CFD study on Energy Promoter insertion for drag reduction in oil pipelines.

by

Teow Jing Min

14967

Dissertation submitted in partial fulfilment of

The requirements for the

Bachelor of Engineering (Hons)

(Mechanical)

JANUARY 2015

Universiti Teknologi PETRONAS

32610 Bandar Seri Iskandar

Perak Darul Ridzuan

TABLE OF CONTENTS

TABLE OF CONTENTS i

LIST OF FIGURES iii

LIST OF TABLES v

CERTIFICATION OF APPROVAL vi

CERTIFICATION OF ORIGINALITY vii

ABSTRACT iii

ACKNOWLEDGEMENT iv

CHAPTER 1: INTRODUCTION 1

1.1 Background of Study 1

1.2 Problem Statement 2

1.3 Objectives 3

1.4 Scope of Study 3

CHAPTER 2: LITERATURE REVIEW 4

CHAPTER 3: METHODOLOGY 8

3.1 Project Flow 8

3.2.1 Solidworks 9

3.2.2 ANSYS CFX 9

3.3 Proposed Design of Energy Promoter 10

3.4 Design of Pipelines 11

3.5 Gantt Chart 12

3.5.1 FYP I 12

3.5.2 FYP II 13

CHAPTER 4: RESULT AND DISCUSSION 14

4.1 Meshing 14

4.2 Mesh Independency Study 15

4.2.1 Without Energy Promoter 16

4.2.2 With Energy Promoter 18

4.3 Verification and Validation 19

4.3.1 Pressure Drop 20

4.3.2 Entrance Region 23

4.3.3 Discussion 23

4.4 Parametric studies on Drag Reduction Efficiency of Energy Promoter 24

4.4.1 Effect of Height of Energy Promoter on Drag Reduction Effect in Pipeline 24

4.4.2 Effect of Direction of Energy Promoter on Drag Reduction Effect in Pipeline 28

4.4.3 Effect of Number of Energy Promoter on Drag Reduction Effect in Pipeline 31

4.4.4 Effect of flow rate on Drag Reduction Effect in Pipeline 34

4.5 Velocity Contour and Velocity Vector 35

CHAPTER 5: CONCLUSION & RECOMMENDATION 37

5.1 Conclusion 37

5.2 Recommendation 38

REFERENCES 39

LIST OF FIGURES

Figure 1 Flow Behaviour With and Without Drag Reduction Additives..................... 4
Figure 2 Volume-based and surface-based models of compliant surfaces by Kramer 7
Figure 3 Project Flow Diagram... 8
Figure 4 Detailed Drawing of Energy Promoter 10
Figure 5 Conceptual Pipeline Design... 11
Figure 6 FYP I Gant Chart 12
Figure 7 FYP II Gantt Chart.. 13
Figure 8 Mesh Generated by ANSYS CFX 14
Figure 9 Graph of Pressure Loss per Length versus Number of Elements 16
Figure 10 Graph of Percentage Error versus Number of Elements 17
Figure 11 Graph of Pressure Loss per Length versus Number of Elements 19
Figure 12 Graph of Percentage Error versus Number of Elements 19
Figure 13 Graph of Pressure Drop per Length versus Location of Pipe 21
Figure 14 Graph of Theoretical Pressure Drop per Length versus Simulation Pressure Drop per Length 22
Figure 15 Graph of Velocity versus Location... 23
Figure 16 Effect of Height of Energy Promoter with normal direction on Drag Reduction Effect in pipeline.. 25
Figure 17 Effect of Height of Energy Promoter with reverse direction on Drag Reduction Effect in pipeline.. 26
Figure 18 Effect of Direction of Energy Promoter with Height=1mm on Drag Reduction Effect in pipeline.. 29
Figure 19 Effect of Direction of Energy Promoter with Height=2mm on Drag Reduction Effect in pipeline.. 30
Figure 20 Effect of Number of Energy Promoter on Drag Reduction Effect in pipeline 32
Figure 21 Correlation between Reynolds Number a and Percentage of Drag Reduction 34

Figure 22 Velocity Contour of Pipeline without Energy Promoter 35
Figure 23 Velocity Contour of Pipeline with Energy Promoter in Normal Direction36
Figure 24 Velocity Contour of Pipeline with Energy Promoter in Reverse Direction
.... 36

LIST OF TABLES

Table 1 Data of Mesh Independency of 20m Pipe Without Energy Promoter 16

Table 2 Data of Mesh Independency of 20m Pipe With Energy Promoter 18

Table 3 Data of Pressure Drop of 20m Pipe without Energy Promoter..................... 22

Table 4 Data of Theoretical Pressure Drop per Length versus Simulation Pressure Drop per Length 22

Table 5 Effect of Height of Energy Promoter on Drag Reduction Efficiency in pipeline at different conditions.. 27

Table 6 Effect of Direction of Energy Promoter on Drag Reduction Efficiency in pipeline at different conditions.. 30

Table 7 Effect of Number of Energy Promoter on Drag Reduction Efficiency in pipeline at different conditions.. 33

Table 8 Percentage of Drag Reduction under different Reynolds Number 34

CERTIFICATION OF APPROVAL

CFD study on Energy Promoter insertion for drag reduction in oil pipelines.

by

Teow Jing Min
14967

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
In partial fulfilment of the requirements for the
Bachelor of Engineering (Hons)
(Mechanical)

Approved by,

____

(AP Dr. Hussain H. Al Kayiem)

Universiti Teknologi PETRONAS
32610 Bandar Seri Iskandar
Perak Darul Ridzuan

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 o done by unspecified sources or persons.

(Teow Jing Min)

ABSTRACT

In oil and gas industry, transportation of crude oil from terminal to terminal has cost companies enormous amount of money in order to restore back pressure which is lost due to the inner surface friction of the pipeline through pumping operation. In this study, it proposed that a small section of pipeline with energy promoter is added to the pipeline in order to achieve drag reduction effect. CFD Simulation was used to study the drag reduction at various number and configurations of energy promoters. Mesh independence study was conducted to ensure the integrity of the result. The dimension and shape of the energy promoter to learn the relationship between the variable and the drag reduction percentage. ANSYS CFX was used to simulate the flow inside the pipeline with a section of energy promoter embedded at the inner wall. The pipeline with energy promoter is modelled using Solidworks and imported to ANSYS CFX Fluid Flow to undergo simulation. The results obtained were compared with the empirical table to ensure the validity of the simulation procedure. Pressure loss at the outlet will become the parameter to be compared in the case of pipeline with energy promoter and energy promote. Through CFD study, drag reduction effect has been discovered with insertion of Energy Promoter and encouraging results are obtained when the Energy Promoters are arranged in reverse direction with 2mm height. The maximum drag reduction efficiency of energy promoter is approximately 7% and it is possible to further push the boundary for drag reduction efficiency limit. In summary, it is feasible to reduce drag in flow through insertion of energy promoter.

ACKNOWLEDGEMENT

In completion of this Final Year Project, I would like to express his utmost gratitude to all parties who have made my Final Year Project a milestone in the learning path.

Firstly, I would like to extend my immense appreciation to my Final Year Project supervisor, Dr. Hussain H. Al Kayiem, associate professor of Mechanical Engineering Department, Universiti Teknologi PETRONAS. His continuous support and guidance help me through all the obstacles I faced in the project. The encouragement from him are the reason why I continue to explore more on field.

Secondly, I would like to thank Mr Javed, a PhD. Student from Mechanical Engineering Department. He is willing to share with me the knowledge he has on simulation software and whenever I had any problems, he is willing to spend his time and brainstorm it together with me. I also like to thank my friend, Victor Tang who always provide valuable suggestion to me.

Last but not least, I would also like to thank my beloved family member for the endless support and motivation.

Thank you so much.

CHAPTER 1

INTRODUCTION

1.1 Background of Study

The first pipelines were introduced in United States in 1859 to facilitate the transportation of crude oil. After one and a half century of pipeline operating practice, it is proven that the pipeline is far more efficient of huge scale transportation of crude oil as well as natural gas compared to conventional means of transportation such as rail and truck that is to be moved on regular basis. (Pipeline Purpose and Reliability, 2014) Rapid development and enormous population growth has increased the demand for petroleum exponentially that will also increase the demand for petroleum pipelines. Typically, the oil is transported from one place to another through the pipelines by a series of pumping stations which usually located at every 50km of pipelines. The pumping stations are needed in the transportation process as the pump is to increase the pressure back in the pipeline due to friction. (Guo et al., 2005).

There are three types of fluid flow, i.e. laminar flow, transitional flow and turbulent flow. The oil flow in the pipelines is preferably turbulent as Sutherland et al. (2009) stated that turbulent oil flow in pipelines has several merits over laminar flow. First, the rate of building up of the deposited material will be reduced as it will be scrubbed away from walls. Second, due to constraint of costs, each grade of oil will be transported in batch by pipelines which will be used repeatedly and the turbulent flow will ensure less mixing of batches of oil compared to the laminar flow. Despite the advantages, in turbulent flow, the fluid behaves as if its viscosity is increased and this results in rise of drag in the flow. However, in turbulent flow, one thing that must be taken into account is the formation of eddies in pipelines which will be causing the output flow rate is relatively lower compared to the input flow rate. Moreover, drag is caused by the friction between the fluid and the pipe wall resulting in pressure drop.
Several of researches have been carried out aiming to improve the flow by reducing the drag. For example, additives, polymeric material, surfactants are added inside the

fluid to alter the viscosity and other fluid properties. However, extra cost will be incurred to remove the drag reducing agents from the material.

In this research, CFD simulation of the oil flows through protrusion ring that is installed to the inner layer of oil pipelines will be conducted and how the protrusion ring affects the flow structure will be studied in details.

1.2 Problem Statement

In oil and gas industry, crude oil is usually required to be transported across long distance and high amount of energy is required to counter the pressure loss due to the friction in the pipelines which will decrease the throughput. After several decades of studies, scientists have come out with several methods which can increase the percentage of drag reduction up to 60% and eventually the cost will be reduced drastically. Adding drag reducing agents, additives, polymers into the oil to achieve drag reduction is practiced a lot in oil and gas industries due to the high performance on drag reduction. However, it is learnt that additional cost and energy is incurred just to perform separation process at the end. Furthermore, it might change the environment unnecessarily. Therefore, a study on how the protrusion ring change the flow structure will be conducted and it is believed that the insertion of protrusion ring is able to save enormous of energy and money from being wasted on the separation process.

1.3 Objectives

The objective for the project are as follows:

1. To investigate the possibility of reduce the pressure drop in pipe flow by Energy Promoter embedded at the inner wall of the pipeline.
2. Perform computational simulation to study the effect of Energy Promoter on drag reduction in horizontal pipe flow.
3. Develop dimensionless parameters to correlate the Protrusion configurations and the percentage of drag reduction.

1.4 Scope of Study

The scope of study for the project follows:

1) The flow is assumed turbulent within Re range of 66482 to 332410 which is corresponds to velocity range of 0.3m/s to 1.5m/s in pipelines with 0.2m diameter.
2) The Pipe diameter is adopted from industrial standards, which is equal to 8in (0.2m).
3) The simulated segment is considered the entry length, the region where energy promoter to be inserted after the fully developed flow.
4) The water, as single phase flow, is considered for the validation and investigation as the properties of water is already well established.
5) The enhancement, or drawback results of the case of the elliptical cross section Energy Promoter are presented in terms of percentage of Drag Reduction (DR%).

CHAPTER 2
LITERATURE REVIEW

The addition of polymer additives to the flow that can reduce the turbulences friction significantly was initially observed by Toms (1949). Even when minute amount of polymer additives is applied to the flow, the drag can be reduced drastically. In laminar flow, the viscosity of the flow could be increased which is caused by the dissolved polymers and hence, the drag is increased instead of decreased (Diamond et al, 1992). Polymer additives, which is also known as drag reducing agent has been applied widely to daily life application due to its drag reducing nature, i.e. oil pipelines, oil well operations, airplane tank fuelling. Without addition of drag reducer in airplane tank fuelling operation, it would take up as much as twice time that with addition of drag reducer (Brostow, 2008). However, polymer additives has its own shortness. In order to reduce drag in flow, it changes the physical and also chemical properties of the flow and this is totally unacceptable in some industries such as pharmaceutical and food industry which requires the properties of the fluid unchanged to prevent any undesirable side effects to human bodies and environments. Despite its remarkable performance, the polymer additives in the flow also undergo mechanical degradation. This phenomenon is due to the polymeric chain that is playing the main role in drag reduction has undergone scission process caused by turbulence flow and hence, the percentage of drag reduction will reduce. Therefore, it is necessary to reintroduce the polymer additives into the flow in order to maintain the desirable drag reduction.

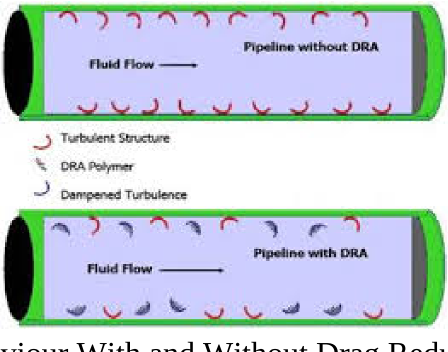
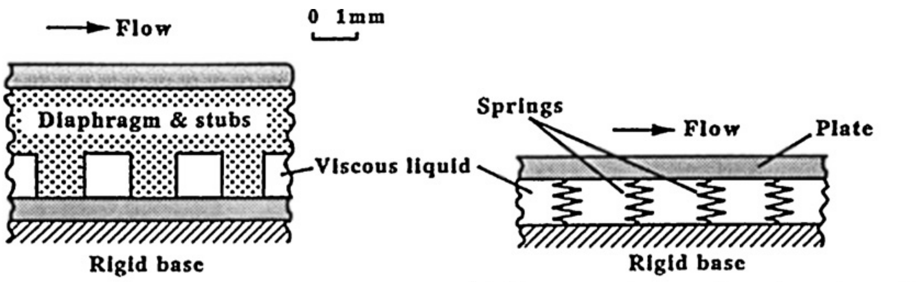


Figure 1 Flow Behaviour With and Without Drag Reduction Additives

Riblets are generally longitudinal microgrooves that etched onto the wall surface. The application of riblets in drag reduction is inspired by the shark's skin which is made up of streamlined V-shaped tooth like grooves. It is learnt that this tiny groove has greatly reduce the drag when this predator is moving underwater and allows it to hunt with less effort. The V-shaped grooves is varied in shape at each parts of sharks which is to optimize the drag reducing effect. Application of this concept on drag reduction technologies needs significant modification but the concept remains the same.There are several theories proposed to explain the mechanism of how riblets reducing the drag. Choi (1985) proposed that the riblets induce restrictions to the span wise movement of quasi-streamwise vortices which leads to shorter period for premature bursts and with reduced intensity and it is believed that the burst plays the main role in Reynolds Number shear stresses. During the burst, the slow speed streaks near the wall region moving toward the center of the flow and resulting in high speed fluid splatter against the wall. On the other hand, Park and Wallace (1993) suggested another theory that has been proposed claimed that the drag reduction by riblets is achieved through viscous interaction. The reduction effect is achieved by increase of viscosity which is using the same concept of polymer additives but with smaller effect. Several researchers claimed that riblets are able to reduce the drag from 7% up to 10% and it is relatively low compared to the drag reduction performance by polymer additives as in some cases, performance of DRA is reported 4 to 8 times higher than performance of riblets. It is found that when riblet spacing has increased more than the threshold limit the drag will increase. Progress in computational studies has introduced a variety of parameters which make comprehensive experimental studies more difficult. Currently, research in the field is trying to push 11% drag reduction limit by introducing innovative riblet design instead of classic designs. The compound geometries has spanwise variation in multiple parameters yet maintaining streamwise uniformity. Moreover, the cost of reconstructing the inner surface of the pipelines is considered ridiculously high and once the low performance barrier of riblets could be overcome, the cost of restructuring inner surface of pipelines would be justified as it is permanent solution to solve the drag issue. (Brostow, 2008)

Same as riblets, compliant surface is considered as one of the non-intrusive as well as passive control of drag reduction method. Compliant surface is made up of elastic walls and it was first discovered by Kramer (1960) in experiment to simulate the drag reduction nature of dolphin's skin. Kramer (1960) claimed that under specific condition, compliant surface is able to reduce the drag up to 60%, however, the sensitivity to the pressure gradients is very high and the results produced is not consistent. It is observed that the transition period from laminar flow to turbulent flow would be delayed with huge factor and it is possible to achieve drag reduction that Kramer has accomplished before. While it is good for application in marine vehicles, it is not feasible to have drag reduction effect in pipe where the flow is already fully turbulent flow. Nevertheless, compliant surface has exhibited possibility to modulate flow noise and prevent boundary layer separation. Compliant surface included an inner skin, outer diaphragm and stubs all made of natural rubber with fluid in cavity and consecutive experiments showed that huge range of drag reduction efficiency and it might be due to slightly different conditions. It is seemed that Kramer's compliant surface could not perform well at high Reynold's number. Despite the several claims by other researchers, Xu et al. (2003) stated that by prolong the averaging interval more than 700 viscous times, the drag reduction phenomenon will be starting to fade and he claimed this as apparent drag reduction. This method is rarely applied due to the complication of the experimental set up and also slightly higher drag reduction compared to what can achieve with riblets.



(a) Kramer's coating. (b) Plate-spring surface-based model.

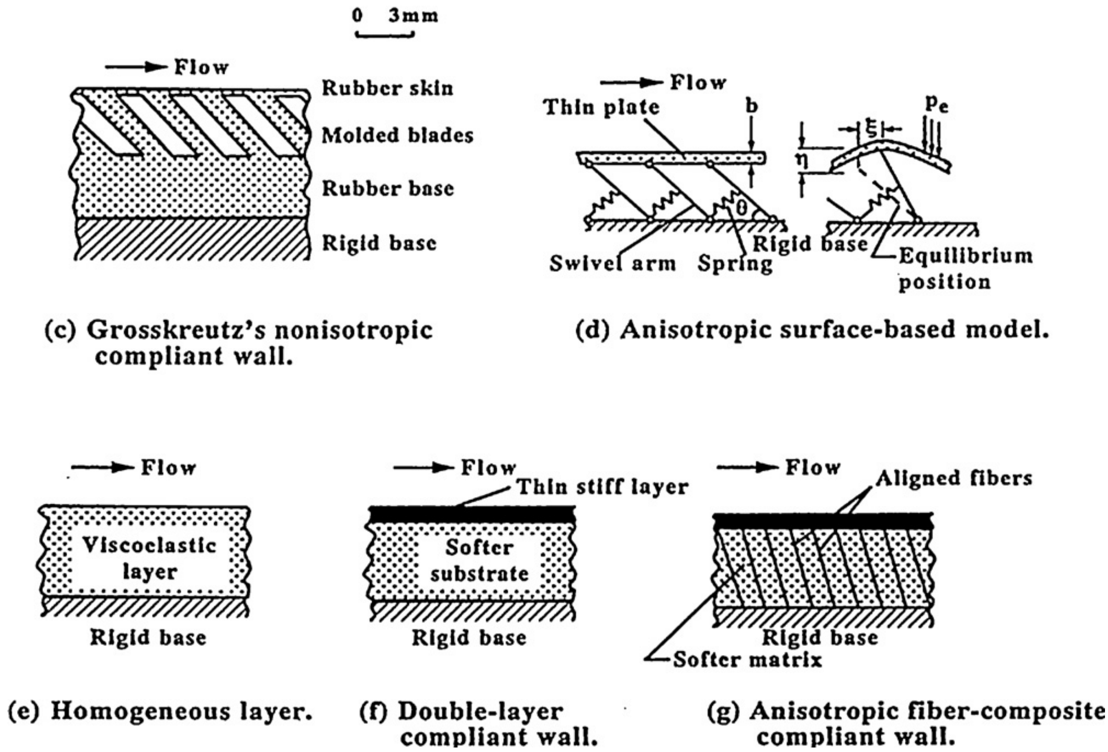


Figure 2 Volume-based and surface-based models of compliant surfaces by Kramer

Summary

The non-additives methods of drag reduction as discussed before could not perform as well as addition of polymer additives in the flow. Researchers all around the world are seeking a permanent solution to solve the pressure loss in pipelines due to surface friction instead of the polymer additives which alters the physical as well as chemical properties of the fluid that might cause negative effect to human body and environment. In this project, the author is to study the effect of insertion of protrusion ring in the pipelines to the drag reduction of the flow as the protrusion ring is to change the structure of the flow from laminar to turbulence. This research aims to provide the industry a new insight on drag reducing in pipelines that can save enormous of energy wasted to overcome the drag in fluid transportation.

CHAPTER 3
METHODOLOGY

3.1 Project Flow

Figure 3 is the flow chart for FYP I and FYP II.

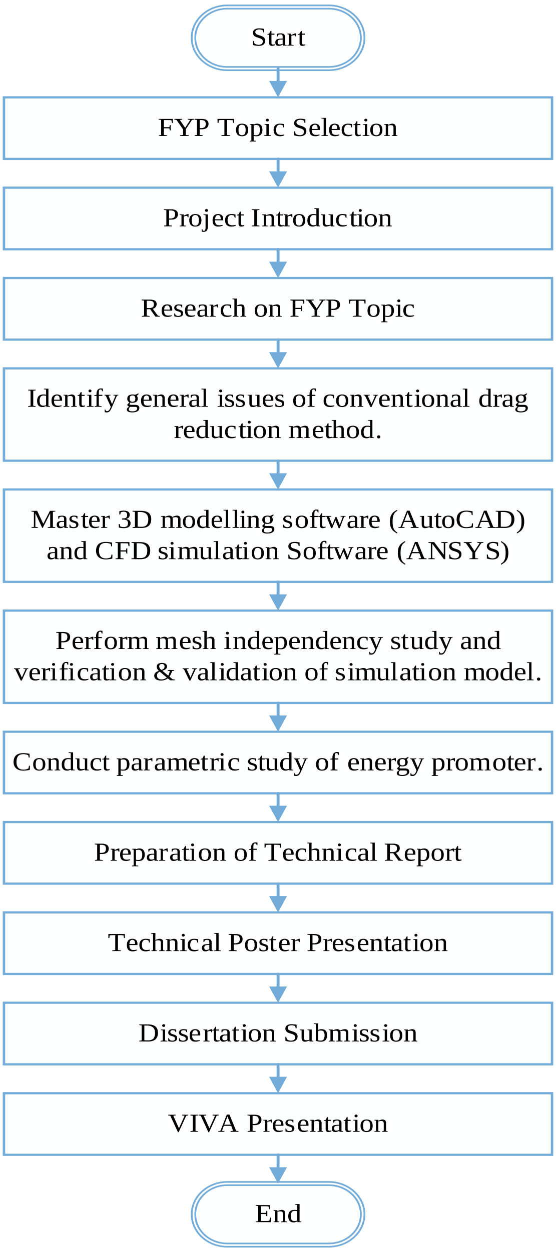


Figure 3 Project Flow Diagram

3.2 Computational simulation

In this project, two types of software which are CAD drawing software and simulation software are required in order to carry out the task. The details of the project are explained as following:

3.2.1 Solidworks

Solidworks is a 3D Mechanical CAD software which is Microsoft Windows base and it is developed by Dassault Systemes Solidworks Corp. It is a parametric solid modeller widely used for 3D modelling. Parametric means that the relationship can be defined between one and another and if a changes made, the software will change each parameter of the objects automatically.

In this project, Solidworks is used to model the energy promoter as well as the pipelines which are to be further studied in CFD Simulation. Solidworks is preferred over AutoCAD in this project as the model generated by AutoCAD is not lean compared to the one generated using Solidworks and it matters much in CFD Simulation which will save a lot of time for pre-processing and obtain a more accurate result.

3.2.2 ANSYS CFX

ANSYS CFX software is integrated into ANSYS Workbench which can provide superior dual connections to all major CAD systems. The geometry created in Solidworks will be imported into ANSYS CFX with the format of .IGS and ANSYS CFX will be used to do the meshing of the model, define the boundary layer and perform iteration in order to simulate the fluid flow across the pipe and the result could be plotted inside the chart section and exported to excel file for further analyzing purpose.

3.3 Proposed Design of Energy Promoter

Figure 4 is the detailed drawing of Energy Promoter.

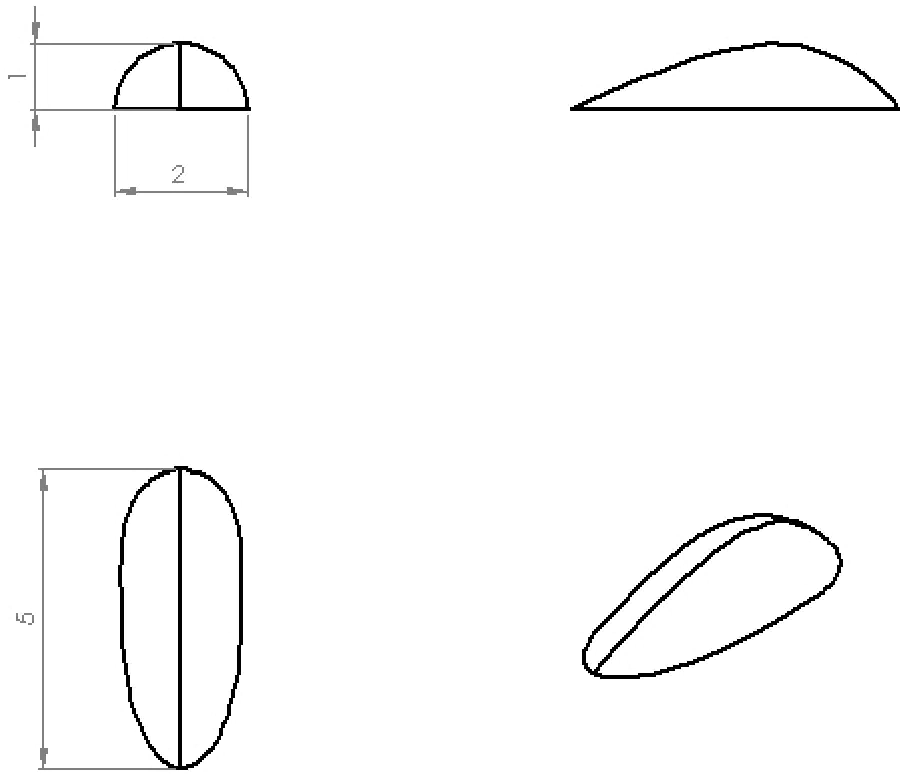


Figure 4 Detailed Drawing of Energy Promoter (dimensions in mm)

3.4 Pipelines configuration

Figure 5 is the conceptual drawing of the pipeline with blue section is where the energy promoter will be located

Length of Pipelines Without Energy Promoter	19800 mm
Length of Pipelines With Energy Promoter	200 mm
Inner Diameter of Pipelines	200 mm
Thickness of Pipelines	10 mm

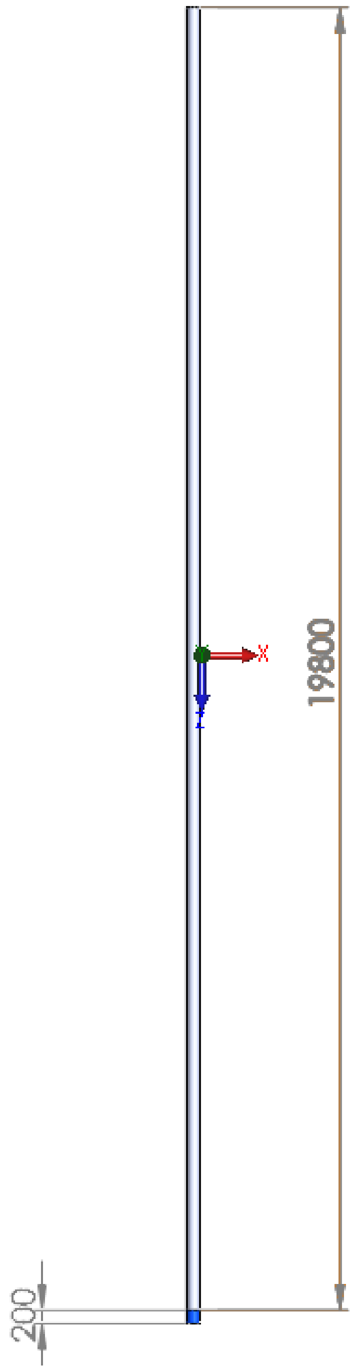


Figure 5 Conceptual Pipeline Design

3.5 Gantt Chart

3.5.1 FYP I

		Semester 1 (FYP 1)													
No	Subject	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	FYP Topic Selection														
2	Project Introduction														
3	Research on FYP Topic / Identify General Issues of Conventional Drag Reduction Method														
4	Master 3D modelling software (AutoCAD) and CFD simulation software (ANSYS)														
5	Modelling of oil pipeline with simple protrusion ring configuration														
6	Perform mesh independency study and verification & validation of simulation model.														

Figure 6 FYP I Gant Chart

3.5.2 FYP II

		Semester 2 (FYP 2)													
No	Subject	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1	Simulation of pipe flow/ Varying protrusion ring dimension						◆								
5	Preparation of Technical Report														
6	Technical Poster Presentation														
7	Dissertation Submission														
8	Viva Presentation														◆

Figure 7 FYP II Gantt Chart

CHAPTER 4
RESULT AND DISCUSSION

4.1 Meshing

Figure 8 shows Face sizing option has been inserted to improve the mesh and the element size for the mesh is set as 0.008 m and the total amounts of the elements generated is 2503689. The quality of the mesh generated in this case is considered good as the aspect ratio is more than 1 which is 1.86, the Jacobian Ratio is 1 and the skewness is less than 0.25 which is 0.2316, however, it cannot be concluded that the mesh will provide accurate result as the status of mesh independent is not yet confirmed. Hence, a mesh dependency study is to be conducted to determine the appropriate meshing size of the elements.

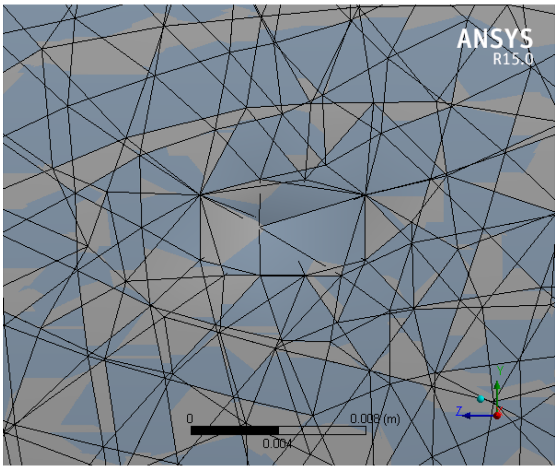


Figure 8 Mesh Generated by ANSYS CFX

Elements	2503689
Aspect Ratio	1.86
Jacobian Ratio	1
Skewness	0.232

4.2 Mesh Independency Study

In order to achieve mesh independence which is to ensure the accuracy of the simulation is close to the reality as the number of elements of the model might be causing huge percentage error to the results, the number of elements of the fluid in pipe is varying and the results obtained is tabulated. Pressure drop per length across 20m pipeline will serve as the main study case for mesh independency study as it is the main focus in the project and it is believed that the pressure drop against number of elements will either be changing in positive or negative trend drastically and until certain point, it can be observed that the trend is changing in the rate which can be ignored and mesh independent study is considered done. Any larger number of elements beyond that point will not have any significant effect on the accuracy and consistency of the simulation results.

In this project, it is required to compare the result between pipe without energy promoter and also pipe with energy promoter and thus, it is necessary to conduct two different set of mesh independent study for both pipe with energy promoter and without energy promoter in order to make sure correct setting of mesh is used in each case so that the result with high accuracy could be obtained. For the pipe without energy promoter, only one variable is being manipulated which is the surface sizing for the wall and on the other hand, for the pipe with energy promoter embedded that the inner surface of pipe, there are two variable are being manipulated which are the surface sizing for the wall and the surface sizing for the energy promoter.

Once mesh independent study for both cases is conducted successfully, the project will be proceeded in to next stage which is the validation of the simulation results that will be discussed in detailed later on.

4.2.1 Without Energy Promoter

A pipe model without energy promoter is created using CAD drawing software and is then imported to CFX simulation software to be conducted CFD study. By changing the element size of the energy promoter from the range of 0.035 to 0.025mm, the pressure loss per length is tabulated as shown in Table 1 and the percentage error for each cases is calculated and included in the table as well.

Table 1 Data of Mesh Independency of 20m Pipe without Energy Promoter

Number of Elements	ΔP/ΔL	[ΔP/ΔL]o-[ΔP/ΔL]n	Percentage Error (%)
16400	0.197	0	0
295924	0.188	0.009	4.569
396400	0.185	0.003	1.596
495500	0.184	0.001	0.541
543500	0.184	0.0003	0.163
679375	0.184	0.0001	0.054

From Figure 9, it is observed that the trend of the graph is decreasing drastically from 100000 elements to 700000 elements and it is starting to get stabilized from 500000 elements onwards. The percentage error of 679375 elements compared with the previous set of data is 0.05%. It can be assumed that as long as number of elements of mesh is more than 600000, mesh independent study for pipe without energy promoter is achieved.

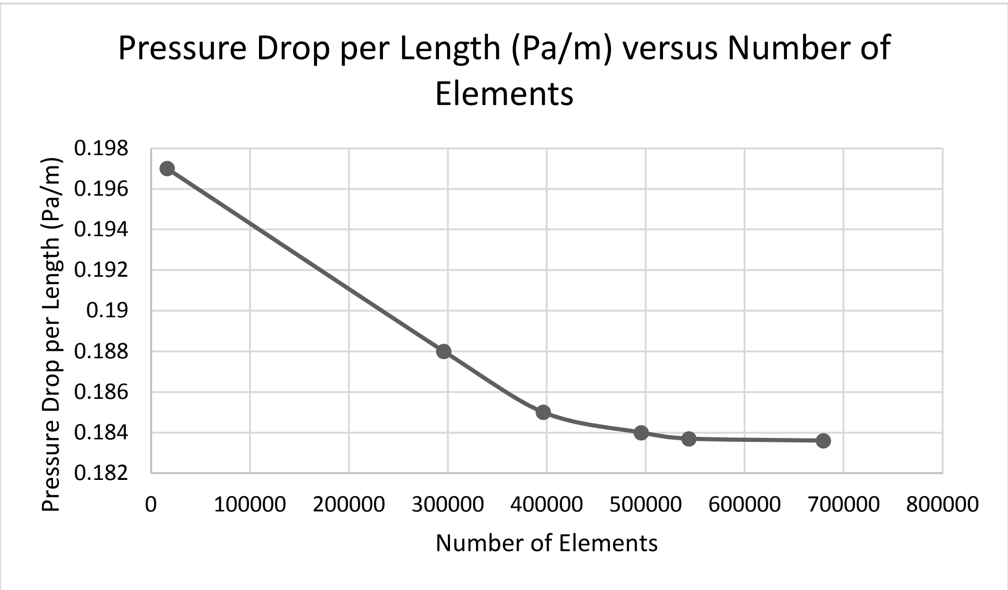


Figure 9 Graph of Pressure Loss per Length versus Number of Elements

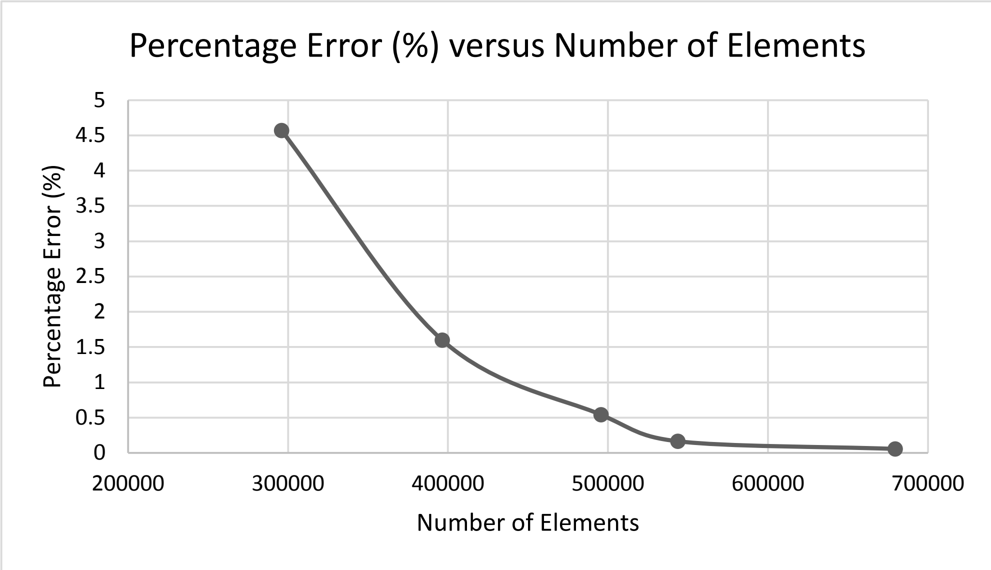


Figure 10 Graph of Percentage Error versus Number of Elements

4.2.2 With Energy Promoter

A pipe model with four energy promoter is created using CAD drawing software and is then imported to CFX simulation software to be conducted CFD study. By changing the element size of the energy promoter from the range of 0.001mm to 0.0001mm, the pressure loss per length is tabulated as shown in Table 2 and the percentage error for each cases is calculated and included in the table as well.

Table 2 Data of Mesh Independency of 20m Pipe with Energy Promoter

Number of Element	ΔP/ΔL	[ΔP/ΔL]o-[ΔP/ΔL]n	Percentage Error (%)
1000000	6.33	0	0
2000000	5.144	1.186	23.056
3000000	4.235	0.909	21.464
4000000	3.727	0.508	13.630
5000000	3.452	0.275	7.966
6000000	3.278	0.174	5.308
7000000	3.152	0.126	3.997
8000000	3.053	0.099	3.243
9000000	2.977	0.076	2.553
10000000	2.917	0.06	2.057
11000000	2.87	0.047	1.638
12000000	2.83	0.04	1.413

From Figure 11, it is observed that the trend of the graph is decreasing drastically from 1000000 elements to 6000000 elements and it is starting to get stabilized from 8000000 elements onwards. The percentage error of 120000000 elements compared with the previous set of data is 1.413%. It is safe to declare that with number of elements which is 12000000 or beyond, mesh independency study for pipe with energy promoter is accomplished. Furthermore, the approximated computation time for number elements around 12000000 is 2 hours and hence, the selection of 12000000 as the baseline for mesh independent study is reasonable as the percentage error is virtually small and the time taken to complete the simulation is within expectation. In summary, by maintaining the same mesh setting, the integrity of the simulation result can be maintained.

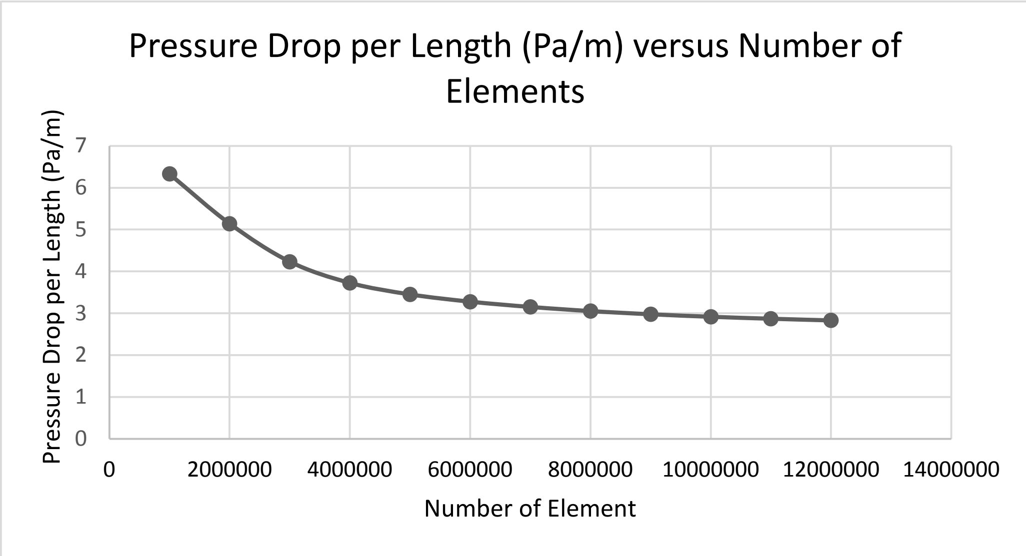


Figure 11 Graph of Pressure Loss per Length versus Number of Elements

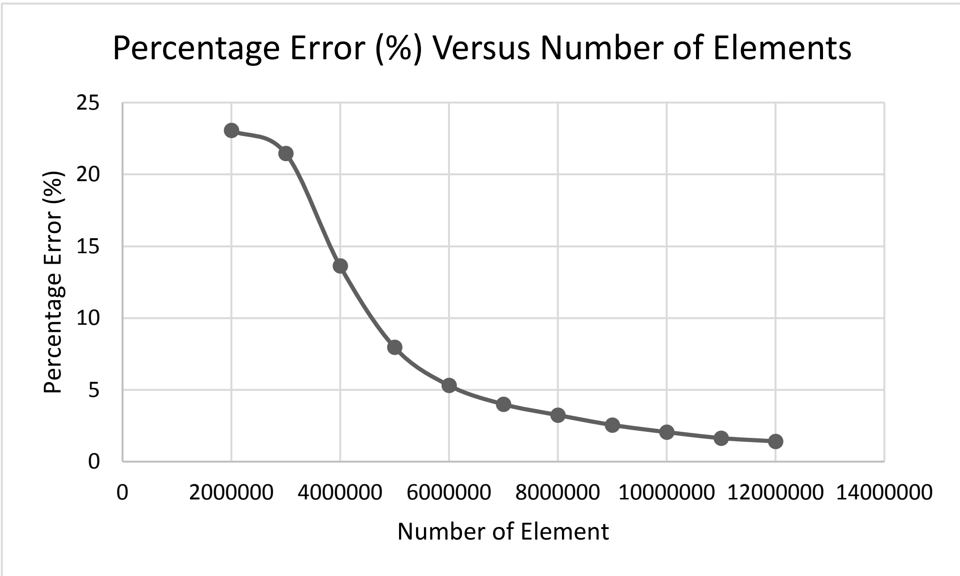


Figure 12 Graph of Percentage Error versus Number of Elements

4.3 Verification and Validation

This process is conducted aiming to produce an accurate simulation model as simulation is playing an important role in decision-making process and therefore, the degree of correct is an utmost concern for the decision maker as simulation process never reflect the real world event exactly. Because of this, it is necessary to make sure of the result gotten from simulation is capable to represent real word system to a certain acceptable degree.

In this project, two variables obtained from simulation model which are pressure loss per length and entrance region are chosen as the parameter to be compared with the theoretical results. To verify and validate the simulation model, only pipe without energy promoter is used to conduct CFD study due to its simplicity as there are mathematical equation to calculate the pressure drop in a simple pipe.

4.3.1 Pressure Drop

4.3.1.1 Theoretical Result

To calculate friction factor for the pipe, the following equation from Colebrook is used which is legit for turbulence range of the moody chart.

$$\frac{1}{\sqrt{f}} = -2.0\log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right)$$

However, the equation is implicit and an alternate form of Colebrook equation is used to estimate the friction factor and by performing several iteration, a result with percentage error being approximately zero could be obtained. The equation is recommended to be first guest, and perform iteration using the first guess obtained.

$$First\ Guess\ of\ Friction\ Factor, f = 0.02$$

Iteration	Friction Factor
1	0.021075264
2	0.021062344
3	0.021063052
4	0.021063013

After fourth iteration, it can be said that the iteration is converged already and the friction factor obtained from fourth iteration which is 0.021 is further used to compute the pressure drop caused by major loss.

$$Pressure\ Drop, \Delta P = f\frac{l}{D}\frac{1}{2}\rho V^2 = 262.48Pa$$

$$Pressure\ Drop\ per\ Length, \frac{\Delta P}{\Delta L} = 13.124Pa/m$$

4.3.1.2 Simulation Result

The pressure drop per length obtained through simulation can be seen from Figure 13 which is getting stable after entrance region which is 6m from calculation and the discrepancy between theoretical result which is 13.124Pa/m and simulation result which is 13.12768Pa/m is relatively small which is 0.028%.

By varying the velocity of the model from 0.3m/s to 1.3m/s with interval of 0.2m/s, the theoretical pressure drop per length and simulation pressure drop per length for different velocities are tabulated as shown in and a linear proportional graph could be plotted with almost exact 45 degree of gradient as shown in Figure 14.

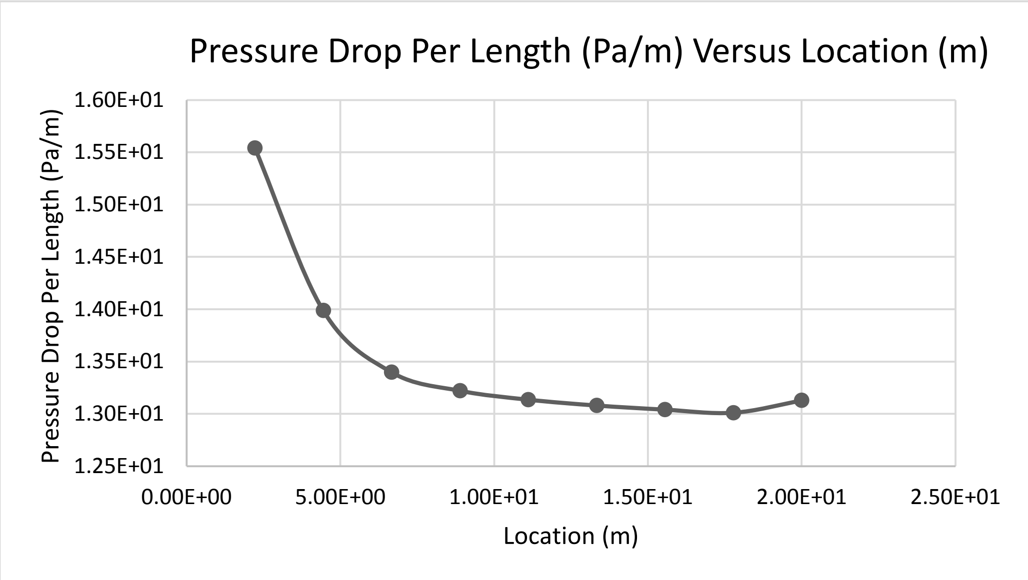


Figure 13 Graph of Pressure Drop per Length versus Location of Pipe

Table 3 Data of Pressure Drop of 20m Pipe without Energy Promoter

Z [m]	Pressure [Pa]	ΔP	ΔP/ΔL	Percentage Error (%)
0.00E+00	2.62E+02	0	0	0
2.22E+00	2.27E+02	34.54	15.54	18.41
4.44E+00	2.00E+02	62.16	13.99	6.57
6.67E+00	1.73E+02	89.32	13.40	2.08
8.89E+00	1.44E+02	117.52	13.22	0.74
1.11E+01	1.16E+02	145.96	13.14	0.09
1.33E+01	8.75E+01	174.40	13.08	0.33
1.56E+01	5.90E+01	202.86	13.04	0.63
1.78E+01	3.06E+01	231.31	13.01	0.86
2.00E+01	1 E-01	262.55	13.13	0.03

Table 4 Data of Theoretical Pressure Drop per Length versus Simulation Pressure Drop per Length

Velocity (m/s)	Theoretical ΔP/ΔL	Simulation ΔP/ΔL
0.3	5.021	5.005
0.5	13.124	13.114
0.7	24.919	24.943
0.9	40.387	40.415
1.1	59.520	59.551
1.3	82.314	82.342

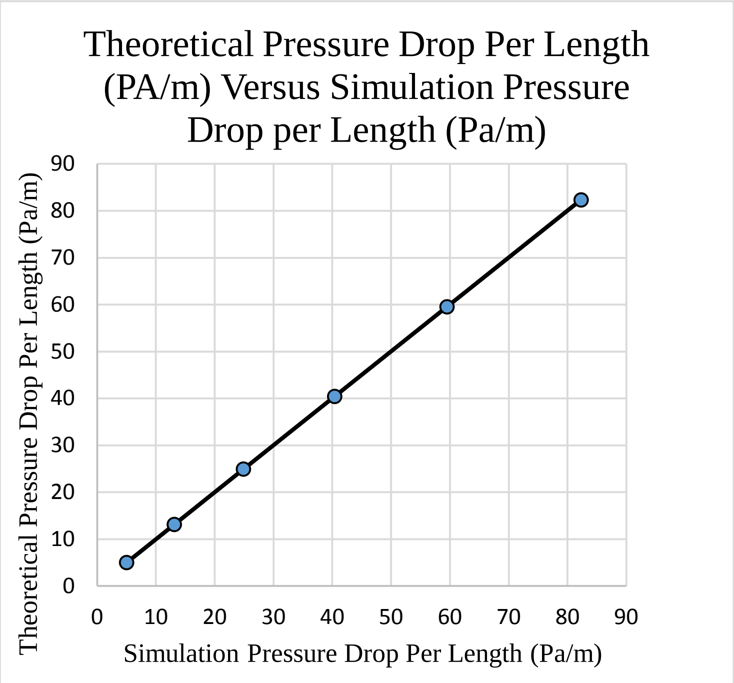


Figure 14 Graph of Theoretical Pressure Drop per Length versus Simulation Pressure Drop per Length

4.3.2 Entrance Region

4.3.2.1 Theoretical Result

For Turbulent Flow,

$$l_e = 6.099m$$

4.3.2.1 Simulation Result

From Figure 15, the velocity of the fluid decreases drastically from 0.3m/s which is inlet speed to 0.2984m/s and starting to stabilize after 6m. The result found matches the entrance region obtained from theoretical result.

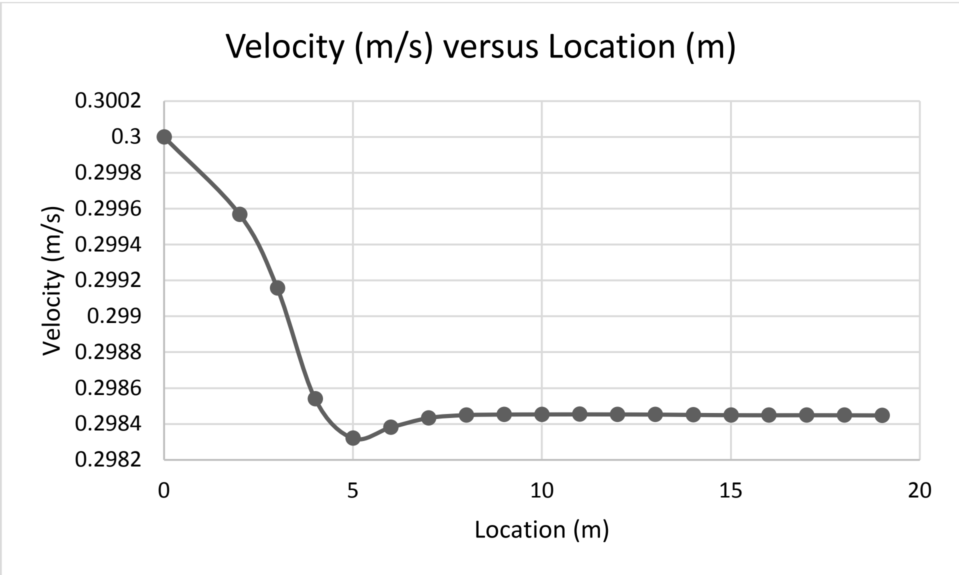


Figure 15 Graph of Velocity versus Location

4.3.3 Discussion

From both validation and verification test, it has been proven that the result obtained from simulation has only minor difference and is convincing enough to ignore difference between simulation result and theoretical result. Hence, the project will be proceeded to next stage which is to study the changes on pressure drop by changing the parameters of energy promoter such as height, dimension, width, number and row of energy promoter since the verification and validation of simulation model has been conducted successfully.

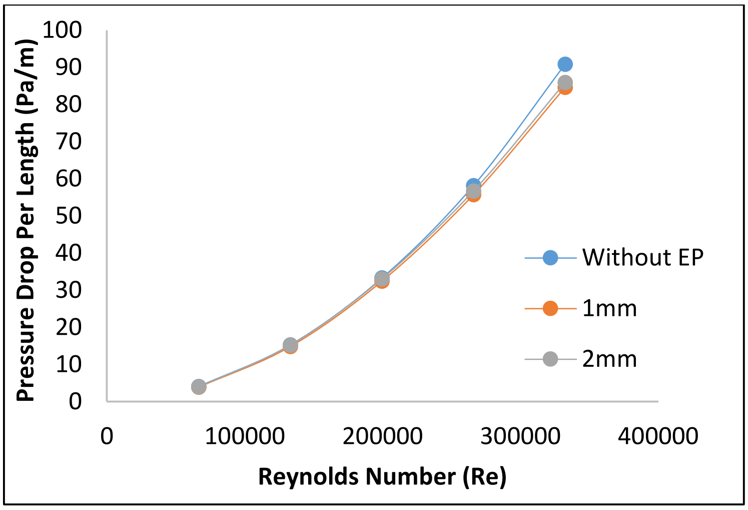
4.4 Parametric studies on Drag Reduction Efficiency of Energy Promoter

There are four factors that are expected to have effect on the drag reduction efficiency of the pipeline had been identified for parametric studies through CFD analysis. The parameters included are: (a) Height of Energy Promoter; (b) Direction of Energy Promoter facing the flow; (c) Number of Energy Promoter and (d) Flow rate in oil pipelines. In this project, the drag reduction efficiency is indicated by the pressure drop per length throughout the whole pipeline. The drag reduction efficiency with respective parameters is clearly shown on the following figures.

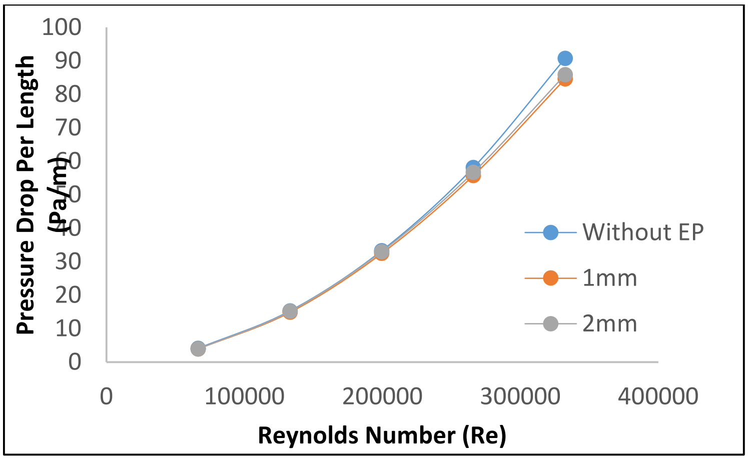
4.4.1 Effect of Height of Energy Promoter on Drag Reduction Effect in Pipeline

Parametric study of height of energy promoter is conducted and two height are used in this project which is 1mm and 2mm height of energy promoter. The results obtained and plotted in the following graph:

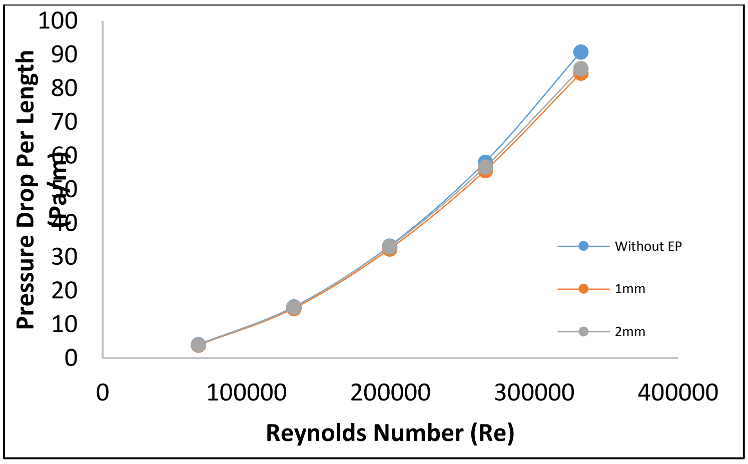
Figure 16 consists of pressure drop per length of pipelines with energy promoter in normal direction with three different section: four energy promoter, eight energy promoter and twelve energy promoter.



(a) 4 Energy Promoter



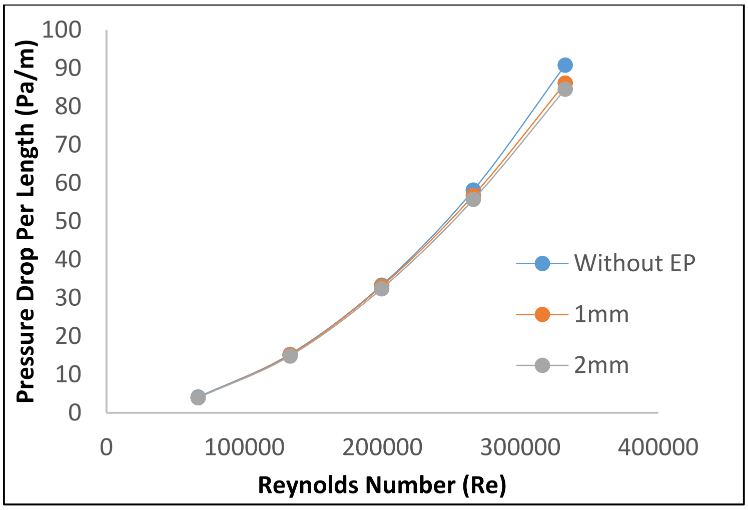
(b) 8 Energy Promoter



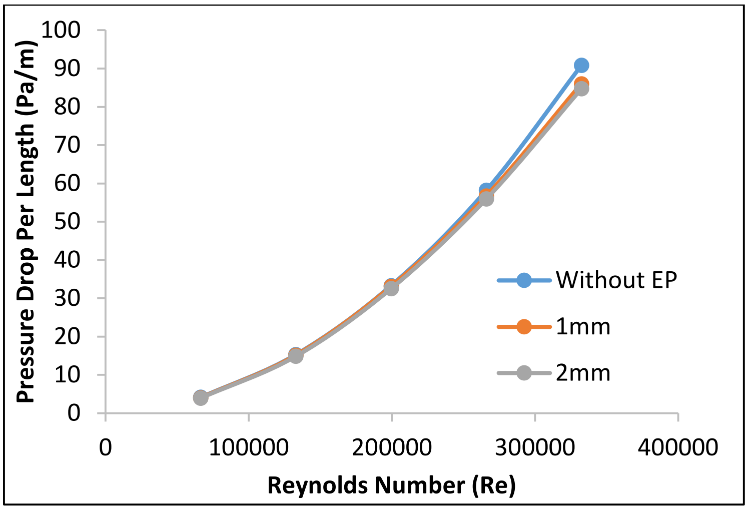
(c) 12 Energy Promoter

Figure 16 Effect of Height of Energy Promoter with normal direction on Drag Reduction Effect in pipeline

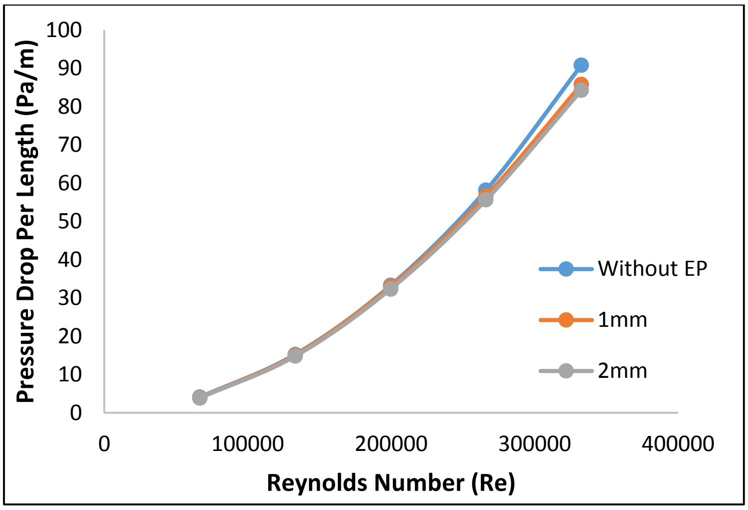
Meanwhile, Figure 17 consists of pressure drop per length with energy promoter in reverse direction with three different section: four energy promoter, eight energy promoter and twelve energy promoter. For each cases, the graph contain of three different pressure drop per length for three cases which are without energy promoter, with 1mm height of energy promoter and with 2mm height of energy promoter. By setting the pressure drop per length of pipeline without energy promoter as the benchmark, drag reduction effect is observed in each cases of pipeline. Detailed results is tabulated in Table 5.



(a) 4 Energy Promoter



(b) 8 Energy Promoter



(c) 12 Energy Promoter

Figure 17 Effect of Height of Energy Promoter with reverse direction on Drag Reduction Effect in pipeline

Based on Table 5, it is observed that for energy promoter in normal direction, the pressure drop per length for height=1mm and height=2mm is irregular which overall drag reduction percentage for 1mm height of energy promoter is higher than 2mm (height) energy promoter for case 4 EP and 12EP, on the other hand, overall drag reduction percentage for case with 8 EP with 2mm (height) energy promoter is higher than 1mm (height) energy promoter. However, for energy promoter in reverse direction, the overall drag reduction of 2mm (height) energy promoter is generally higher than 1mm (height) energy promoter. In summary, pipeline with energy promoter in reverse direction with height=2mm has shown better drag reduction effect.

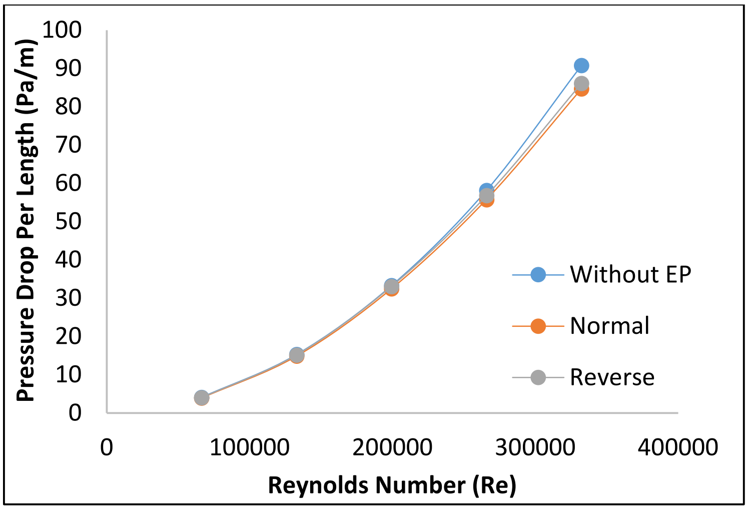
Table 5 Effect of Height of Energy Promoter on Drag Reduction Efficiency in pipeline at different conditions.

Section	Height	DR (%)
16a) 4 Energy Promoter	1mm	4.03
	2mm	2.64
16b) 8 Energy Promoter	1mm	3.90
	2mm	3.95
16c) 12 Energy Promoter	1mm	4.25
	2mm	2.43
17a) 4 Energy Promoter	1mm	2.45
	2mm	4.04
17b) 8 Energy Promoter	1mm	2.46
	2mm	4.04
17c) 12 Energy Promoter	1mm	2.78
	2mm	4.45

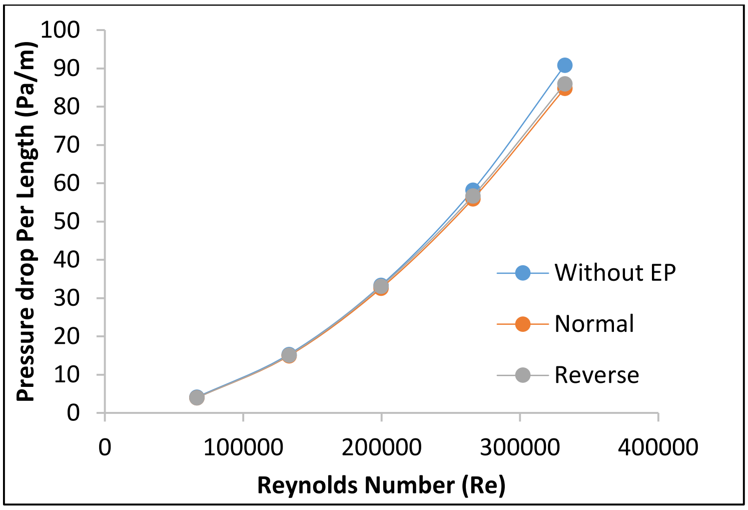
4.4.2 Effect of Direction of Energy Promoter on Drag Reduction Effect in Pipeline

Parametric study of height of energy promoter is conducted and two direction are used in this project which is in normal direction and reverse direction. The results obtained and plotted in the following graph:

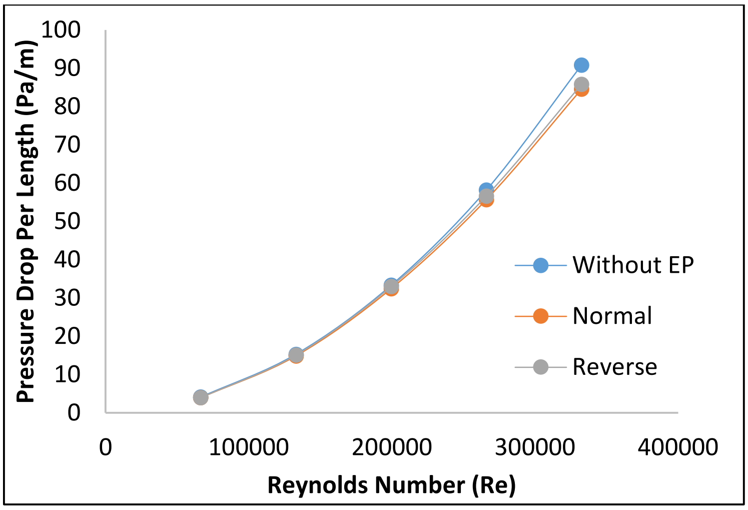
Figure 18 and Figure 19 represents the pressure drop per length in pipeline with energy promoter 1mm and 2mm height of Energy promoter respectively and each figure consists of three different section: four energy promoter, eight energy promoter and twelve energy promoter. With the pressure drop per length of pipeline without energy promoter being set as benchmark, it can be seen that each case with energy promoter embedded inside the pipeline has shown drag reduction effect. The pressure drop per length of each cases are summarized in Table 6.



(a) 4 Energy Promoter

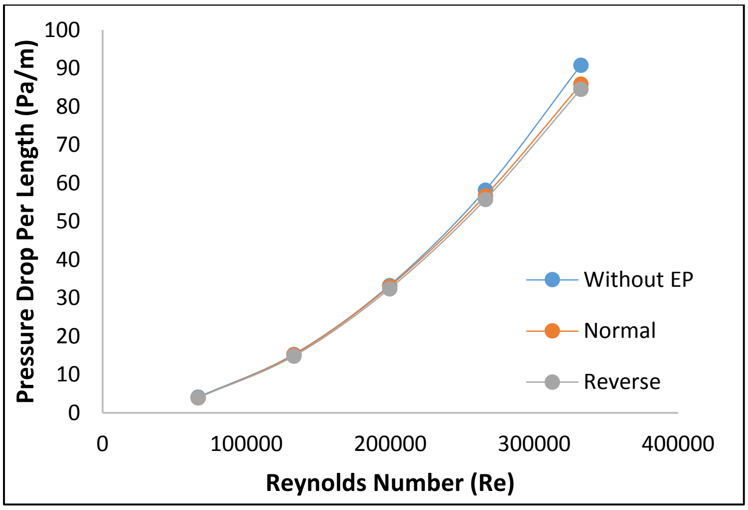


(b) 8 Energy Promoter

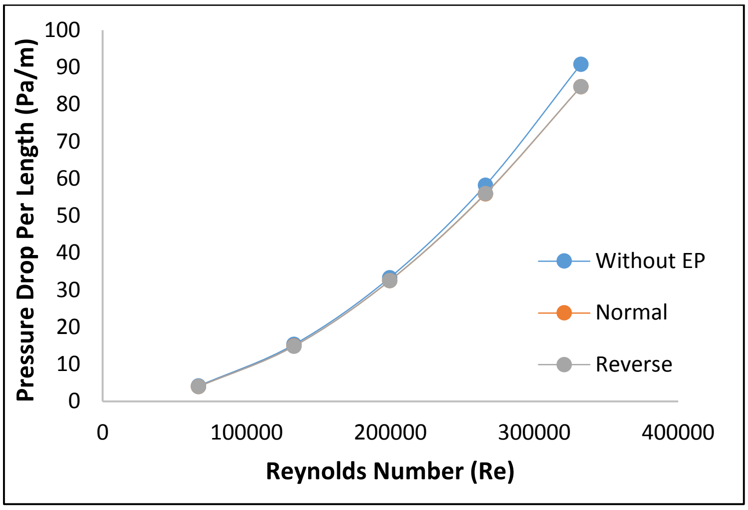


(c) 12 Energy Promoter

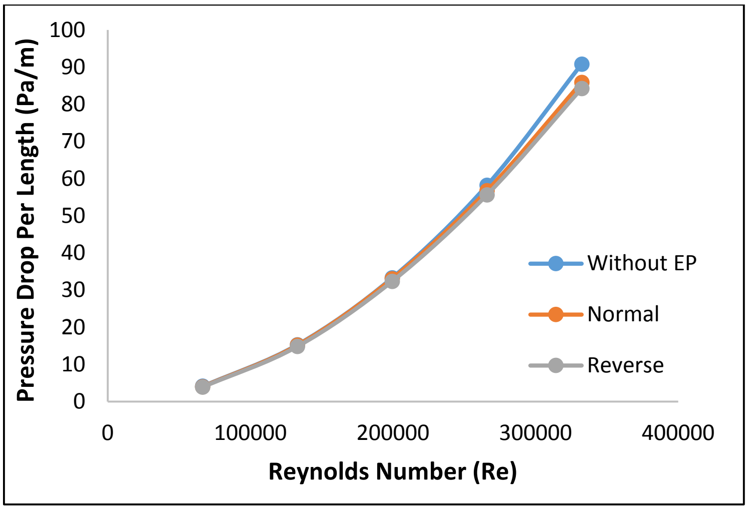
Figure 18 Effect of Direction of Energy Promoter with Height=1mm on Drag Reduction Effect in pipeline



(a) 4 Energy Promoter



(b) 8 Energy Promoter



(c) 12 Energy Promoter

Figure 19 Effect of Direction of Energy Promoter with Height=2mm on Drag Reduction Effect in pipeline

Table 6 Effect of Direction of Energy Promoter on Drag Reduction Efficiency in pipeline at different conditions.

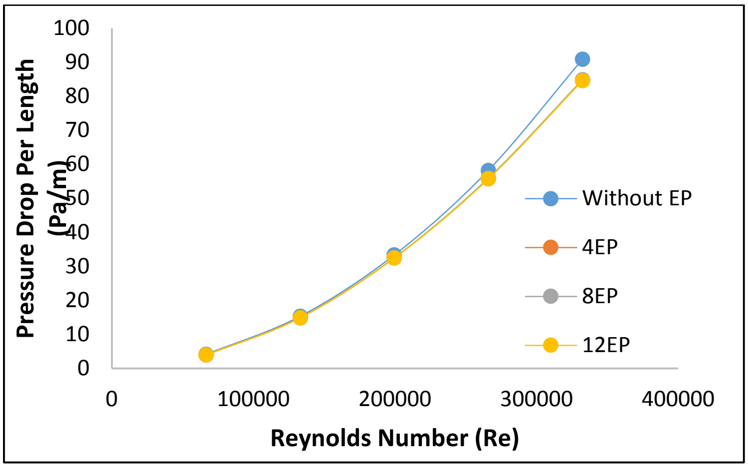
Section	Direction	DR (%)
18a) 4 Energy Promoter	Normal	4.03
	Reverse	2.45
18b) 8 Energy Promoter	Normal	3.90
	Reverse	2.46
18c) 12 Energy Promoter	Normal	4.25
	Reverse	2.78
19a) 4 Energy Promoter	Normal	2.64
	Reverse	4.04
19b) 8 Energy Promoter	Normal	3.95
	Reverse	4.04
19c) 12 Energy Promoter	Normal	2.43
	Reverse	4.45

It is observed from Table 6 that overall drag reduction for pipeline with 1mm (height) Energy Promoter in normal direction is better than Energy Promoter in reverse direction. Meanwhile, for the pipeline with 2mm (height) Energy Promoter, the Energy Promoter in reverse direction has shown better effect in drag reduction.

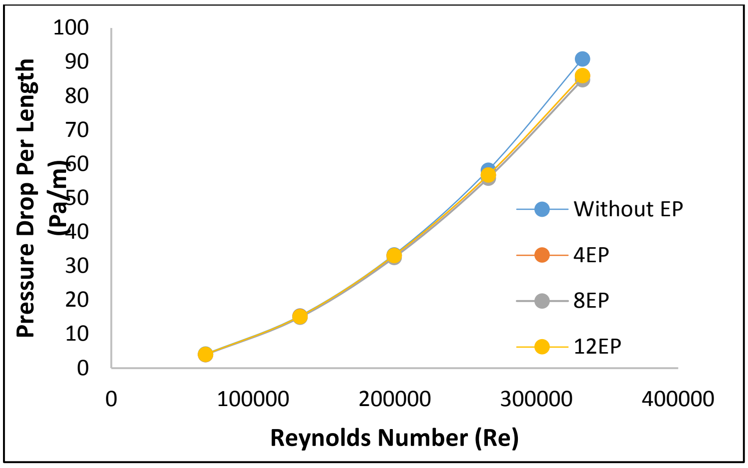
4.4.3 Effect of Number of Energy Promoter on Drag Reduction Effect in Pipeline

Based on Figure 20, each graph is plotted under respective section, which are (a) Normal Direction, 1mm (Height) Energy Promoter; (b) Normal Direction, 2mm (Height) Energy Promoter, (c) Reverse Direction, 1mm (Height) Energy Promoter and (d) Reverse Direction, 2mm (Height) Energy Promoter. Although reduction effect is observed, nearly identical lines observed from Figure 24 indicates the number of energy promoter is insignificant to pressure drop per length.

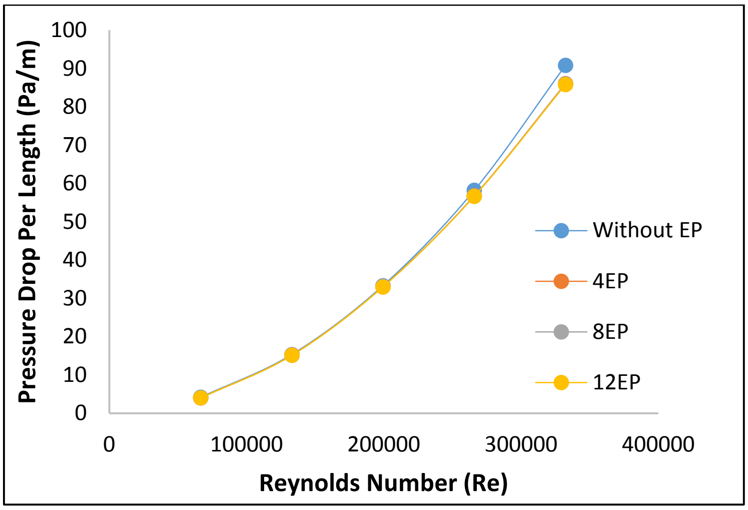
Parametric study of number of energy promoter is conducted and 3 number are used in this project which is 4, 8 and 12 energy promoter. The results obtained and plotted as follows:



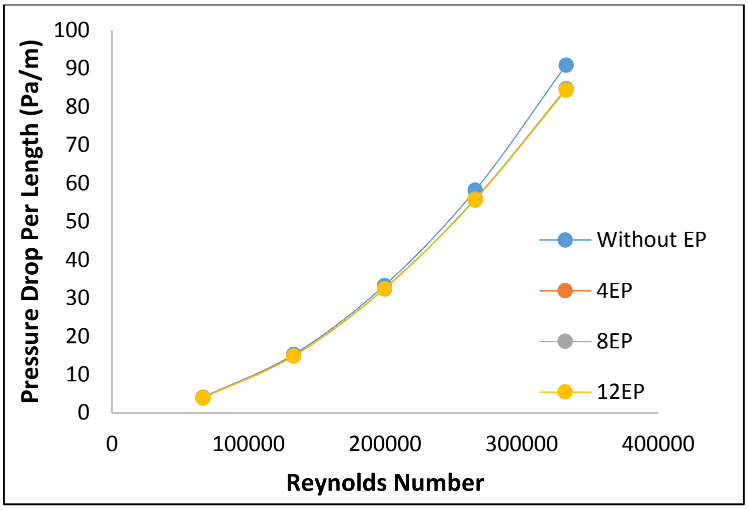
(a) Normal Direction; 1mm (Height)



(b) Normal Direction; 2mm (Height)



(c) Reverse Direction; 1mm (Height)



(d) Reverse Direction; 2mm (Height)

Figure 20 Effect of Number of Energy Promoter on Drag Reduction Effect in pipeline

By referring to Table *7*, it is found that Case 20a and 20d which are 1mm (height) energy promoter in normal direction and 2mm (height) energy promoter in reverse direction exhibit drag reduction of 4%. Figure 20 illustrates that decrement of pressure drop per length is present in each cases from 2% to 4.5% regardless of the number of energy promoter. Inconsistency of the results have proved that the number of energy promoter is not significant to the drag reduction level.

Table 7 Effect of Number of Energy Promoter on Drag Reduction Efficiency in pipeline at different conditions.

Section	Number of Energy Promoter	DR (%)
20a) Normal Direction; 1mm (height)	4	4.03
	8	3.90
	12	4.25
20b) Normal Direction; 2mm (height)	4	2.64
	8	3.95
	12	2.43
20c) Reverse Direction; 1mm (height)	4	2.45
	8	2.46
	12	2.78
20d) Reverse Direction; 2mm (height)	4	4.04
	8	4.04
	12	4.45

4.4.4 Effect of flow rate on Drag Reduction Effect in Pipeline

Based on Figure 21, the curvy line represents the percentage of Drag Reduction for four conditions, which are: (a) Normal Direction, 1mm (height) Energy Promoter; (b) Normal Direction, 2mm (height) Energy Promoter; (c) Reverse Direction, 1mm (height) Energy Promoter; (d) Reverse Direction, 2mm (height) Energy Promoter versus range of Reynolds Number. It is clearly shown that, the percentage drag reduction is decreasing gradually until Re=190000, beyond this point, the percentage of drag reduction is increasing steadily up to Re=350000. The results has shown that drag reduction effect of energy promoter is strong at low velocities and high velocities. The results suggest that the application of energy promoter is most suitable at Reynolds number smaller than 60000 and Reynolds number higher than 260000.

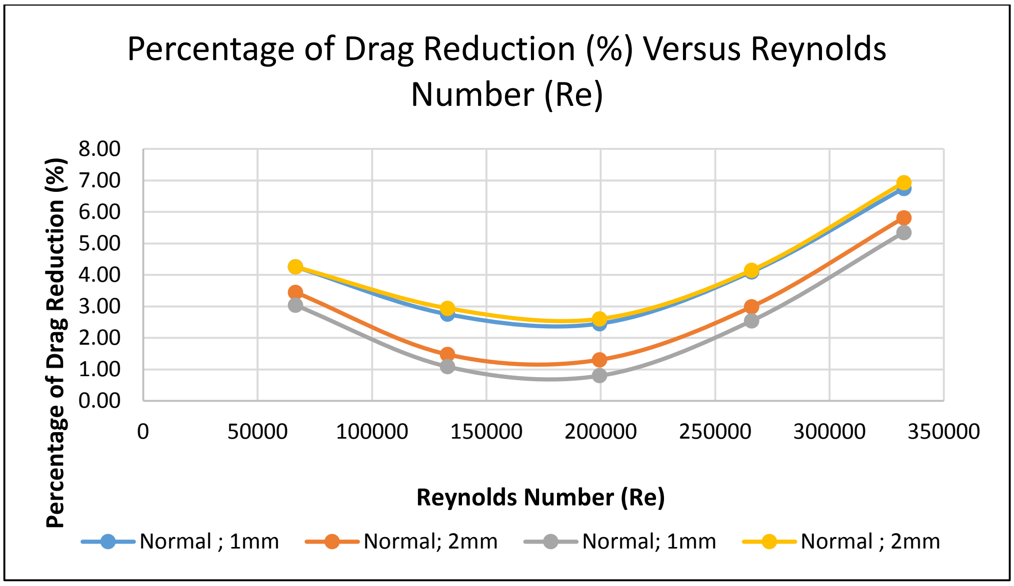


Figure 21 Correlation between Reynolds Number a and Percentage of Drag Reduction

Table 8 Percentage of Drag Reduction under different Reynolds Number

	Percentage of Drag Reduction (%)			
	Normal		Reverse	
Reynolds Number	1mm	2mm	1mm	2mm
66482	4.26	3.44	3.04	4.26
132964	2.75	1.47	1.09	2.94
199446	2.45	1.30	0.80	2.60
265928	4.10	2.99	2.54	4.15
332410	6.75	5.81	5.35	6.93

4.5 Velocity Contour and Velocity Vector

Figure 22, Figure 23 and Figure 24 shows the velocity contour of three different pipelines which (i) Without Energy Promoter, (ii) With Energy Promoter in normal Direction, and (iii) With Energy Promoter in reverse direction. It can be seen that the velocity contour of pipeline without energy promoter displays a fully developed flow which is stabilized throughout the section while the other two cases which pipeline with Energy Promoter has shown that the presence of Energy Promoter has caused disruption in near wall fluid flow and it is believed that Energy Promoter managed to restructure the turbulent level of the flow by interacting with the near wall fluid flow.

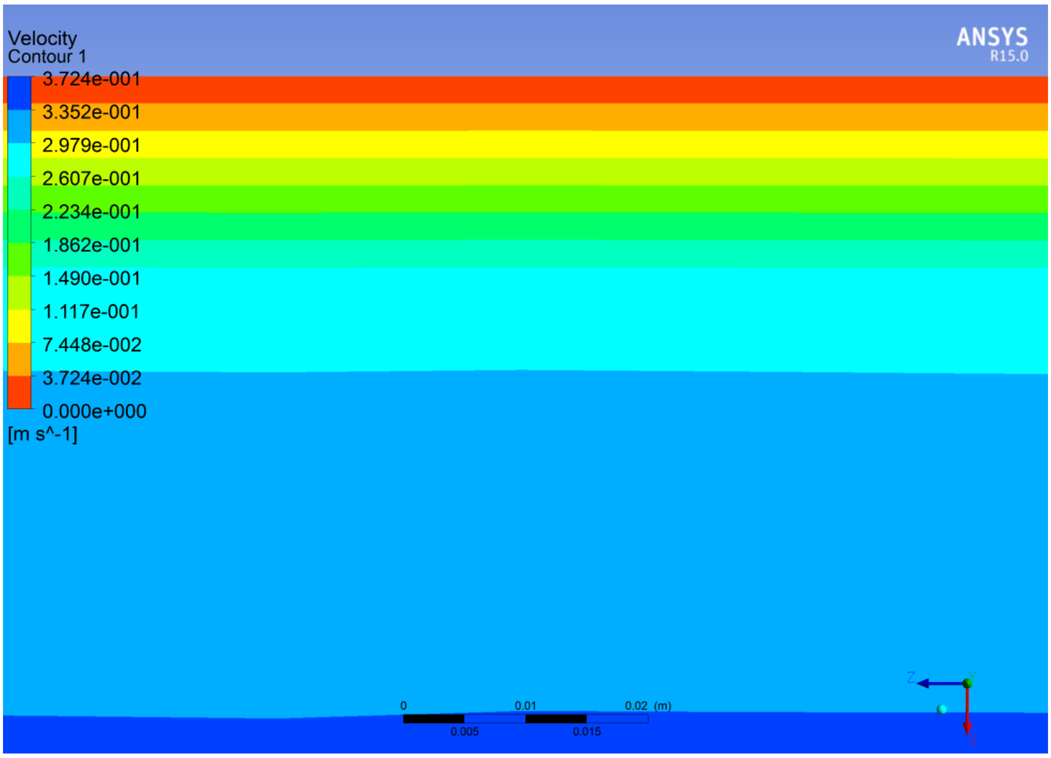


Figure 22 Velocity Contour of Pipeline without Energy Promoter

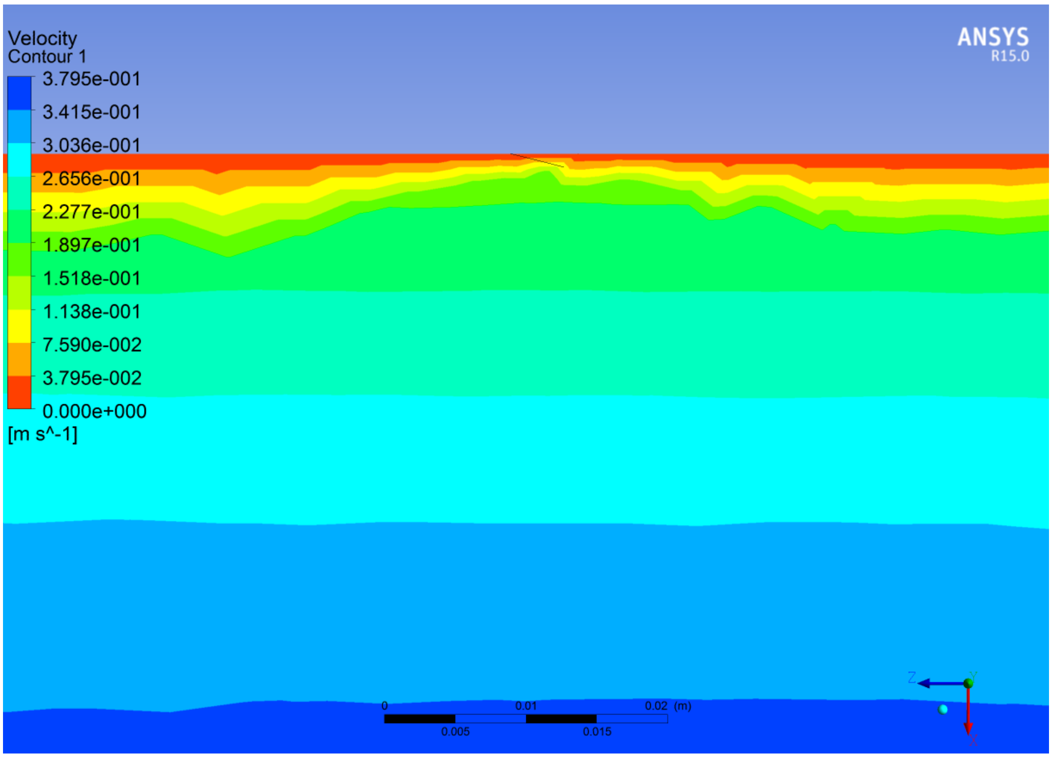


Figure 23 Velocity Contour of Pipeline with Energy Promoter in Normal Direction

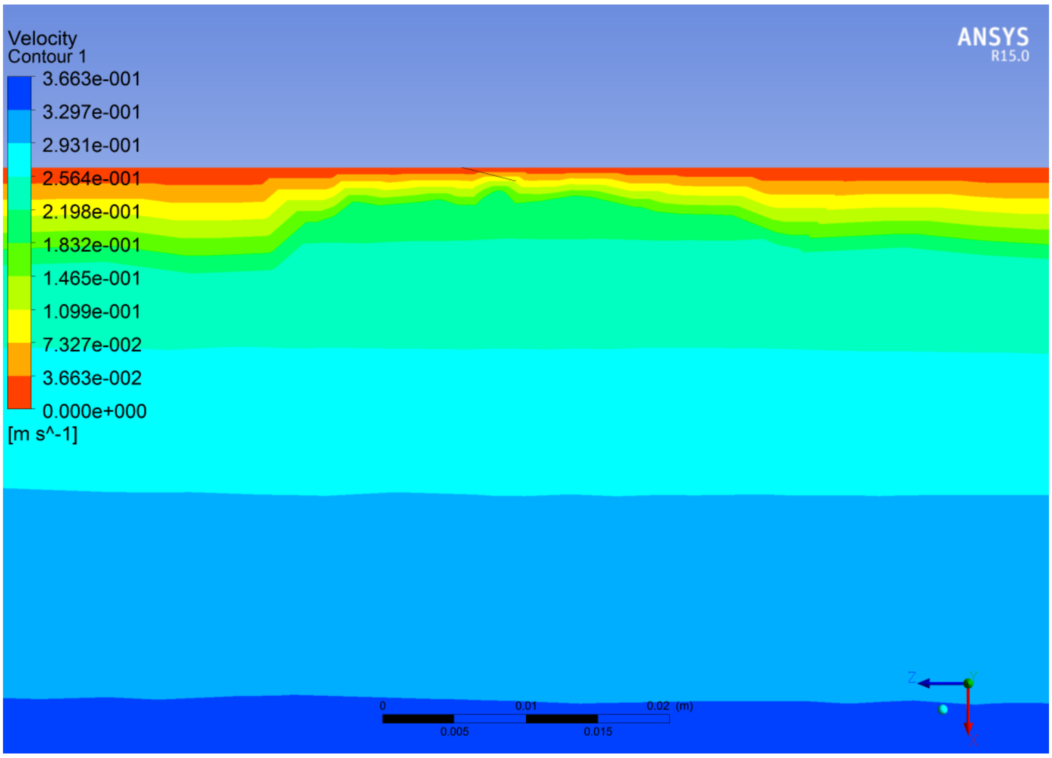


Figure 24 Velocity Contour of Pipeline with Energy Promoter in Reverse Direction

CHAPTER 5
CONCLUSION & RECOMMENDATION

5.1 Conclusion

In short, by selecting the correct parameters for the dimension of Protrusion Ring in the pipeline, it is possible that the drag caused by the friction between the fluid and inner wall of pipelines be reduced to the desirable level. Implementation of this technology into the pipelines that are used to transport natural gas and crude oil will certainly reduce the cost and energy that is used to restore the pressure.

In order to ensure the accuracy of the result obtained from simulation model is close to the real world model, validation and verification of simulation model has been carried out and two parameters have been chosen to be compared with the result calculated by the formula and the percentage error from the simulation model is less than 5%.

From parametric studies, it is observed that pipeline with energy promoter in reverse direction with height=2mm has shown higher percentage of drag reduction while the number of energy promoter does not have significant effect on drag reduction. Then, it is found out that energy promoter has shown good drag reduction effect when Reynolds Number is smaller than 60000 or larger than 260000 in 20m long pipe with 0.2m diameter.

The objectives of the project have been accomplished which drag reduction effect has been discovered by insertion of Energy Promoter in pipeline and parametric study of Energy Promoter on drag reduction efficiency has been conducted.

5.2 Recommendation

There are several recommendations would like to be proposed:

i) Experimental measurements to compare with & validate the theoretical and simulation prediction

ii) Include more parameters in the study:

   a) Change the shape of Energy Promoter,
   b) Dimension of Energy Promoter: Length, Height, Width
   c) Configuration of Energy Promoter
   d) Increase the length of the pipeline
   e) Change the medium for flow
   f) Change the shape of the pipeline, e.g. more bent

REFERENCES

Abdulbari, H., Yunus, R., Abdurahman, N., & Charles, A. (n.d.). *Going against the flow- A reviewof non-additive means of drag reduction.* Pahang.

Boyun Guo;Shanhong Song;Jacob Chacko;Ali Ghalambor. (2005). *Offshore Pipelines.* Elsevier.

Brostow, W. (2008). Drag reduction in flow: Review of applications, mechanism and prediction. *Journal of Industrial and Engineering Chemistry*, 409-416.

Choi, S. (1985). *Near-wall turbulence structure on a riblet wall.* UK.

Diamond, P., Harvey, J., Katz, J., Nelson, D., & Steinhardt, P. (1992). *Drag Reduction by Polymer Additives.* Virginia: The Mitre Corporation.

Kramer, M. (1960). Boundary layer stabilization by distributed damping. *Am. Soc. Navig. Eng.*, 25-33.

Park, S., & Wallace, J. (1993). Near-Wall Turbulent Flows. *B.E. Launder*, 749.

Pipeline Purpose and Reliability. (2014). Retrieved from Buckeye Partners, L.P.: http://www.buckeye.com/pipelineawareness/pipelinepurposeandreliability/tabid/109/default.aspx

Sutherand, B.R.;Van Vliet, D.;Dohan, K;Moodie,T.B. (n.d.). Interfacial Mixing in Viscous Pipe Flows. *Final Report to Imperial Oil*.

Toms, B. (n.d.). *In Proc. of 1st Int. Congress on Theology.*

Xu, S., Rempfer, D., & Lumley, J. (2003). Turbulence over a compliant surface: numerical simulation and analysis. *J. Fluid Mech*, 11-34.