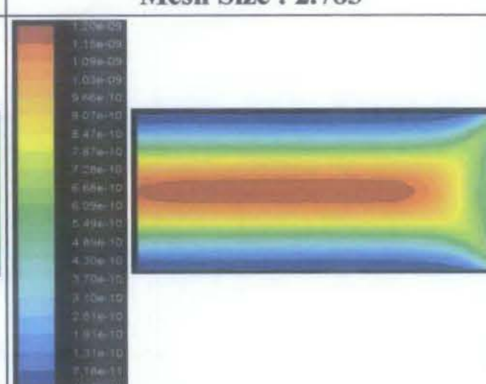
No.	Re	Velocity (ms^{-1})
1	600	2.9699E-05
2	1100	5.4448E-05
3	1600	7.9197E-05
4	2100	1.0395E-04

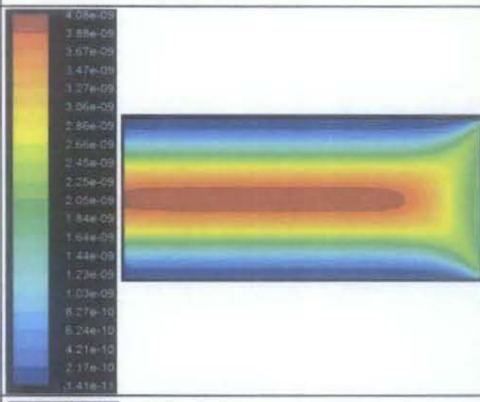
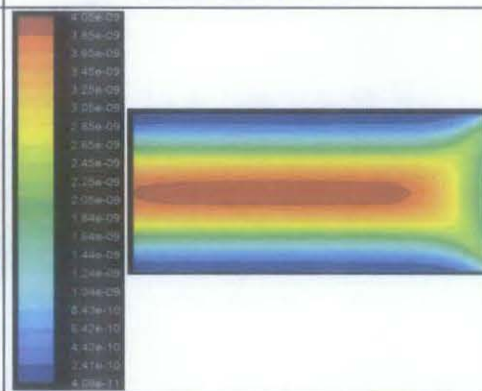
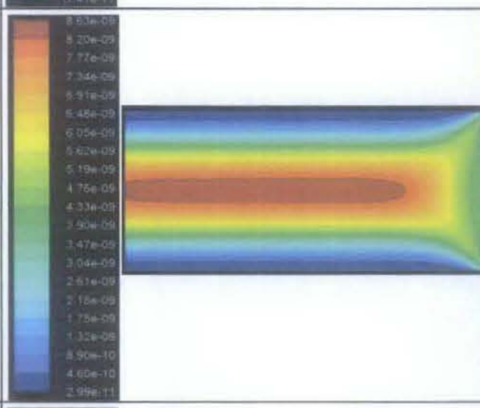
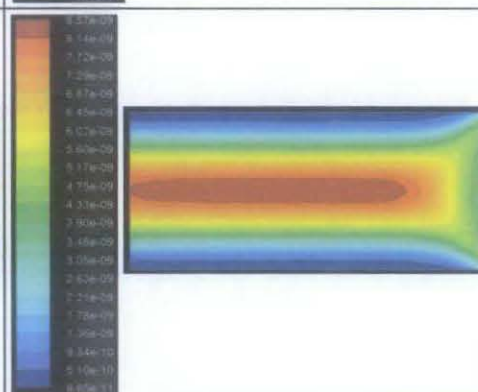
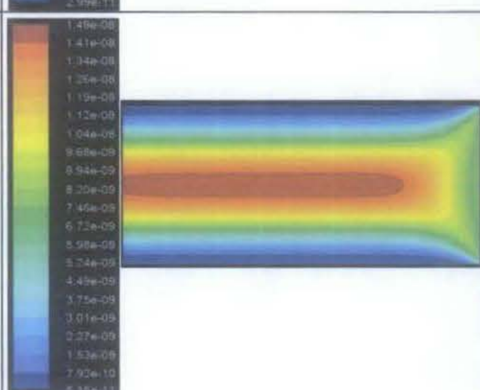
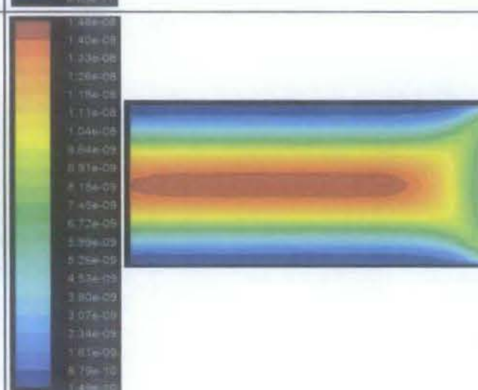
Dimensions for all geometries are based on the geometry used in J.M. Nouri, H.Umur, and J.H. Whitelaw (1993) and the computational length is based on J. L. Vieira Neto, A. L. Martins, A. Silveira Neto, C. H. Ata´ıde1 and M. A. S. Barrozo (2010).

4.1 Pipe Flow Development

4.1.1 Simulation Results for Pipe

Table 3 : Simulation results for pipe

Re	VELOCITY (m/s)	RESULTS	
		Mesh Size : 1.783	Mesh Size : 2.783
600	2.9699E-05		

Re	VELOCITY (m/s)	RESULTS	
		Mesh Size : 1.783	Mesh Size : 2.783
1100	5.4448E-05		
1600	7.9197E-05		
2100	1.0395E-04		

Based on the FLUENT Manual (2010), it has highlighted the inter-relation between velocity and dynamic pressure. Dynamic pressure is known as *velocity pressure* for incompressible fluids and can be defined as below:

$$Q = 0.5pV^2$$

Where q= dynamic pressure (Pascal)

P= Density (kg/m3) and V= Velocity (m/s)

The relationship between velocity and dynamic pressure is parallel based on the equation above. Theoretically, at a higher velocity, we can predict that the dynamic pressure will be high at respective design zone with constant density throughout the simulation process. The flow starts to develop the profile based on the contour color. The color development in the pipe represents the smallest to highest value of flow in the pipe which represents by blue to red respectively.

Blue color of contour near the wall is due to the internal resistance of flow in that area or the no-slip condition. Experimental evidence indicates that when any fluid flows over a solid surface the velocity is not uniform at any cross section; it is zero (no slip) at the solid surface and progressively approaches the free stream velocity in the fluid layers far away from the solid surface. This is evidently confirms by the pressure loss experienced by the flow for each Re values Since pattern of pressure loss is the same, examples of the pressure loss of the flow in pipes are represented in Figure 9 & 10.

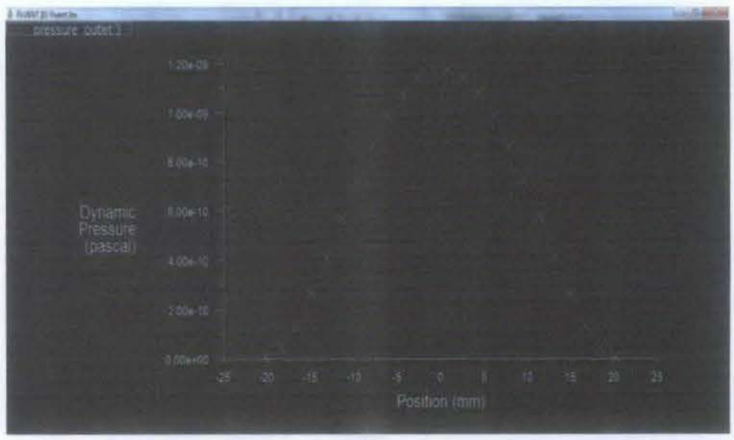


Figure 7 : Pressure loss in the pipe of flow with Re = 600

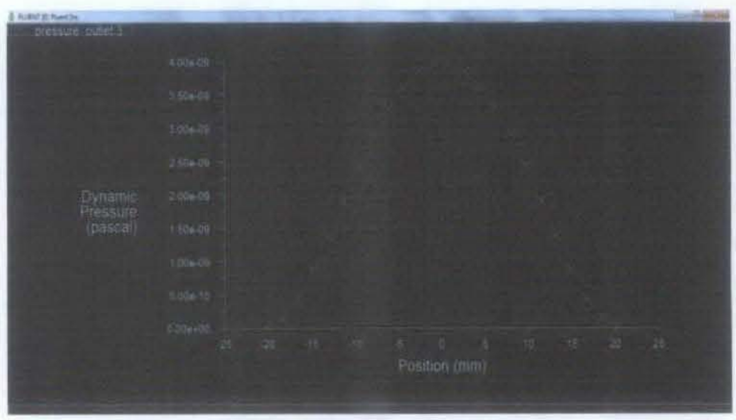
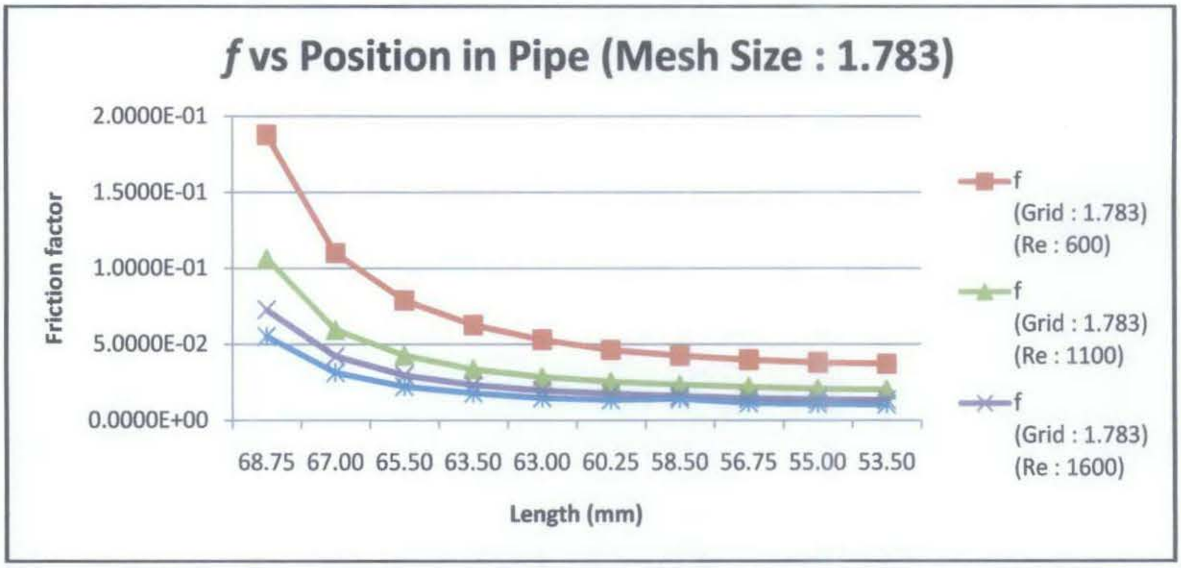


Figure 8 : Pressure loss in the pipe of flow with Re = 1100

The fluid enters the free stream flow which represented by the red color in the contour. At the first red contour starts to develop is the first point of the fluid be in fully developed flow. In the free stream flow area, the flow reaches the Re value for laminar flow as desired since there is no internal resistance to the flow in this area. It is proven by the validation made based on the friction factor versus length constitutes in the next part of this discussion.

4.1.2 Validation of Results : Friction Factor vs Position (L) in Pipe



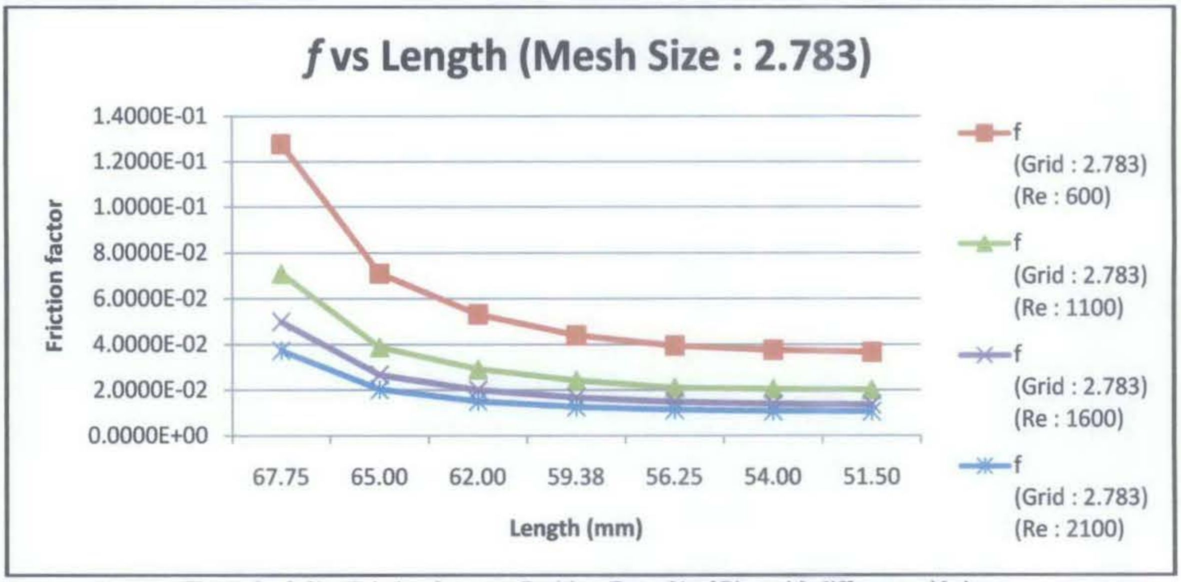


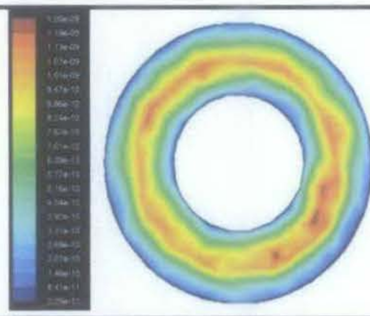
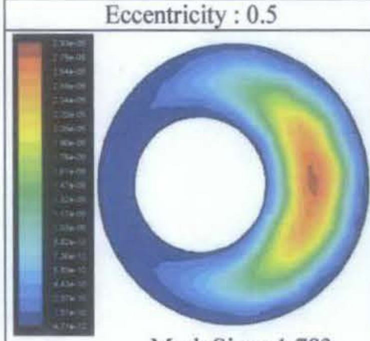
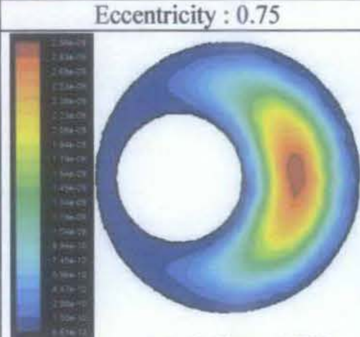
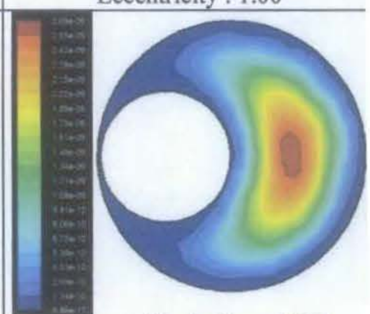
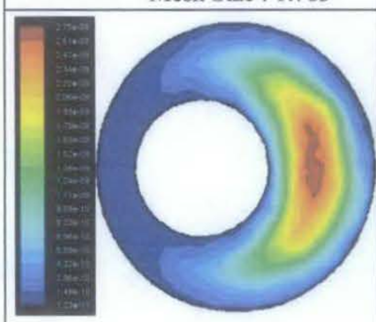
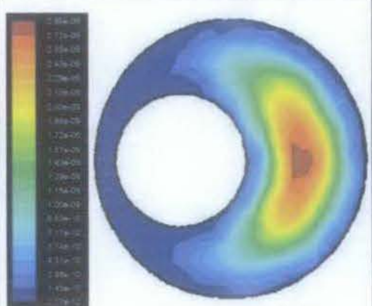
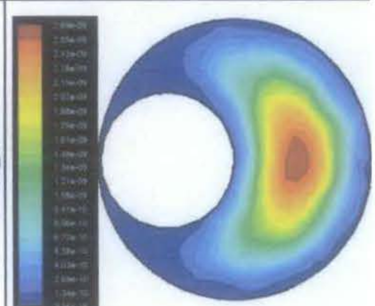
Figure 9a & 9b : Friction factor vs Position (Length) of Pipe with different grid sizes

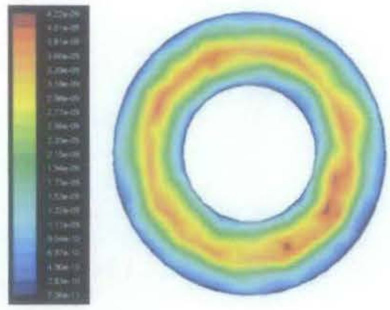
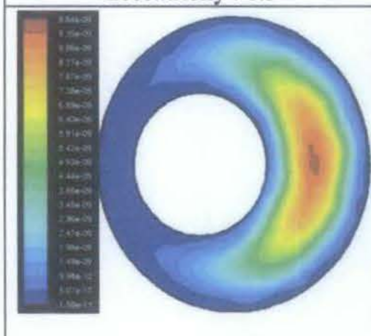
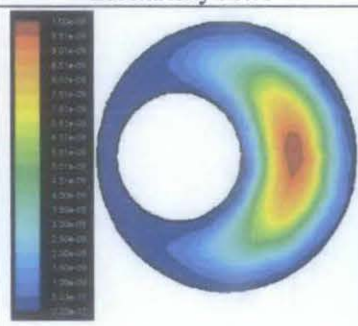
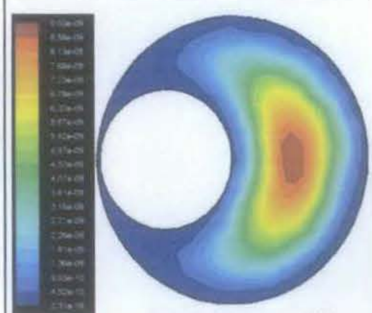
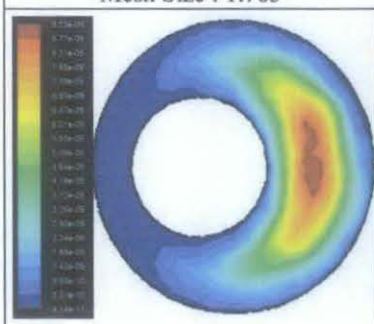
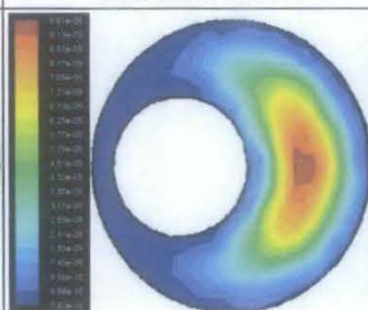
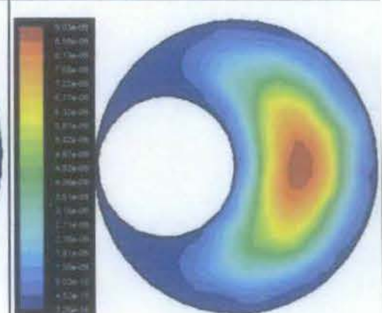
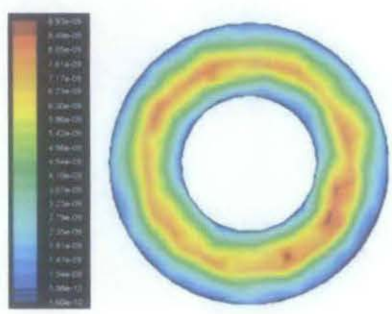
Based on graph plotted to compare the behavior of flow simulated in CFD and the analytical results for pipe flow, we found that grid sizes do not affect the flow behavior in terms of flow development of the flow itself but it refines the accuracy. Graph in Figure 9a & 9b showed that at smaller Re number, the friction factor increases. This is due to higher friction experienced by the fluid to flow with smaller Re. The decrement curve in the graph indicates the values when the fluid enters the pipe and starts to be in constant value as the flow is becoming a fully developed flow. Calculations for friction factor are based on the Darcy Weisbach friction factor and we also have compared our calculated friction factor values to Moody Charts values. The values are confirmed to each other.

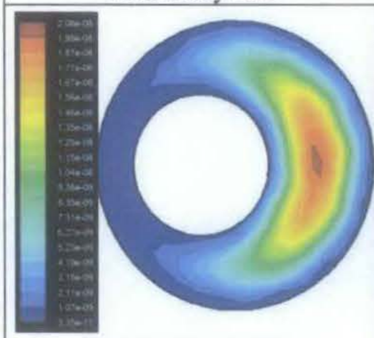
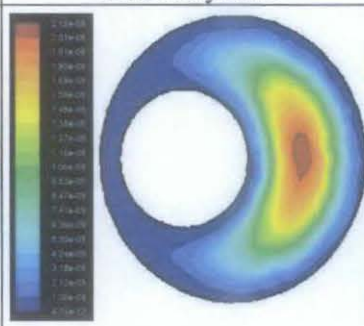
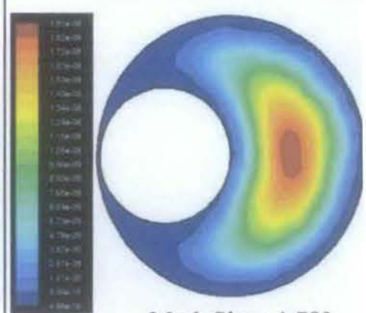
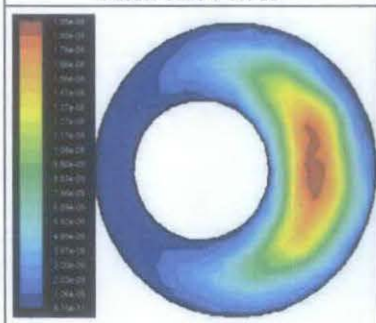
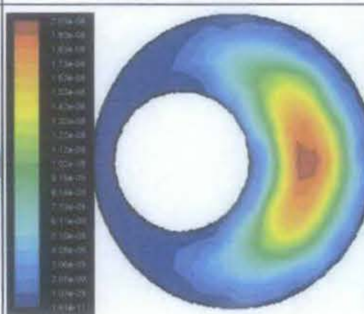
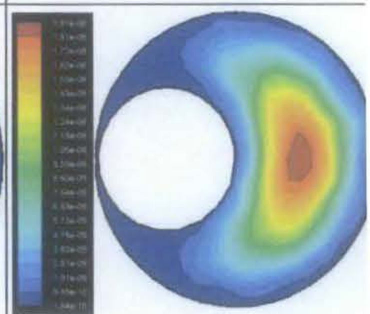
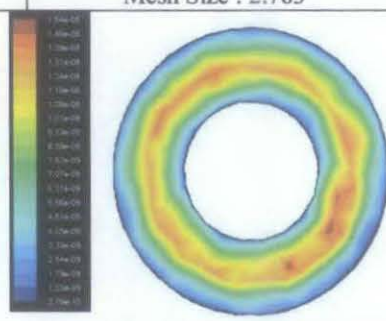
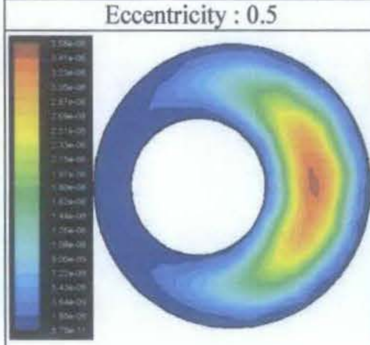
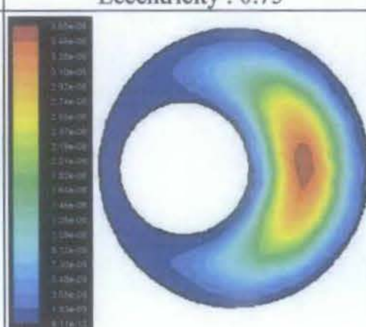
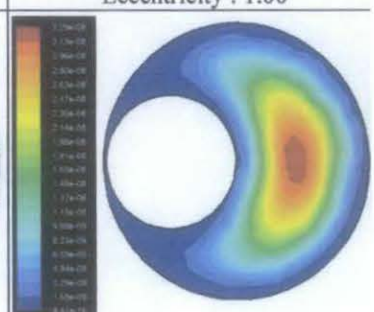
4.2 Non-Circular Geometry Flow Development

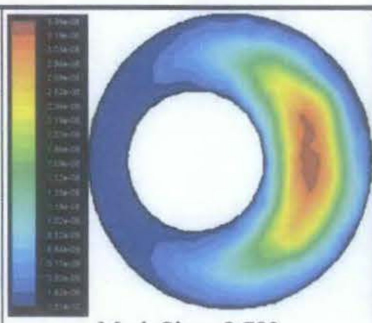
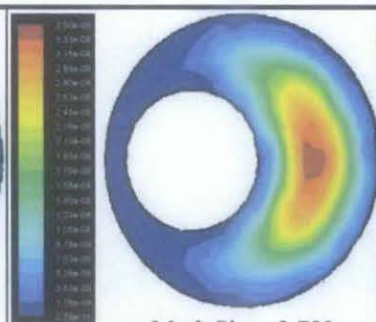
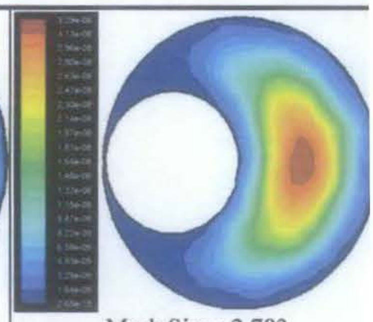
4.2.1 Simulation Results for Concentric & Eccentric Annulus

Table 4 : Simulation results for concentric & eccentric annulus

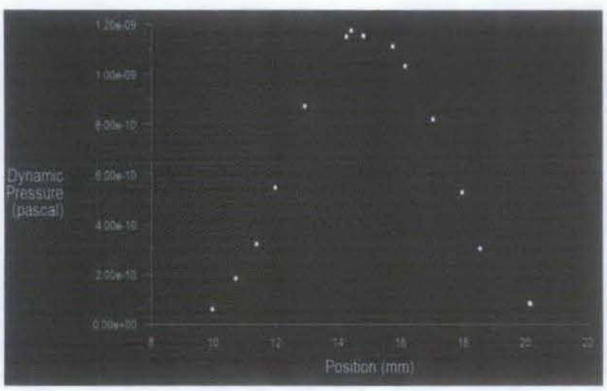
Re	VELOCITY (m/s)	RESULTS		
600	2.9699E-05	Concentric Annulus		
		Eccentricity : 0.5	Eccentricity : 0.75	Eccentricity : 1.00
		 Mesh Size : 1.783	 Mesh Size : 1.783	 Mesh Size : 1.783
		 Mesh Size : 2.783	 Mesh Size : 2.783	 Mesh Size : 2.783

1100	**5.4448E-05**	Concentric Annulus 		
		Eccentricity : 0.5	Eccentricity : 0.75	Eccentricity : 1.00
		 Mesh Size : 1.783	 Mesh Size : 1.783	 Mesh Size : 1.783
		 Mesh Size : 2.783	 Mesh Size : 2.783	 Mesh Size : 2.783
1600	**7.9197E-05**	Concentric Annulus 		

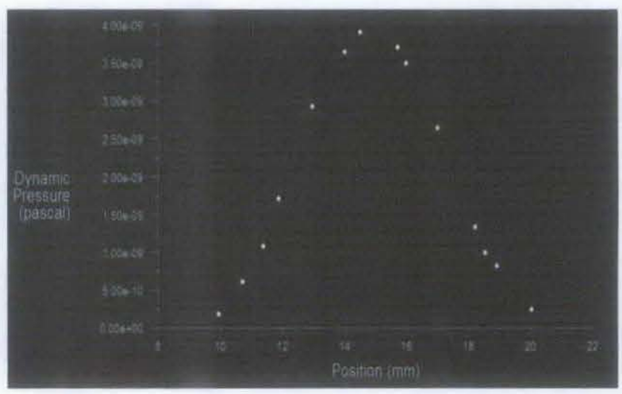
		Eccentricity : 0.5	Eccentricity : 0.75	Eccentricity : 1.00
		 Mesh Size : 1.783	 Mesh Size : 1.783	 Mesh Size : 1.783
		 Mesh Size : 2.783	 Mesh Size : 2.783	 Mesh Size : 2.783
2100	1.0395E-04	Concentric Annulus		
		Eccentricity : 0.5	Eccentricity : 0.75	Eccentricity : 1.00
		 Mesh Size : 1.783	 Mesh Size : 1.783	 Mesh Size : 1.783

		 Mesh Size : 2.783	 Mesh Size : 2.783	 Mesh Size : 2.783

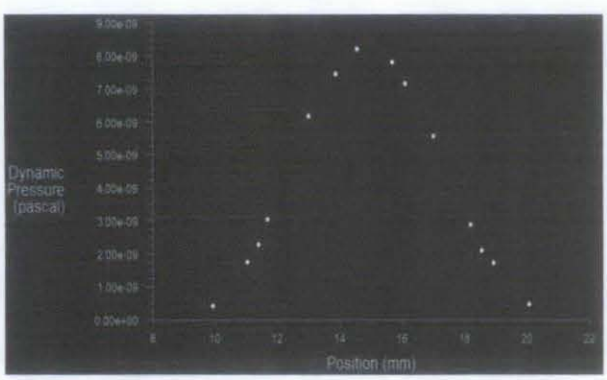
The above are the results obtained for CFD simulation for concentric and eccentric annulus. The eccentric annulus simulations are done with different grid sizes, Re values and the eccentricity of used are 0.5, 0.75 and 1.00. The simulation were carried out using Semi-Implicit Method for Pressure-Linked Equation (SIMPLE) algorithm for the pressure-velocity coupling, standard scheme was adopted for the pressure discretization and the First Order Upwind algorithm was used for the momentum equations discretization and for the flow model equations.



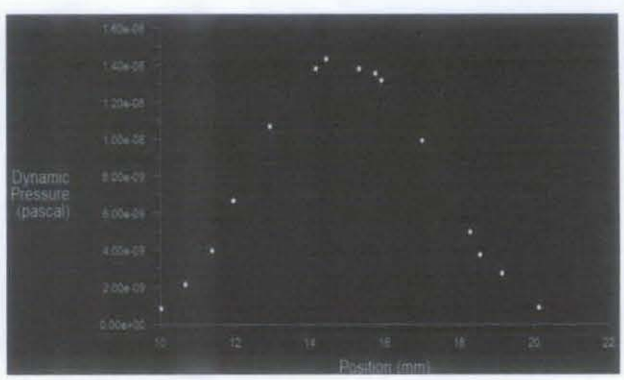
(a)



(b)



(c)



(d)

The above are the pressure loss profiles for concentric annulus of flow with different Re values. Each represents Re value of 600, 1100, 1600 and 2100 for a, b, c and d respectively. We could see the variation of the maximum pressure losses in the annulus as the Re values increase. The fluid in the concentric annulus dispersed from the outer wall to the inner wall of the annulus. Fully developed laminar flow is concentrated in the center of the annuli which conforms to the flow in circular ducts. Shah and London (1978) showed analytically that with diameter ratio of 0.5 which we used for our simulation, the flow resistance in the laminar region in concentric annulus was 30% higher than that of the pipe flow and this decreased with the eccentricity took place in the design.

As for the eccentric annulus, the analysis should be done at different sector in the outlet to measure the effect of the eccentricity to the flow. Hence, the outlet cross sectional area is divided into four (4) sectors; defined as A, B, C and D as Figure 10 to allow us to study pressure dropped and velocity profile of each sector. This allows us to understand the effect of eccentricity to the flow.

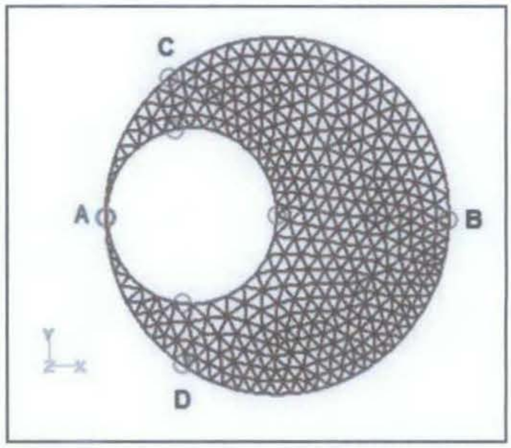


Figure 10 : Sectors divided for analysis $e = 1.0$

From the results obtained, flow higher pressure loses in towards the inner of the annulus, deviating from the walls due to the resistance to flow is increased as the gap between the two pipes decreases. This clearly shown in Figure 11 a & b.

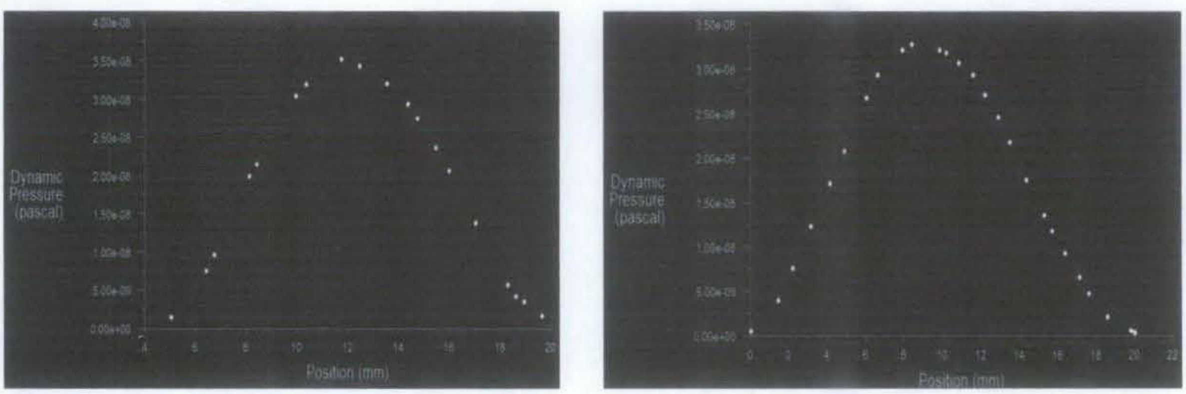


Figure 11 a & b : Dynamic pressure profile of annulus with eccentricity 0.5 and 1.0 respectively.

CHAPTER 5 : CONCLUSION

The main objectives of this project are to model different type of geometry and study its flow behavior when a laminar, Newtonian fluid flowing inside them. The basic circular geometry is developed using GAMBIT and simulations using FLUENT were successfully depicts the behavior of a circular geometry based on the analysis done and comparison done with other previous study. We also modeled the non-circular geometries to study the effect of eccentricity of the geometry. With zero eccentricity, concentric annulus has flow development to the center of the annulus gap. This is consistent with previous studies done by numbers of authors. As expected, based on previous understanding of fluid behavior, we found that as the eccentricity values increase, Newtonian, laminar flow has the tendency to have higher pressure loses in towards the inner of the annulus, deviating from the walls. This phenomenon happened because of the resistance to flow is increased as the gap between the two pipes decreases. The velocity also decreases when flow entering the narrow gap as compared to the wider gap in the annulus. With this phenomenon presents in the flow behavior of eccentric annulus, the narrow gap will be vulnerable to drilling cuttings under drilling situation.

CHAPTER 6 : REFERENCES

[1] Azouz, I., Shirazi, S. A., Pilehvari, A. A., and Azar, J. J., 1993, "Numerical Simulation of Turbulent Flow in Concentric and Concentric Annuli," Paper presented at the AIAA 24th Fluid Dynamics Conference, Orlando, FL, Jul.6–9.

[2] C. Chin, Wilson, 2001, *Computational Rheology for Pipeline and Annular Flow,* MA : Gulf Professional Publishing

[3] Cengel, Yunus A., Cimbala, John M., 2006, *Fluid Mechanics: Fundamentals and Applications,* IL : McGraw-Hill Higher Education.

[4] Dogde, N A, 1963, Friction Losses in Annular Flow (Resistance To Flow Through Smooth Eccentric And Concentric Annuli In Turbulent Flow Regime) Presented at Ann. Meeting, ASME, Phila- Delphia, 17-22 Nov. 1963

[5] Hansen, S. H., Rommetveit, R., Sterri, N., and Aas, B., 1999, "A New Hydraulics Model for Slim Hole Drilling Applications," Presented at the 1999 SPE/IADC Drilling Conference, Abu Dhabi, UAE, Nov. 8–10, Paper No. SPE/IADC 57579.

[6] J. L. Vieira Neto, A. L. Martins, A. Silveira Neto, C. H. Ataide, M. A. S. Barrozo, CFD Applied to turbulent flows in concentric and eccentric annuli with inner shaft rotation, The Canadian Journal of Chemical Engineering.

[7] Jones, O. C. Jr.: "An Improvement in the Calculation of Turbulent Friction in Rectangular Ducts," ASME Journal of Fluids Engineering, Vol. 98 (1976), pp. 173.

[8] Jones, O. C. Jr. and Leung, J.C.M.: "An improvement in the Calculation of Turbulent Friction in Smooth Concentric Annuli," Journal of Fluids Engineering, Vol.103 (1981), pp. 615-623.

[9] McCann, R. C., Quigley, M. S., Zamora, M., and Slater, K. S., 1993, "Effects of High-Speed Pipe Rotation on Pressures in Narrow Annuli," Presented at the 1993 SPE/IADC Drilling Conference, Amsterdam, The Netherlands, Feb. 22–25, Paper No. SPE 26343.

[10] Mete Avcı, Orhan Aydın, 2006, Laminar forced convection with viscous dissipation in a concentric annular duct, Elsevier Sciences Inc.

[11] M. P. Escudier, I. W. Gouldson, P. J. Oliveira, F. T. Pinho, 1999, Effects of Inner cylinder rotation on laminar flow of a Newtonian fluid through an eccentric annulus, Elsevier Sciences Inc.

[12] Ning Hsing Chen, 1979, An Explicit Equation for Friction Factor in Pipe, Ind. Eng. Chemical Fundamental.

[13] Nouri, J. M. and J. H. Whitelaw, "Flow of Newtonian and Non-Newtonian Fluids in an Eccentric Annulus With the Rotation of the Inner Cylinder," Int. J. Heat Fluid Flow 18, 236–246 (1997).

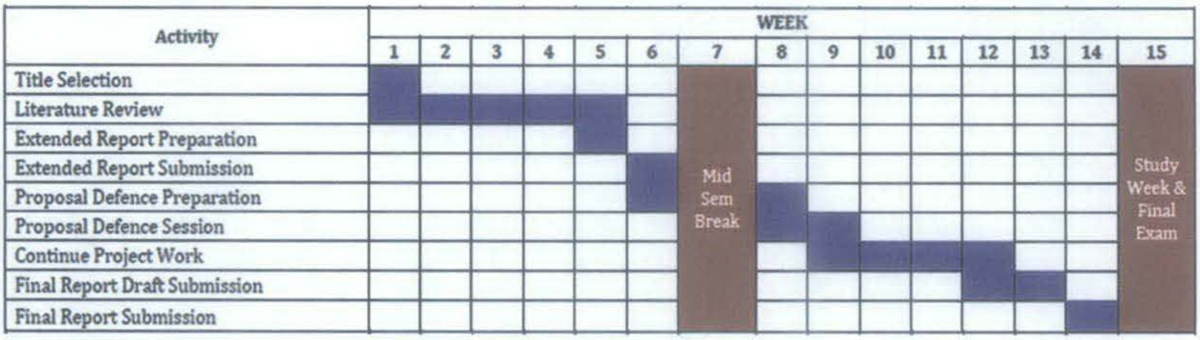
[14] Nouri, J. M., H. Umur and J. H. Whitelaw, "Flow of Newtonian and Non-Newtonian Fluids in Concentric and Eccentric Annuli," J. Fluid. Mech. **253**, 617–641 (1993).

[15] Singhal, N., Shah, S. N., and Jain, S., 2005, "Friction Pressure Correlations for Newtonian and Non-Newtonian Fluids in Concentric Annuli," Presented at the 2005 SPE Conference, Oklahoma City, OK, Apr. 17–19, Paper No. SPE 94280.

[16] Yu-Quan Liu, Ke-Qin Zhu, 2010, Axial Couette–Poiseuille flow of Bingham fluids through concentric annuli, Elsevier Sciences Inc.

APPENDIX A : PROJECT GANTT CHART

Final Year Project 1 : Time Line



Final Year Project 2 : Time Line