INVESTIGATING THE EFFECT OF GAS FLOW ON THE DRAG REDUCTION PERFORMANCE IN MULTIPHASE FLOW SYSTEM

NOOR ASHIKIN BINTI OHYANUDDIN

Thesis submitted in fulfillment of the requirements
for the award of the degree of
Bachelor of Chemical Engineering (Gas Technology)

Faculty of Chemical & Natural Resources Engineering
Universiti Malaysia Pahang

JANUARY 2012

ABSTRACT

The study on drag reduction towards multiphase flow system which in the fluid flow systems become interest towards the academic and industrial during the past several decades as it can give benefit to many industrial applications such as oil well operations, fire fighting, heating and cooling water circuits and biomedical system when drag reduction is reduce. The flow transitions inside the pipeline itself can give the effect to the drag reduction performance. It can offer large economic advantages and larger effectiveness in the transportation of the fluid. The objectives of the research are to investigate the effect of gas flow rate on drag reduction performance and to investigate the drag reduction ability from the effect of gas flow rate in the multiphase flow system. The research is carried out by sieving the sand to get different sand. Then, the main reservoir is filled with ≈ 450 kg water and the sand is mixed together. The solution is passed through 1.5 inch pipe and compressed air is supplied to the solution at a point of pipeline The turbulent drag reductions are measured by reading the value of pressure drop along the re-circulatory flow. Experimental works are repeated with the other liquid velocity, sand concentration, sizes and without presence of air. From the research, the results show that occurrence of gas flow will increase the drag reduction in all conditions. Drag reduction ability increase when effect of gas flow occurs in multiphase flow system. Drag reduction is believed can be strongly influenced by the concentration of particles The influence of drag reduction on the turbulence worked better at higher Reynolds number. The percentage of drag reduction is also affected by the concentration and size of the particle, where it is more favored towards higher concentration and larger particle sizes.

ABSTRAK

Kajian mengenai pengurangan daya seret ke arah sistem aliran berbilang fasa yang dalam sistem aliran bendalir menjadi minat terhadap akademik dan industri dalam tempoh beberapa dekad yang lalu kerana ia boleh memberi manfaat kepada banyak aplikasi industri seperti operasi telaga minyak, proses pemadaman api, pemanasan dan penyejukan aliran air dan sistem bioperubatan apabila pengurangan seretan dapat dikurangkan. Peralihan aliran di dalam saluran paip itu sendiri boleh memberi kesan kepada prestasi pengurangan daya seret. Ia boleh menawarkan kelebihan ekonomi yang besar dan keberkesanan yang lebih besar dalam pengangkutan bendalir. Objektif kajian adalah untuk menyiasat kesan kadar aliran gas pada prestasi pengurangan seretan dan menyiasat keupayaan seretan pengurangan dari kesan kadar aliran gas ke dalam sistem aliran berbilang fasa. Penyelidikan ini dijalankan dengan pengayakan pasir untuk mendapatkan saiz pasir yang berbeza. Kemudian, takungan utama dipenuhi dengan ≈ 450 kg air dan pasir dicampurkan bersama-sama. Campuran disalurkan melalui paip 1.5 inci dan gas disalurkan kepada campuran pada satu titik saluran paip serta pengurangan seretan diukur dengan membaca nilai kejatuhan tekanan sepanjang saluran paip. Kerja-kerja eksperimen diulangi dengan halaju cecair lain, kepekatan pasir, saiz dan tanpa kehadiran udara. Dari kajian, keputusan menunjukkan bahawa berlakunya aliran gas akan meningkatkan pengurangan seretan dalam semua keadaan. Pengurangan seretan dapat meningkatkan keupayaan apabila kesan aliran gas berlaku dalam sistem aliran berbilang fasa. Pengurangan seretan dipercayai boleh dipengaruhi oleh kepekatan zarah pengaruh pengurangan daya seret pada aliran lebih baik pada nombor Reynolds yang lebih tinggi. Peratus pengurangan daya seret juga dipengaruhi oleh kepekatan dan saiz zarah, di mana ia lebih digemari ke arah penumpuan yang lebih tinggi dan saiz zarah yang lebih besar.

CHAPTER 1

INTRODUCTION

1.0 OVERVIEW

The study on drag reduction phenomenon in the fluid flow systems become interest towards the academic and industrial during the past several decades. Many industrial applications such as oil well operations, fire fighting, heating and cooling water circuits and biomedical system can have the benefits by decreasing the drag reduction. It has been more than 60 years since Toms (1948) first discovered the tremendous reduction in wall shear stress (Ahmed Kamel and Subhash N. Shah, 2009) which can cause reduction of energy loss in turbulent flow in the pipeline. After that, many other scientists and also researchers find the other ways to reduce the drag reduction.

Multiphase flow system can be occurred inside the pipeline system over a long distance when the crude oil is transporting from the upstream to the processing plant. The flow system may contain oil, gas (combination of sour gas and sweet gas), water, sludge, mud and also rock. The flow transitions in gas–liquid–solid phase can depend on many different variables such as liquid and gas velocities, liquid and gas physical properties, solid features, gas distributor design and the pipeline structure. The flow transitions inside the pipeline itself can give the effect to the drag reduction performance. In some cases, drag reduction performance can be observed by measuring the gas flow rate within the multiphase flow system. It can offer large economic advantages and larger effectiveness in the transportation of the fluid. In general, drag reduction can occur between an object which is exposed to the fluid flow and the fluid can be in gas or liquid phase.

Pressure drop is one of the problem that may encountered to the fluid transportation is the pipeline. Variety of flow patterns will be formed or the value of pressure gradient will be vary depending on the physical properties of the object involve. Decreasing of the pressure drop is one of the practical importance from an economic viewpoint as it can reduce the pumping energy of the fluid in the long pipeline. In some conditions, sufficient gas quantities to the liquid could be transported by not building new pipeline and also by not increasing the pump horsepower is required. As a future engineer, the study of drag reduction is very helpful to design the process equipment later and to explore the effect of gas injection on the hydraulic transport of fluid in the pipeline.

1.1 PROBLEM STATEMENT

There are some problems encounters inside the pipeline that carry out the fluid flow.

i. The used of pipelines to transport the fluid are mainly practice in chemical process and petroleum industries due to the effectiveness result.

ii. Turbulence flows occur inside the transmission pipeline and this phenomenon contribute to the eddy flow movement of the fluid. Therefore, huge amount of energy is required to be used to move the fluid in the pipeline.

iii. Resistance between the fluid molecules itself, between the other material molecules and the pipe wall always occur in the pipeline during the fluid transportation. The resistance in pipeline causes pressure loss.

iv. Pressure drop can occur in the pipelines and also in the individual units such as pump during the fluid transportation. It is a problem that needs to be focus in this research.

v. Pumps are used to provide a pressure head to the flow to overcome the wall friction and also drag in the flow. More pumping energy is required when the pressure head increase and can cause in high operation cost.

1.2 OBJECTIVES

These are the objectives that should be accomplished by the end of this research of 'Investigating the effect of gas flow rate on the drag reduction performance in multiphase flow system'. The objectives are taken with consideration of the knowledge in chemical engineering study and due to the industrial problems.

i. To investigate the effect of gas flow rate on drag reduction performance.

ii. To investigate the drag reduction ability from the effect of gas flow rate in the multiphase flow system.

1.3 SCOPES OF RESEARCH

In order to investigate the effect of gas flow rate performance in multiphase flow system, some of restriction factors that involve have been manipulated to observe the pressure drop effect. The restrictions that involve in this research are listed like below:

Table 1.1: Research Scope

ITEM	DESCRIPTION
Gas flow rate	• Compressed air at 1 psi is used.
Liquid flow rate	• Newtonian fluid (tap water) is used. • Ten different liquid flow rate are measured in the research.
Particles size	• Two different sand sizes (71µm and 315µm) are being used in every set of research. • One of the effects in three phase flow system.

Particle concentration	• Based on weight basis. • Four different sand concentrations (111ppm, 223ppm, 446ppm and 892ppm) are considered to be used in the research.
Percentage of drag reduction (%DR)	• Consideration of pressure drop that occur in the pipeline system.

1.4 SIGNIFICANCE OF THE RESEARCH

The significance of the study is about to concern of energy consumption as huge amount of energy is needed to transport the fluid especially in oil and gas industry. This matter may decrease the consumption of fossil fuel as this non-renewable source is the main source to generate power to be used in various sectors. Other than that, the methods that practiced are to maximize the flow inside the pipeline by reduce the cost of energy generation and the operations. Thus, the methods might be applied by the industries to control the pressure drop in the pipeline problems and also to increase the drag reduction performance.

CHAPTER 2

LITERATURE REVIEW

2.0 OVERVIEW

The addition of small concentrations of high molecular weight polymer to water or other solvent can produce large reductions in frictional pressure drop for turbulent flows past a surface, leading to the possibility of increased pipeline capacities and faster ships since (Toms, 1948) discovered the technique in late 1940s. Multiphase flow is frequently encountered in many industrial units such as distillation columns, pipelines, boiler tubes, condensers, evaporators, and chemical reactors. Offshore production has necessitated transportation of both gas and liquid phases over long distances before separation. The flow type has many unique features, which must be evaluated in each situation. However, one phenomenon which is always undesirable is the high axial pressure gradient that can result substantial energy consumption per unit volume of liquid throughput.

Addition of small additives quantity such as a few parts per million (ppm), can greatly reduce the turbulent friction factor of a fluid. The aim for the drag reduction is to improve the fluid-mechanical efficiency by using active agents, known as drag reducing agent (DRA). In multiphase flow, percent drag reduction (%DR) is defined as the ratio of reduction in the frictional pressure drop when the flow rates are held constant to the frictional pressure drop without DRA, multiplied by 100, as shown in Eq. (1).

$$\%DR = \left(\frac{\Delta P_b - \Delta P_a}{\Delta P_b}\right) \times 100 \qquad (1)$$

In this equation P_b is the pressure difference before adding additives and P_a is the pressure difference after adding additives.

2.1 TYPES OF FLUID FLOW

Fluids are pumped and passed through straight pipes during operations such as hydraulic fracturing, acidizing, wellbore cleanup, cementing and drilling in the petroleum industry. The fluid inside the transmission pipeline usually moves under turbulent flow conditions. High friction pressure losses are encountered when fluids are pumped through straight pipes and will limit their pumping capacity.

2.1.1 Laminar Flow

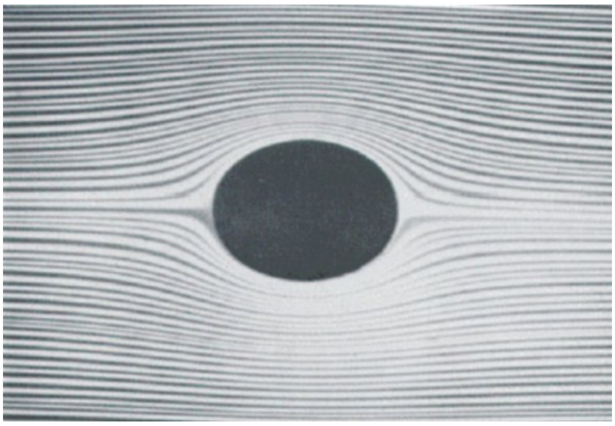

Figure 2.1: Laminar Flow

Laminar flow can be defined as state of flow where the various fluid do not mix with each other (Hoener, 1965). The value of Reynolds number less than 2000 can be defined as laminar flow regime for pipes. Laminar flow occurs when the fluid particles are travelling in straight parallel lines. It can be consider as a smooth motion or steady motion of the fluid. This type of flow is known as low friction or viscous flow.

Laminar flow as a series of co-axial cylinders oriented along the flow direction such a flow structure which is known as telescopic shear. The central part of the fluid which includes the axis has the highest velocity u_0. The velocity at the wall is necessarily zero, with intermediate velocities in-between. A schematic representation is shown in Fig. 2.2.

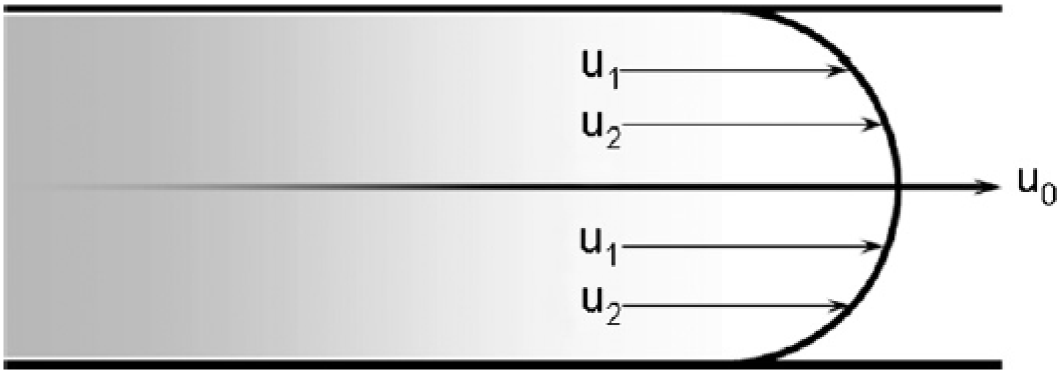

Figure 2.2: Laminar flow at velocity, u in a cylindrical conduit.

Laminar flow can be created between two parallel plates as shown in Fig. 2.3.

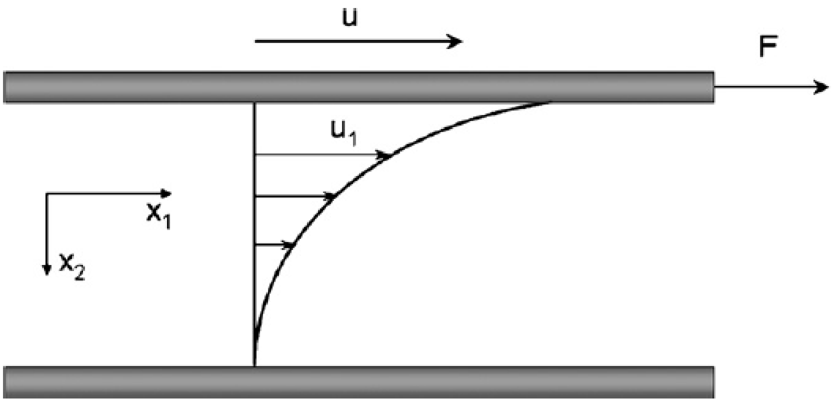

Figure 2.3: Laminar flow between parallel plates. The shearing force, F acts on the top plate as indicated. The velocity, u decreases going down along the vector x_2 since the velocity at the bottom plate is necessarily 0.

2.1.2 Turbulent Flow

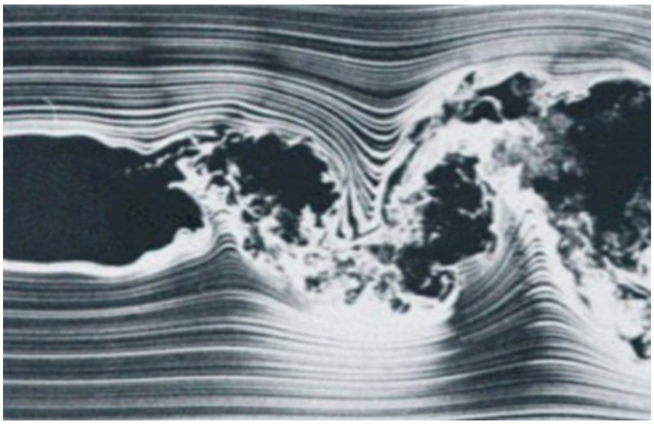

Figure 2.4: Turbulent Flow

In most petroleum pipelines, the flow is turbulent (Kamel and Shah, 2009). Non-linear currents and frictional resistance occur in this type of flow and consume much of energy to be applied to move the fluids. The value of Reynolds number increase when the flow changes from laminar to turbulence. With the increases of Renold's number, pipe flow are transitions from laminar to turbulent over a range of values from 2,000 to 10 500 and is fully turbulent above 10 500 (Stanford P. Seto, 2005). Eddies motions and disturbances are formed and will cause friction between the liquid molecules and also between the liquid molecules and the pipe wall.

2.2 DRAG FORCE

A drag force is produced in flow direction when fluid moves or exerts over a solid body and will result of two forces. Force will be generated due to skin friction drag or friction drag which is directly related to the wall shear stress and the other force is due to result of pressure drag which is same as normal stress. Skin friction which is a consequence of no-slip boundary on the surface can be either laminar at low Reynolds number or turbulent at high Reynolds number (Bushnell and Moore, 1991). Turbulent flow possesses higher total shear stress than laminar flow and can be defined as,

$$\tau_{tot} = \tau + \tau_{turb} = \mu \frac{dU}{dy} - \rho\overline{uv} \tag{2}$$

where y is distance from the wall, U is local mean velocity, μ is absolute viscosity (dynamic viscosity), ρ is fluid density and $\overline{uv}$ is Reynolds stress.

Turbulent boundary layer structures cause most of dissipation of energy. These structures can be incoherent and coherent (vortices), which parallel and close to the wall. Usually, the coherent structures are oriented in the stream wise direction and can cause up to 80% of turbulent fluctuating energy (Lumley and Blossey, 1998).

2.3 PRESSURE DROP

Generally, there are two major types of fluid flow exist which is laminar and turbulent flow. The movement of the fluid is easily to determine as a flow where any of the fluid particles are not travelling in straight parallel lines. These flow regimes are defined by Reynolds number which is a dimensionless number which can define the ratio of inertial forces to viscous forces measurement.

$$\mathrm{Re} = \frac{\rho VD}{\mu} \tag{3}$$

2.3.1 Pipe Flow Characteristics

A fully developed turbulent flow through a straight pipe with diameter, D and mean shear stress at wall, τ_{ω}, for Newtonian and non-Newtonian fluids for all regimes is given by,

$$\tau_{\omega} = \frac{D}{4}\frac{\Delta P}{\Delta x} \tag{4}$$

where ΔP/Δx is the constant pressure gradient. The wall shear stress can be expressed in terms of Fanning friction factor f,

$$f = \frac{\tau_{\omega}}{\frac{1}{2}\rho {U_b}^2} \tag{5}$$

where U_b is mean velocity in the pipe and ρ is density of fluid. For instance, the expression of f for laminar and fully turbulent pipe flow of a Newtonian fluid is respectively,

$$f = \frac{16}{Re} \tag{6}$$

$$f^{-1/2} = 4.0 \log\left(Ref^{\frac{1}{2}}\right) - 0.4 \tag{7}$$

$Re = \rho U_b D/\mu$ is Reynolds number based on the constant viscosity, μ of the fluid.

For polymeric liquid, the viscosity is in general shear rate dependent, so that the usual Reynolds number cannot be used. The Reynolds number is based on viscosity at the pipe wall as obtained,

$$\mu_\omega = \frac{\tau_\omega}{\dot{\gamma}_\omega} \tag{8}$$

where $\mu_\omega = \mu(\dot{\gamma}_\omega)$ and $\dot{\gamma}_\omega$ is local shear rate at the wall.

At maximum drag reduction of non-shear-thinning fluids the friction law approaches an empirical asymptote. Maximum drag reduction asymptote is given by,

$$f^{-1/2} = 19.0 \log\left(Ref^{\frac{1}{2}}\right) - 32.4 \tag{9}$$

For shear-thinning fluids, it is showed that plotting of the friction factor vs. The wall Reynolds number collapses the data near Virk asymptote. The problem will not occur when Reynolds number based on viscosity of water is used.

The amount of drag reduction (percentage) is defined as the percentage reduction of pressure drop due to the addition of drag reduction agents at the same wall Reynolds number.

$$\%DR = \frac{\Delta P_b - \Delta P_a}{\Delta P_b} \times 100\% = \frac{f_b - f_a}{f_b} \times 100 \tag{10}$$

The suffices ‘a’ and ‘b’ stand for after adding additives and before adding additives.

The pressure loss in a pipe is due to fluid-frictional resistance can be classed in terms of laminar and turbulent flows by the fluid Reynolds number. In the engineering sense of the flow exceeding a critical Reynolds number (Re), turbulent flow can be defined:

which is for pipes,

$$Re = \frac{VD}{v} > 2300 \tag{11}$$

for an external flow such as over a ship hull,

$$Re = \frac{VD}{v} > 500\,000 \tag{12}$$

and for a rotating disc,

$$Re = \frac{VR}{v} > 250\,000 \tag{13}$$

where V is the flow velocity, R is the radius), D is the pipe diameter and ν is the kinematic viscosity of the drag reducing solution.

Drag reduction (DR) can be also expressed in terms of friction factor, *f*. The relationship between *f* and Re (Hoyt, 1986) can be expressed by,

$$f = \frac{64}{Re} \tag{14}$$

for the laminar regime, and

$$\frac{1}{(f)^{\frac{1}{2}}} = 2\, log_{10}\, Re\, (f)^{\frac{1}{2}} - 0.8 \qquad (15)$$

for the turbulent flow.

Fig. 2.5 shows a typical relationship between DR and poly(ethylene oxide) (PEO) concentration at a Reynolds number of 14 000 in a small pipe which indicates a flow in the turbulent regime.

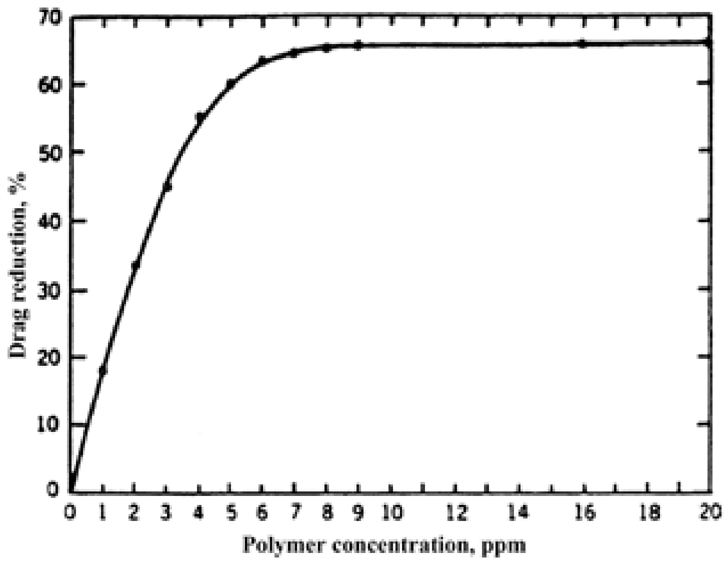

Figure 2.5: Drag reduction of poly(ethylene oxide) in water, at a Reynolds number of 14000, in a small pipe.

Source: Hoyt (1986)

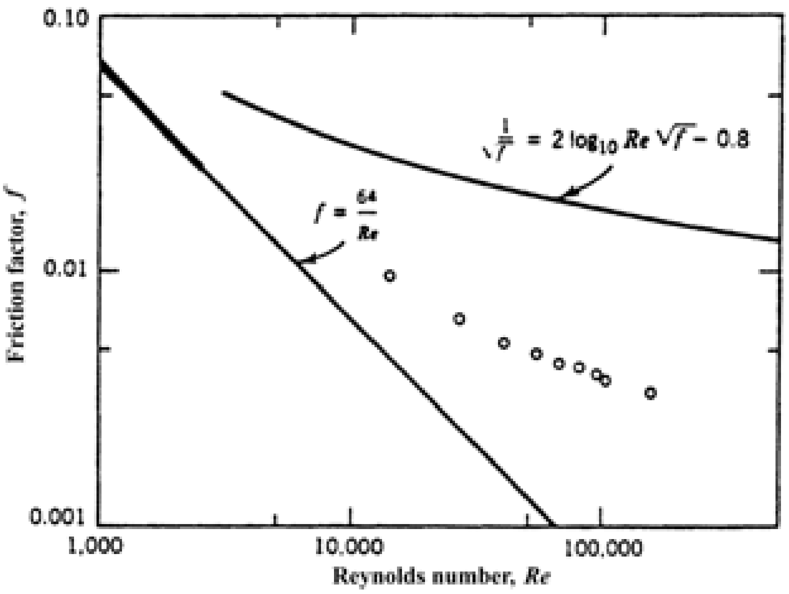

Figure 2.6: Typical data for drag reducing polymer solutions fall between the turbulent friction line for pipe flow, and the laminar line, 64/Re, extended beyond its usual limit of a Reynolds number of 2300, where f = pipe friction coefficient in engineering terms, equal to pressure drop per length times the diameter, divided by $\frac{1}{2}\rho V^2$, and ρ is fluid density.

Source: Hoyt (1986)

Fig. 2.6 shows that typical drag reduction data fall between solvent values for laminar flow and the curve for turbulent smooth pipe flow. The effect of drag reduction is to reduce the friction to a value considerably lower than the turbulent flow of the solvent, but not approaching that corresponding to laminar conditions.

2.3.2 Flow Patterns

There is a wide range of possible flow patterns or flow regimes. The distribution of each phase can be determined relative to the other phase based on flow pattern. Important physical parameters in determining the flow pattern are surface tension which keeps pipe walls always wet and which tends to make small liquid drops and small gas bubbles spherical and also gravity which tends to pull the liquid to the bottom of the pipe. Many researchers have attempted to predict the flow patterns that exist for various sets of conditions. In certain applications, for example in two-phase flow lines from offshore platforms to on-shore facilities, there is concern which is grown regarding the prediction of the flow pattern and also expected liquid slug sizes.

2.3.2.1 Vertical Flow Patterns

Common flow patterns for vertical upward flow are illustrated in Fig. 2.7.

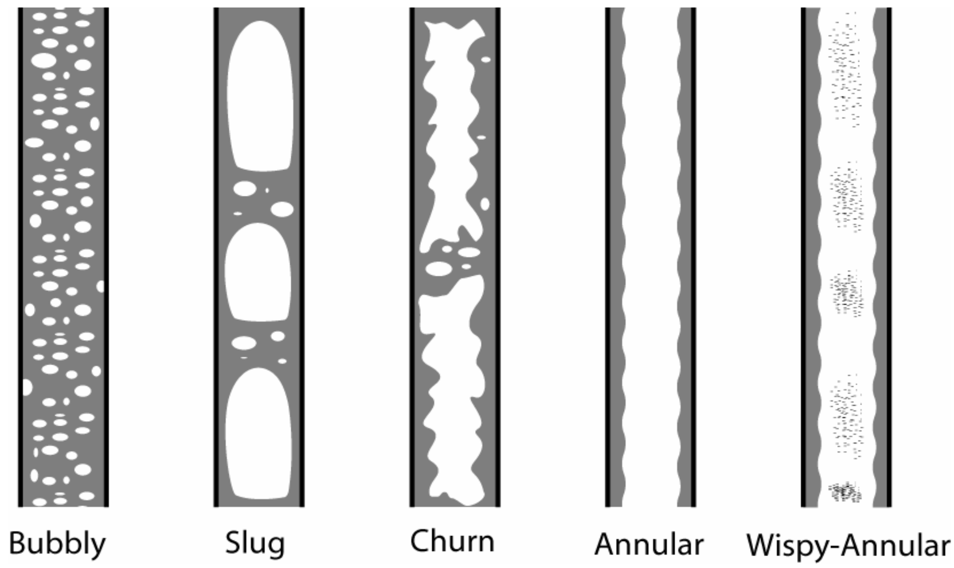

Figure 2.7: Flow patterns in vertical upward flow in a tube.

Bubbly flow is when the gas (or vapor) bubbles are of approximately in uniform size. Slug flow pattern occurs when the gas flows as large bullet-shaped bubbles and there are also some small gas bubbles distributed throughout the liquid. This flow pattern sometimes is called plug flow. Churn flow pattern occurs when highly unstable flow of an oscillatory nature and the liquid near the tube wall continually pulses up and down. Annular flow pattern is when the liquid travels partly as an annular film on the walls of the tube and partly as small drops distributed in the gas which flows in the center of the tube. Wispy-Annular flow phase forms as the liquid flow rate is increased in annular flow and the concentration of drops in the gas core increases, droplet coalescence in the core leads to large lumps or streaks (wisp) of liquid in the gas core. This flow pattern is a characteristic of flows with high mass flux and was proposed by Hewitt (1982). In addition, Froth is sometimes used to describe a very finely divided and turbulent bubbly flow approaching an emulsion, while on other conditions it is used to describe churn flow(Chisholm, 1973).

2.3.2.2 Horizontal and Slightly Upward Inclined Flow Patterns

The phases tend to separate in horizontal flow due to differences in density. The heavier (liquid) phase tends to accumulate at the bottom of the pipe. When the flow occurs in a pipe inclined at some angle other than vertical or horizontal, the flow patterns take other forms. In this situation, a form of slug flow can occur. The effect of gravity on the liquid precludes stratification. The common flow patterns for horizontal and slightly upward inclined flows in a round tube are illustrated in Fig. 2.8.

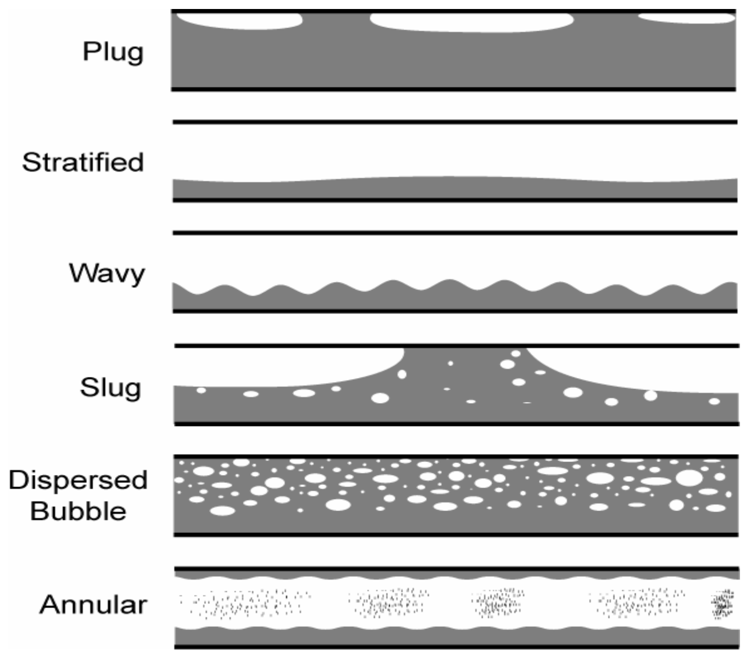

Figure 2.8: Flow patterns in horizontal and slightly upward inclined flow in a tube.

Plug flow phase is a phase where the individual small gas bubbles have coalesced to produce long plugs. Sometimes the flow pattern observed at very low flow quality, prior to the plug flow, is referred to as bubbly flow. In this situation the gas bubbles tend to flow along the top of the tube. Stratified flow phase which the gas-liquid interface is smooth. This flow pattern does not usually occur and the interface is almost always wavy as in wavy flow. Wavy flow is the wave amplitude increases as the gas velocity increases. Slug flow phase occurs as the wave amplitude is so large that the wave touches the top of the tube. Dispersed bubble flow occurs when many small gas bubbles are distributed uniformly across the entire tube cross section when the gas and liquid velocities are high. Annular flow occurs at similar to vertical annular flow except when the liquid film is much thicker at the bottom of the tube than at the top. Sometimes, the term intermittent flow is also to refer to the presence of plug and slug flows together.

2.4 DRAG REDUCTION

Skin friction drag is major factor to the total resistance in transportation systems moving in a fluid. More than 60% of a ship's total resistance can be contributed when Froude number is on the order of 10-1. As a result, skin friction reduction techniques have been investigated for several decades ago with used of active (polymer and gas injection) and passive (hydrophobic coatings) methods. Drag reduction by injection of air has attracted the interest the interest of many investigators over the years since it has been shown to produce drag reduction up to 80%.

Drag reduction can be formed when DRA is added into a fluid that can lead to a reduction in the pressure drop when the gas and liquid flow rates are fixed. By the end of the 19th century and beginning of the 20th century, (McCormick and Bhattacharya, 1973) found that drag reduction can occur by applying bubbles through the flow fluid. Net drag reduction approaching 40% at speeds flow of 2.6 m/s occur with bubbles produced by electrolysis near the leading edge of an axisymmetric body. Few years later, (Bogdevich & Evsees, 1976) found that observed drag reduction decrease while increasing the distance of injection point until original drag levels are achieved. Additional research found that bubble drag reduction (BDR) also sensitive to gas flow rates, free stream speed, plate orientation (buoyancy), injector geometry, surface tension and surface roughness.

Researchers at the Pennsylvania State University have been conducted BDR test in laboratory scale testing. These studies (Madavan et al., 1984; Clark & Deutsch, 1991;

Fontaine & Deutsch, 1992; Merkle & Deutsch, 1992; Deutsch et al., 2003) measured drag reduction on a small flat plate. Based on their findings, types of gas has minimal impact on BDR, injector pore size does not affect drag reduction (pore size from 0.5 to 100 μm), bubble size is most sensitive to free-stream velocity and gas injection rates, gravitational force can affect the performance of BDR and favorable pressure gradients can reduce BDR efficiently (Kawikita & Takano, 2000).

A major issue that has persisted in the study of BDR is about the bubble size. This is important to be studied because most of BDR experiments are conducted in fresh water and saltwater has been shown to be capable of reducing bubble diameters by approximately a factor of four (Winkel et al., 2004). Most studies have shown either small bubble are better (Kawamura et. Al, 2003) or the results are independent on bubble size (Takashi et al., 2001). Direct measurement of skin friction with gas injection into fresh saltwater have been made by Shen et al. (2006) and observed there is no significant change in the drag reduction. Shen et al. also used a method which was injection of lipid-stabilized bubbles that were on the order of 10 viscous wall units in diameter into the boundary layer. However, there is still no measureable improvement in drag compared to equivalent void fractions with large bubbles. From the result obtained, Shen et al. indicated that saltwater effects will either improve or not affect BDR.

A group of Japanese researchers have been conducted series of experiments on towed large scale ship models (12 to 15 m in length). They have done investigations on long, slender (0.6 to 1.0 m wide) flat-bottom ship models at speeds to 7 m/s (Watanabe et al., 1998; Kodama et al., 1999; Kodama et al., 2002). Span wise uniformity of the injected gas has been problematic with these studies due to air being injected over only the center 50% of the model span and no ‘skegs’ or ‘strakes’ used to contain gas underneath the model likely resulting in air escaping from beneath. On the 50 m long model, overall skin friction reduction approaching 20% has been reported as drag reduction decreased along the model until only few percent observed at its end. Then, the same group conducted some BDR experiments on a full scale vessel (over 100m in length). From the first trial at sea (Kodama et al., 2000), there was no net power savings observed. However, the lack of net power saving was suspected to be the result of gas entrainment into the propeller intake which offset any reduction in skin-friction drag. Since the first sea trial, additional work has been conducted (Nagamatsu et al., 2002; Kodama et al., 2006) and shown that net friction and power savings on the order of a few percent with gas injection. Unfortunately, the complexity such as scale, repeatability of sea conditions, surface curvature and roughness and difficulty in producing high fidelity measurements of fluid parameters at sea trial has prevent much information about either the gas distribution or the corresponding drag reduction. Thus, by applying air injection drag reduction on a ship, the groups offer little information on the physical mechanisms involving BDR. These problems stress the need for high Reynolds number testing which can close the gap between the small scale laboratory experiments and the full scale ship studies.

The first experiment aimed to close the gap was the high Reynolds number BDR experiment reported by Sanders et al. (2006), which was conducted on the same 12.9 m test model which is used in the current study. Drag reduction was measured directly on this near-zero pressure gradient turbulence boundary layer at Reynolds numbers over 108 based on downstream distance. Significant levels of drag reduction only present within the first couple meters of injection and that negligible drag reduction persisted downstream. BDR quite impractical for most real applications due to poor downstream persistence. Further investigation on the same model by Winkel (2007) and Elbing et al. (2008) has shown additional insight into BDR but significant improvements to downstream persistence have not been realized. However, it has been suspected that further insight into the physical mechanism governing BDR would allow for more efficient injection methods which can improve the result in downstream persistence.

The physical mechanism of BDR is still remaining unclear. However, generally agreed that the bubbles in some way reduce the exchange of turbulent momentum within the buffer region of the boundary layer, which can result in a drag reduction. Meng & Uhlman (1998) proposed that ‘bubble splitting’ could be responsible for the observed reduction in drag by arguing that energy is extracted as large bubbles split into two or more smaller bubbles which more small bubbles have larger surface area and thus higher surface energy. A reduction in bulk density is also thought to be a possible cause for drag reduction. Reduction in bulk density will decrease Reynolds shear stress in the near wall region. If the drag reduction is the product of a reduction in bulk density then the drag reduction should scale with the near wall void fraction.

Lumley (1973, 1977) hypothesized that bubbles will increase viscosity and thus reduced turbulent fluctuation levels and increased the thickness of the sub-layer and buffer region. The theory has been supported by Pal et al. (1989), which observed fluctuations reduction in skin friction is a factor to reduction in mean skin friction. However, Nagaya et al. (2001) observed increasing the turbulence fluctuations with gas injection is contra to the theory.

2.5 BENEFITS OF INCREASING DRAG REDUCTION PERFORMANCE

2.5.1 Fluid Flow

Pipeline flow rates of a system will be increased when less drag reduction occur in the pipeline. In order to deliver more hydrocarbon fluid in the pipeline especially in oil and gas industry, the effect of gas flow rate inside the system can be one of the subjects that need be focused. Usually, there are many pipelines are used to carry out the crude oil from the offshore. By decreasing friction between the fluid flow and the wall of the pipe can assure the fluid flow inside the pipe will be at optimum rate. It can also be applied when the fluid is transferred to the tanker and it may increase the tanker loading rate.

2.5.2 Pressure Reduction

Pipeline operations usually depend on pumping pressure as it is a lifeline to deliver flow capacity. Other than friction between the fluid flow and the pipe wall inside the pipeline, there are many factors such as aging systems, corrosion, abrasions and also pipe line bottleneck that can affect pressure constrains. Many researchers and scientists focus on this matter to manage pressure constraints and enable typical capacity at lower pressures as it has been recognized can make the cost of operation at low value.

2.5.3 Energy Consumption

Reducing the drag reduction in the pipeline will make less energy to be used to maintain the flow rate. This show an innovative approach that allowed the operation to bypass half of the pumping stations on the system and significantly lower the overall energy use. The energy hedge delivers a significant cost savings each year, reduces maintenance and repairs costs and provides a green alternative of energy usage.

CHAPTER 3

METHODOLOGY

3.0 INTRODUCTION

Various industries have their objectives to increase pipeline capabilities, production rates and able to produce faster transportation. However, internal friction between the fluid inside the system and pressure should be reduced to accomplish the purpose. The importance to decrease pressure gradient in pipeline or system may reduce pump horsepower. It is very importance from economic viewpoint. In this experiment, the effects of drag reduction in three phase flow system are studied by controlling fluid and solid parameters between the walls of pipe. The testing apparatus of drag reduction is build-up to measure the pressure loss when fluid passes through the pipe.

3.1 MATERIALS

The experiment consists of three phases flow system which involve solid, gas and liquid. Sand is easily to get around the university area. The sand with different size and concentration will be used as solid medium. Compressed air which is supplied via the pipeline in the FKKSA laboratory is to be used as gas medium and tap water is to be used as liquid medium. Tap water with approximately 450 kg is used for one circulation system.

3.1.1 General

The experimental works is carried out in the 1.5 inch pipe diameter. The sand is sieved to get different sizes (71 µm and 315 µm) to be used in the experiment. The sand is mixed together with tap water in the tank and passes through the pipeline. At one point of the pipeline, by using a orifice tube type device, compressed air is supplied to the solution (sand and tap water). The flow rate of liquid is adjusted by controlling the valve on the system. Pressure loss data according to the time are detected and recorded by pressure transmitter which can transmit the pressure data to the computer system.

3.2 RESEARCH METHODS

The sand is used as solid medium in this experiment and it is easily to get around the university area. The preparations to get different size of sand particles are done at the open FKKSA laboratory. Availability of air and change in flow rate of tap water via the pipeline in laboratory is considered during the experiment at the experimental rig.

3.2.1 Preparation of Different Sand Size

Sieve shaker machine is used to sieve the sand to attain fine particle size of 71 µm and 315 µm.

Figure 3.1: Sieve Shaker

3.2.2 Experimental System

Fluid friction process plant as shown in Figure 3.2 is used to carry out the experiment. The pipe inner diameter is 0.0381 m and made from transparent PVC to ensure the flow pattern can be observed during the experiment. The pipe (PT100) diameter is 0.0381 m and consists of pressure transmitter (PT-101 ~ PT-105) to detect pressure loss at four test sections. The first test section is installed quite distance from second test section to ensure turbulence flow has been generated in the pipeline. Valves (MV-100 and MV-101) are used to control the flow rate of liquid pass through the pipe. Centrifugal pump (P101) is connected to the entrance of the pipe and the other pipe which is used to connect the reservoir tank (TK-100) and drainage pipe. The system is closed loop pipeline design.

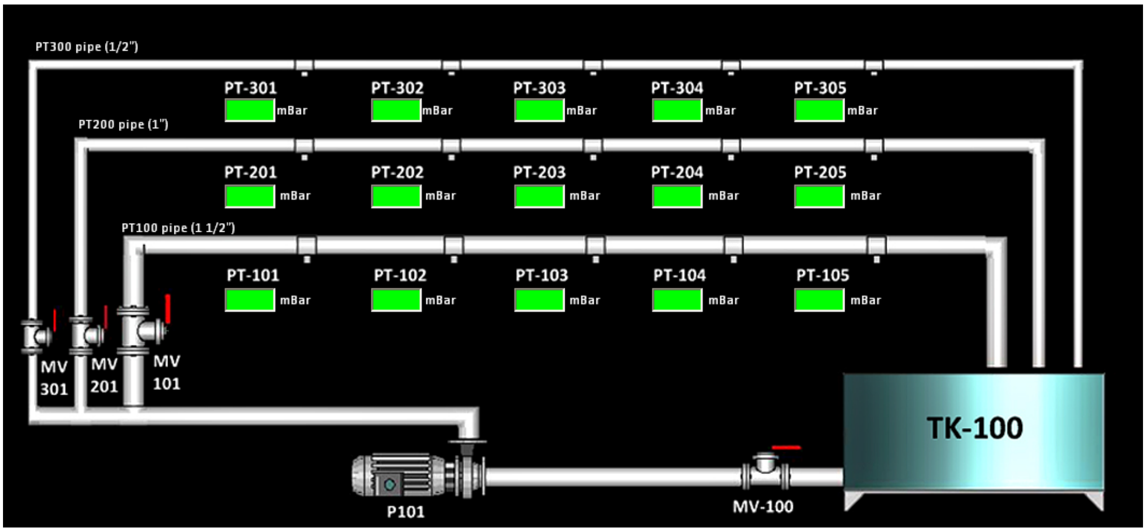

Figure 3.2: Fluid friction process plant

An ultrasonic flow meter, Ultraflux Portable Flow Meter Minisonic P which has accuracy up to 0.001 cubic meter per hour is used to indicate the volumetric flow rate of fluid flow in the pipeline system. The purpose of using exterior portable ultrasonic flow meter is to avoid any disturbance that may occur to the fluid flow.

A compressed air pipe is connected to the pressure regulator (0~1.2 psi) which is installed at side of pipeline system. The regulator is connected to the custom-made