ABSTRACT

This research project is focused on the fluid flow analysis of gas flow in oil and gas pipeline. Flow inside pipeline can cause vibration to the pipeline structure. The objective of this project is to analyze gas flow in oil and gas transmission pipeline by using FEA (Finite Element Analysis). In order to analyze the gas flow a rig has to be made and then do the modal analysis and operational deflection shape analysis (ODS) will be conducted. Modal analysis is done by attaching an accelerometer at different place of the rig and then knocks the rig by using impact hammer to simulate external forces. The data are analyze by using the MEscope software. The result is then compared with the result obtained from the Ansys software. After that the ODS analysis is done by conducting the experiment when there is flow inside the pipe. The modal analysis will show the result when there are external force acting on the rig and ODS analysis will show the result of the rig under operating condition. The result will show the mode shape of the rig, the natural frequencies and the damping of the rig. From the result we can see that in each of the mode shape that the rig has it will have its own natural frequency. The result of ODS will show the mode shape, natural frequency and the damping of the rig under operating condition. The data are compared with the simulation is because in real world the oil and gas pipeline are buried underground, so experiment cannot be done. By comparing these two results we can obtain the error of the result obtained from the simulation.

ABSTRAK

Projek penyelidikan ini memberi tumpuan kepada analisis aliran cecair aliran gas dalam minyak dan saluran paip gas. Aliran dalam saluran paip boleh menyebabkan getaran struktur perancangan. Objektif projek ini adalah untuk menganalisis aliran gas dalam minyak dan saluran paip penghantaran gas dengan menggunakan FEA (Analisis Unsur Terhingga). Dalam usaha untuk menganalisis aliran gas pelantar perlu dibuat dan kemudian membuat analisis modal dan operasi pesongan analisis bentuk (ODS). Analisis Modal dilakukan dengan melampirkan pecutan di tempat yang berbeza pelantar dan kemudian mengetuk pelantar dengan menggunakan tukul untuk meniru kesan kuasa-kuasa luar. Selepas itu bacaan pecutan adalah menganalisis dengan menggunakan perisian Mescope itu. Hasilnya kemudian dibandingkan dengan keputusan yang diperolehi balik perisian ANSYS. Selepas itu analisis ODS itu dilakukan dengan menjalankan eksperimen apabila terdapat aliran di dalam paip. Analisis modal akan menunjukkan hasil apabila ada daya luaran yang bertindak ke atas rig dan analisis ODS akan menunjukkan hasil daripada pelantar di bawah keadaan operasi. Hasilnya akan menunjukkan bentuk mod pelantar itu, frekuensi semulajadi dan redaman rig. Dari hasil yang kita dapat melihat bahawa dalam setiap bentuk mod yang mempunyai pelantar ia akan mempunyai kekerapan sendiri semula jadi. Hasil ODS akan menunjukkan bentuk mod, kekerapan semulajadi dan redaman pelantar di bawah keadaan operasi. Alasan bahawa keputusan membentuk eksperimen dibandingkan dengan simulasi adalah kerana dalam dunia sebenar minyak dan saluran paip gas telah disemadikan di bawah tanah, jadi untuk mendapatkan bentuk mod dan frekuensi semulajadi perancangan eksperimen tidak boleh dijalankan supaya hasilnya boleh diperolehi daripada simulasi. Dengan membandingkan kedua-dua keputusan yang kita boleh mendapatkan kesilapan keputusan yang diperolehi daripada simulasi.

TABLE OF CONTENTS

CHAPTER 5: CONCLUSION AND RECOMMENDATION

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATION

FRF	Frequency response function
ODS	Operational deflection shape
Re	Reynolds number
PVC	Polyvinyl chlorine
FEM	Finite element method
FFT	Fast fourier transform
APS	Auto power spectra
XPS	Cross power spectra
NIDAq	National instruments data acquisition
ρ	Density
μ	Viscosity
υ	Velocity
d	Diameter
Re	Reynolds number
f	Friction factor
L	Length
τ_w	Shear stress
ε	Surface roughness
K_L	Minor losses
h_L	Irreversible head loss
g	Gravity
P_L	Pressure loss

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

Oil and gas are the most important source of energy in this world nowadays. This natural energy source exist inside the earth and commonly located deep under the ocean floor. In order to transport them to the shore, pipeline are the main are the main solution used to solve this problem instead of carrying them by ship.

Theses pipeline are usually large in diameter and can operate at high pressure so that high transportation rate can be achieved. The design and material used for the pipeline must be selected properly so that it can suit with the requirements above because raw fossil fuel dug from the earth are hazardous to the surrounding population and environment.

Steel are the conventional material selected for the pipeline. Initially, all the steel pipes that is used to construct the pipeline must be threaded together but due to the difficult to perform for large pipes and leaking problem due to high pressure, the pipeline nowadays are welded together

This pipeline are usually located under the sea or buried on land. This pipeline must be able to withstand high pressure cause by the flow inside the pipeline. This high pressure can cause many failures to the pipeline such as burst, puncture, overload, structural overload (buckling), fatigue, and fracture. The high pressure from the inside of the pipeline also will cause the pipeline to vibrate that will eventually cause failure as stated above to the pipeline

1.2 PROBLEM STATEMENT

Due to the high demand of this natural energy resource, the oil and gas company have to increase the flow rate of the gas inside the pipeline so that customer demand can be fulfill. By increasing the flow, the pressure inside the pipeline will increase significantly.

In order to conduct this analysis, two parameter must be analyzed that is the speed of the gas flow across the pipeline, the vibration of the pipeline and the design of the pipeline.

To understand on how fluid flow effects the vibration of the pipeline, an experiment has to be conducted. Nowadays the effects of fluid flow on vibration of the pipeline are not completely understood. By doing the experiment on how fluid flow effect vibration on the pipeline, we can reduce the problem that is cause by vibration on the pipeline in the oil and gas industry.

1.3 OBJECTIVE OF PROJECT

The overall objective of this project is to analyze gas flow in oil and gas transmission pipeline by using FEA (Finite Element Analysis). Therefore the main objective of this project is to make a rig on the gas flow in the oil and gas pipeline and compare the data from the test rig with the data obtained from the software.

1.4 SCOPE OF PROJECT

The scopes of project are

1) Carry out the experiment and analyze the data from the experiment
2) Compare the data obtained from the experiment with the CFD software

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Fluid around a structure can significantly alter the structure's vibrational characteristic. The presence of a quiescent fluid decreases the natural frequencies and increases the damping of the structure. A dense fluid couples the vibration of elastic structures which are adjacent to each other. Fluid flow can induce vibration. A turbulent fluid flow exerts random pressures on a structure, and these random pressures induce a random response. The structure can resonate with periodic components of the wake. If a structure is sufficiently flexible, the structural deformation under the fluid loading will in turn change the fluid force.

The ways of the structure vibrate and the natural frequency can be determined by doing analysis on the structure. The analyses that can be done are the modal analysis and operational deflection shape (ODS) analysis. By doing this analysis the mode shape, natural frequency and damping can be determine.

2.2 FLUID FLOW

In general there are three types of fluid that is liquid, vapour, and gas. The motion of fluid occurs due to the unbalanced forced or stresses subjected to them. As long as the imbalanced force is applied, the fluid will continuously move.

Between this three state, gases have the most weak intermolecular forces. Gaseous will expand to fill any container. Gases also are highly compressible. This means if the pressure is doubled at a constant temperature, the density of the gases will increase.

In contrast, liquid have the strongest intermolecular force compare to the other two states. These strong molecular forces will cause liquid to retain constant volume if we put it inside a container compare to gases which tend to expand. Liquid are nearly incompressible.

Vapour is a state which happen when a type of gaseous interacting with its own liquid. The vapour and liquid can flows together as a mixture. There are two type of flow in fluid that is laminar flow and turbulent flow. This type of flow can be determined by calculating the Reynolds number by using equation 2.1 (Yunus, 2006).

Reynolds number equation:

$$Re = \frac{\rho v d}{\mu} \qquad (2.1)$$

2.2.1 Entry Length

The hydrodynamic entry length are normally taken to be the distance from the pipe entrance to where the wall shear stress reaches about two percent of the fully develop valve. The hydrodynamics entry length foe a laminar flow is calculated from the equation 2.2 (Yunus, 2006).

Entry length for laminar flow:

$$L_{h,laminar} \cong 0.05 ReD \qquad (2.2)$$

For the turbulent flow, the hydrodynamics entry lengths are shorter compare to laminar flow. This is due to the smaller Reynolds number for turbulent flow. This can be calculated from equation 2.3 (Yunus A. Cengel 2006).Figure 2.1 show the flow in a pipe from the entrance to the fully develop region.

Entry length for laminar flow:

$$L_{h,turbulent} = 1.359 D Re_D^{\frac{1}{4}} \quad (2.3)$$

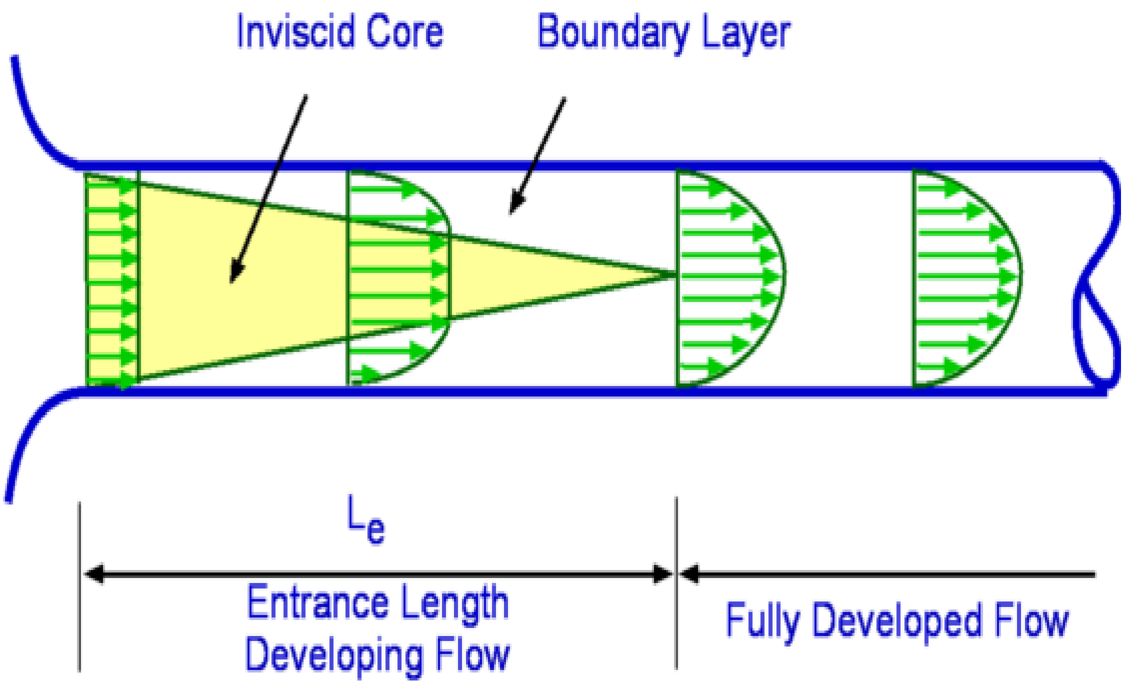


Figure 2.1: Entry length

Source: mdp.eng.cam.ac.uk, 2003

2.2.2 Fully Develop Laminar Flow in Pipe

Fully develop laminar flow in pipe have the Reynolds number, Re≤ 2300. The particle in fully develop laminar flow moves at a constant axial velocity that is across the pipe. The velocity profile for fully develop laminar flow remain unchanged, the acceleration is equal to zero because the flow is steady and fully develop (Yunus, 2006).

The entrance effect in fully develop laminar flow is negligible because in order to have a fully develop laminar flow, the pipe must be long enough approximately about the same length from the calculated length from the formula above. The steady laminar flow also can be consider as incompressible fluid with constant properties in the fully develop region from the flow. Figure 2.2 show the fully develop laminar flow velocity profile

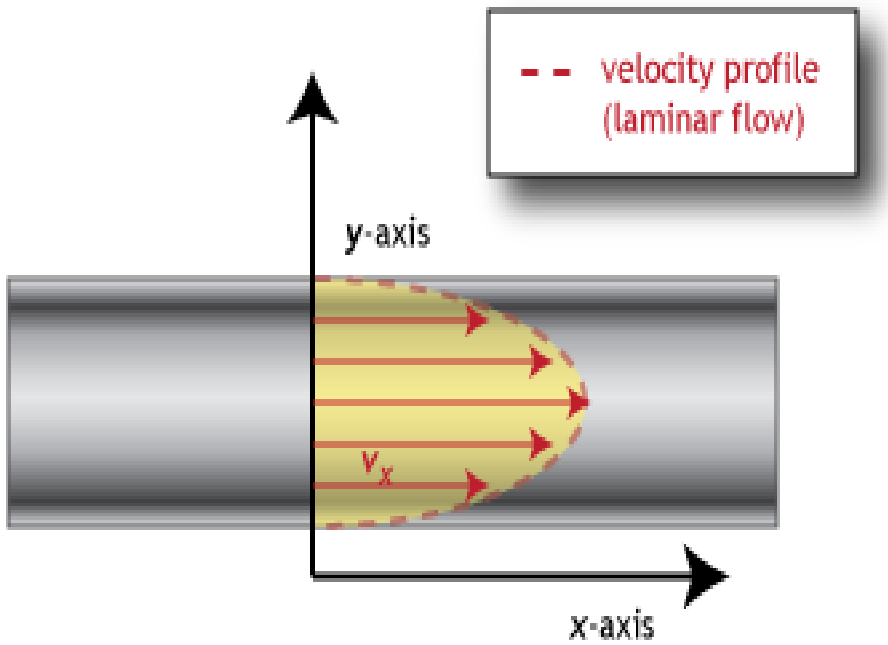


Figure 2.2: Laminar flow velocity profile

Source: accessibility.psu.edu, 2005

2.2.3 Fully Develop Turbulent Flow in Pipe

Fully develop turbulent flow in pipe have the Reynolds number, Re$\geq$ 4000. Throughout the turbulent flow, there exist some random and rapids fluctuations of swirling region of fluid that is called eddies. This swirling eddies transport mass, momentum, and energy rapidly than molecular diffusion to other region. This will cause the friction, heat transfer, and mass transfer coefficient to increase (Yunus, 2006).

When the average flow is steady, the eddie motions in turbulent will continuously cause significant fluctuation in the values of velocity, temperature, pressure, and even density in compressible flow.

There are four region of flow can be consider in turbulent flow. The regions are characterized by the distance of the flow from the wall. The region is known as viscous layer, buffer layer, overlap layer, and turbulent layer. The first layer which is very thin and closest to the wall is known as viscous sublayer. In this region, the velocity profile is nearly linear and the flow is streamlined (John M. Cimbala, 2006).The second layer which have the second closest length to the wall are known as buffer layer. In this layer, the turbulent effects are becoming significant but the flow is still dominated by viscous effect (Yunus, 2006).

Overlap layer or more known as inertial sublayer are located right above the buffer layer. In this layer, the turbulent effects are much more significant. However the turbulent effect is still not dominant, it still dominated by the viscous effect (Yunus, 2006). Above the overlap layer is the turbulent layer. In this layer the turbulent effect are much more dominant than the viscous effect. In this layer also the turbulent effect can be seen clearly.

Even the thickness of the viscous sublayer is very small which is about less than one percent of the pipe diameter, it have a very important role on the flow characteristic because of the large velocity gradient is involved. The eddies motion is dampened by the wall, this will cause the flow in this layer is laminar and the shear stress is also consist of laminar shear stress which is proportional to the fluid viscosity (Yunus, 2006). Figure 2.3 shows the velocity profile for turbulent flow in pipe.

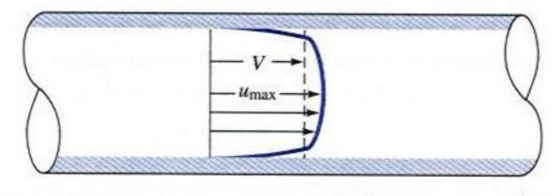


Figure 2.3: Turbulent flow velocity

Source: pofileme.queensu.ca, 2001

2.2.4 Pressure Lost and Friction Factor

The pressure lost can be calculated from the equation 2.4

$$\Delta P_L = f \frac{L}{D} \frac{\rho V^2{}_{average}}{2} \tag{2.4}$$

The friction factor, f can be calculated from the equation 2.5

$$f = \frac{8\tau_w}{\rho V^2{}_{average}} \tag{2.5}$$

This equation 2.4 and 2.5 be applied for both fully develop laminar and turbulent flow. But for this equation we do not taken the roughness of the material used for the pipe as a factor. If we take the surface roughness as a factor, the friction factor equation will be in equation 2.6. The surface roughness can be determine from the table 2.1

$$\frac{1}{\sqrt{f}} = -1.8 \log\left[\frac{6.9}{Re} + \left(\frac{\varepsilon/D}{3.7}\right)^{1.11}\right] \tag{2.6}$$

Table 2.1: Surface roughness

Pipe Material	**Roughness ε**
	feet
drawn brass or copper	0.000005
PVC pipe	0.000005
commercial steel	0.000150
wrought iron	0.000150
asphalted cast iron	0.000400
galvanized iron	0.000500
cast iron	0.000850
concrete	0.001 - 0.01

Each material have its own surface roughness. With this we can determine the pressure lost for the flow (Munson, 2002)

2.2.5 Minor Loss

In pipeline, there are many components that make it a system such as elbows, valve and many more. These components can effects the flow of the fluid inside the pipe. Although these components will affect the flow just a little but in some cases these components will cause significant effects to the flow. For example valve, if we fully opened the valve the effect of the fully open valve to the flow will be minor. If the valve was half open, this will cause major effect to the flow. These losses are called minor losses. These minor losses are usually expressed in the term of los coefficient, K_L. The minor losses can be calculated from the equation 2.7, (Yunus, 2006).

$$K_L = \frac{h_L}{V^2/2g} \qquad (2.7)$$

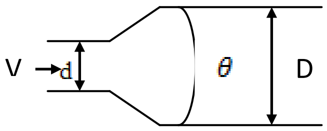
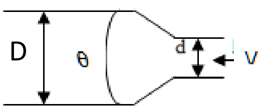
The head loss can be calculated from the equation 2.6 where ΔP_L is equal to presser different when the pipe have components attached to it minus the pipe when there is no components attached to it. The majority of irreversible head loss occurs near valve but some of it may occur downstream of the valve. This happen due to the induce swirling turbulent eddies that are produce in the valve and continues downstream. This eddies effect will dissipated into heat while the flow in the downstream section will eventually become fully develops turbulent flow. The total head lost can be calculated by adding all the losses calculated earlier. The loss coefficient can also be determined from table 2.2

$$h_L = \frac{\Delta P_L}{\rho g} \tag{2.8}$$

This loses in the elbow happen due to flow separation on the inner side and the swirling secondary flow that is caused by the different path length (Yunus A. Cengel 2006). The method used to reduce the losses from happen is by making the turn into arc shape. This will make the fluid to flow easily. If sharp 90° bend were used, the losses will be large. But these losses also can be reduce by installing vanes inside the elbow near the sharp elbow. This will make the fluid to flow easily compare to the flow if we do not installed vanes. However these losses do not include the frictional losses along the bend. These losses are calculated as in straight pipe by using the length of the centreline of the pipe length.

The purpose of the usage of valve is to control the flowrate by simply altering the head loss until the desire flowrate is achieved. Valves have a low loss coefficient when it is fully open. However when it is not fully opened, the losses will increase. Each valve has its own advantage and disadvantages. Gate valve slide up and down like a gate, globe valve closed a hole place in the valve, angle valve is a globe valve with 90°turn, and a check valve function like a diode which allowed fluid to flow in one direction only.

Table 2.2: Minor losses, K_L

90° smooth bend	Flanged = 0.3
	Threaded = 0.9
90° miter bend	Without vanes = 1.1
	With vanes = 0.2
45° threaded bend	0.4
180° returned bend	Flanged = 0.2
	Threaded = 1.5
Tee (branched flow)	Flanged = 1.0
	Threaded = 2.0
Tee (line flow)	Flanged = 0.2
	Threaded = 0.9
Expansion	$\theta \rightarrow 20^\circ = 0.02$
	$\theta \rightarrow 45^\circ = 0.04$
	$\theta \rightarrow 60^\circ = 0.07$
Contraction	For $\theta = 20^\circ$
	$\frac{d}{D} \rightarrow 0.2 = 0.30$
	$\frac{d}{D} \rightarrow 0.4 = 0.25$
	$\frac{d}{D} \rightarrow 0.6 = 0.15$
	$\frac{d}{D} \rightarrow 0.8 = 0.10$
Globe valve (fully open)	10
Angle valve (fully open)	5
Ball valve (fully open)	0.05
Swing check valve	2
Gate valve	Fully open = 0.2
	$\frac{1}{4}$ open = 0.3
	$\frac{1}{2}$ open = 2.1
	$\frac{3}{4}$ open = 17

2.3 MODAL ANALYSIS

Modal analysis is a method to determine the dynamic properties of a structure by doing experiment. There are other ways to determine the dynamics properties of a structure such as FEM modal simulation (Finite Element Method). Modal analysis will determine the frequency response function (FRF) of a system. The relationship between the measured output and input in FRF is linear (Jenneskens, 2006). For example when there is an applied force which is the input, the response such as displacement which is output will be measured simultaneously and the FRF can be calculated. The mode shape, natural frequency and modal damping can be obtained from the FRF.

The most important thing in modal analysis is the structure must be isolated from its surrounding. This can be done by suspending it by elastic springs and there is more other ways that can be done to isolate the structure from the surrounding. The isolation is important so that there are no external forces acting on the system (Jenneskens, 2006).

There are some basic assumptions that must be done and normally the assumptions are done before performing the analysis. The assumptions are the structures linear, the structure must be time variant, structure must obey Maxwell's reciprocity and the structure is stable. In modal analysis, force is applied as an input at a specific coordinate on the structure is defined as the excitation mechanism (Maia & Silva, 1997).

There are two ways that can be used as the excitation mechanism which is the impact hammer excitation and shaker excitation. Impact hammer will only applied in contact with the structure in order to gives excitation in a short period of time while the shaker will remain in contact with the structure throughout the experiment. Each excitation mechanism has its own advantages and disadvantages but the data obtained from the analysis are not wrong.

2.4 OPERATIONAL DEFLECTION SHAPE (ODS) ANALYSIS

Operating deflection shape (ODS) is a vibration analysis on a structure under operating condition. The output of the system can be any number of things such as displacement, acceleration and many more. ODS is different from modal analysis due to the force acting on the system for ODS are not measured (Jenneskens, 2006).

Vibration measurement are performed at different points and direction on the structure known as degree of freedom (DOF) and the vibration pattern can be shown in animated geometry model. An ODS can be defined from any forced motion, either in time or at a specific frequency (Schwarz & Richardson, 1999). The one that are defined in time are called time domain ODS and the one that are defined in frequency are called frequency domain ODS. An ODS that can be obtained from a set of measured time domain response are random, impulsive, sinusoidal and ambient. An ODS that obtained from a set of computed frequency domain measurement are linear spectra (FFT's), auto power spectra (APS's), cross power spectra (XPS's), and frequency response function (FRFs).

Real continuous have an infinite degree of freedom and infinite number of modes but for experimental purpose we can limit the frequency value of the structure that we want to analyse. We can measure small subset of the measurement and yet we can accurately define the resonances that are within the frequency range of the measurement.

ODS have its own advantage and disadvantage. The advantages of ODS analysis are there is no assumption of a linear model, the structure will experience actual operating forces which means that the result will be accurate to the situation in the real world, and true boundary condition will be apply in this analysis. The disadvantages of ODS analysis are the dynamic modelling obtained are not complete which means there are no natural frequency, mode shape and damping properties. The other disadvantages of ODS analysis is the operational deflection shape only reflect the cyclic motion at a specific frequency, but there will be no conclusion can be make for the behaviour of the analyse structure at different frequency.

CHAPTER 3

RESEARCH METHODOLOGY

3.1 INTRODUCTION

Research methodology act asguidance in order for student to complete the project given.Experiment setup is important so that we can compare the analysis of the fluid flow from the real world and with the analysis that we obtain from the software.

3.2 FLOW CHART OF METHODOLOGY

The flow chart of methodology is the basic guidelines for us to follow in order to achieve the objective of our project. Figure 3.1 show the flow chart for this project. For this final project, we must follow the flow chart that we have made so that the progress of our project can be seen.

In order to understand more about the topic for the *Projek Sarjana Muda* (PSM), journals and books are the main references. The journals are obtains from the internet and books are borrowed from the library. By reading these references, the basic parameter to do the analysis of fluid flow in oil and gas transmission pipe can be determined. The review on the journal and books that have been read are summarized in chapter 2.