DESIGN AND ANALYSIS OF PIPING SYSTEM FOR AN OPEN CHANNEL FLUME'S FLOW

HAMDI ABDULLAH MOHAMMED MUDHSH

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

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2013

SUPERVISOR'S DECLARATION

I hereby declare that I have examined the final copy of this report and in my opinion, it is fully adequate in terms of language standard, and report formatting requirement for the award of the degree of Mechanical Engineering.

Signature : ………………………………………
Name of Supervisor : Mr. AZIM BIN MOHD ARSHAD
Position :
Date :

Signature : ………………………………………
Name of Panel : Mr. AMIR BIN AZIZ
Position :
Date :

STUDENT'S DECLARATION

I declare that this thesis entitled "Design and Analysis of Piping System for an Open Channel Flume's Flow" is the result of my own research except as cited in the references. The thesis has not been accepted for any bachelor's degree concurrently submitted in candidature of any other degrees.

Signature : …………………………………………..
Name : HAMDI ABDULLAH MOHAMMED MUDHSH
ID Number : MG 09033
Date : 11 | JULY | 2013

ACKNOWLEDGEMENT

All Praises be to Allah S.W.T Almighty who blessed me in the success of this research and gave me the strength and patience throughout my journey of studies.

First of all, I would like to present my sincere gratitude to my supervisor Mr. Azim bin Mohd Arshad and my co-supervisor Mr. Ahmad Basirul Subha bin Alias for their constant guidelines and motivation. I am very grateful for their patience who never gives up showing me the right way throughout the research. Their guidance helped me in all the time of research and writing of this thesis. I am also thankful to Energy Sustainable Focus Group (ESFoG) for giving me the advice and constructive criticism on my research to the extent of its completion.

Furthermore I would like to thank and show my great appreciation to my group members in Flume Design Research Group: Sazwan bin Sahar, Irwan bin Misalam, Muhammad Adlin bin Hasmar, Ahmad Bazli bin Mohd Sibri, Mohd Firdaus bin Mohd Rosli, Nurul Hidayah Binti Husin and Nabil Fikri bin Zuiful Zubri, for the time and effort to co-operate together, group discussions, for the sleepless nights we were working before deadlines, and for all the encouragement along the time to finish the project.

Last but not least, I am also in debt to Faculty of Mechanical Engineering for the usage of workstation computer, equipment's, administration, and references for analytical study purpose. My sincere appreciation also extends to all my colleagues whom had provided assistance at various occasions.

Finally to individuals who had involved either directly nor indirectly in succession of this thesis. Thank you.

ABSTRACT

The increase on the usage of non-renewable energy has caused high demanding on energy supplies. This demand is highly needed for the industrial, transportation, domestic usage and etc. Shortage in the non-renewable energy had caused human to find other alternatives to generate energy. Hence, this research is aim to develop a flume, that later use to generate electricity. However, researcher only focuses on the design and analysis of the piping system of the flume. The objectives of this research is to design piping system of an open channel with less head losses and simulate it using CFD for only single phase perform (not free service), to evaluate the pressure drop in the pipes and to analyses and determine the head losses in the pipe. The instrument used is Solid Work 2012 for technical drawing of the piping system and Computational Fluid Dynamics (CFD) for simulation of the design and analysis. There are four designs of piping system that had been produce which is Design A, Design B, Design C and Design D, the properties measured for all four designs are pressure drop, percentage pressure drop, head loss and velocity. Design A; average percentage of pressure drop is 40.2%, average head loss which is 2.7 m, velocity; inlet velocity is 8.84 m/s and average outlet velocity is 8.75 m/s. Design B; average percentage of pressure drop is 53.6 %, average head loss which is 5.8 m, velocity; inlet velocity is 8.84 m/s and average outlet velocity is 14.31 m/s. Design C; percentage of pressure drop is 38.4 % ,head loss which is 4.4 m, velocity; inlet velocity is 8.84 m/s and outlet velocity is 11.76 m/s. Designs D; percentage of pressure drop is 30.7%, head loss which is 3.1 m, velocity; inlet velocity is 8.84 m/s and outlet velocity is 12.03 m/s. The best design is determined by less percentage of pressure drops, hence from the research findings Design D is the best design since it has the less percentage of pressure drop.

ABSTRAK

Peningkatan pada penggunaan tenaga tidak boleh diperbaharui telah menyebabkan permintaan tinggi pada bekalan tenaga. Permintaan ini amat diperlukan untuk perindustrian, pengangkutan, penggunaan domestik dan lain-lain. Kekurangan dalam tenaga tidak boleh diperbaharui ini telah menyebabkan manusia mencari alternatif lain untuk menjana tenaga. Oleh itu, kajian ini adalah bertujuan untuk membangunkan satu flum, yang kemudiannya digunakan untuk menjana elektrik. Walau bagaimanapun, penyelidik hanya memberi tumpuan kepada reka bentuk dan analisis sistem paip flum. Objektif kajian ini adalah untuk membina sistem paip dari saluran terbuka, dengan kurang kehilangan turus pada paip dan simulasi dengan menggunakan CFD satu fasa, untuk menilai penurunan tekanan dalam paip, dan menentukan kerugian turus kepala paip. Instrumen yang digunakan adalah *Solid Work 2012* untuk lukisan teknikal sistem paip dan *Computational Fluid Dynamics (CFD)* untuk simulasi reka bentuk dan analisis. Terdapat empat reka bentuk sistem paip yang telah dihasilkan iaitu *Design A, Design B, Design C* dan *Design D*, ciri-ciri yang diukur pada empat reka bentuk paip tersebut adalah penurunan tekanan, peratus penurunan tekanan, kerugian pada kepala paip dan kelajuan (halaju). *Design A*; purata penurunan peratusan tekanan adalah 40.2%, purata kehilangan pada kepala paip ialah 2.7 m, halaju; halaju masuk adalah 8.84 m / s dan halaju keluar adalah 8.75 m / s. *Design B*; purata penurunan peratus tekanan adalah 53.6%, purata kehilangan pada kepala paip ialah 5.8 m, halaju; halaju masuk adalah 8.84 m / s dan halaju keluar adalah 14.31 m / s. *Design C*; purata penurunan peratus tekanan adalah 38.4%, purata kehilangan pada kepala paip ialah 4.4 m, halaju; halaju masuk adalah 8.84 m / s dan halaju keluar adalah 11.76 m / s. *Design D*; purata penurunan peratus tekanan adalah 30.7%, purata kehilangan pada kepala paip ialah 3.1 m, halaju; halaju masuk adalah 8.84 m / s dan halaju keluar adalah 12.03 m / s. Reka bentuk terbaik ditentukan oleh penurunan peratusan tekanan yang paling sedikit, maka dari hasil penyelidikan, *Design D* adalah reka bentuk yang terbaik kerana ia mempunyai penurunan peratusan yang paling kurang.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

ε	Roughness
kg	Kilogram
s	Second
mm	Millimetre
Pa	Pascal
in	Inch
m	Meter
Re	Reynolds number
ρ	Density
V	Velocity
D	Diameter
μ	Dinamic viscosity
A	Area
ΔP_L	Drop Pressure loss
K_l	Major loss (K_l for elbow = 0.9)
L	Length
h_l	Headloss
g	Gravity
P_t	Total Pressure

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamics
DM	Design Modeller

CHAPTER 1

INTRODUCTION

1.0 Introduction

In this chapter reader can get information on background of study, problem statement as well as objectives, and scope of study.

1.1 Background of study

The increase on the usage of energy since Industrial Revolution until now has caused high demanding on energy supplies. This demand is highly needed for industrial, manufacturing as well as daily usage. Energy sources from petroleum, gas and coil is a natural resources, and it is being used since two hundred years ago. Shortage on these energy sources and continuous highly usage had burdened up in order to fulfil this demand.

Moreover, poor management on the energy usage become one of the factors of energy depletion. Less attention on the renewable energy research which can be generated from wind, water and sun ray.

Hence, this need has created ‘desperation’ on the research of renewable energy. Realizing this, facilities to supports the application of renewable energy to then be applied in the real world should be developed. This is to prevent the existing energy sources to be depleting fast and as a replacement to the existing energy sources one they are all gone in the future. The reality is, the world is keeping developing from day to day, and energy sources will be more crucial

1.2 Problem statement

i. Disturbance on electric supply has become a serious issue, especially when it comes to industrial and manufacturing. This issue had cause lost of millions ringgit.
ii. Other than that, disturbance on electric supply also occur in housing area either urban or rural, which electricity is the most crucial basic daily needs in powering up the houses and utilities.
iii. Lack of testing areas and equipment to conduct the tests for hydro turbines or micro hydro turbines.
iv. Lack of testing apparatus to analyze the river flow which may show variability of river properties such as river velocity, mass flow rate and volume flow rate.

1.3 Objective of study

Objective of the research is very important thing that we must be considering before we conduct research; it can avoid us from out of topic from its aim of research. So, in this research paper, there are several objectives are aim to be achieving, such as:

i. To design piping system of an open channel with less head losses and simulate it using CFD for only single phase perform.
ii. To evaluate the pressure drop and head losses in pipes.
iii. To analyse and determine the velocity changes in pipes.

1.4 Hypothesis

The expected result for this research is that there will be a difference in terms of analyzing the suitable finalizes design of flume within all the references, analysis, and simulation.

1.5 Scope of study

In order to achieve the research objectives, there are a few research scopes that are important and necessary to be revised and followed, such as:

i. The specification of pipe size (length, diameter).
ii. Pressure losses in pipe.
iii. Isothermal (constant temperature).
iv. CFD simulation for piping system.
v. For the sake of simulation only single phase is perform (not free surface flow).
vi. Free surface will be considered during the work stake.
vii. The properties of water:
    a. Mass flow rate
    b. Volume flow rate
    c. Velocity of water
    d. Reynolds number

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

Selection of a piping system is an important aspect of system design in any energy consuming system. Piping system design in large industrial complexes like Refineries, Petrochemicals, Fertilizer Plants and many more done nowadays with the help of design software, which permits the designer or engineer to try out numerous possibilities. The main objectives in building a piping system is, it pays to do it economically as possible which is minimize the work that have to do and match the pump requirements. A suitable piping system for its operation can be determined by considering the three components that make up the total resistance to water movement in the pipe. Those components are the sum of lift, the velocity head and the friction head. Piping system will be build and attach to the flume, so that the fluid or specifically water can flow through it. Although pipes and flumes are not classified as a lining, they are consider as an alternative to the remediation of a channel to reduce the seepage where supported by other advantages. A pipeline is described as the most effective lining option as it also prevents losses due to evaporation and allows pressurized delivery if required.

2.1 Design

According to Oxford Dictionary 4th Edition (2008), design means a plan or drawing produced to show the look and function or working of a building, garment or other object before it is made. It is a decorative pattern, purpose or planning that exists behind an action, fact or object. Hence, in this project, a piping system is designed so it can fit to the flume. The design process consists of the establishment of the design criteria for the piping system- For a proper piping design, it is essential that the client and the contractor agree on a design basis, process, and mechanical, civil and electrical control and instrumentation.

2.2 Analysis

According to Oxford Dictionary 4th Edition (2008), analysis means detailed examination of the elements or structure of something. Analysis is a process of separating something into constituent elements. It is the identification and measurement of the chemical constituents of a substance or specimen.

Analysis of data is a process of inspecting, cleaning, transforming and modeling data with the goal of highlighting useful of information, suggesting conclusions and supporting decision-making. Data analysis has multiple facets and approaches, encompassing diverse techniques under a variety of names, in different business, science and social science. For engineering field, quantitative data is the most suitable data to analyze. Quantitative data is analyzed with a statistical data analysis. This involves three major steps:

- Data preparation: logging the data, checking the data for accuracy, developing a database structure, entering the data into the computer, data transformations.
- Descriptive statistics: the basic features of the data (distribution, central tendency, dispersion or variability, correlations).
- Statistical analysis of the research design: here the hypotheses are tested.

Hence, this project, the water properties such as pressure drop, velocity and head loss analyzed.

2.3 Open Channel Flow

Open channel hydraulics is the study of the physics of fluid flow in conveyances in which the flowing fluids forms a free surface is driven by gravity (Strum. T.W, 2001). Natural open channels include brooks, streams, rivers and estuaries. Artificial open channels are exemplified by storm sewers, sanitary sewers and culverts flowing partly full, as well as drainage ditches, irrigation canals, aqueducts and flood diversion channels. Applications of open channel hydraulics range from the design of artificial channel for beneficial purposes such as irrigation, drainage, water supply and wastewater conveyances to the analysis of flooding in natural waterways to delineate floodplains and assess flood damages for a flood of specified frequency. Principles of open channel are also utilized to describe the transport of environmental contaminants, including those carried by sediments in motion, as well as to predict flood surges caused by dam breaks or hurricanes.

2.3.1 Characteristics of open channel flow

The relevant forces causing, resisting motion, and the inertia must form a balance, such that the pressure of a free surface of streamline must be constant and equal to atmospheric pressure.

- Extreme variability encountered in cross sectional shape and roughness. Condition changes from a circular gravity sewer flowing partly full to a natural river channel with a floodplain subject to overbank flow.
- Roughness height in the gravity sewer corresponds to those encountered in closed conduit flow, while roughness such as brush, vegetation and deadfalls in natural open channel make the roughness extremely difficult to quantify.

Table 2.1: Pipe flow versus open channel flow

	Pipe Flow	**Open Channel Flow**
Flow driven by	Pressure work	Gravity (potential energy)
Flow cross section	Known, fixed	Unknown in advance because the flow depth is unknown
Characteristics flow parameters	Velocity deduced from continuity	Flow depth is deduced simultaneously from solving both continuity and momentum equations
Specific boundary conditions	-	Atmospheric pressure at the free surface.

Table 2.2: Types of open channel flow

Steady flow	- when discharge (Q) does not change with time
Uniform flow	- when depth of fluid does not change for a selected length or section of the channel
Uniform steady flow	- when discharge does not change with time and depth remains constant for a selected section - Cross section should remain unchanged-referred to as a prismatic channel
Varied steady flow	When depth changes but discharge change along a channel length of interest
Varied unsteady flow	When both depth and discharge change along a channel length of interest
Rapidly varying flow	Depth change is rapid
Gradually varying flow	Depth change is gradual

Source: Open channel problem can be solved through a combination of theory and experiment (Sturm. T.W, 2001)

2.4 Flume

Flume has been used to transport water since Roman times. The word flume is derived from Latin word means ‘to flow’ and today it means ‘an inclined channel for conveying water from a distance’ (Arthur. R, M. Nowell and Peter A. Jumars, 1987). Flume is an open artificial water channel, in the form of a gravity chute, which leads water from a division dam or weir completely aside a natural flow. Flume have been used extensively in hydraulic mining and working placer deposits for gold, tin and other heavy metal. They also used in the transportation of logs in the logging industry, electric power generation and to power various mill operations by the use of a waterwheel.

2.5 Piping

Pipe is a pressure tight cylinder used to convey a fluid or to transmit a fluid pressure, ordinarily designated pipe in applicable material specifications. Materials designated tube or tubing in the specifications is treated as pipe when intended for pressure service. Piping is an assembly of piping components used to convey, distribute, mix, separate, discharge, meter, control or snub fluid flows. Piping also includes pipe-supporting elements but does not include support structures, such as building frames, bents, foundations, or any equipment excluded from Code definitions. Piping components are mechanical elements suitable for joining or assembly into pressure-tight fluid containing piping systems. Components include pipe, tubing, fittings, flanges, gaskets, bolting, valves and devices such as expansion joints, flexible joints, pressure hoses, traps, strainers, in-line portions of instruments and separators. Piping is typically round.

2.5.1 Piping system

Piping is a system of pipes that used to convey fluids (fluids or gases) from one location to another location. Piping can be manufactured from wood, fiberglass, glass, and steel, aluminum, plastic, copper and concrete. The in-line components, known as fittings, valve and other devices, typically sense and control the pressure, flow rate and temperature of the transmitted fluid and usually are included in the field of piping design.

2.5.2 Piping Design

Before proceeding with the design of the pipelines, some restrictions or assumptions about the characteristics of the production wells, re-injection wells, power plant location need to be considered. The output characteristics, mass flow rates, well head pressure, temperature and chemistry of the wells enable the selection of optimum production values, which will be considered for the entire life of the project (Henriquez and Aguirre, 2011) however, in this project well is replaced with flume but the characteristics remained.

José Luis Henríquez Miranda and Luis Alonso Aguirre López (2011) have stated that the problem of design procedure is to find a pipeline configuration and size within the constraints, which is safe and economical. The design process consists of the establishment of the design criteria for the piping system- For a proper piping design, it is essential that the client and the contractor agree on a design basis, process, and mechanical, civil and electrical control and instrumentation, (Henríquez and Aguirre, 2011)

Table 2.3: Design criteria

General	**Process**	**Mechanical**	**Civil/Struture**
Design life	Steamfield layout	Design parameters-Process conditions-Design loads	Design codes and procedures
Meteorological & other local data	Economic analysis	Design codes and procedures	Project layout
Environmental requirements	Piping criteria: pressure drop line sizing pipe routing design pressure	Piping system design	Access
Operating and maintenance criteria	Draining & venting philosophy	Pipes	General civil constructions
Cost minimization	Silica deposition	Valves	Thermal ponds
Avoiding uphill two phase flow	Insulation	Fittings	Retaining walls
	Control valve types	Vessels	Foundation design
	Pressure relief devices	Mechanical equipment	Structure design loads
	Pumps	Other components	Pipe support and anchors
	System isolation philosophy	Constructability and maintainability	Structures
	Instrument air-source and materials		Concrete design
	Sampling and testing requirements		Steel design

Source: José Luis Henríquez Miranda and Luis Alonso Aguirre López. 2011

Table 2.3 shows the design criteria that need to be consider when designing the piping system. The steps in pipeline design are as follows:

I. The determination of the problem, which includes:
   a. The characteristics of the fluid to be carried, including the flow rate and the allowable headloss;
   b. The location of the pipelines: its source and destination, and the terrain over which it will pass, the location of separator station and the power plant;
   c. The design code to be followed; and
   d. The material to be used.
II. The determination of a preliminary pipe route, the line length and static head difference.
III. Pipe diameter based on allowable head loss;
IV. Structural analysis:
   a. Pipe wall thickness; and
   b. Stress analysis.
V. The stress analysis is performed in pipe configuration until compliance with the code is achieved.
VI. Support and anchor design based on reaction found in the structural analysis.

The pipe specifically for water supply system must be design to achieve appropriate water pressure and flow, and to avoid contamination to potable water. By minimizing water use and making good materials choices, running cost, cut demand on community infrastructure can be reduce as well as reduce harm to environment. To avoid contamination and obtain the right pressure and flow, the system must be suitable for the temperature of water carried. A good designed and installed system is a durable, minimize noise from water flow and from problems such as water hammer and support efficient use of water. All water supply system use a combination of pipes (different dimensions and materials), valves and outlets to deliver water to building users. Designing a water supply system also use storage tanks and pumps. Designing a water supply system include all of these elements so that clean water can be delivered to the user at the appropriate rate and temperature.

CHAPTER 3

METHODOLOGY

3.0 Introduction

This chapter discuss about the methodology that used in conducting the project and to achieve the objectives of the project. To design the piping system, researcher have used the computer software in order to generate technical drawing that gives the visual of piping system in full structure. Solid Work 2012 is selected for design and drawing the piping system.

Data acquisition using Computational Fluid Dynamics (CFD) Simulation is the backbone of this project after technical drawing of the piping system is done. From CFD Simulation, velocity, pressure drop and heat loss are obtained.

. Finally the analysis of the whole project is tabulated, further discussed, and concluded in the following chapter after all of the data acquired being verified. The flow of the research process is simplified in Figure 3.1.

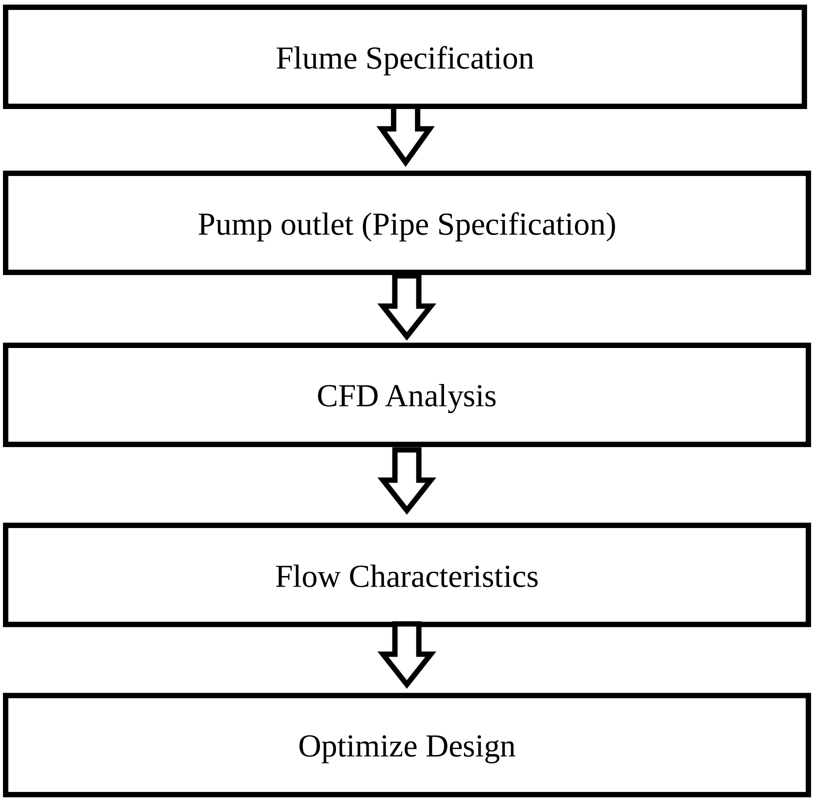

Figure 3.1: Project's flow chart

3.1 FLUME SPECIFICATION

Before design piping system for the flume there are some theories considerations have to be made, to maintain the pressure drop and head loss in the pipe at lowest value. The volume flowrate that needed to be maintained is 9 m^3/s because the velocity that wanted in the flume is 3 m/s with depth of water 1.5 m. The flume specification was set to be 2 meter width, 2 meters height and 30 meters length.

The inlet for each pipe can be assumed as fix volume flowrate 1.125 m^3/s because the pump that chosen has flowrate 1.125 m^3/s, so there will be 8 pumps to delivered 9 m^3/s volume flowrate to the flume.

The main objective of the whole project is to design the flume so that later in future it can be develop significantly. The whole structure of the flume has been designed and translated into technical drawing by using computational software named

Solid Work 2012. However, the researcher only focuses on the design of piping system for the flume. The part of the piping system is marked by red line, as shown below (Figure 3.2). The full structure of the flume is shown in Figure 3.2.

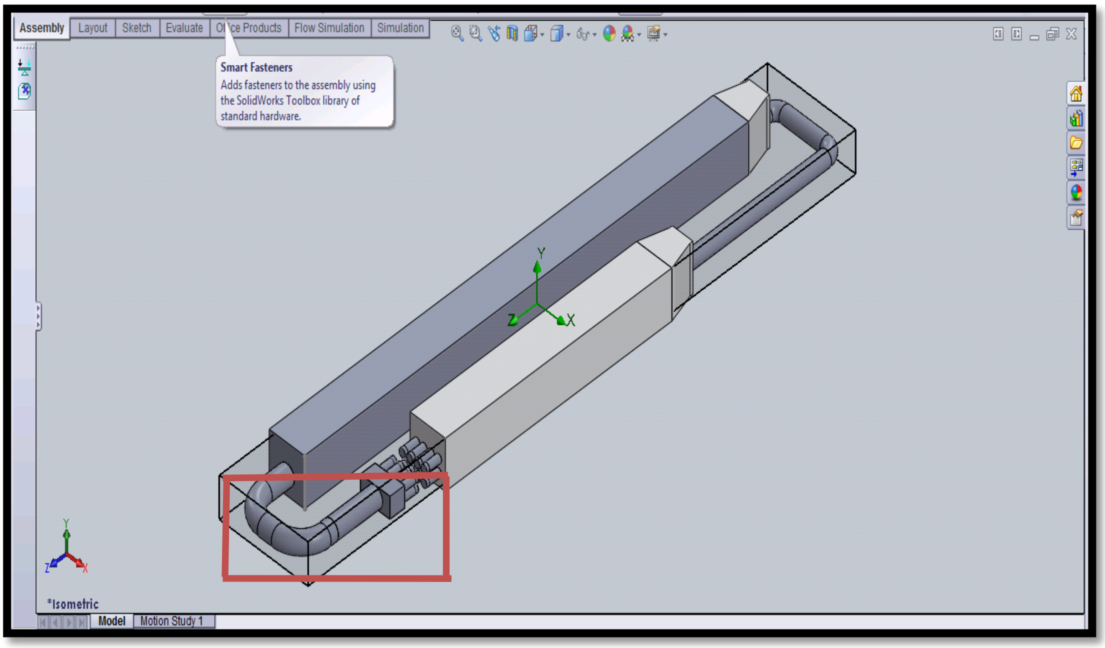

Figure 3.2: Full structure of flume

3.2 Pipe Specification

In this study Computational Fluid Dynamic Simulation (CFX) will be performed to verify the pressure drop, head loss and velocity changes along the pipe, the design chosen that has lowest percentage of pressure drop.

In CFX, it has three main elements which is preprocessing, solving and postprocessing. For the preprocessing, it contains three steps which is geometry, meshing and setting (boundary condition). The steps on how the researcher did the simulation by using CFD Simulation Software (Refer Appendix A).

3.2.1 Materials

The material selection proposed for the piping system is stainless steel. Steel comprises iron and other elements such as carbon, manganese, phosphorus, sulfur, nickel, chromium and more. There are few properties that need to be consider by a designer when specifying steel construction products, which is; strength, toughness, ductility, weldability and durability. Stainless steel is a highly corrosion resistant material that can be used structurally, particularly where a high quality surface finish is required.

Steel piping may be used in water transmission mains due to the cheap initial construction cost of the system. However, care must be taken in the design of the wall thickness of the steel pipe for the particular systems that exist. Steel pipes are more commonly used for raw water mains.

3.2.2 Losses in pipe

3.2.2.1 Pressure Drop(Pressure Loss)

Pressure Drop a quantity of interest in the analysis of pipe flow is the pressure drop ΔP since it is directly related to the power requirements of the fan or pump to maintain flow (Yunus A. Cengel and John M. Cimbala, 2010).

$$\Delta \boldsymbol{P_L} = \left(\boldsymbol{f}\frac{\boldsymbol{l}}{\boldsymbol{D}} + \sum \boldsymbol{K_l}\right)\frac{\boldsymbol{\rho V^2}}{\boldsymbol{2}} \tag{3.2}$$

3.2.2.2 Head loss

The head loss h_l, represents the additional height that the fluid needs to be raised by a pump in order to overcome the frictional losses in the pipe. The head loss is caused by viscosity, and it is directly related to the wall shear stress. Equation above and below are valid for both laminar and turbulent flows.

$$\boldsymbol{h_{l,\,total}} = \left(\boldsymbol{f}\frac{\boldsymbol{l}}{\boldsymbol{D}} + \sum \boldsymbol{K_l}\right)\frac{\boldsymbol{V^2}}{\boldsymbol{2g}} = \frac{\boldsymbol{\Delta P}}{\boldsymbol{\rho g}} \tag{3.3}$$

3.2.2.3 Percentage of Pressure Drop

$$\text{Percentage of Pressure Drop}= \frac{\boldsymbol{\Delta P}}{\boldsymbol{pressure\ inlet}}\,\boldsymbol{x}\ 100\% \quad (3.4)$$

3.2.3 Fluid Flow in Pipe

3.2.3.1 Fluid flow in pipes

Flow can be classified as two types which is laminar or turbulent flow (with a small transitional region between these two). The non-dimensional number, the Reynolds number, Re, is used to determine which type of flow occurs:

$$Re = \frac{\rho V D}{\mu} \quad (3.1)$$

For a pipe

Laminar flow: Re < 2000

Transitional flow: 2000 < Re < 4000

Turbulent flow: Re > 4000

It is important to determine the flow type as this governs how the amount of energy lost to friction relates to the velocity of the flow. And hence how much energy must be used to move the fluid.

3.2.3.2 Laminar and turbulent flow in pipes

It is often useful to be able to determine whether a given pipe flow is laminar or turbulent. This is necessary, because different methods of analysis or different equations are sometimes needed for the two different flow regimes.

Laminar flow takes place for flow situations with low fluid velocity and high fluid viscosity. In laminar flow, all of the fluid velocity vectors line up in the direction of flow. Turbulent flow, on the other hand, is characterized by turbulence and mixing in the flow. It has point velocity vectors in all directions, but the overall flow is in one direction. Turbulent flow takes place in flow situations with high fluid velocity and low fluid viscosity. The figures below illustrate differences between laminar and turbulent flow in a pipe.

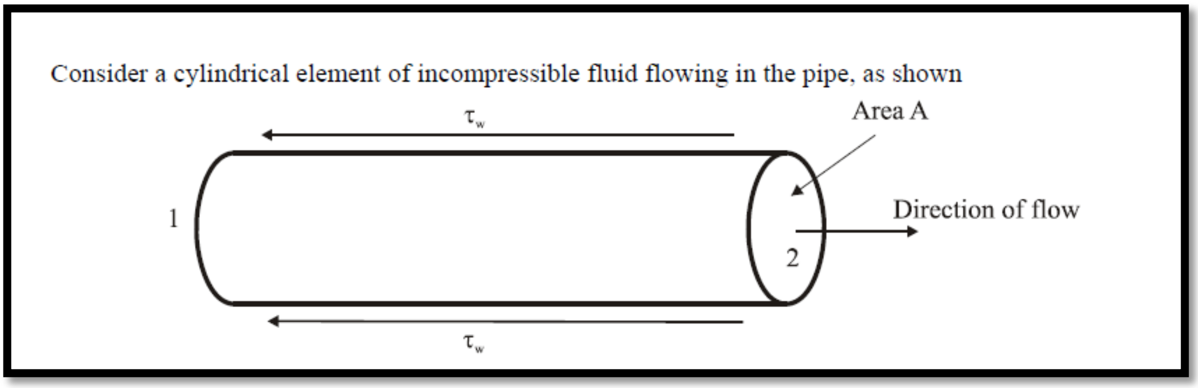

Figure 3.3: Element of a fluid in pipe

3.2.4 Design Parameters

3.2.4.1 Designs A

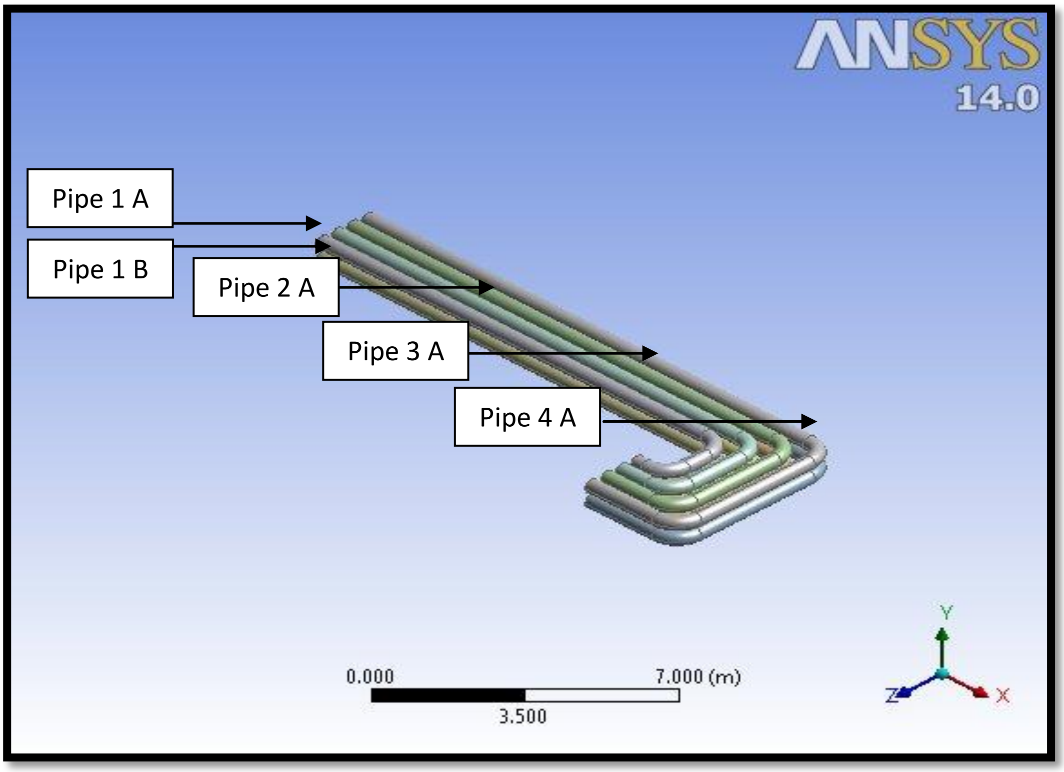

Figure 3.4: Design A

Eight pipes connected from pump to flume inlet

Table 3.5: Parameters Design A

Pipe	Length (m)	Diameter (m)
1 (A & B)	15	0.4064
2 (A & B)	17.50	0.4064
3 (A & B)	20.3	0.4064
4 (A & B)	22.2`	0.4064

3.2.4.2 Design B

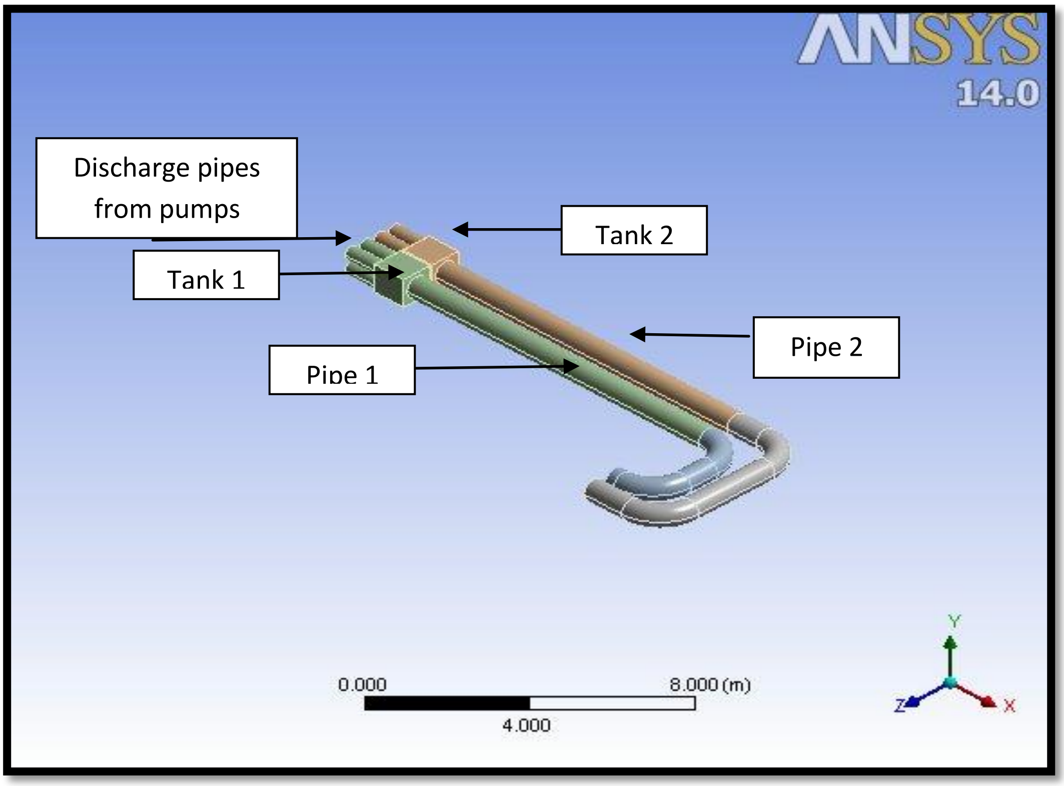

Figure 3.6: Design B

Each four pipes attached to tank and then the tank connected to pipe

Table 3.2: Parameters Design B

Pipe	Length (m)	Diameter (m)
Discharge pipe	1	0.4064
Tank 1 & 2	0.91	Width x height (0.91 x 1)
Pipe 1	14.2	0.6604
Pipe 2	17.88	0.6604

3.2.4.3 Design C

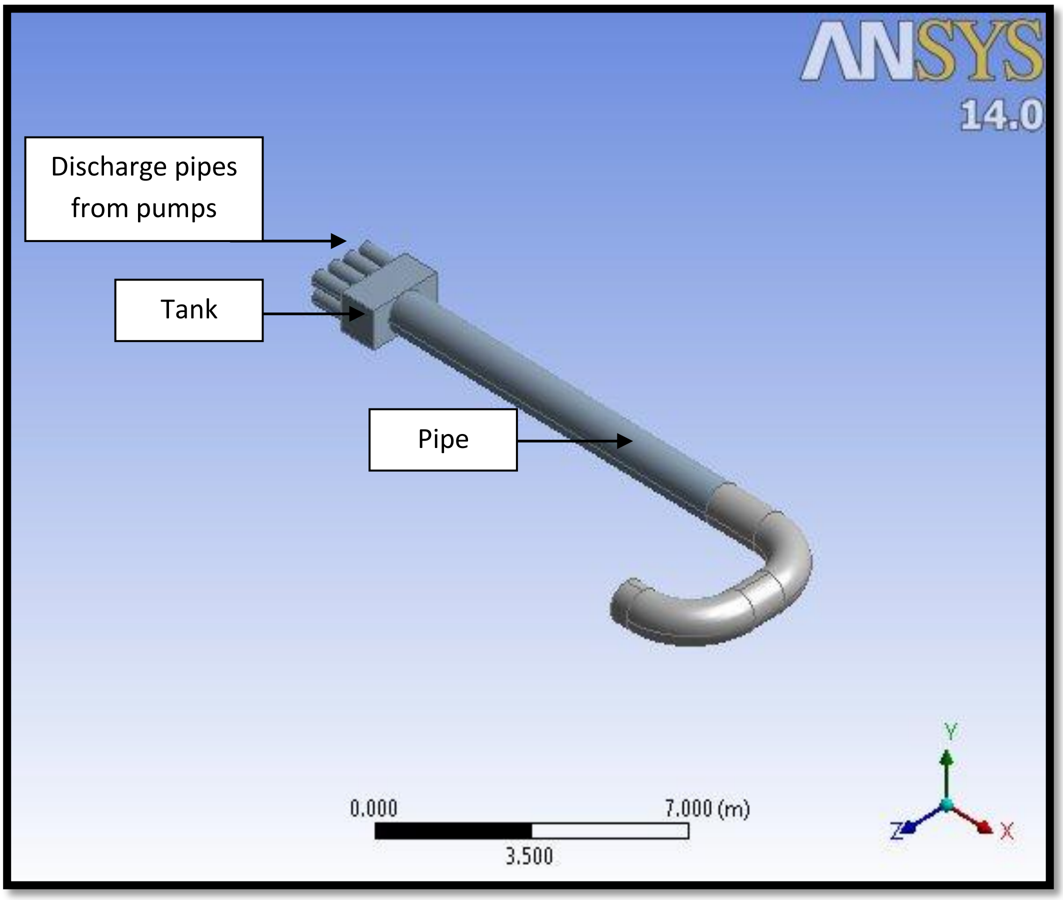

Figure 3.7: Design C

Eight pipes attached to tank, and then the tank connected to one big pipe

Table 3.3: Parameter Design C

Pipe	Length (m)	Diameter (m)
Discharge pipe	1	0.4064
Tank	2	Width x height (2 x 1.02)
Pipe	19	1.016

3.2.4.4 Design D

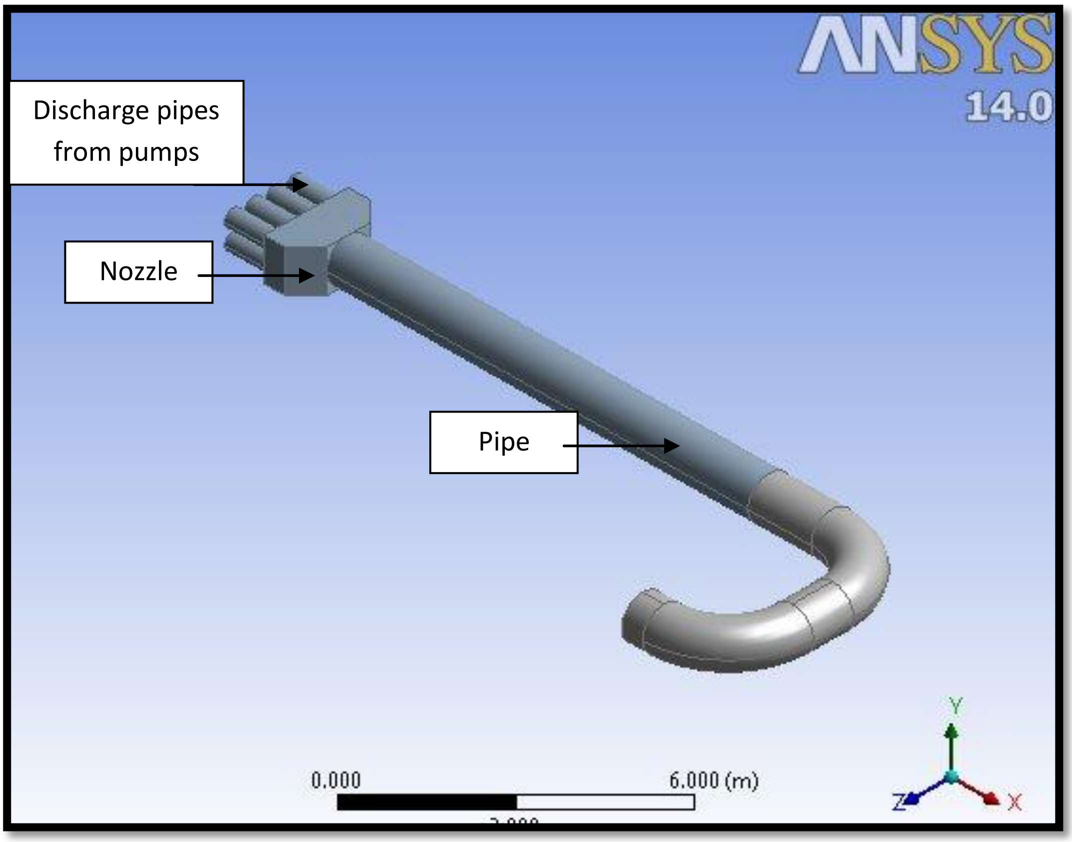

Figure 3.6: Design D

Eight pipes attached to tank (nozzle), then the tank connected to one big pipe

Table 3.4: Parameters Design D

Pipe	Length (m)	Diameter (m)
Discharge pipe	1	0.4064
Tank	2	Width x height (2 x 1.02)
Pipe	19	1.016
Nozzle: 45°	Edge: 0.490 m	

3.3 Boundary condition

The set of conditions specified for the behavior of the solution to a set of diffetential equations at the boundary of its domain. Boundary conditions are important in determining the mathematical solutions to many physical problems. The boundary condition used in the simulation of piping system design is : mass flow rate at the inlet = 1125 kg/s, wall roughness = 0.002 mm because the material used is stainless steel, static pressure at the outlet = 0 Pa .

CHAPTER 4

RESULTS AND DISCUSSIONS

4.0 Introduction

In this chapter, researcher discussed the result from the experiment and simulation. Process of data analysis is discussed in detail to meet the objectives of the study and it is being done gradually involving the elements that needed for the researcher to analyze the data collected. Researcher directly used computer software known as Computational Fluid Dynamic (CFD) to present the research findings in order to meet the objectives of study. Findings that involved pressure drop, head loss and velocity are simplified and presented in the form of tables and graphs.

4.1 Experimental analysis result

From the experiment analysis, a set of data collected when the simulation of pipe is running at the specific boundary condition which is mass flow rate for each pipe is 1125 kg/s. There are four pipe designs (A, B, C, and D) that have been done and each one of it simulated in order to get pressure drop, head loss and velocity. The findings is analysed and compared in order to select the best design that has less percentage of pressure drop. The result for all designs that simulated as follows.

4.1.1 Design A

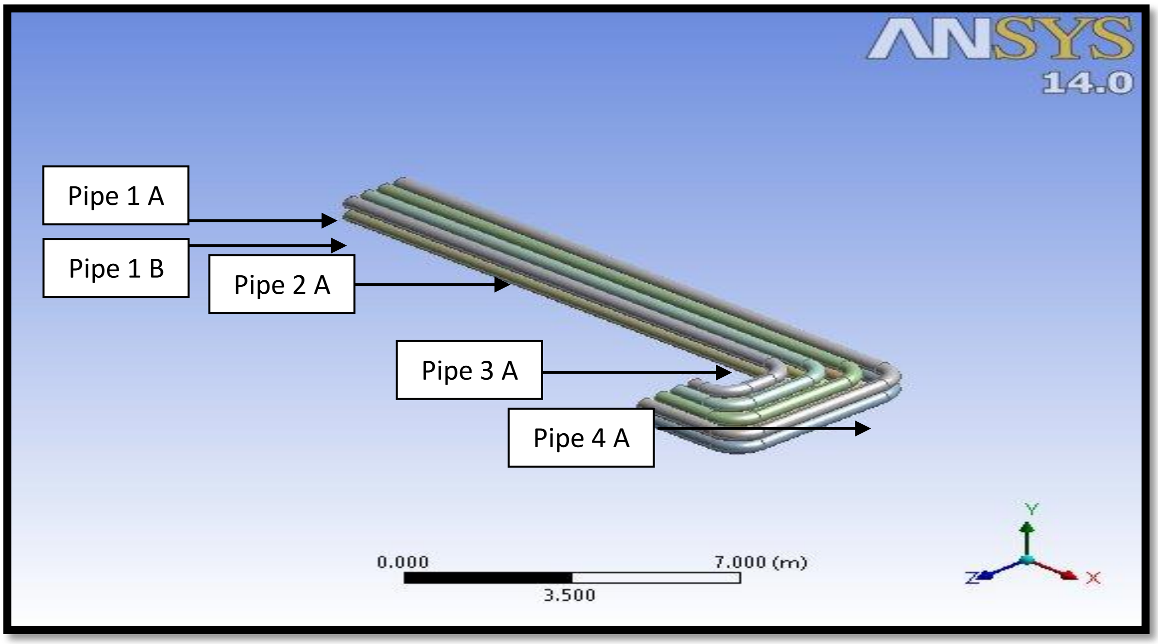

Figure 4.1: Design A Eight pipes

Table 4.1: Data of Design A

Pipe 1 [A&B]			Pipe 2 [A&B]			Pipe 3 [A&B]			Pipe 4 [A&B]		
L [m]	P_t [kP]	V [m/s]	L [m]	P_t [kPa]	V [m/s]	L [m]	P_t [kP]	V [m/s]	L [m]	P_t [kP]	V [m/s]
0.00	62.3	8.84	0.00	64.1	8.84	0.00	65.2	8.84	0.00	68.1	8.84
1.33	62.3	9.02	1.39	64.1	9.03	1.00	65.2	8.97	1.56	68.1	9.05
2.67	62.3	9.20	2.78	64.1	9.21	3.00	65.2	9.24	3.11	68.1	9.25
4.00	62.3	9.36	4.17	64.1	9.38	5.00	65.3	9.47	4.67	68.1	9.44
5.33	62.4	9.53	5.56	64.1	9.55	7.00	65.3	9.70	6.22	68.1	9.61
6.67	62.6	9.71	6.94	64.3	9.73	9.00	65.6	9.91	7.78	68.2	9.79
8.00	62.9	9.90	8.33	64.5	9.91	10.0	65.8	10.00	9.33	68.4	9.94
9.33	63.5	10.07	9.72	65.0	10.07	11.0	65.9	10.08	10.9	68.6	10.06
10.7	63.9	10.21	11.1	65.3	10.19	12.0	65.6	10.12	12.4	68.0	10.12
12.0	62.3	10.16	12.5	63.6	10.13	13.0	64.1	10.04	14.0	64.9	9.93
15.0	41.7	9.15	17.5	39.7	8.92	20.3	37.9	8.72	22.3	35.4	8.42

Table 4.1 shows the data distribution of Design A, which consist of four pipes A & B. Where L is length, P_t is total pressure and V is velocity. The size of pipe 1-4, A and B is in the same size which is length and diameter (refer to Table 4.1). Thus, for each pair of the pipe is represented as one line in the graph (Figure 4.2). The data distribution from the table above is translated in to graph below.

4.1.1.1 Total pressure drop and head loss

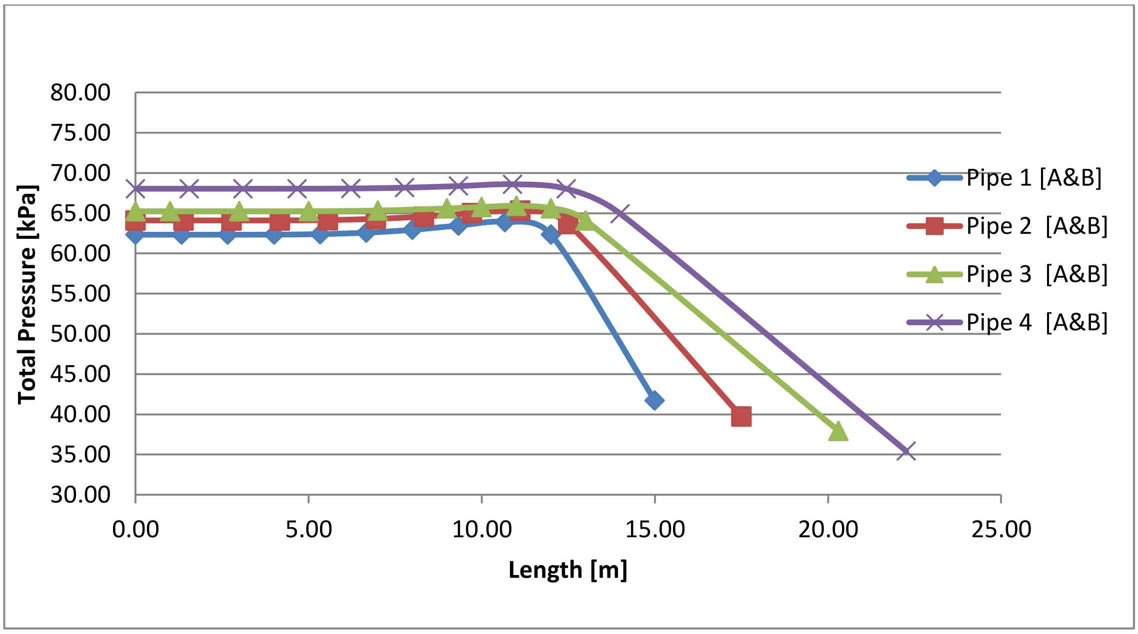

Figure 4.2: Total Pressure Design A

a) For Pipe 1 A and B

Since pipe 1A is the same size with pipe 1B so, pipe 1A equal Pipe 1B.

In pipe 1 (A and B) after simulation and calculated, researcher get the total pressure drop and percentage of both pipes are 20.622 kPa and 33.1% respectively.

The head loss (h_l) is calculated by using head loss equation (3.3), and the value obtained is 2.1 m.

b) Pipe 2 A and B

Since pipe 2A is the same size with pipe 2B so, pipe 2A equal Pipe 2B.

In pipe 2 (A and B) after simulation and calculated, researcher get the total pressure drop and percentage of both pipes are 24.38 kPa and 38.1% respectively.

The head loss is calculated by using head loss equation (3.3), and the value obtained is 2.8 m.

c) Pipe 3 A and B

Since pipe 3A is the same size with pipe 3B so, pipe 3A equal Pipe 3B.

In pipe 3 (A and B) after simulation and calculated, researcher get the total pressure drop and percentage of both pipes are 27.301 kPa and 41.85% respectively.

The head loss is calculated by using head loss equation (3.3), and the value obtained is 2.5 m.

d) Pipe 4 a and b

Since pipe 4A is the same size with pipe 4B so, pipe 4A equal Pipe 4B.

In pipe 4 (A and B) after simulation and calculated, researcher get the total pressure drop and percentage of both pipes are 32.63 kPa and 47.94 % respectively.

The head loss is calculated by using head loss equation (3.3), and the value obtained is 3.34 m.

From the calculation (refer to appendix B), the pressure drop and head loss difference from pipe to another pipe. These differences due to the differences in pipe length, which mean the pressure drop and head loss, are proportional to the pipe length (the longer pipe length, the higher pressure drop and head loss).

The average pressure drop and head loss of Design A calculated below, to make comparison with other design in order to select which design has less pressure drop and head loss.

Average pressure drop, Percentage of pressure drop and head loss

By using average pressure drop (ΔP_{avg}), percentage of pressure drop(3.4) and head loss(3.2) equations we got ΔP_{avg}= 26.23 kPa, average percentage of pressure drop is 40.4% and average head loss is 2.7 m.

(Calculation on how researcher get pressure drop, percentage of pressure drop, head loss and their average refer to appendix B)

4.1.1.2 Velocity

Figure 4.3 below, shows the velocity changes in Design A, the velocity is proportional to the pipe length however, the velocity increase when the pipe in straight line and after the elbow, the velocity decrease, due to minor loses.

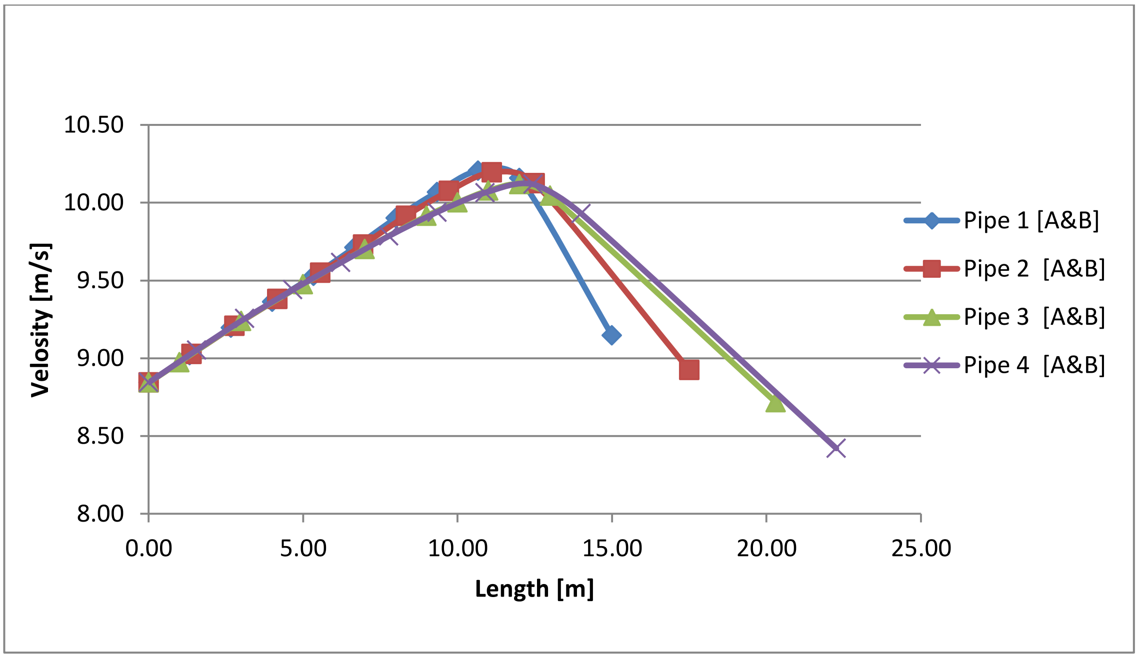

Figure 4.3: Velocity Design A

Table 4.1a: Inlet and Outlet velocity Design A

Pipe	Velocity inlet (m/s)	Velocity outlet (m/s)
1 (A&B)	8.84	9.15
2 (A&B)	8.84	8.92
3 (A&B)	8.84	8.72
4 (A&B)	8.84	8.42

From the simulation, velocity changes from the inlet to outlet obtained. Inlet velocity starts at the same point which is 0.00 m in length with velocity 8.84 m/s. In the outlet, the velocity changes from pipe to another due to pipe length.

4.1.2 Design B

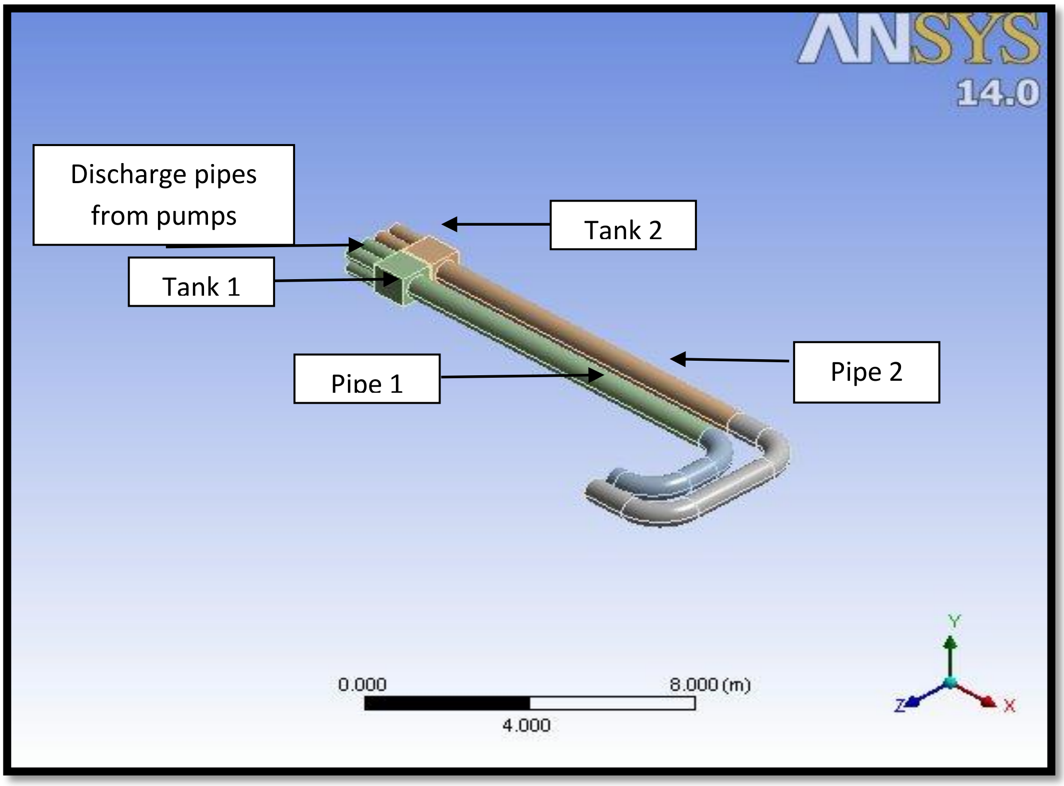

Figure 4.4: Design B

Design B, consists of two pipes (pipe 1 and pipe 2). The size of pipe 1 and pipe 2 is difference in terms of pipe length. Design B has eight discharge pipes from pump, where each four pipes is attached to one tank, and the tank is attached to one pipe that has longer diameter than the four discharge pipes. Table 4.2 shows the data distribution of Design B. The properties measure for each pipe is length, total pressure and velocity

Table 4.2: Data Distribution Design B

Pipe 1			Pipe 2		
Length [m]	Total Pressure [kPa]	Velocity [m/s]	Length [m]	Total Pressure [kPa]	Velocity [m/s]
0.00	155.74	8.84	0.00	162.60	8.84
0.89	156.01	8.88	0.84	162.80	8.92
1.00	123.79	2.04	1.00	131.22	2.34
1.56	138.90	4.14	1.66	143.47	3.61
2.00	144.12	11.02	1.99	151.65	10.72
3.11	143.49	15.30	3.19	147.88	15.09
4.22	132.66	14.50	4.38	135.38	14.19
5.32	125.89	14.09	5.58	130.00	13.85
6.43	122.15	13.88	6.77	127.18	13.73
7.54	119.89	13.80	7.97	125.56	13.71
8.65	118.12	13.77	9.16	123.92	13.69
9.75	116.94	13.79	10.18	122.62	13.71
10.86	115.41	13.77	11.37	120.91	13.66
11.97	114.53	13.70	12.57	120.51	13.75
14.08	110.97	13.96	17.14	111.08	13.99
16.20	105.81	14.55	19.88	98.75	14.07

The data distribution from the table above is translated in to graph below.

4.1.2.1 Total Pressure Drop and Head loss

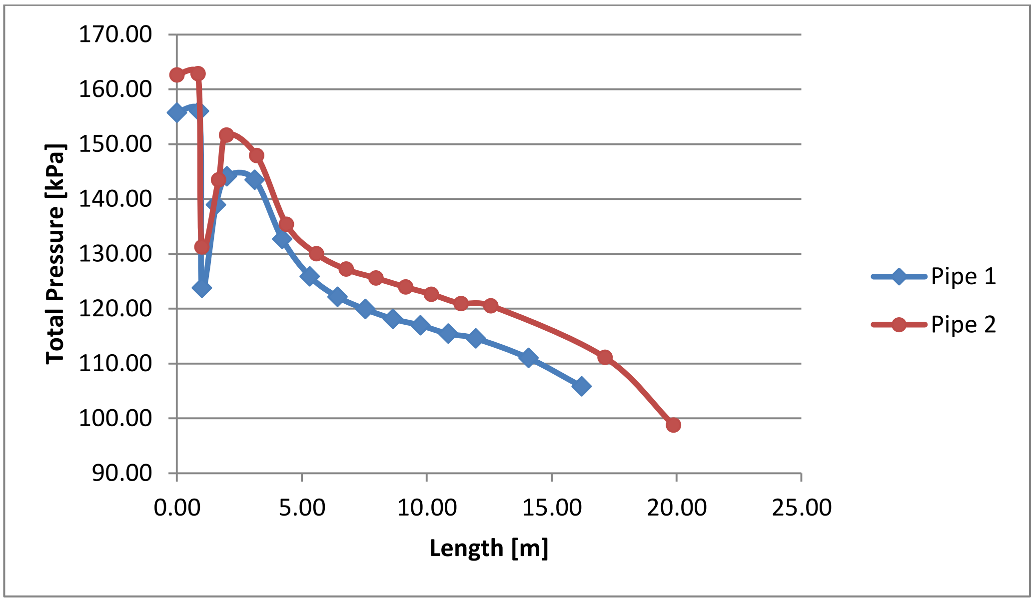

Figure 4.5: Total Pressure Design B

From the graph, when the length of pipes is 0.00 m , the total pressure inlet for Pipe 1 is 155.74 kPa and Pipe 2 is 162.60 kPa. At 1.00 m, total pressure for pipe 1 and pipe 2 is drop to 123.79 kPa and 131.22 kPa respectively. The total pressure drop is caused by wide area of the tank. However, the total pressure increases again at the length of 1.56 m until 3.11 m and 1.66 m until 3.19 m for pipe 1 and pipe 2. After this point of pipe, the total pressure is decrease due to pipe length. The total pressure drop and head loss calculated as below.

a) Pipe1

In pipe 1 after simulation and calculation, researcher get the total pressure drop and percentage of pipe is 49.93 kPa and 32.05 % respectively

Head loss (h_l)

The head loss is calculated by using head loss equation (3.3), and the value obtained is 5.1 m.

b) Pipe 2

In pipe 2 after simulation and calculation, researcher get the total pressure drop and percentage of pipe is 63.851 kPa and 39.2 % respectively

The head loss is calculated by using head loss equation (3.3), and the value obtained is 6.5 m.

From the calculation above, the pressure drop and head loss difference from pipe to another pipe. These differences due to the differences in pipe length, which mean the pressure drop and head loss, are proportional to the pipe length (the longer pipe length, the higher pressure drop and head loss).

The average pressure drop and head loss of Design B calculated below, to make comparison with other design in order to select which design has less pressure drop and head loss.

Average Pressure Drop, Percentage of Pressure Drop and Head Loss for Design B

From the simulation and calculation we got average Pressure Drop (ΔP_{avg}) equal 56.8905 kPa, Average Percentage equal 35.7 % and aAverage head loss for Design A is 5.8 m.

(Calculation on how researcher get pressure drop, percentage of pressure drop, head loss and their average refer to appendix C)

4.1.2.2 Velocity

Figure 4.6 below**,** shows the velocity changes in Design B.

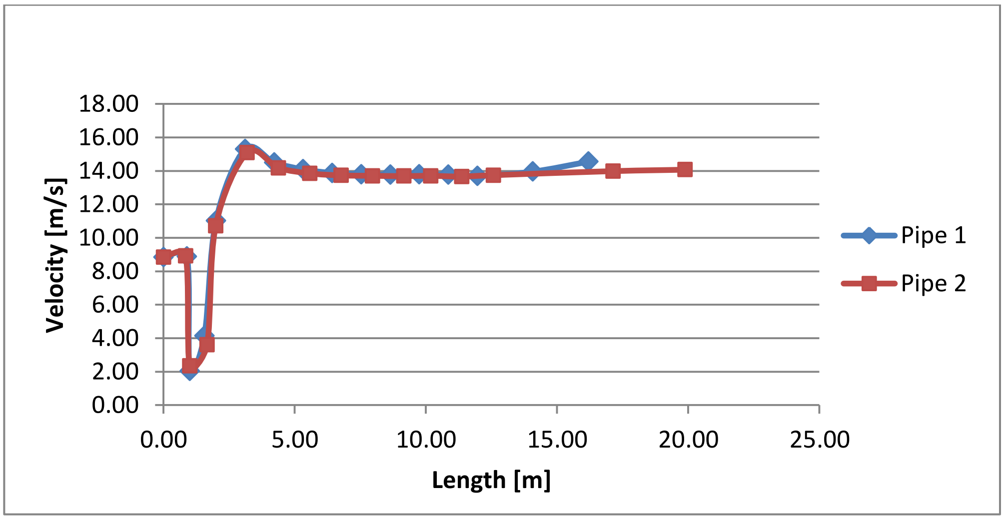

Figure 4.6: Velocity Design B

For the velocity in Design B, when both pipe at 0.00 m, the velocity is 8.84 m/s, as the length of pipe increase to 1.00 m, the velocity is drop to 2.04 and 2.34 because of the wide area of tank then it increase back because the tank is attached to pipe that has smaller diameter than tank. Then the velocity is maintained higher along the pipe. Summary of velocity at the inlet and outlet for both pipes as shown below.

Table 4.2a: Velocity Design B

	Inlet Velocity [m/s]	**Outlet Velocity [m/s]**
Velocity pipe 1	8.84	14.55
Velocity pipe 2	8.84	14.07

From the simulation, velocity changes from the inlet to outlet obtained. Inlet velocity starts at the same point which is 0.00 m in length with velocity 8.84 m/s. In the outlet, the velocity changes from pipe to another due to pipe length and diameter.

4.1.3 Design C

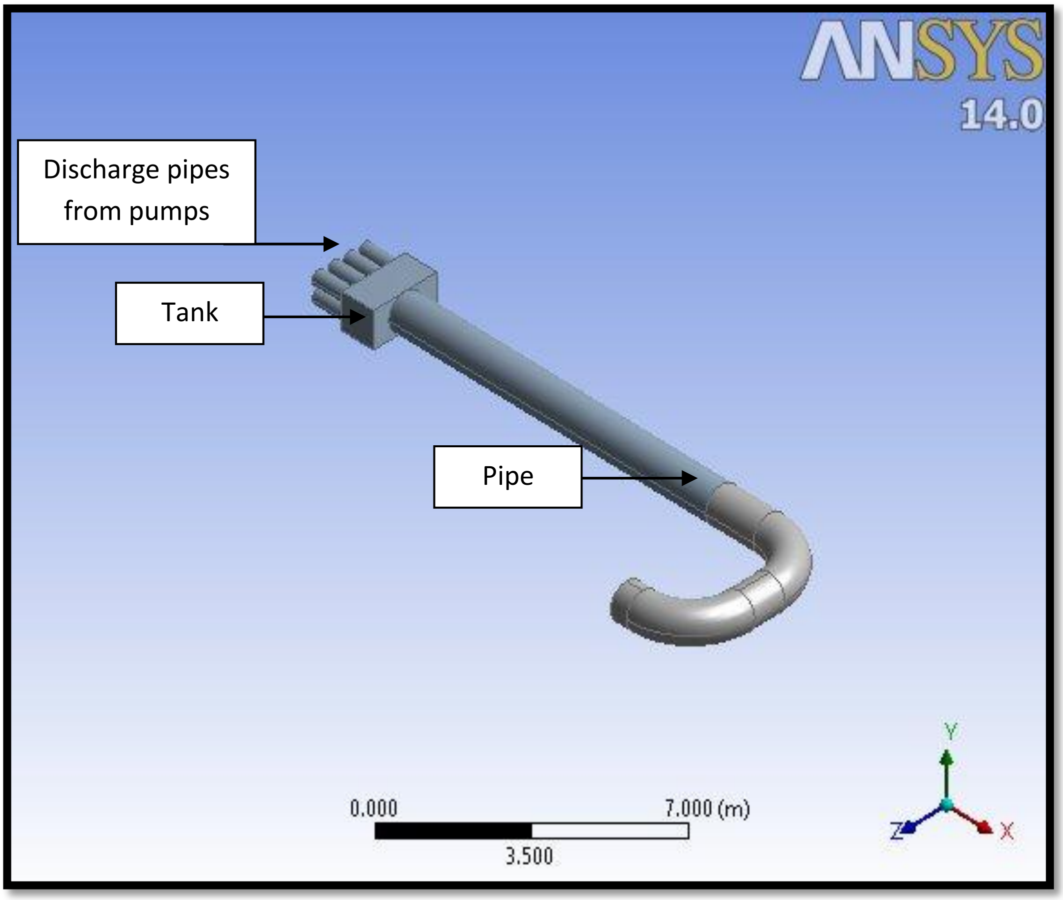

Figure 4.7: Design C

Design C has eight discharge pipes from pump, and then attached to one tank, and the tank is attached to one pipe that has longer diameter than the eight discharge pipes. Table 4.3 shows the data distribution of Design C. The properties measure for Design C is length, total pressure and velocity

Table 4.3: Data Distribution Design C

Design C		
Length [m]	**Total Pressure [kPa]**	**Velocity [m/s]**
0.00	112.04	8.84
0.84	112.34	8.93
1.00	71.74	1.46
1.56	90.39	5.22
2.00	96.70	9.63
3.22	99.48	13.07
4.44	93.06	12.52
5.67	89.33	12.22
6.89	85.63	11.91
8.11	83.32	11.74
9.33	81.65	11.62
10.56	80.67	11.60
11.78	79.93	11.58
13.00	79.69	11.63
16.74	75.98	11.77
21.00	69.00	11.76

The data distribution from the table above is translated in to graph as follow.

4.1.3.1 Total Pressure Drop and Head loss

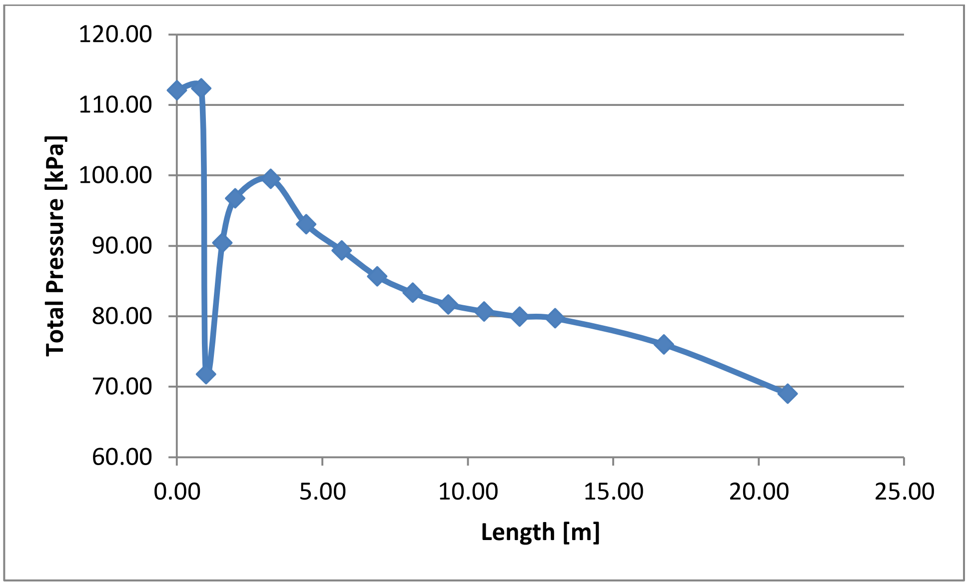

Figure 4.8: Total Pressure Design C

From the graph, when the length of pipes is 0.00 m , the total pressure inlet for Pipe is 112.04 kPa. At 1.00 m, total pressure is drop to 71.74 kPa. The total pressure drop is caused by wide area of the tank. However, the total pressure increases again at the length of 1.56 m until 3.22 m. After this point of pipe, the total pressure is decrease continuously due to pipe length and diameter. The total pressure drop and head loss calculated as below.

After simulation and calculation, researcher get the total pressure drop and percentage of pipe is 43.04 kPa and 38.4 % respectively

The head loss is calculated by using head loss equation (3.3), and the value obtained is 4.4 m. (Calculation on how researcher get pressure drop, percentage of pressure drop and head loss for Design C refer to appendix D)

4.1.3.2 Velocity

Figure 4.9 below, shows the velocity changes in Design C.

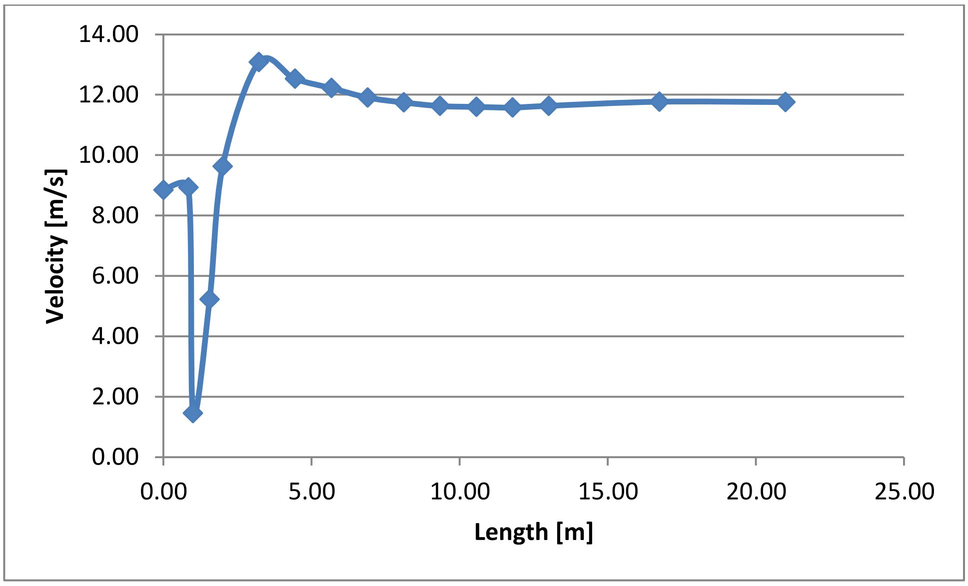

Figure 4.9: Velocity Design C

For the velocity in Design C, when pipe at 0.00 m, the velocity is 8.84 m/s, as the length of pipe increase to 1.00 m, the velocity is drop to 1.46 m/s because of the wide area of tank then it increase back because the tank is attached to pipe that has smaller diameter than tank. Then the velocity is maintained higher along the pipe. Summary of velocity at the inlet and outlet in pipe are:

i. Inlet Velocity 8.84 m/s
ii. Outlet Velocity 11.76 m/s

From the simulation, velocity changes from the inlet to outlet obtained. Inlet velocity starts at 8.84 m/s when 0.00 m in length and outlet velocity 11.76 m/s at 21.0 m in length. The velocity changes in pipe due to pipe length and diameter.

4.1.4 Design D

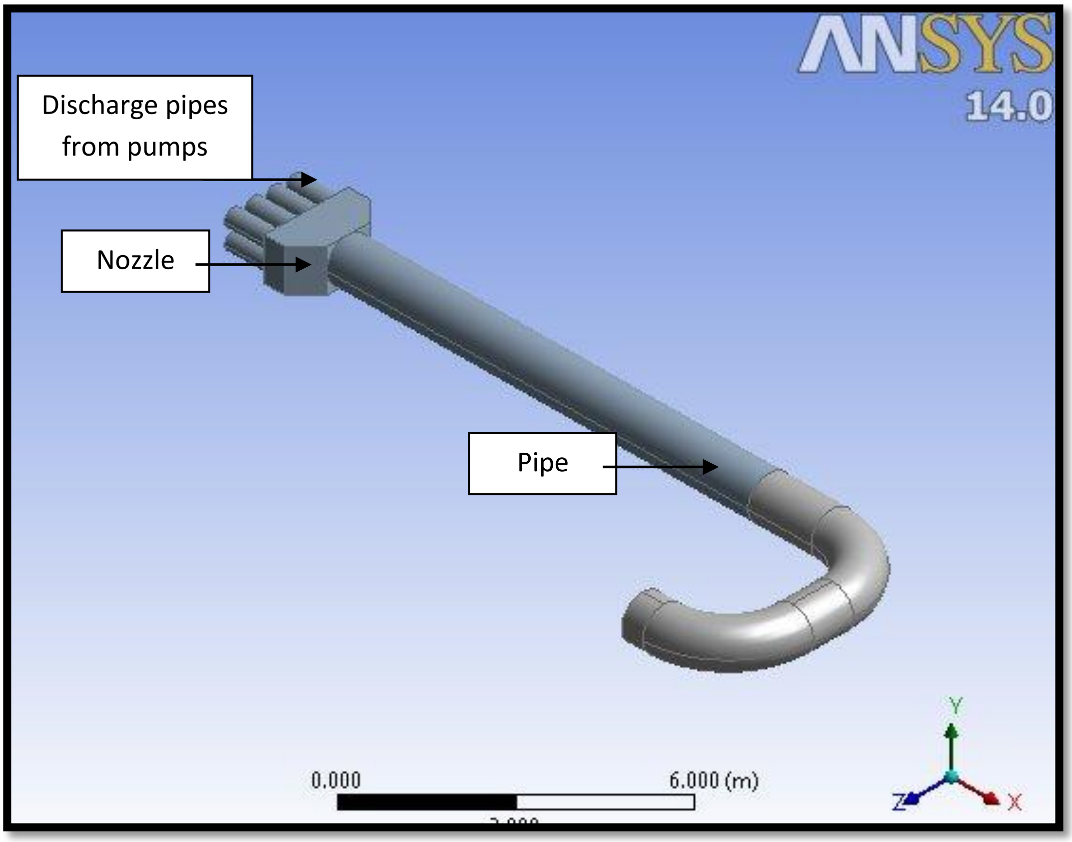

Figure 4.10: Design D

Design D has eight discharge pipes from pump, and then attached to Nozzle, and the nozzle is attached to one pipe that has longer diameter than the eight discharge pipes. Table 4.4 shows the data distribution of Design C. The properties measure for Design C is length, total pressure and velocity

Table 4.4: Data Distribution Design D

Design D		
Length [m]	**Total Pressure [kPa]**	**Velocity [m/s]**
0	104.83	8.84
0.7	105.02	8.94
0.9	104.89	8.83
1.2	76.60	1.20
1.6	90.43	6.37
2	94.22	10.30
3.12	96.76	12.62
4.24	92.68	12.32
5.37	88.91	12.00
6.49	86.31	11.80
7.61	84.85	11.73
8.73	83.58	11.67
9.86	82.96	11.65
10.98	82.41	11.65
12.1	81.80	11.66
17.13	78.64	11.96
21	72.60	12.03

The data distribution from the table above is translated in to graph below

4.1.4.1 Total Pressure and Head loss

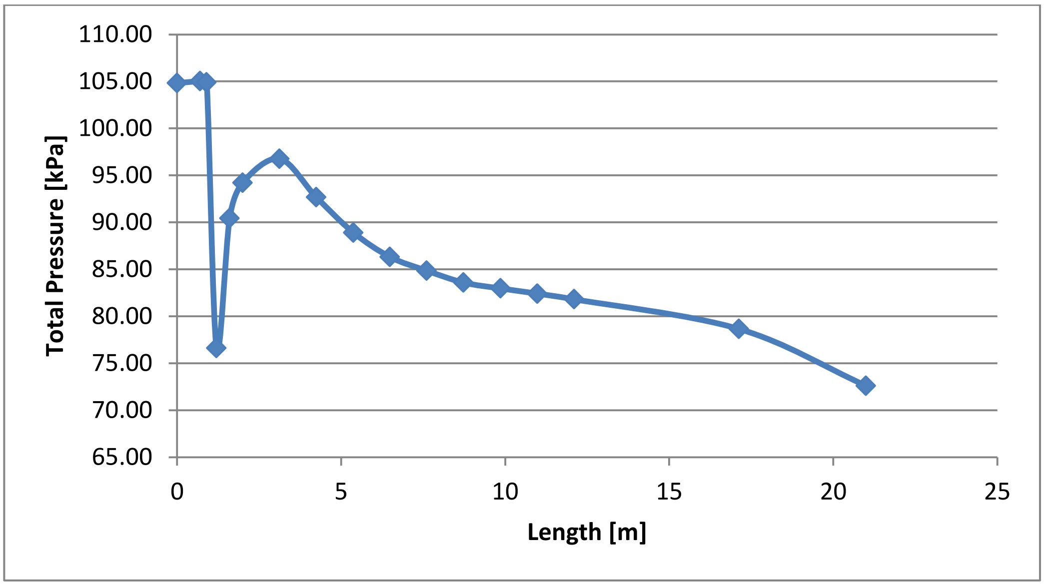

Figure 4.11: Total Pressure Design D

From the graph, when the length of pipes is 0.00 m , the total pressure inlet for Pipe is 104.83 kPa. At 1.2 m, total pressure is drop to 76.60 kPa, the total pressure drop is caused by wide area of the nozzle. However, the total pressure increases again at the length of 1.6 m until 3.12 m. After this point of pipe, the total pressure is decrease continuously due to pipe length and diameter. The total pressure drop and head loss calculated as below.

After simulation and calculation, researcher get the total pressure drop and percentage of pipe is 32.23 kPa and 30.7 % respectively

The head loss is calculated by using head loss equation (3.3), and the value obtained is 3.1 m. (Calculation on how researcher get pressure drop, percentage of pressure drop and head loss for Design D refer to appendix E)

4.1.4.2 Velocity

Figure 4.12 below**,** shows the velocity changes in Design C.

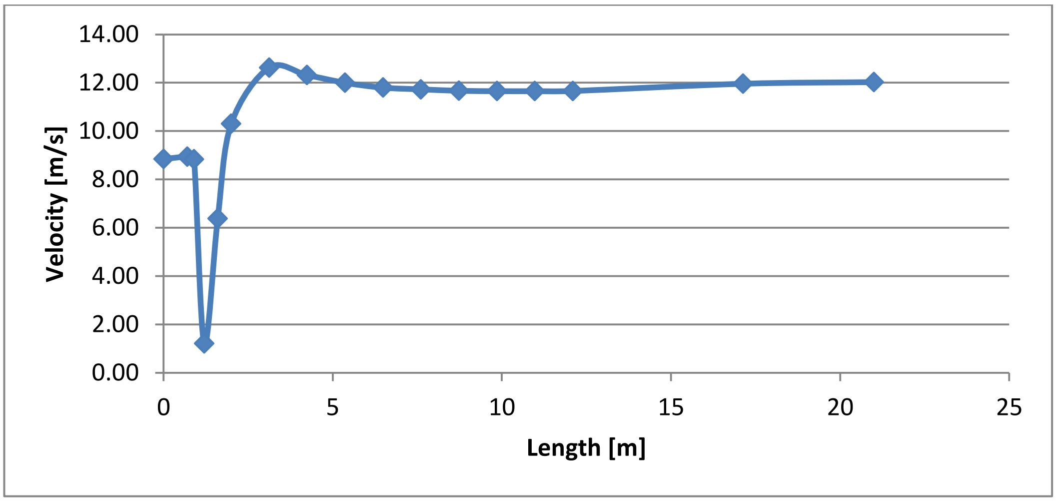

Figure 4.12: Velocity Design D

For the velocity in Design D, when pipe at 0.00 m, the velocity is 8.84 m/s, as the length of pipe increase to 1.2 m, the velocity is drop to 1.20 m/s because of the wide area of nozzle then it increase back because the nozzle is attached to pipe that has smaller diameter than nozzle. Then the velocity is maintained higher along the pipe. Summary of velocity at the inlet and outlet in pipe are:

i. Inlet Velocity 8.84 m/s
ii. Outlet Velocity 12.01 m/s

From the simulation, velocity changes from the inlet to outlet obtained. Inlet velocity starts at 8.84 m/s when 0.00 m in length and outlet velocity 12.01 m/s at 21.0 m in length, the velocity changes in pipe due to pipe length and diameter.

4.2 Comparison between designs.

Table 4.5: Design Comparison

Design Comparison			
Design	$\Delta P\ [kPa]$	ΔP %	$h_L[m]$
A	26.23	40.2	2.7
B	56.9	35.6	5.8
C	43.04	38.4	4.4
D	32.23	30.7	3.1

In comparison between four designs (A, B, C, D), research findings shows that the design that has less percentage of pressure drop is the best design among others. Hence, from the table below, it can be seen that Design D has less percentage of pressure drop which is 30.7%.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.0 Introduction

This chapter sums up the findings from the study analysis of Chapter 4. Scope of data analysis from chapter 4 is to answer the problem statement and also the objectives of research. This chapter also includes recommendation for future researcher, if they have to do this field.

5.1 Conclusion

In the conclusion, all the objectives of the research are achieved. From the design of piping system until analysis of the pressure drop, head loss and velocity has been determined.

The result obtained from the simulation of Design A is average percentage of pressure drop and average head loss which is 40.2% and 2.7 m respectively. For the velocity, inlet velocity is 8.84 m/s and average outlet velocity is 8.75 m/s

The result obtained from the simulation of Design B is average percentage of pressure drop and average head loss which is 53.6 % and 5.8 m respectively. For the velocity, inlet velocity is 8.84 m/s and average outlet velocity is 14.31 m/s

The result obtained from the simulation of Design C is percentage of pressure drop and head loss which is 38.4 % and 4.4 m respectively. For the velocity, inlet velocity is 8.84 m/s and outlet velocity is 11.76 m/s.

The result obtained from the simulation of Designs D is percentage of pressure drop and head loss which is 30.7% and 3.1 m respectively. For the velocity, inlet velocity is 8.84 m/s and outlet velocity is 12.03 m/s.

To choose the best design is determined by less percentage of pressure drops, hence from the research findings Design D is the best design since it has the less percentage of pressure drop.

5.2 Recommendation

My recommendation for the future of this project is:

i) Student to be familiar, fully prepared, and sharpened with their CFD skills and knowledge before taking up this kind of analytical title which requires a vast amount of patient and independence
ii) For Design A I recommend to increase the diameter at outlet of from pipe to another to make the pipe velocity outlet at the same value.
iii) Reduce the size of Nozzle tank to reduce the percentage of pressure drop.

REFERENCES

Arthur, R, M, Nowell and Jumars, A. P 1987. Flumes: theoretical and experimental considerations for simulation of benthic environments. Rev, 1987, 25, 91-112. University of Washington, USA.

Bankston, J. D and Baker, F. E 1994. Piping systems. *Southern Regional Aquaculture Center*: Publication No. 373.

Bengston, H. H 2005. Pipe flow-friction factor calculation with excel. *Continuing Education and Development, Inc*. New York, USA.

Cengel, Y. A and Cimbila. J. M 2010. *Fluid mechanics fundamentals and applications*. USA: McGraw-Hill.

Devki Energy 2006. Best practice manual- fluid piping systems.

Edwards, J. E 2010. Piping workbook. *Chemstations Engineering Advance*. Version 6.2: Tesside, UK.

Engineering & piping design guide- fiber glass systems reinforced piping system, 2007 Manual No. E5000. USA.

Haapala, A. 2010. Paper machine white water treatment in channel flow. University of Oulu, Finland.

Haktair, T. and Ardichoglu, M. 2004. Numerical modeling of Darcy-Weisbach friction factor and branching pipes problems. Elsevier, Advanced in Engineering Software 35 (2004) 773-779.

Liu, H 2003 . *Pipeline Engineering*. USA: LEWIS PUBLISHERS.

Menon, E . S 2005. *Piping Calculations Manual*. USA: McGRAW-Hill.

Merkley, G. P 1995 . flume for open- channel flow measurement.

Miranda, J. L. H and Lopez, L. A. A 2011. Piping design: the fundamentals. *United Nation University*. January 16-22, 2011.

Practical piping course. *Engineering Design & Analysis Ltd.*

Replogle, J. A and Clemments, A. J. Hydraulics design of flow measuring structures. USA.

Subulakshmi, A 2004. Open channel hydraulics-applied hydraulics engineering. Einstein College of Engineering.

Silowash, B 2010 .*Piping System Manual*. USA: McGraw-Hill.

Savic, D. A and Walters, G. A 1996. Genetic operators and constraint handling for pipe network optimization.

Sturm, T. W 2001. Open channel hydraulics. *McGraw Hill Higher Edeucation*. USA.

APPENDIXES A

Simulation Steps

The steps on how the researcher did the simulation by using CFD Simulation Software.

Step 1: open the program > choose the Fluid Flow (CFX) from Analysis System menu.

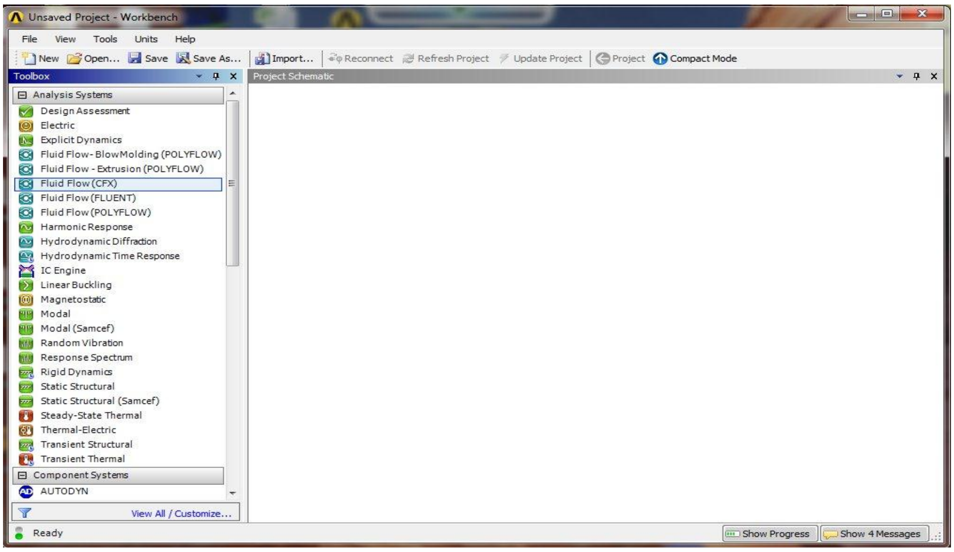

Figure 1: Click Fluid Flow (CFX) from Analysis System menu.

Step 2: Drag the Fluid Flow (CFX) to the Project Schematic

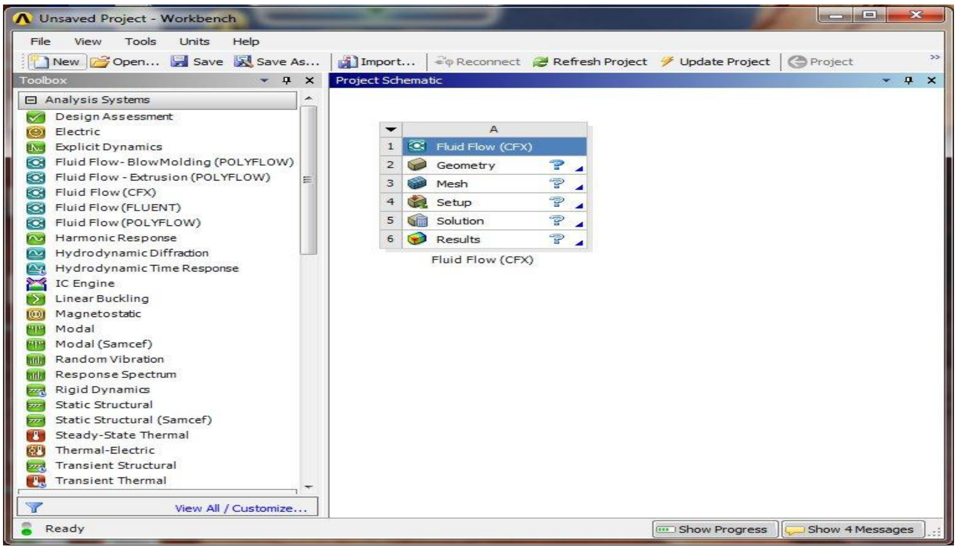

Figure 2: CFX Workbench

Step 3: From the Project Schematic, choose Geometry and right click, select New Geometry. New page of DesignModeler (DM) will appear. From the DesignModeler page, the ANSYS Workbench menu will appear, then choose Meter on the desire length unit then click ok as shown in figure 3.

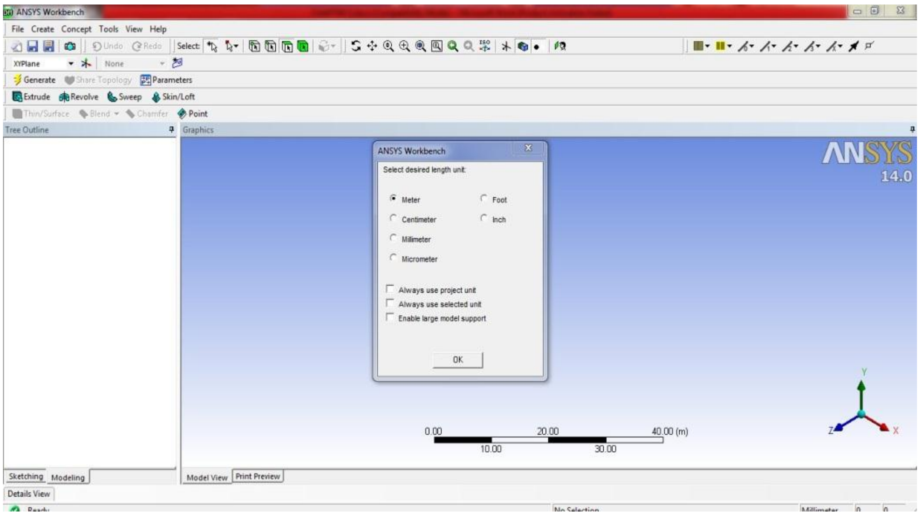

Figure 3: ANSYS Workbench

Step 4: Go to File at the menu bar, then select Import External Geometry File, > choose the designs that already draw in Solid Work 2012 that save as IGS file. Then click Generate at the menu bar, the design will appear at the DM sheet (Figure 4)

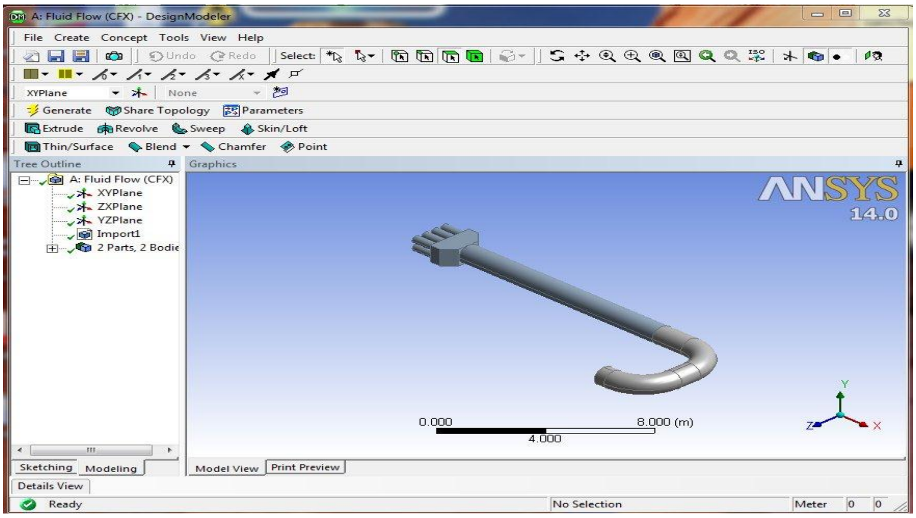

Figure 4: CFX DesignModeller

Step 5: From Project Schematic, double click on Mesh, Meshing [ANSYS ICEM CFD] page will appear > at the Generate Mesh > update. As shown in figure 5

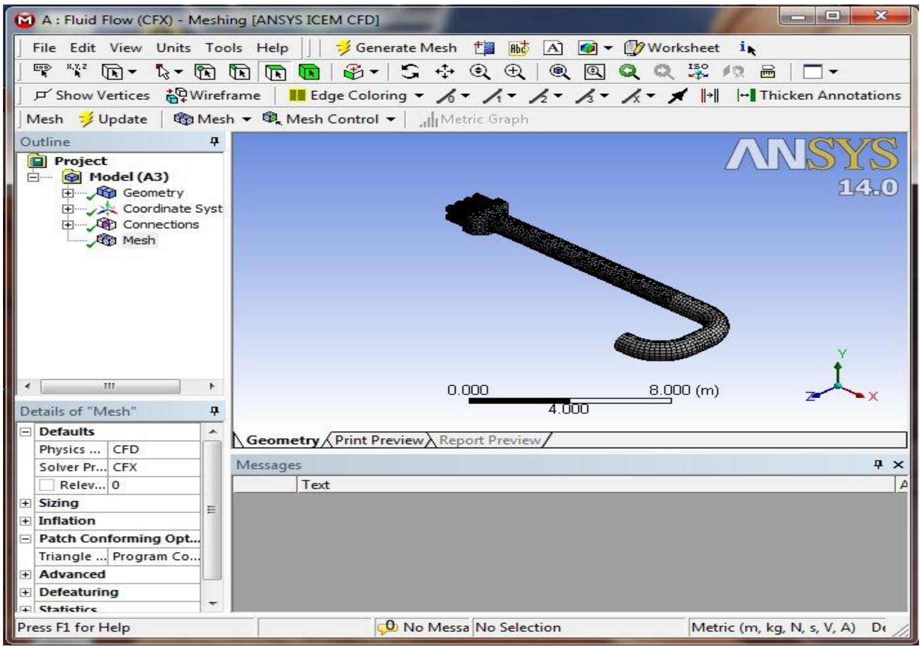

Figure 5: CFX Meshing

Step 6: From the Project Schematic, double click on Setup > CFX Pre appeared > go to Outline at the left side of menu bar > Default Domain > Material > water > ok. Then at the Outline menu, double click on Default Domain Default > then select Boundary Details > Option > Rough Wall. Sand Grain Roughness appeared then put the value 0.002 mm > ok. Again, from the Outline, right click at the Default Domain > insert > boundary > Insert Boundary > Name: inlet

Basic Setting > select the location of the inlet > Boundary Details > Option > Mass Flow Rate > put the value 9000 kg/s > ok.

Again, from the Outline, right click at the Default Domain > insert > boundary > Insert Boundary > Name: outlet

Basic Setting > select the location of the outlet > Boundary Details > Option under Mass and Momentum > Static pressure > put the value of relative pressure 0 Pa > ok. As shown in Figure 6.

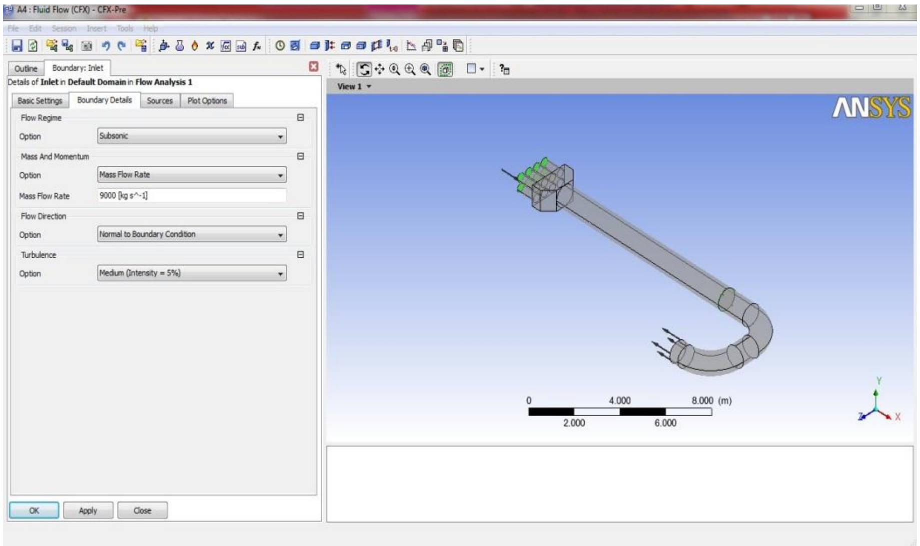

Figure 6: CFX Preprocessing (boundary condition)

Step 7: From the Project Schematic, double click on solution > Start Run > ok. As shown in Figure 7.

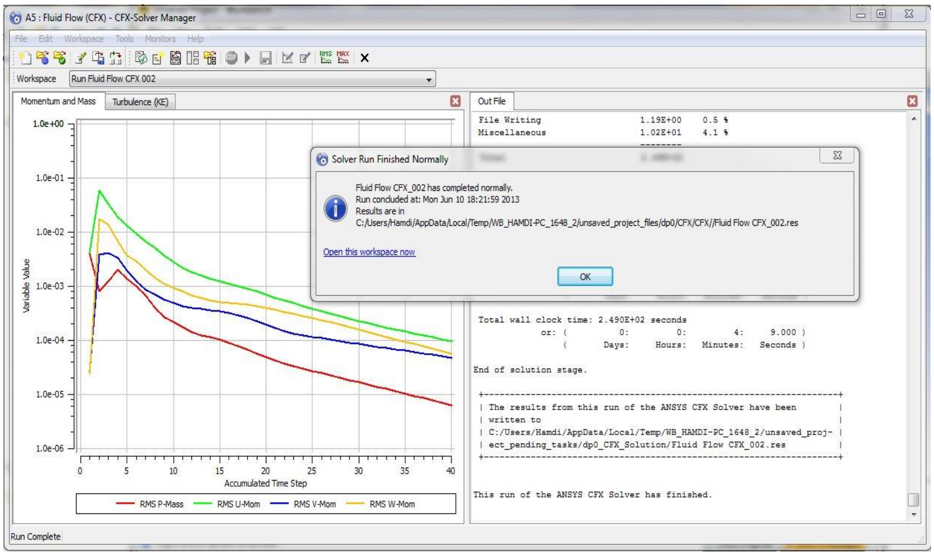

Figure 7: CFX Solver Manager (Solving)

Step 8: From the Project Schematic, double click on Result CFD-Post page appeared > click on streamline at the top menu bar > insert streamline > streamline1 > ok. Details of streamline appeared > Geometry > Start From > Inlet > Apply. Insert

Line from center inlet to center outlet > selected total pressure or velocity data. As shown in Figure 8.

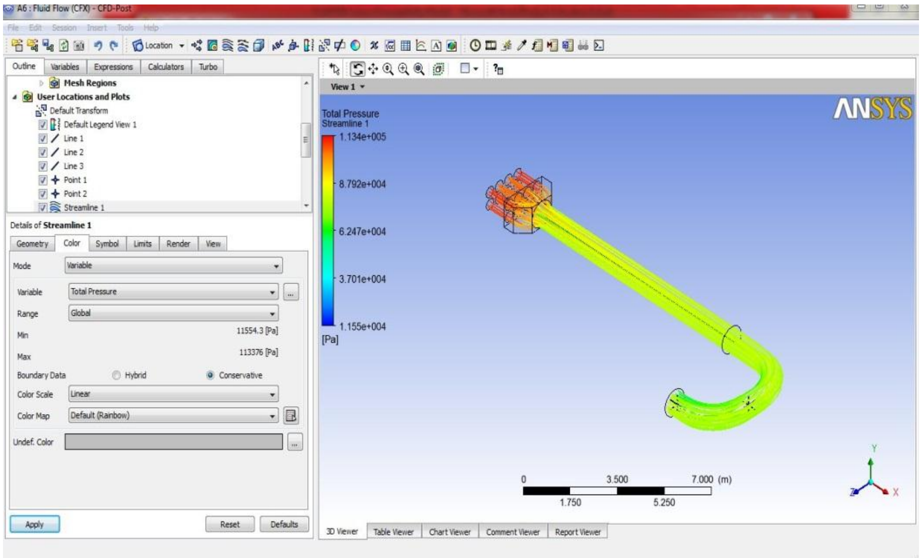

Figure 8: CFX Postprocessing (Result)

APPENDIXES B

Calculation Design A

a) For Pipe 1 A and B

since pipe 1A is the same size with pipe 1B so, pipe 1A equal Pipe 1B

Pressure Drop (ΔP) = Total Pressure inlet - Total Pressure outlet

= 62.332 kPa - 41.709 kPa = 20.622 kPa

Percentage of Pressure Drop = $\frac{20.622\ kPa}{62.332\ kPa}$ X 100% = 33.1 %

In pipe 1 (A and B) after simulation and calculated, researcher get the total pressure drop and percentage of both pipes are 20.622 kPa and 33.1% respectively.

Head loss (h_l)

$$\text{Head loss } (h_l) = \frac{\Delta P}{\rho g}$$

$$= \frac{20.622\ kPa\ x\ 10^3}{(997\ kg/m^3)(9.81\ m/s^2)} = 2.1m$$

The head loss is calculated by using head loss equation as stated above, and the value obtained is 2.1 m

b) Pipe 2 A and B

Since pipe 2A is the same size with pipe 2B so, pipe 2A equal Pipe 2B

Pressure Drop (ΔP) = Total Pressure inlet - Total Pressure outlet

= 64.106 kPa - 39.726 kPa = 24.38 kPa

Percentage of Pressure Drop = $\frac{24.38\ \text{kPa}}{64.106\ \text{kPa}}\ x\ 100\% = 38.1\%$

In pipe 2 (A and B) after simulation and calculated, researcher get the total pressure drop and percentage of both pipes are 24.38 kPa and 38.1% respectively.

Head loss (h_l)

$$\text{Head loss } (h_l) = \frac{\Delta P}{\rho g}$$

$$= \frac{27.301\ kPa\ x\ 10^3}{(997\ kg/m^3)(9.81\ m/s^2)} = 2.8 \text{ m}$$

The head loss is calculated by using head loss equation as stated above, and the value obtained is 2.8 m

c) Pipe 3 A and B

Since pipe 3A is the same size with pipe 3B so, pipe 3A equal Pipe 3B

Pressure Drop (ΔP) = Total Pressure inlet - Total Pressure outlet

= 65.241 kPa - 37.941 kPa = 27.301 k Pa

Percentage of Pressure Drop = $\frac{27.301\ kPa}{65.241\ kPa}$ X 100% = 41.85 %

In pipe 3 (A and B) after simulation and calculated, researcher get the total pressure drop and percentage of both pipes are 27.301 kPa and 41.85% respectively.

Head loss (h_l)

$$\text{Head loss } (h_l) = \frac{\Delta P}{\rho g}$$

$$= \frac{24.38\ kPa\ x\ 10^3}{(997\ kg/m^3)(9.81\ m/s^2)} 2.5\ m$$

The head loss is calculated by using head loss equation as stated above, and the value obtained is 2.5 m

d) Pipe 4 a and b

Since Pipe 4A is the same size with pipe 4B so, pipe 4A equal Pipe 4B

Pressure Drop (ΔP) = Total Pressure inlet - Total Pressure outlet

= 68.046 kPa - 35.426 kPa = 32.62 kPa

Percentage of Pressure Drop = $\frac{32.62\ kPa}{68.046\ kPa}$ X 100 % = 47.94 %

In pipe 4 (A and B) after simulation and calculated, researcher get the total pressure drop and percentage of both pipes are 32.63 kPa and 47.94 % respectively.

Head loss (h_l)

$$\text{Head loss } (h_l) = \frac{\Delta P}{\rho g}$$

$$= \frac{32.62\ kPa\ x\ 10^3}{(997\ kg/m^3)(9.81\ m/s^2)} = 3.34\ m$$

The head loss is calculated by using head loss equation as stated above, and the value obtained is 3.34 m

From the calculation above, the pressure drop and head loss difference from pipe to another pipe. These differences due to the differences in pipe length, which mean the pressure drop and head loss, are proportional to the pipe length (the longer pipe length, the higher pressure drop and head loss).

The average pressure drop and head loss of Design A calculated below, to make comparison with other design in order to select which design has less pressure drop and head loss.

Average pressure drop for Design A

Average Pressure Drop (ΔP_{avg}) =

$$\frac{\Delta P_{\text{pipe 1 a and b}} + \Delta P_{\text{pipe 2 a and b}} + \Delta P_{\text{pipe 3 a and b}} + \Delta P_{\text{pipe 4 a and p}}}{\text{Total Pipes}}$$

$$= \frac{(20.622 \text{ kPa x } 2) + (24.38 \text{ kPa x } 2) + (\ 27.301 \text{ k Pa x } 2)\ + (\ 32.62 \text{ kPa x } 2)}{8}$$

$$\Delta P_{avg} = 26.23 \text{ kPa}$$

Hence, average pressure drop for Design A is 26.23 kPa.

Average Percentage of Pressure Drop for Design A

Average Percentage = $\frac{\text{Average Total Pressure Drop}}{\text{Average total Pressure inlet}}$

Average total pressure inlet = 64.925 kPa

Average Percentage $= \frac{26.23kPa}{64.925\ kPa}\ x\ 100\%$

= 40.4 %

Hence, average percentage of pressure drop for Design A is 40.4%

Average Head loss

$$h_{l\ avg} = \frac{h_{l\ pipe\ 1\ a\ and\ b} + h_{l\ pipe\ 2\ a\ and\ b} + h_{l\ pipe\ 3\ a\ and\ b} + h_{l\ pipe\ 4\ a\ and\ b}}{total\ pipe}$$

$$= \frac{(2.1\text{m x } 2) + (2.5 \text{ m x } 2) + (2.8 \text{ m x } 2) + (3.34 \text{ m x } 2)}{8}$$

= 2.7m

Hence, average head loss for Design A is 2.7m.

APPENDIX C

Calculation Design B

a) Pipe1

Pressure Drop (ΔP) = Total Pressure inlet - Total Pressure outlet

= 155.741 kPa - 105.81 kPa = 49.93 kPa

Percentage of Pressure Drop = $\frac{49.93\ kPa}{155.741\ kPa}$ X 100% = 32.05 %

In pipe 1 after simulation and calculation, researcher get the total pressure drop and percentage of pipe is 49.93 kPa and 32.05 % respectively

Head loss (h_l)

$$\text{Head loss } (h_l) = \frac{\Delta P}{\rho g}$$

$$= \frac{49.93\ kPa\ x\ 10^3}{(997\ kg/m^3)(9.81\ m/s^2)} = 5.1\ m$$

The head loss is calculated by using head loss equation as stated above, and the value obtained is 5.1 m

b) Pipe 2

Pressure Difference (ΔP) = Total Pressure inlet - Total Pressure outlet

= 162.60 kPa - 98.75 kPa = 63.851 kPa

Percentage of Pressure Drop = $\frac{63.851\ kPa}{162.6\ kPa}$ X 100% = 39.2 %

In pipe 2 after simulation and calculation, researcher get the total pressure drop and percentage of pipe is 63.851 kPa and 39.2 % respectively

Head loss (h_l)

$$\text{Head loss } (h_l) = \frac{\Delta P}{\rho g}$$

$$= \frac{63.851\ kPa\ x\ 10^3}{(997\ kg/m^3)(9.81\ m/s^2)}\ 6.5\ m$$

The head loss is calculated by using head loss equation as stated above, and the value obtained is 6.5 m.

Average Percentage of Pressure Drop for Design B

$$\text{Average Pressure Drop } (\Delta P_{avg}) = \frac{\Delta \text{P}_{\text{pipe 1}} + \Delta \text{P}_{\text{pipe 2}}}{\text{Total Pipes}}$$

$$= \frac{(49.93\ \text{kPa}) + (63.851\ \text{kPa})}{2}$$

$$\Delta P_{avg} = 56.8905\ \text{kPa}$$

Hence, average total pressure drop for Design B is 56.8905 kPa

$$\text{Average Percentage} = \frac{\text{Average Total Pressure Drop}}{\text{Average total Pressure inlet}}$$

Average total pressure inlet = 159.17 kPa

$$\text{Average Percentage} = \frac{56.8905\ kPa}{159.17\ kPa}\ x\ 100\%$$

$$= 35.7\ \%$$

Hence, average percentage of pressure drop for Design B is 35.7 %

Average head loss:

$$h_{l\ avg} = \frac{h_{l\ pipe\ 1} + h_{l\ pipe\ 2}}{Total\ pipe}$$

$$= \frac{(5.1\text{m}) + (6.5\ \text{m})}{2} = 5.8\ m$$

Hence, average head loss for Design A is 5.8 m.

APPENDIX D

Calculation Design C

Total Pressure [kPa] at the inlet = 112.04 kPa

Total Pressure at Outlet = 69 kPa

Pressure Drop (ΔP) = Total Pressure inlet - Total Pressure Outlet

= 112.04 kPa - 69 kPa

= 43.04 kPa

Percentage of Pressure Drop = $\frac{43.041kPa}{112.04kPa}$ x 100% = 38.4%

After simulation and calculation, researcher get the total pressure drop and percentage of pipe is 43.04 kPa and 38.4 % respectively

Head loss (h_l)

$$\text{Head loss } (h_l) = \frac{\Delta P}{\rho g}$$

$$= \frac{43.04\ kPa\ x\ 10^3}{(997\ kg/m^3)(9.81\ m/s^2)} = 4.4\ m$$

The head loss is calculated by using head loss equation as stated above, and the value obtained is 4.4 m.

APPENDIX E

Calculation Design D

Total Pressure [Pa] in the inlet = 104.829 kPa

Total Pressure at Outlet = 72.599 kPa

Pressure Drop = Total Pressure inlet - Total Pressure Outlet

= 104.829 kPa – 72.599 kPa

= 32.23 kPa

Percentage = $\frac{32.23kPa}{104.829kPa}$ x 100% = 30.7%

After simulation and calculation, researcher get the total pressure drop and percentage of pipe is 32.23 kPa and 30.7 % respectively

Head loss (h_l)

$$\text{Head loss } (h_l) = \frac{\Delta P}{\rho g}$$

$$= \frac{32.23 \;\; kPa\ x\ 10^3}{(997\ kg/m^3)(9.81\ m/s^2)} = 3.1\ m$$

The head loss is calculated by using head loss equation as stated above, and the value obtained is 3.1 m.