



UNIVERSITI TEKNIKAL MALAYSIA MELAKA

**EFFECT OF INLET VELOCITY ON FLUID FLOW IN
T-JUNCTION PIPE WITH EDGE FILLET FOR DIVIDING
FLOW**

This report submitted in accordance with requirement of the Universiti Teknikal Malaysia Melaka (UTeM) for the Bachelor Degree of Engineering Technology Manufacturing (Process and Technology) (Hons.)

by

FATIN NAJWA BINTI MASTOR

B071310552

FACULTY OF ENGINEERING TECHNOLOGY

2016

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: EFFECT OF INLET VELOCITY ON FLUID FLOW IN T-JUNCTION PIPE WITH EDGE FILLET FOR DIVIDING FLOW

SESI PENGAJIAN:

Saya **FATIN NAJWA BINTI MASTOR**

mengaku membenarkan Laporan PSM ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Laporan PSM adalah hak milik Universiti Teknikal Malaysia Melaka dan penulis.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja dengan izin penulis.
3. Perpustakaan dibenarkan membuat salinan laporan PSM ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. ****Sila tandakan (✓)**

☐

SULIT

(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia sebagaimana yang termaktub dalam AKTA RAHSIA RASMI 1972)

☐

TERHAD

(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)

☒

TIDAK TERHAD

Disahkan oleh:

Alamat Tetap:

Cop Rasmi:

Tarikh: _____

Tarikh: _____

**** Jika Laporan PSM ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali sebab dan tempoh laporan PSM ini perlu dikelaskan sebagai SULIT atau TERHAD.**

DECLARATION

I hereby, declared this report entitled “EFFECT OF INLET VELOCITY ON FLUID FLOW IN T-JUNCTION PIPE WITH EDGE FILLET FOR DIVIDING FLOW ” is the results of my own research except as cited in references.

Signature :

Author's Name : FATIN NAJWA BINTI MASTOR

Date :



اونيورسيتي تيكنيكل مليسيا ملاك

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

APPROVAL

This report is submitted to the Faculty of Engineering Technology of UTeM as a partial fulfillment of the requirements for the degree of Bachelor of Electrical Engineering Technology (Process and Technology) with Honours. The member of the supervisory is as follow:



ABSTRAK

Paip yang mempunyai penyambungan T di gunakan secara meluas. Sebagai contoh di gunakan dalam sektor industry, sistem rumah,automotif dan sebagainya. Sistem bagi setiap aliran paip adalah berbeza mengikut kegunaan dan perimeter. Dalam kajian ini, fokus utama adalah untuk mengkaji kesan had laju dalaman terhadap paip penyambungan T bagi aliran yang berbahagi. Kajian terhadap kesan had laju bendalir di dalam paip berkemungkinan juga akan meninggalkan kesan terhadap tekanan pada paip bergantung kepada nilai had laju yang di berikan. Paip yang mempunyai penyambungan T ini terlebih dahulu di reka menggunakan perisian CATIA V5 . Di mana, pada perisian ini, perimeter bagi paip ini telah di tentukan dan menghasilkan rekaan 3D. Manakala bagi mengkaji kesan had laju dalaman terhadap paip, perisian Hyperwork telah di gunakan bagi menjalankan simulasi terhadap paip.

Di akhir kajian ini, melalui simulasi yang telah di jalankan telah memberikan hasil keputusan yang berbeza terhadap ketiga-tiga paip yang berbeza. Di mana, paip-paip tersebut masing-masing mempunyai nilai berbeza (lengkungan tajam, lengkungan 5mm dan lengkungan 10mm). selain dari berbeza nilai lengkungan,nilai had laju juga berbeza (2.56 m/s, 2.0 m/s dan 1.5 m/s). Berdasarkan ketiga-tiga paip,keputusan menunjukkan bahawa lengkungan 10 mm memberikan bacaan tekanan dan had laju magnitud yang paling rendah. Manakala had laju 1,5 m/s memberikan bacaan nilai tegasan ricih dan tekanan yang paling rendah

ABSTRACT

T-Junction pipes are widely used, such as in industries sector, home systems, automotive and others. For each pipe flow system are different according to the usability and perimeter. In this study, the main focus was to study the effect of inlet velocity of fluid flow in T-Junction pipe with edge fillet for dividing flow. In addition, the studies on the effects of inlet velocity of fluid flow in pipe may give the result on pressure in the pipe depends on the given velocities. CATIA V5 software is used in order to create a design of pipe. Whereas, by using this software, perimeters of the pipe have been determined and could create 3D designs. As for determine the effect of inlet velocity in pipe, simulation upon the pipe model is done by using Hyperwork software.

At the end of this study, based on the simulation that has been run has shown a different result against on the three different pipes. Whereas each pipe has different values (sharp edge, edge 5mm and edge 10mm). Apart from the different edge's value, the simulation also runs on the various value of velocities (2.56 m / s, 2.0 m / s and 1.5 m / s). Based on the three pipes, the results shows that the pipe with edge value of 10 mm gives the lowest pressure readings and velocity magnitude. Also the velocity value of 1.5 m / s that shows the lowest reading of pressure and shear stress.

DEDICATION

Dedicated to my father, Mastor Bin Manan. To my supervisor, Mr. Iskandar Bin Waini, co-supervisor, Mr. Mohd Suffian bin Ab Razak, lecturers and friends for all of their help and friendship



ACKNOWLEDGEMENT

I would like to express my deepest thanks and gratitude's to Allah S.W.T who give me the spirit and the soul throughout the duration of my final year project. I have completed this project even though there are many difficulty and hardship along the way.

I would like to express my appreciation and deep respects to my supervisor, Mr Iskandar Bin Waini for the guidance and encouraged during finishing this project. Not to forget, my co-supervisor Mr. Mohd Suffian bin Ab Razak for his constant guidance and support during my thesis writing is invaluable to me and continuous direction and opinion regarding the flow of the project has invaluable contribution to achieve the objectives of the project.

Thanks to my family, who has been the loveliest advisor to give constant support and inspiration through my campus life. Their supports are meant so much to me in finishing this project. Last but not least, I thank everyone who involved directly and indirectly in this project. Without them, this PSM report will not good completed like this

TABLE OF CONTENT

Abstrak	v
Abstract	vi
Dedication	vii
Acknowledgement	viii
Table of Content	ix
List of Tables	xi
List of Figures	xii

CHAPTER 1: INTRODUCTION 1

1.1	Introduction	1
1.2	Background of study	1
1.3	Statement of Problem	2
1.4	Objective of study	3
1.5	Scope of Study	3
1.6	Layout of Project	4
1.7	Conclusion	5

CHAPTER 2: LITERATURE REVIEW 6

2.1	Introduction	6
2.2	Previous Researches of Fluid Flow in Pipe	6
2.3	T-junction	12
2.4	Dividing Flow	13
2.5	Effect of Inlet Velocity	14
2.6	Effect of Pressure in Pipe	18

CHAPTER 3: METHODOLOGY

6

3.1	Introduction	
		22
3.2	Overall Process Design of pipe	22
3.3	Flowchart of the process	23
3.4	Design pipe model	24
3.5	Catia v5 software	24
	3.5.1 CATIA modelling T-junction pipe	26
3.6	Simulation	28
	3.6.1 Simulation on the Fluid Flow in T-Junction Pipe	29

CHAPTER 4: RESULT & DISCUSSION

36

4.1	Introduction	38
4.2	General Procedure for Pipe Simulation	36
4.3	Effect on Velocity with Various Edges and Inlet Velocity	39
4.4	Effect on Pressure with Various Edge and Inlet Velocity	41
4.5	Effect on Wall Shear Stress with Various Inlet Velocity	44
4.6	Effect on Velocity Magnitude with Various Velocity and Edge of Pipe	47

CHAPTER 5: CONCLUSION & FUTURE WORK

51

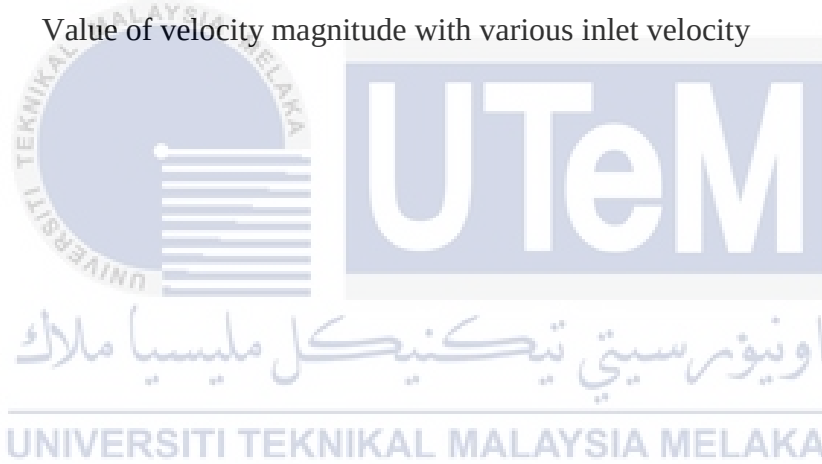
5.1	Conlusion	51
5.2	Future Studies	52

REFERENCES

53

LIST OF TABLES

1.1	Chapter Organization	4
4.1	Parameter set for each pipe	39
4.2	Value of axis-velocity	41
4.3	Pressure value for each pipe	42
4.4	Pressure value in various velocity	44
4.5	Wall shear stress on the axis of each type of pipe	46
4.6	Value of velocity magnitude	48
4.7	Value of velocity magnitude with various inlet velocity	50



LIST OF FIGURES

2.1	T-junction pipe	12
2.2	Fluid flow varieties, dividing and joining	13
2.3	Dividing Flow	14
2.4	Dividing flow	14
2.5	Head loss for dividing flow: Radius of branches is 0.5 cm, Inlet velocity vary from 1 cm/sec to 3 cm/sec, at both outlet pressure is 100 Pascal and Classical pressure loss formula by A. Gardel	16
2.6	Velocity profile for laminar flow	17
2.7	Velocity contours and vectors for laminar flow (a) base model (b) corner radius 0.6mm.	17
3.1	Flowchart for design and development of the structure	23
3.2	2D drawing show dividing flow of T-Junction Pipe	24
3.3	DS CATIA V5	25
3.4	The initial step	26
3.5	Choosing 'xz plane' for sketching	26
3.6	Sketch circle and constraint	26
3.7	Determine the length of horizontal pipe	27
3.8	'Pad' on the drawing	27
3.9	Make another junction of the pipe	27
3.10	3D drawing for the vertical pipe as the junction	28
3.11	Menu Bar to create new database	29
3.12	Data tree show 'Global' (Mesh/geometry independent) & 'Model' (Mesh/geometry dependent)	29
3.13	Problem Description	30
3.14	Auto Solution Strategy setup	30
3.15	Data Tree checklist before importing CAD drawing	31

3.16	Import Geometry to check the units of x, y and z bounds and display Resolution	31
3.17	This figure shows the example of pipe model in order to show how to define the fluid flow pattern and the wall of the pipe.	32
3.18	Simple Boundary condition with 'flow type' is mass flux	32
3.19	Simple Boundary Condition with 'flow type' is velocity	33
3.20	Surface Mesh Attribute at wall is specified	33
3.21	Surface Mesh Attribute for both inflow and outflow	34
3.22	Volume Manager dialog box	34
3.23	'Launch AcuFieldView' dialog	35
3.24	AcuTail dialog box	35
3.25	AcuFieldView when opened	36
3.26	Unchecking 'Visibility' in boundary surface panel	36
3.27	Coordinate Surface in AcuFieldView	37
4.1	The velocity (m/s) of x, y and z axis of pipe with sharp edge	40
4.2	The velocity (m/s) of x, y and z axis of pipe with edge 5mm	40
4.3	The velocity (m/s) of x, y and z axis of pipe with edge 10mm	40
4.4	Effect of pressure on T-junction Pipe with different edge	42
4.5	Pressure effect with inlet velocity 1.5 m/s	43
4.6	Pressure effect with inlet velocity 2.0 m/s	43
4.7	Pressure effect with inlet velocity 2.56 m/s	44
4.8	Shear stress at wall of pipe with sharp edge	45
4.9	Shear stress at wall of pipe with 5mm edge	46
4.10	Shear stress at wall of pipe with 10mm edge	46
4.11	Velocity magnitude for T-junction pipes	47
4.12	Velocity Magnitude in various inlet velocity	49

CHAPTER 1

INTRODUCTION

1.0 Introduction

This introductory chapter has provided some background of study and also explains the objective or purpose of this study based on the problem statement as well as the scope of the study. Then it also gives an overview and structure of this report and lastly conclusion from the content..

1.1 Background Study

T-Junction pipes are often necessary and widely used for industrial piping system, chemical engineering, home application, automotive and etc. T-Junction pipe normally has three branches which also give the pattern on the fluid flow. Those branches consist of two inlets and one outlet for combining flow and one inlet and two outlets for dividing flow. According to the studies conducted by Štigler et. al (2014), T-junction consists of the one straight pipe and the adjacent branch which can having an preference under various angle. In T junction, there are three branches it means there are two entrances and one exit in case of combining flow or one entrance and two exits in case of dividing flow. Moreover, anything that flows in the pipe (fluid and gas), they may cause many types of effects against the pipe such as pressure, velocity, and friction. . In the pipe network normally have two types of fluid flow that caused by the junction device. For each type of fluid flow which are laminar flow and turbulent flow, has their own characteristic which able to affect the fluid properties

and the pressure. For pipe 3D model, CATIA V5 software is used to design T-junction pipe also this software will follow the scale and the real dimension which follow geometry from previous study.

Text

1.2 Statement of Problem

The flow pattern inside the pipe could be in laminar, transition or fully turbulent. Parameters affect the flow regime mostly on length, diameter and velocity. The acceleration velocities will generate higher pressure and pressure rise rate will cause large friction to the inner pipe and reduce the ability of the pipe.

Pipe networks are mainly used for transportation and supply of fluids or gasses from one location to the other location. During the process it may cause loss in pressure due to change in momentum of the flow caused to friction and pipe component. Therefore, in this project the understanding of the behavior of fluid and pressure loss that occur in pipe is required. In order to get clear view of the problem, simulation of fluid flow in pipe is conducted using software. Problems that are occurring in pipe which has been designed can be identified by doing simulation on the pipe. Simulation software such as Hyperwork is used to identify the suitable velocity and pressure in order to prevent less friction in the pipe as well as to produce better pipe for any application. This simulation software will provide the data of pressure loss at outlet pipe due the various velocities on the branch pipe (inlet). This study is expected to identify and analyze the suitable velocity that is used to minimize pressure loss and its flow properties in the T-junction pipe for dividing flow.

1.3 Objective of Study

In order to determine the effect of inlet velocity of fluid flow in pipe, there are several objectives of this study need to be achieved:

- i. To create 3D modeling pipe by using CATIA v5 software
- ii. To simulate the fluid flowing by using Hyperwork software (ACuSolve)
- iii. To investigate the effects of inlet velocity in T-junction pipe for dividing flow

1.4 Scope of Study

This paper studies about the effect of inlet velocity on the fluid flow for dividing flow at the T-junction pipe. Simulation of the fluid flow will give data of the pressure loss and information of the fluid properties:

- i. CATIA v5 software is used to design T-junction pipe
- ii. Simulation of the fluid flow using Hyperwork (AcuSolve) software
- iii. Using Spalart Allmaras as a type of Turbulent flow
- iv. Dividing flow as a fluid flow pattern while water is a type of fluid used during simulation.
- v. Simulate the effect of inlet velocity with the inner diameter used for pipe is 10mm

1.5 Layout of Project

Table 1.1: Chapter Organization

Chapter	Content	Division
1	Introduction	PSM 1
	This chapter contains the initial information regarding the project which is background including historical background. It also contains objective based on the problem statement and also the scope of this study that act as boundaries of the project.	
2	Literature Review	
	This chapter contains the information need to have for the study. The information from various kinds of source such as paper research and case studies on topic relate to fluid flow in pipe and also the software used for the design and simulation. It also contains comparison on the case studies based on their method and result. So based on the information of the research paper, the methodology for this study can be developed.	
3	Methodology	
	This chapter is the methodology that is done based on the finding in the literature review. The methodology is for the procedure guideline that needs to be done during the PSM 2 later. It also contains the method how to design and simulate for the particular case in this study.	

Chapter	Content	Division
3	Methodology	PSM 2
	In this chapter contains the information that has been decided before running the simulation in order to get the result. Studies about the characteristic of the fluid flow is obtained and the parameter for both pipes and the fluid flow are decided. Those pipes are then been through the simulation.	
4	Result and Discussion	
	This chapter contains the information about the result which is obtained from the simulation. Results shows the reading of pressure, velocity magnitude, velocity in different axis of the pipes and shear stress of the pipe.	

1.6 Conclusion

This chapter concludes the overview of this study by stating the objective, scope of the study, and methodology. The objective has addressed the direction of the study while from the scope of study and the critical parameter was specified. Lastly the arrangement of the study was briefly explained that also act as guideline for the study done.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

This chapter is made based on reviews of the existing research that related to project title such type of pipe junction, flow properties, type of fluid flow, simulation and others important factor that correlate to project title. Besides, in this chapter also contain information and discussion that were relevant and need to consider before beginning the project..

2.1 Previous Researches of Fluid Flow in Pipe

Peng *et.al* (1994) conducted a study of dividing steam and water flow in T-Junction. In this study also provide detailed experimental data of phase redistribution and associated pressure changes in T-junctions having horizontal inlet are presented for both annular and stratified inlet flows. Two phenomenological phase redistribution models for annular and stratified flows were refined. The objectives of the study are to improve the available data bank on the subject through the experimental measurements and strengthen the current understanding of the phenomenon through model development and analysis.

Al- Wazzan (2003) has been conducted a study based on the objective; to study the characteristic of split of horizontal two-phase flow at a T-junction by using a CFD simulation. The experimental results were compared achieve by the investigators and

with theoretical models prepared by previous investigators. The velocities of both phases should be controlled in order to avoid the slug flow that is unfavourable from both an operational also for a safety point of view. This could also help in reducing the pressure drop especially at the junction. The sudden change in the velocity profile represents an important change in the flow patterns which need more attention during the time of operation. PHOENICS can provide the main objectives of the two-phase flow simulation and can predict the misdistribution of the phases in the pipe during the flow. The capabilities of PHOENICS include the ability to predict all the important factors in the two-phase flow phenomenon like the pressure drop, void fraction and phase distribution. The simulation results acquired from PHOENICS are quite close to the results acquired from experiments and the theoretical models.

Saffari (2003) have calculated the pressure drop inside condensing vertical pipes in new inlet pressures using a new modified three-fluid model. The objective of his study is to study the effect of variation of inlet pressure on predictions of pressure drop in the downward condensing annular flow of steam inside vertical pipes. The pressure drop is calculated in two new inlet pressures (i.e., 1.5 and 2.5 (MPa) by using the new modified three-fluid model and Stevanovic et al.'s correlation for the steam-liquid film interfacial friction coefficient.

Bertani *et.al* (2006) has studied about the 2 phase flow in a horizontal T-junction. The objective of the study is to investigate the dividing flow rates and pressure drops in a Plexiglas tee junction with horizontal inlet, run, and branch sides using air-water mixtures.

Paal *et al.* (2006) has identified the energy losses and the size and strength of the recirculation region in the branch pipe. They also have investigated the turbulent flow in 90° T-junctions with sharp and rounded corners. They using an extensive numerical parametric (LDA-measurements and flow simulations) for the simulation. From the finding, by increasing the radius of curvature of the corner reduces the total energy loss especially because of the reduction in the branch flow loss related to flow separation. We can see that big radius will give a big loss for the flow and it will make the flow become slow after the branch. The intuitively expected influence of rounding

the edge has been confirmed both by the simulations and the experiments: the branch pipe loss coefficient significantly decreased.

Costa *et al.* (2006) have made experiment how the effect on flow is characteristic and compared the flows in two similar tees having different edges at the junction: One has a sharp-edged connection between the branch pipe and the main straight pipes whereas the second tee has round edges. The fluid use in the experiment is water. The water flow was continuous and was driven by a volumetric **Mohno** pump supplied by a constant head tank. A fluctuate speed controller operated the pump and three valves controlled the flow distribution to the two outflow pipes.

In both geometries, the loss coefficient of the branched flow was higher than for the straight flow, as expected, because of flow segregation in the branch pipe and the void of separation in the main outlet duct. Round edge on the corner of the junction, lead to higher turbulence in the branch pipe which resulted in a shorter, thinner, and weaker recirculation bubble region, thus reducing the loss coefficient of the branched flow.

Round corner did not affect the characteristics of the flow going into the outlet straight pipe, indicating the current behaviour of common turbulent flows. Moreover, the increase in dissipation in the branched flow coefficient and in terms of the total energy loss is enough to justify that the rounded tee is certainly more efficient for all the investigated Reynolds numbers.

Meanwhile, the edge effect on the flow characteristic in 90 degree T-Junction has been studied by Pinho (2006). These studies aim to determine whether the angle of corners effect energy and turbulence dissipation. CAD-CAM system has been used along with the K-epsilon ($k-\epsilon$) and standard Reynolds stress as a turbulence model. Rounding the edge of junction lead to higher turbulence in the branch pipe which resulted in a shorter, thinner and weaker recirculation bubble region, thus reduce the loss coefficient of branch flow which justify that rounded tee is obviously more efficient for all investigated Reynolds numbers.

Vasava (2007) was conducted study using a software to study the flow properties at T-junction of pipe, pressure loss suffered by the flow after passing

through T-junction and to study reliability of the classical engineering formulas used to find head loss for T-junction of pipes. In this study, they have compared their own results with CFD software packages with classical formula and made an attempt to determine accuracy of the classical formulas. The other objective of this study is also to study the change in pressure losses with change in angle of T-junction.

Study was directed by utilizing programming and not utilizing the real trial. The liquid use is considered water with typical properties at room temperature. This study is utilizing CFD strategies Finite Element technique (FEM) and Finite Volume Method (FVM). Limited Volume Method (FVM) is a numerical strategy taking into account Integral protection law. These methods are used for solving partial differential equations that calculates the values of the conserved variables averaged across the volume.

From the observation, there is difference between head loss in T-junction of pipes observed by calculations from software packages Fluent and Comsol. Also from the study, they were observing the difference between observations by 2D simulations of software and classical formula was considerably larger than the difference between observations by 3D simulation of software and classical formula. Work presents a prediction of pressure loss of fluid with turbulent incompressible flow through a 90° tee junction was carried out and compared with analytical and experimental results.

The dispersion funnel is displayed as a perfect 3D channel with two gulfs and one way out. The hypothetical relationship for stream at channel intersections has been produced by joining the preservation condition of mass, protection condition of energy and Bernoulli's rule to a control volume of the stream over a funnel intersection. This study was utilizing CFD recreation to mimic the analysis and the information gathered will contrast and the genuine test information.

Studies about dividing flow at a 90Degree open channel junction also has been conducted by Rashwan *et.al* (2008). In this study, a theoretical model for division of flow through T-junction over a horizontal bed was obtained for subcritical steady flow through main, extension and branch channels of equal widths. The new proposed model was derived with the aid of continuity, energy and momentum equations. For a given inflow discharge, the water depth and the width of the channels, using the present

model, the downstream depth and discharge could be determined. Experimental data from previous studies were used to verify the proposed new model. The deduced model was found in good agreement with the observations.

It was found that a linear relationship has been existed between the experimental data of the inflow water depth with the branch water depth. The equation of trend line was given and it could then be used to compute the branch water depth by knowing the inflow water depth. The energy head-loss coefficient of a junction was approximated and expressed only in terms of discharge ratio. In the design of dividing flow in rectangular open channels a theoretical model was developed to relate the discharge ratio $R_q = Q_2/Q_1$ with the Froude number F_1 and the depth ratio $1.2 R_y/y =$. The proposed model was validated by the experimental data and appeared in good agreement.

Zhou and Zheng (2009) studied the mean flow and turbulence characteristics of open-channel dividing flows, a hybrid LES-RANS model, which combines the large eddy simulation (LES) model with the Reynolds-averaged Navier-Stokes (RANS) model. The unstable RANS model was used to simulate the upstream and downstream regions of a main channel, as well as the downstream region of a branch channel. The LES model was used to simulate the channel diversion region, where turbulent flow characteristics are complicated. Isotropic velocity fluctuations were provided at the inflow interface of the LES region to trigger the generation of resolved turbulence. A method depend on the virtual body force is proposed to impose Reynolds-averaged velocity fields near the outlet of the LES region in order to take downstream flow effects computed by the RANS model into account and dissipate the excessive turbulent fluctuation.

A relatively new three-dimensional hybrid LES-RANS model has been developed to simulate open-channel T-diversion flows (Zhou and Zhang, 2009). The model was established with a classic case of fully developed open-channel turbulent flow. The model used to simulate the flow in an open-channel T-diversion.

Xin and Shaoping (2013) has been conducted the studies about the flow field and pressure loss analysis of junction and its structure optimization of aircraft hydraulic pipe system. The authors investigate flow fields of T-junction and Y-

junction using shear stress transport (SST) model. ANSYS/CFX is used for the simulation in this process as well as K-epsilon ($k-\epsilon$) turbulence model and SST model have been used to describe CFD simulation which used to simulate mean flow characteristics for turbulent flow conditions. The result shows that, the variation rule of the velocity peak is obtained and the eddy current does exist in the corner of T-junction.

Studies of numerical prediction of pressure loss of fluid in a T-junction by Abdulwahhab *et.al* (2013), presents a prediction of pressure loss of fluid with turbulent incompressible flow through a 90° tee junction was carried out and compared with analytical and experimental results. . This work is part of a wider research program and here preparatory results of numerical computations of the turbulent flow in a 90° T-junction are presented and compared with experimental data as previous study with analytical. The flow configuration is that of a convergence flow in a 90° T-junction with sharp corners. Predictions of the turbulent flow in a 90° T-Junction were carried out and compared with theoretical and experimental data for two cases as the pipe area ratio (i.e. $A_3/A_2=1.0$ and 4.0) for sharp edged. The pressure loss coefficient given by the numerical results is higher than those obtained from theoretical and experimental results. The higher the flow rate ratio is the higher the difference between them. The behaviour of the curve of pressure loss coefficient for pipe area ratio between the two different value of curve (i.e. 1.0 and 4.0) are different."

Štigler *et al.* (2014) has studied characteristics of the T-junction with the equal diameters of all branches for the variable angle of the adjacent branch. These studies aim to bring out the T-junctions which consists straight and adjacent branch pipe which can be inclined under various angle by numerical calculation and experiment activity. Two types of method are used such as experiments and comparison with numerical calculations. The mathematical model can be used for solution of fluid flow in pipe systems and also for comparison of the different shapes of the T-junctions.

2.2 T-junction

Pipe networks are mainly used for transportation and supply of fluids and gases. These networks vary from fewer pipes to thousands of pipes. In addition to pipes, the network also consists of elbows, T-junctions (refer Figure 2.1), bends, contractions, expansions, valves, meters, pumps, turbines and many other components. All these components cause loss in pressure due to change in moment-tum of the flow caused due to friction and pipe components. This means conversion of flow energy in to heat due to friction or energy lost due to turbulence.



Figure 2.1: T-junction pipe

Pipe networks are very common in industries, where fluid or gases are to be transported from one location to the other. In industries the networks are usually large and require very precise pressure at certain points of network. It is also sometimes essential to place valves, pumps or turbines of certain capacity to control pressure in the network. The placement of valves, pumps and turbines is important to overcome pressure losses caused by other components in the network. This is one of the important reasons why this study was conducted.

2.3 Dividing Flow

In this work, we have concentrated our attention to a very small and common component of pipe network: T-junction (some also refer as 'Tee'). T-junction is a very common component in pipe networks, mainly used to distribute (diverge) the flow from main pipe to several branching pipes and to accumulate (converge) flows from many pipes to a single main pipe. Depending on the inflow and outflow directions, the behaviour of flow at the junction also changes. The following figure shows some possibilities of fluid entering and leaving the junction.

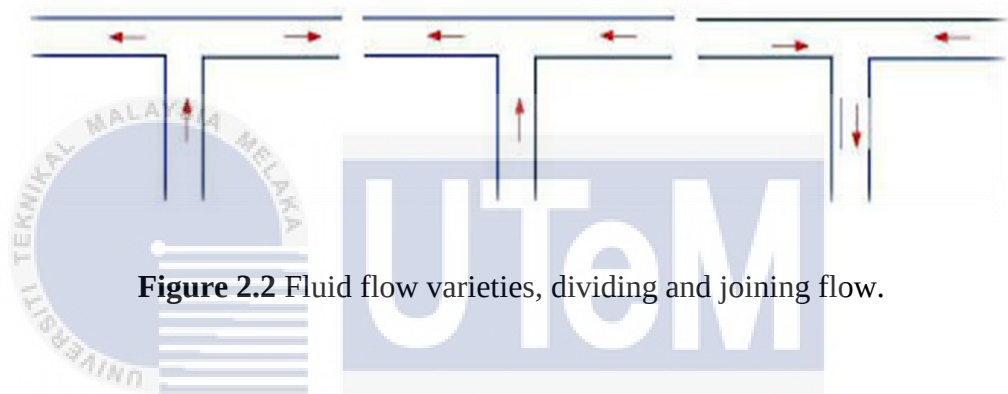


Figure 2.2 Fluid flow varieties, dividing and joining flow.

When a two-phase mixture flows through a dividing T-junction, there is an almost inevitable, misdistribution of the phases between the outlets. The unequal splitting of gas and liquid at T-junction was observed to create problems in the industry where it may be found. For example, in gas distribution networks, condensate can be formed in pipelines in winter due to low temperature. It was found that the condensate appears at some delivery stations while the other stations receive only dry gas. This kind of uneven splitting may result in creating operational and separation problems.

From Paritosh (2007) case study, the dividing flow has two conditions. First, it flows in coming toward the junction from the perpendicular branch and leaving from the junction from two branches in main pipe (figure 6.7).

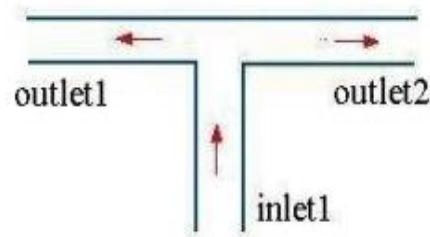


Figure 2.3: Dividing flow.

The other case is where the flow is coming toward the junction from one branch. In main pipe and perpendicular branch and leaving from the junction from the remaining branch in the main pipe (Figure 2.4) [The other situation is exactly the mirror image]

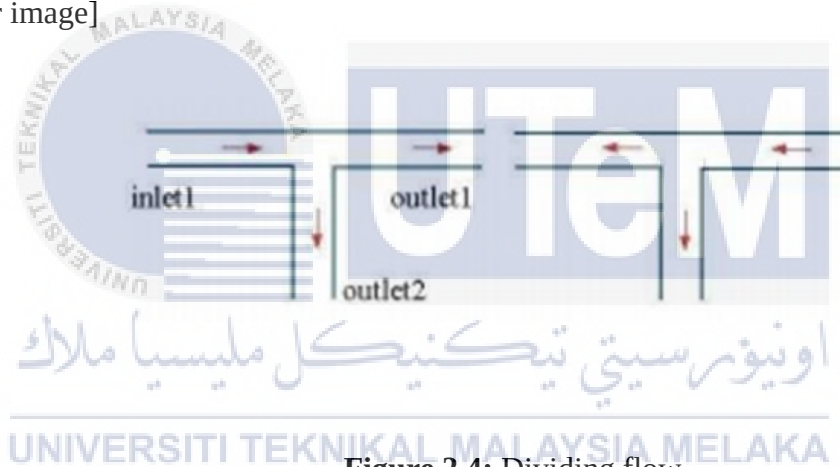


Figure 2.4: Dividing flow.

2.4 Effects of Inlet Velocity

To study the contribution, Costa, *et al.* (2006) have made experiment how the effect on flow is characteristic and compared the flows in two similar tees having different edges at the junction: One has a sharp-edged connection between the branch pipe and the main straight pipes whereas the second tee has round edges. The fluid use in the experiment is water. The water flow was continuous and was driven by a

volumetric Mohno pump supplied by a constant head tank. A fluctuate speed controller operated the pump and three valves controlled the flow distribution to the two outflow pipes. : In both geometries, the loss coefficient of the branched flow was higher than for the straight flow, as expected, because of flow segregation in the branch pipe and the void of separation in the main outlet duct. Round edge on the corner of the junction, lead to higher turbulence in the branch pipe which resulted in a shorter, thinner, and weaker recirculation bubble region, thus reducing the loss coefficient of the branched flow.

By Peng *et.al* (1994) studied, the two-phase redistribution in T-junctions was found to be considerably affected by the inlet flow conditions, inlet flow pattern, inlet quality and inlet flow rates. The phase redistribute phenomenon also depends on the junction geometry. The experimental results showed that for annular flow in horizontal T-junctions an increase of inlet quality reduces the degree of phase redistribute while the inlet mass flux was found to be less incomparably. However, in stratified flow the increase of either the inlet superficial vapours or liquid velocities increases the degree of phase division. The experimental results also showed that decreasing the branch diameter will increase the degree of phase division. Furthermore; downward orientation of the branch can reduce the branch flow quality. The pressure changes in T-junctions were associated using simple momentum and energy balances for the run and branch appropriately using measured void fractions. Comparison of the present data on pressure changes in T-junctions with some available models showed that those models which accounts for phase reorganize effects were better than the others in correspond the present data, confirming the strong interconnection between the pressures.

There is also a studied the effect of corner radius on the energy loss in 90° T-junction turbulent flow. Paal, *et al.* (2006) has identified the energy losses and the size and strength of the recirculation region in the branch pipe. They also have investigated the turbulent flow in 90° T-junctions with sharp and rounded corners. They using an extensive numerical parametric (LDA-measurements and flow simulations) for the simulation. From the finding, by increasing the radius of curvature of the corner reduces the total energy loss especially because of the reduction in the branch flow loss related to flow separation. We can see that big radius will give a big loss for the

flow and it will make the flow become slow after the branch. The intuitively expected influence of rounding the edge has been confirmed both by the simulations and the experiments: the branch pipe loss coefficient significantly decreased.

From (Figure 2.5), we can observe that the head loss by software and classical formulas also do not agree in this case (Paritosh, 2007). Though the curves seem to get along with the increase in inlet velocities, but they do not exactly match for any combination of velocities. Regarding from Paritosh, there is about 4.5 to 6.1 % error between results by software and classical formula. The inlet velocity increase influence the increasing of the pressure in the pipe. We can see that, in the graph although the result from simulation and formulae is difference.

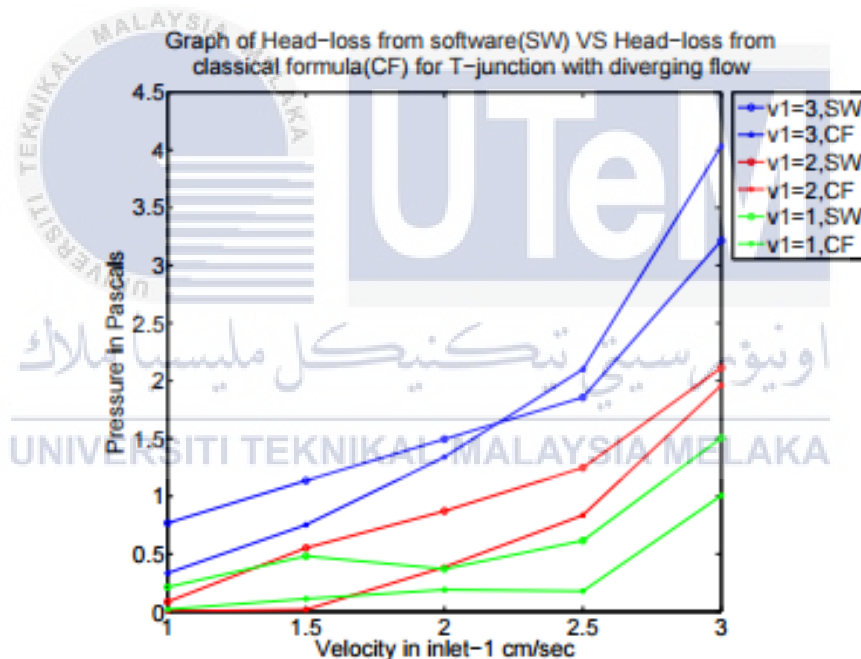


Figure 2.5: Head loss for dividing flow: Radius of branches is 0.5 cm, Inlet velocity vary from 1 cm/sec to 3 cm/sec, at both outlet pressure is 100 Pascal and Classical pressure loss formula by A. Gardel.

Difference in corner radius also gives effect on the velocity of the fluid flow in the pipe. According to Abhik and Sambit (2015), corner that have radius will increase the velocity of the fluid. Refer figure 2.6, the velocity profile taken at different point in horizontal channel for laminar flow. It shows, increasing velocity at the corner radius.

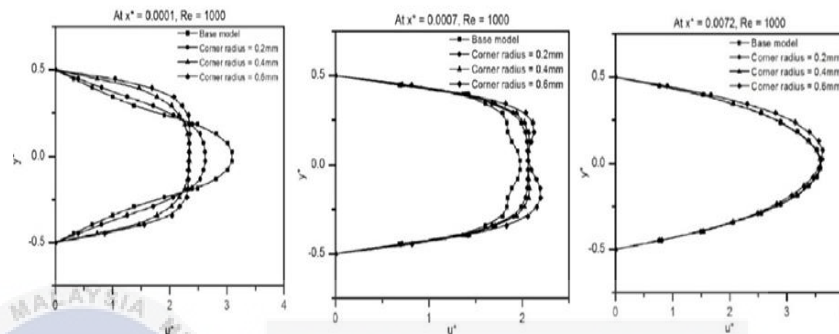


Figure 2.6 Velocity profile for laminar flow.

There is vortices develop near the junction and clearly seen on the figure 2.7. We can see that the vortices profile decrease in corner radius.

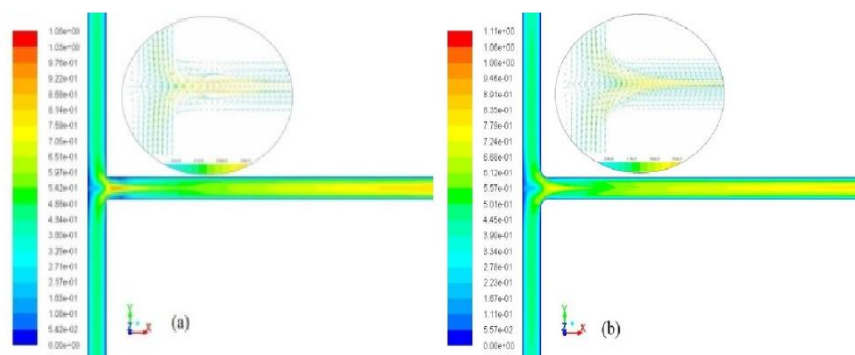


Figure 2.7 Velocity contours and vectors for laminar flow (a) base model (b)

corner radius 0.6mm.

2.5 Effect of Pressure in Pipe

Vasava (2007), from her study, there is difference between head loss in T-junction of pipes observed by calculations from software packages Fluent and Comsol. In case of combining flow, the difference between observations obtained by Comsol (3D experiments) and classical formula were in the range of 3.2 to 5.1 %. For dividing flow, this difference was in the range of 4.5 to 5.5 %. In the case, they were varied the angle of the T-junction from 87 degrees to 93 degrees, difference between observations by Comsol (3D experiments) and classical formula was in the range of 4.6 to 6.7 %. From the study, they were observed the difference between observations by 2D simulations of software and classical formula was considerably larger than the difference between observations by 3D simulation of software and classical formula. We also recommend 3D simulation for such calculations, since 3D simulation are more near to the reality and also effect of turbulence can be modeled and observed in 3D simulations. Also, 3D simulations give more clear view of swirl movements, streamlines and turbulence in the fluid.

Work presents a prediction of pressure loss of fluid with turbulent incompressible flow through a 90° tee junction was carried out and compared with analytical and experimental results. This work is part of a wider research program and here preliminary results of numerical computations of the turbulent flow in a 90° T-junction are presented and compared with experimental data as previous study with analytical. The flow configuration is that of a convergence flow in a 90° T-junction with sharp corners.

The distribution pipe is modelled as an ideal 3D pipe with two inlets and one exit. The theoretical relationship for flow at pipe junctions has been developed by combining the conservation equation of mass, conservation equation of momentum and Bernoulli's principle to a control volume of the flow across a pipe junction. This

study was using CFD simulation to simulate the experiment and the data collected will compare with the actual experiment data.

Predictions of the turbulent flow in a 90° T-Junction were carried out and compared with theoretical and experimental data for two cases as the pipe area ratio $A_3/A_2=1.0$ and 4.0 for sharp edged. The pressure loss coefficient given by the numerical results is higher than those obtained from theoretical and experimental results. The higher the flow rate ratio is the higher the difference between them. The behaviour of the curve of pressure loss coefficient for pipe area ratio 1.0 is different from curve for pipe area ratio 4.0 especially after the flow rate ratio $q=0.4$ because the value of velocity at inlet 2 is greater than the velocity at inlet 1 and this causes the recirculation of the downstream fluid of the main pipe.

From Bertani *et al* (2006), they made phase separation by using the map of Buell and Azzopardi which show that Azzopardi *et al.* developed a map of flow patterns in the inlet pipe of a 90° T-junction while map of Buell *et al.* shows that, at constant gas superficial velocity, the liquid preferably flows into the branch if its superficial velocity is low, while at high liquid superficial velocities the gas phase preferably runs into the branch pipe. Then, they identified the pressure drop for two different pipe diameters. The finding for this studies have succeeded in identifying important flow phenomena: the flow quality in the branch and in the run pipes are usually different from each other and also different from the inlet quality; the flow rate of the two phases splitting through the run and branch streams affects strongly the pressure drop across the channels downstream of the mixing tee.

Saffari (2003) have calculated the pressure drop inside condensing vertical pipes in new inlet pressures using a new modified three-fluid model. The objective of his study is to study the effect of variation of inlet pressure on predictions of pressure drop in the downward condensing annular flow of steam inside vertical pipes. The pressure drop is calculated in two new inlet pressures (i.e., 1.5 and 2.5 (MPa) by using the new modified three-fluid model and Stevanovic *et al.*'s correlation for the steam–

liquid film interfacial friction coefficient. The study also made a correction on new modified three-fluid model, which are neglected by all previously developed three-fluid models; (the virtual mass (added mass) force term is taken into account, and the friction stress of droplets with liquid film is considered). From this studies, the results that has been obtained are; At the inlet pressure of 1.08 (MPa), the new modified three-fluid model provides a much better match with experimental data compared to all other correlations. The boundary mass flux of the low and high mass flux ranges increases with the increase of inlet pressure in condensing vertical pipes. At a constant high mass flux in a vertical pipe with steam condensation, when the inlet pressure increases, the magnitude of total pressure change decreases. In condensing vertical pipes in passive reactors, a specified amount of total pressure drop requires a specific amount of steam inlet mass flux, in determination of which the presented new modified three-fluid model can be very helpful at 1.5 and 2.5 (MPa) inlet pressures.

Peng (1994) have studied a dividing steam water flow in T-junction. Phase redistribution is a complicated physical phenomenon which occurs in dividing two-phase flow in T-junctions. In this study detailed experimental data of phase redistribution and associated pressure changes in T-junctions having horizontal inlet are presented for both annular and stratified inlet flows. Two phenomenological phase redistribution models for annular and stratified flows were developed. The objectives of the study are enriching the available data bank on the subject through the experimental measurements and enhancing current understanding of the phenomenon through model development and analysis. Two-phase redistribution in T-junctions was found to be significantly affected by the inlet flow conditions, i.e. inlet flow pattern, inlet quality and inlet flow rates. The phase redistribution phenomenon also depends on the junction geometry, i.e. branch orientation and diameter. The experimental results showed that for annular flow in horizontal T-junctions an increase of inlet quality reduces the degree of phase redistribution while the inlet mass flux was found to be less significant. However, in stratified flow the increase of either the inlet superficial vapors or liquid velocities increases the degree of phase separation. The experimental results also showed that decreasing the branch diameter will increase the degree of phase separation. Moreover, downward orientation of the branch can reduce

the branch flow quality significantly. The pressure changes in T-junctions were correlated using simple momentum and energy balances for the run and branch respectively using measured void fractions. The run momentum correction factor was found to be independent of inlet flow conditions but was dependent on the junction geometry. The branch two-phase multiplier was found to depend on both the inlet flow conditions and junction geometry.

Comparison of the present data on pressure changes in T-junctions with some available models showed that those models which account for phase redistribution effects were better than the others in correlating the present data, confirming the strong interdependence between the pressure changes and phase redistribution. A general phenomenological phase redistribution model was derived based on the analysis of available models. This general model was extended to two phenomenological models for annular flow and stratified inlet flows. Each of the models included two sub models to account for two phase distribution in the inlet tube and phase redistribution in the junction. Comparisons of the present experimental data and some available models were made and the results indicate that most of the available models can predict 70% of annular flow data and 80% of stratified flow data within +40% of the measurements. The newly developed models in this study can predict 90% of the data within +40% for both annular and stratified flows.

CHAPTER 3

METHODOLOGY

3.0 Introduction

This chapter basically is an explanation of working procedure to execute during the whole project. Methodology is an important to define a direction, guideline and method that need to perform later. The design of the 3D model of component and simulation of fluid flow in pipe are discussed. Every method regarding the process of research will be explained including the standard, software and specification of the component that are being use for the research. All the process involved should follow the standard guide line to validate the upcoming data.

3.1 Overall Process Design of pipe

The flow starts with the title selection for Projek Sarjana Muda (PSM). After the title is confirmed, the proposal is prepared and submitted to supervisor. The proposal contains the background of the project, problem statement objective and scope of work.

As the proposal being approved, the process starts with data collection. The data collected through the reliable source such as reference books, journal and internet. All data are important during writing the literature review.

Next process is project methodology. In this chapter, the methods and approaches that being used in order to complete this project will be explain in detail. The process is continues with simulate design through simulation software. Simulation is important to find the best result for the project outcome especially for produce a good quality of pipe. As the best result is obtained and the objectives are achieved, the project will be finalized and complete the final report writing.

3.2 Flowchart of the process

Regarding this project, a few charts and diagrams needed to show the overall process. A flow chart is necessary as it show the planning in flow chart process.

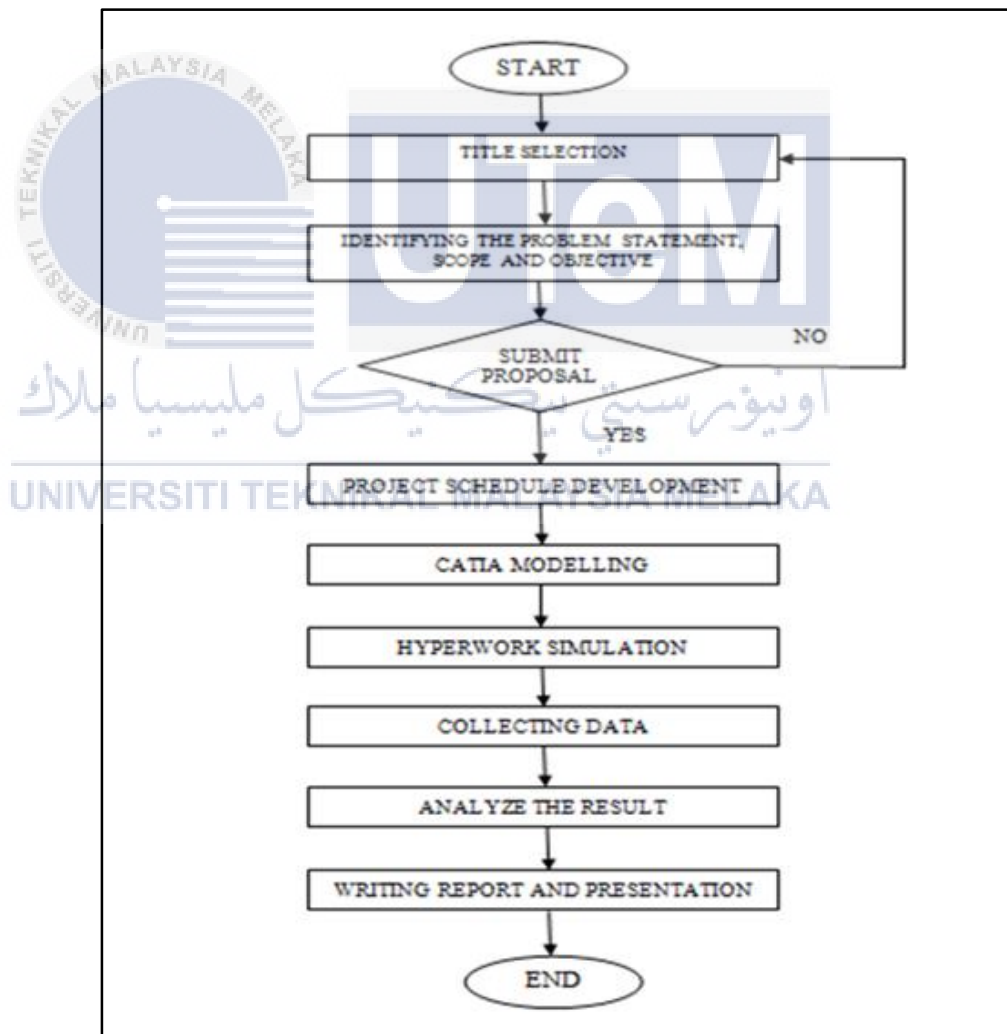


Figure 3.1: Flowchart for design and development of the structure

3.3 Design pipe model

Before designing 3D modeling for a pipe, 2D drawing is sketched to identify the parameters of pipe. Drawing also identifies the fluid flow (inlet and outlet).

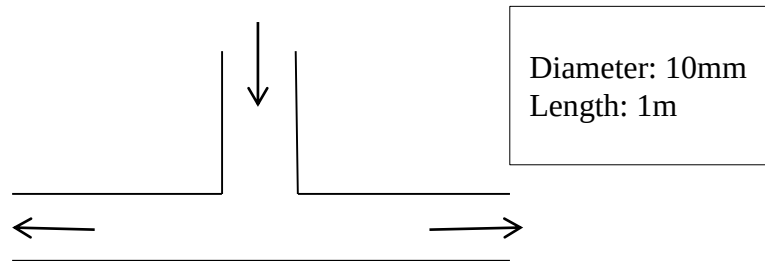


Figure 3.2: 2D drawing show dividing flow of T-Junction Pipe

3D modelling of pipe has been designed by using CATIA v5 software. Whereas this software chosen due to precisely in scaling and dimensioning which follow the actual dimension.

3.4 CATIA v5 Software

CATIA (computer aided three-dimensional interactive application) is a multi-platform computer-aided design (CAD)/computer-aided manufacturing (CAM)/computer-aided engineering (CAE) software suite developed by the French company Dassault Systèmes. CATIA started as an in-house development in 1977 by French aircraft manufacturer Avions Marcel Dassault, at that time customer of the CAD/CAM CAD software to develop Dassault's Mirage fighter jet. It was later adopted in the aerospace, automotive, shipbuilding, and other industries. Commonly referred to as 3D Product Lifecycle Management software suite, CATIA supports multiple stages of product development (CAx), including conceptualization, design (CAD), engineering (CAE) and manufacturing (CAM).

CATIA facilitates collaborative engineering across disciplines around its 3DEXPERIENCE platform, including surfacing & shape design, electrical fluid & electronics systems design, mechanical engineering and systems engineering. CATIA facilitates the design of electronic, electrical, and distributed systems such as fluid and HVAC systems, all the way to the production of documentation for manufacturing.

In designing CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify and validate complex innovative shapes from industrial design to Class-A surfacing with the ICEM surfacing technologies. CATIA supports multiple stages of product design whether started from scratch or from 2D sketches. CATIA v5 is able to read and produce STEP format files for reverse engineering and surface reuse.



Figure 3.3: DS CATIA v5

3.4.1 CATIA modelling T-Junction pipe

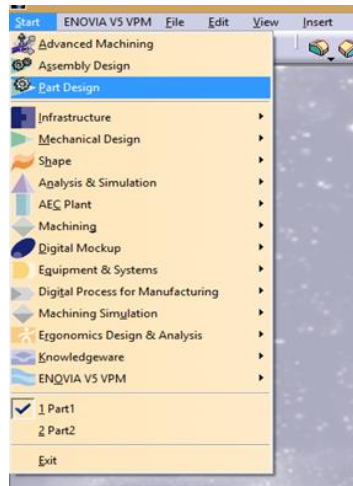


Figure 3.4: The initial step

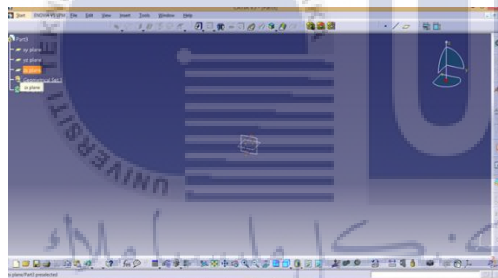


Figure 3.5: Choosing 'xz plane' for sketching

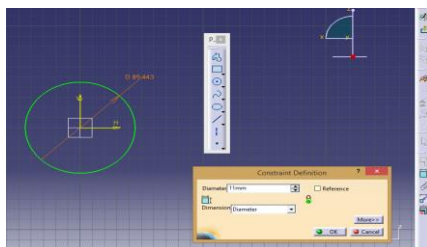


Figure 3.6: Sketch circle and constraint

- CATIA is used in designing 3D model of T-junction pipe. Before start on sketching, click on 'Part Design' to ensure that we can sketch our pipe model. Rename the 'part' before start the modeling process.
- Once the sketch display is appeared (Figure 3.5), choose 'zx plane' as a starting plane for sketching. Click on the 'sketch' icon to enable sketch then 'zx plane' is chosen to create a horizontal pipeline before creating the vertical pipe.
- After enabling the sketch, click 'circle' icon at the right side and draw a circle with diameter 11 millimetre. Thus, click 'constraint' icon at the right side and enter the value of the circle diameter as shown in Figure 3.6.

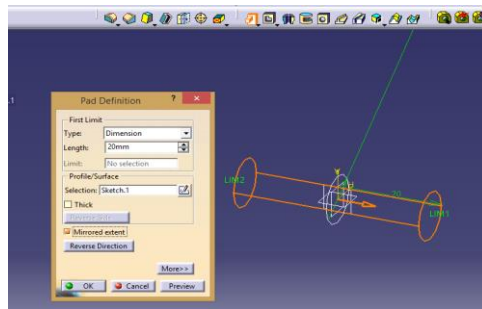


Figure 3.7: Determine the length of horizontal pipe



Figure 3.8: 'Pad' on the drawing

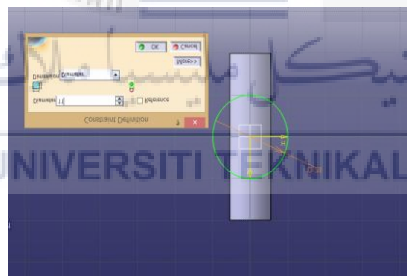


Figure 3.9: Make another junction of the pipe

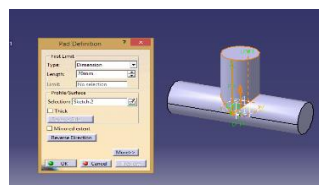


Figure 3.9: 3D drawing for the vertical pipe as the junction

- Create the 3D drawing by click 'exit workbench' icon at the right side and click 'pad' icon to form the drawing from 2D to 3D design. Enter the value of the pipe length which is 1m and diameter of the pipe is 10mm as shown on Figure 3.7 and 3.8

- For a vertical pipe for the junction in middle of straight pipe, 'xy plane' is chosen. Using the same method to draw the horizontal pipe, draw a circle by click 'circle' icon at the right side (Figure 3.9). Enter the value of the circle and do the previous step to form the drawing from 2D to 3D design

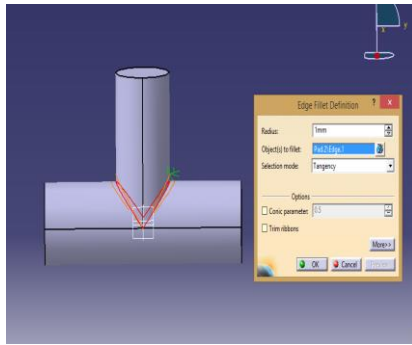


Figure 3.10: Determine the suitable radius and click on the object to fillet

- After the 3D pipe drawing is completed, we need to make it as a 3D model which is have three holes at each branch and make an angular junction at the intersection between the vertical and horizontal pipe. Then, create the angular junction using the 'edge fillet'. Click on the 'edge fillet' icon and determine the suitable radius.

3.5 Simulation



After completing the 3D modeling of T-Junction pipe, continue with simulate the fluid flow of the pipe by using the Hyperwork software whereas the ACuSolve program is used to complete the simulation. Other than that, the parameter, fluid properties and the type of flow used also can define using simulation software. Hyperwork software in other words also called Altair Engineering, is an American product design and development, engineering software and cloud computing software company. Altair Engineering is the creator of the HyperWorks suite of CAE software products. The principal product that offerings from Altair's Commercial Software division is its HyperWorks line of software, including ; MotionSolve - Multi-body Solver; an integrated solution to analyze and optimize multi-body system performance, HyperMesh, HyperCrash, Simlab, HyperView, HyperGraph - CAE Pre & Post Processing, AcuSolve - General-purpose Finite Element Based CFD Solver and etc.

3.5.1: Simulation on the Fluid Flow in T-Junction Pipe

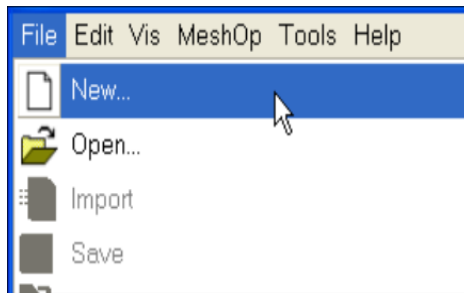


Figure 3.11: Menu Bar to create new database

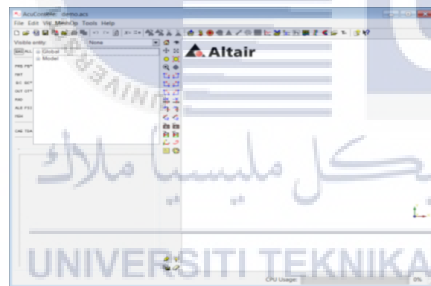


Figure 3.12: Data tree show 'Global' (Mesh/geometry independent) & 'Model' (Mesh/geometry dependent)

- For the simulation process, the process only run until imported the CAD drawing into the simulation software. For the simulation process, we use the ACuSolve program. After launching the ACuConsole, the new database needs to be created before start any simulation. At the menu bar click on 'File' then 'New' then the new database dialog will opens (Figure 3.11).
- Then, navigate to the folder in which the simulation files are to be stored and rename the File name and click 'Save' until the file name seen in the bar.
- At the 'Data Tree' (Shown in Figure 3.12) the 'visible entity' is set to „None“ as there is no Geometry/Mesh

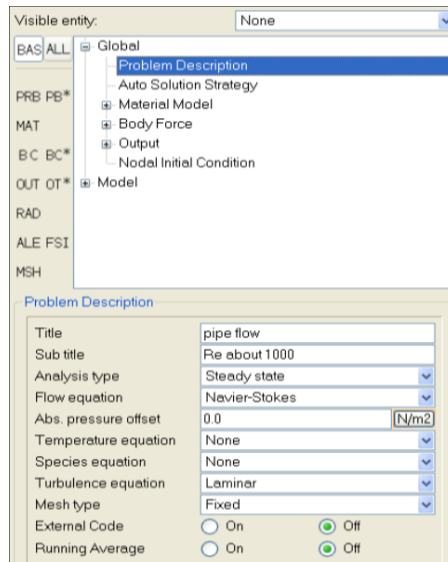


Figure 3.13 : Problem Description

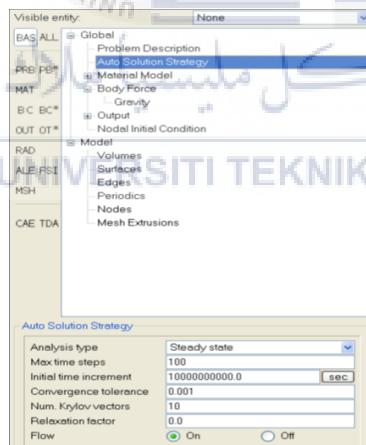


Figure 3.14: Auto Solution Strategy setup

- The next step is to make the problem description. Double-click or right-click to open on Problem Description beneath Global in the Data tree. In the Panels area, set problem parameters of:

•Title: Pipe Flow

•Sub Title: Re about 1000

•Turbulence equation:

Spallart Allmaras

•Mesh type: Fixed

- The Problem Description table also shown on the Figure 3.13

- Also setup the Auto Solution Strategy (shown in figure 3.14) which we used the defaults for steady state analysis, Maximum time of steps which is 100 and flow only.

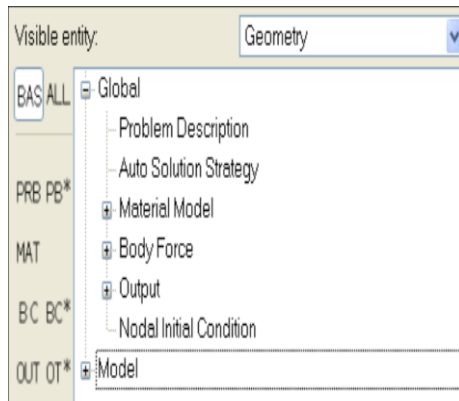


Figure 3.15: Data Tree checklist before importing CAD drawing

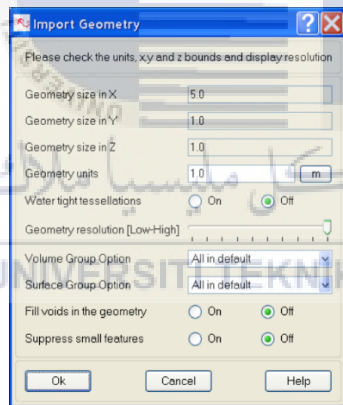


Figure 3.16: Import Geometry to check the units of x, y and z bounds and display resolution

- The next step is importing CAD drawing into the simulation software. Import the CAD drawing file by selecting on the 'Import' tool at the 'File' bar. 'Choose a file to open' to opens the dialog and change the 'Files of Type' to "Acis File" or "Parasolid File". Navigate to the directory in which the CAD model is present and select "pipe.sat " or "pipe.x_t".
- Then click open 'Import Geometry' until the dialog opens. Noted, if Acis file is loaded, need to change the 'Geometry units' from 1000 mm to 1 m. Click "Ok" to load the geometry

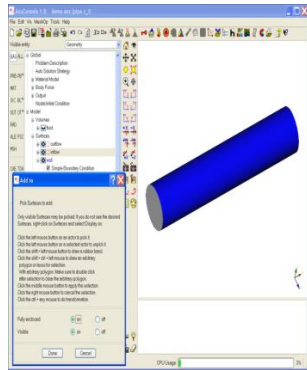


Figure 3.17: This figure shows the example of pipe model in order to show how to define the fluid flow pattern and the wall of the pipe.

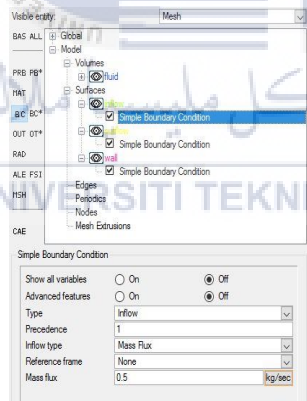


Figure 3.18: Simple Boundary condition with 'flow type' is mass flux

- After the CAD drawing is successfully import, the region of data tree is in 'default' volume group and three (3) faces in 'default' Surface. Rename 'default' volume to 'Fluid'. Meanwhile, for the Surface Grouping, create new three (3) groups at the 'surfaces' which is this three group consist of two 'outflow' and one 'inflow'. Then, rename "default" to "wall". To define the fluid flow pattern and the wall of pipe, right click on the 'inflow' and 'Add To'. Then, pick the inflow face. Repeat with 'outflow' and 'wall'.

- At the 'surface' tree, there are inflow, outflow and wall. For each surface, the condition must be decided at 'simple boundary condition'. Condition can be either mass flux (Figure 3.18) or velocity (Figure 3.19) depends on case studies that need to be stimulated. In this section is important because it will give an affect on the simulation result.

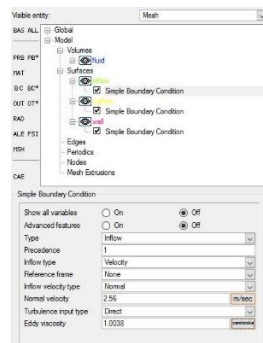


Figure 3.19: Simple Boundary Condition with 'flow type' is velocity

- If the 'flow type' is set with 'velocity', then the 'velocity type' must be set as 'normal'. Because, in this case the fluid flow is present in inflow surface.

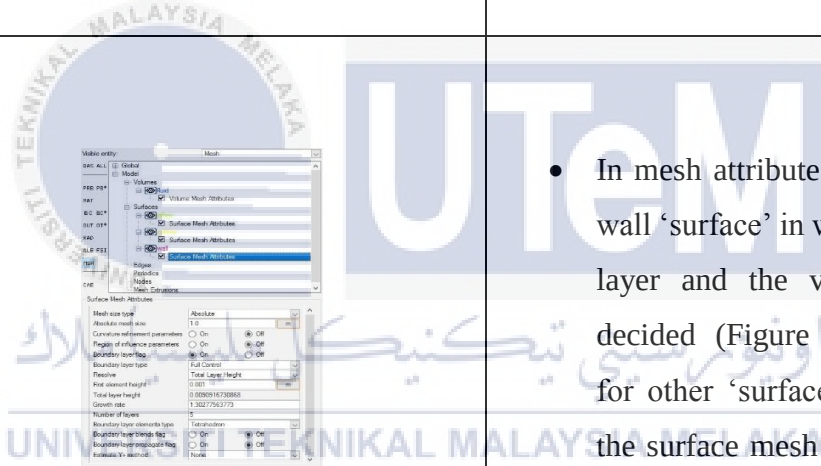


Figure 3.20: Surfaced Mesh Attribute at wall is specified

- In mesh attribute section is more on wall 'surface' in which the number of layer and the value of mesh are decided (Figure 3.20). Meanwhile, for other 'surface' (inflow,outflow), the surface mesh attribute only need to be able(Figure 3.21).

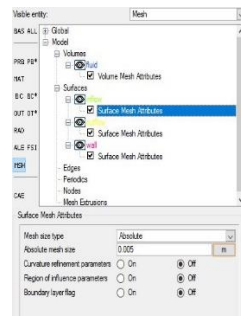


Figure 3.21: Surface Mesh Attribute for both inflow and outflow

- Before proceeding the simulation, right click on 'volume' and select 'volume manager'. The 'volume manager' box will appear and the type of 'material model' is decided then click 'add to' to select on the wall of the pipe to able the simulation giving the result (Figure 3.22)

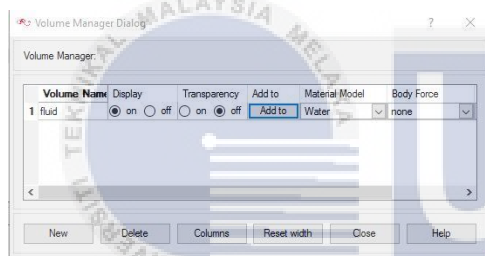


Figure 3.22: Volume Manager dialog box

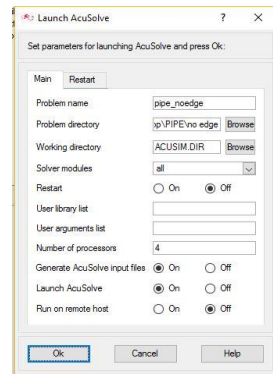


Figure 3.21: ‘Launch AcuFieldView’ dialog

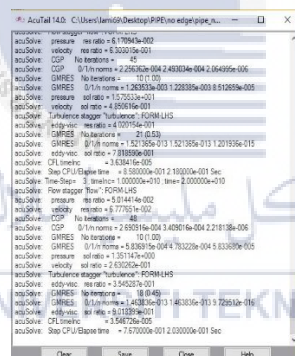


Figure 3.22: AcuTail dialog box

- By clicking on the ‘generate’ and ‘run’ button,  AcuField view can be proceed. AcuFieldView offers the interactive review of transient data with sweep caching, along with CFD data management capabilities. Also, it can handle steady and unsteady data of any size with high speed.

- Click on ‘AcuFieldView’ button  to start the simulation. Ensure that the path of the log file is provided (Figure 3.21). Then click ‘ok’. ‘AcuTail’ is appear to give the respon either the ‘AcuFieldView’ process can be proceed or not (Figure 3.22).

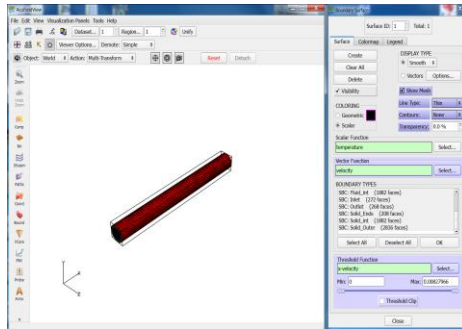


Figure 3.23: AcuFieldView when opened

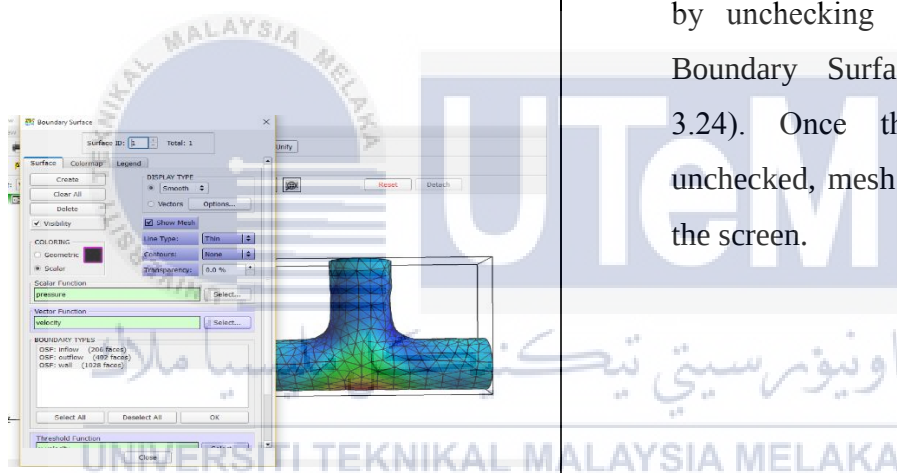


Figure 3.24: Unchecking 'Visibility' in boundary surface panel

- 'AcuField' is open with the 'Boundary Surface' displayed (Figure 3.23). For the first, 'AcuField' background may appear in all black. But the background colour may be changed by clicking 'view' button.
- Turn off the display of Surface ID 1 by unchecking 'Visibility' in the Boundary Surface Panel (Figure 3.24). Once the 'Visibility' is unchecked, mesh will not appear on the screen.

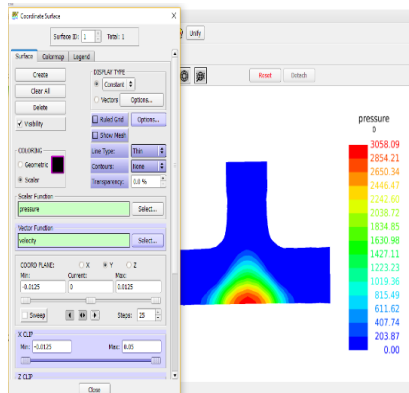


Figure 3.25 : Coordinate Surface in
AcuFieldView

- Next is 'Coordinate Surface' (Figure 3.25). There is three section in 'Coordinate Surface' which is surface, colourmap and legend.

- For the 'surface' section, ensure the 'Visibility' is able and click 'create'. The mesh of pipe will appear. Then;

- Display type is set with 'constant'
- Choose y-axis at the 'coordinate plane'
- 'coloring' is set with 'scalar'

- In order to start the simulation, choose the element that needs to be calculated at 'scalar function'.

- In 'colourmap' section normally for the display of the result that appear next to the mesh pipe

- For the 'Legend' section, 'Legend' must be able to ensure the result can appear and shows the reading of the result.

CHAPTER 4

RESULT & DISCUSSION

4.0 Introduction

In this chapter, the results from the project are discussed. The design of the pipe has same parameter (diameter, length and velocity). In section 4.3 and 4.4 discuss the effect of velocity for each axis and the effect of pressure in pipe. While in section 4.5 and 4.6 discuss how wall shear stress will affect the pipe and the velocity magnitude giving the result in pipe.

4.1 General Procedure for Pipe Simulation

- Steps that are involved; pre-processing, solving and post-processing. For pre-processing, the model need to define the main problem, the parameter in which will set as constant or as responding parameter and define the model's behaviour. In solving step, the value of the constant parameter is finalised also the value of the responding parameter in which is needed in order to get the result.
- Finally, in post processing the analysis and evaluation of the result is conducted in this step. Examples of operations that can be done include define the type of fluid flow and pipe conjunction, defining the parameter of the pipe and the elements to investigate. Specific procedures of pre and post are different based on the software used. Table 4.2 shows the parameter used for investigation on each pipes.

Table 4.1: Parameter set for each pipe

Pipe	Diameter (mm)	Velocity 1 (m/s)	Velocity 2(m/s)	Velocity 3(m/s)
Sharp Edge	25	2.56	2.00	1.50
5mm Edge	25	2.56	2.00	1.50
10mm Edge	25	2.56	2.00	1.50

Hyperwork Accusolve is the simulation software in which it is often used throughout the procedure to get the result. The verifying result may have some difficulties and not giving an exact reading if the parameter of the model is not properly setup. Also, several mistakes from the import drawing (CATIA) may affect the simulation and result.

4.2 Effect on Velocity with various edges and inlet velocity

Flow velocity is a vector quantity used to describe the motion of a fluid. It can be easily determined for laminar flow but complex to determine for turbulent flow. In this case, the velocity for each axis which is x, y and z of the pipe has been examined. Based on the studies, there is three type of pipes used; pipe without edge, pipe with edge 5 mm and pipe with edge 10mm. Figure below shows the effect of velocity on T-junction pipe with three (3) different edges.

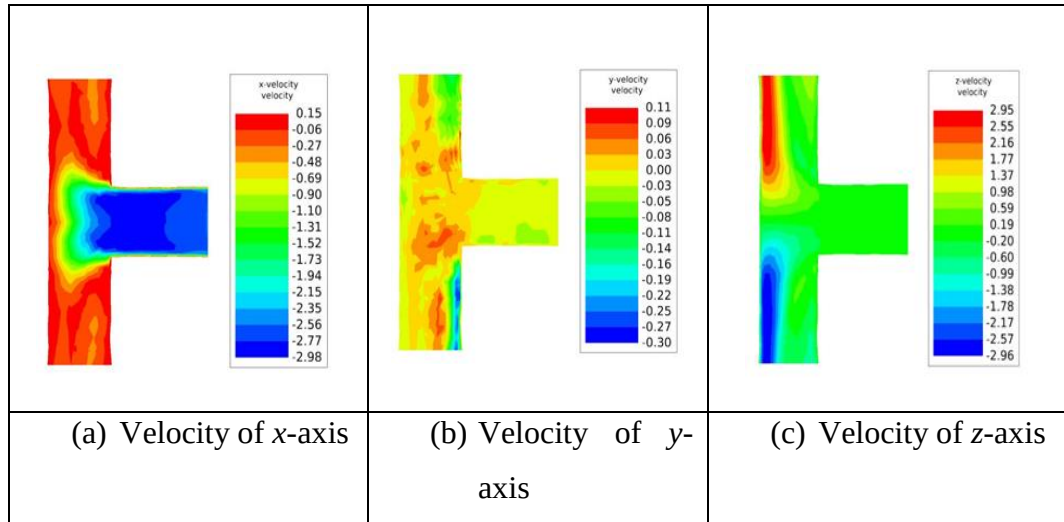


Figure 4.1: The velocity (m/s) of x, y and z axis of pipe with sharp edge

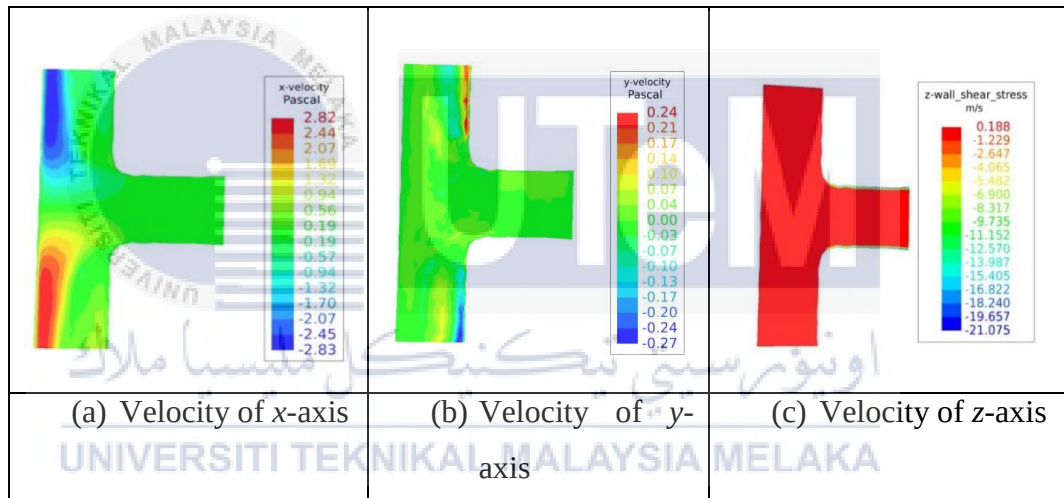


Figure 4.2: The velocity (m/s) of x, y and z axis of pipe with edge 5mm

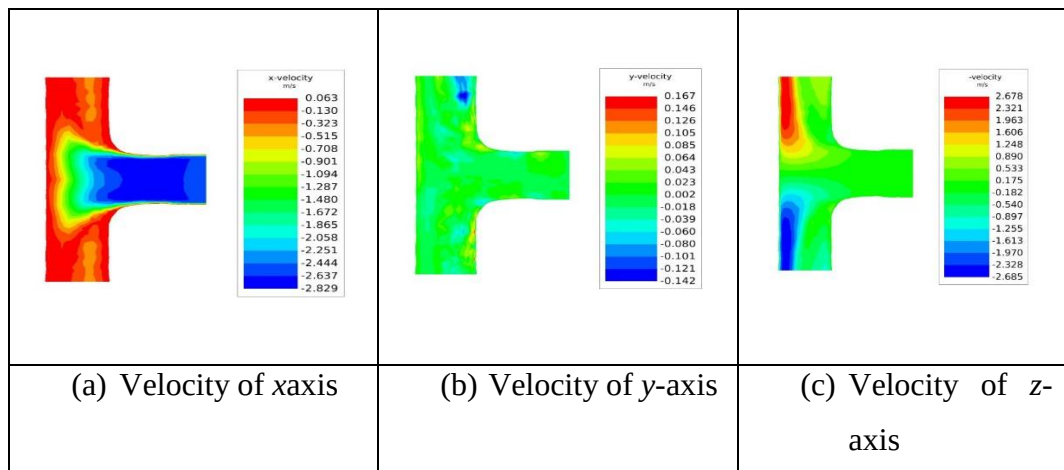


Figure 4.3: The velocity (m/s) of x, y and z axis of pipe with edge 10mm

The value of velocity are different depends on the axis selected and the edge value of the pipes. Based on the simulation that has been conducted, result shows the biggest value of edge have the minimum reading of velocity of the fluid flow.

Table 4.2: Value of axis-velocity

Velocity	Pipe with no-edge	Pipe with edge 5mm	Pipe with edge 10mm
x-axis	0.15	0.08	0.063
y-axis	0.06	-0.03	-0.018
z-axis	2.95	2.82	2.67

4.3 Effect on Pressure with various edge and inlet velocity

The pressure for each type of pipe gives different reading due to the different value of edge as shown in Figure 4.4. It is because, the greater edge of pipe may reduce the pressure effect on the pipe. According to the studies and result from Bertani (2007) and Shaoping (2012). The study state that the pressure loss resulted at minimum value in pipe with large edge or in arc pipe compared with pipe with sharp edge. It is proved in both theoretical and simulation.

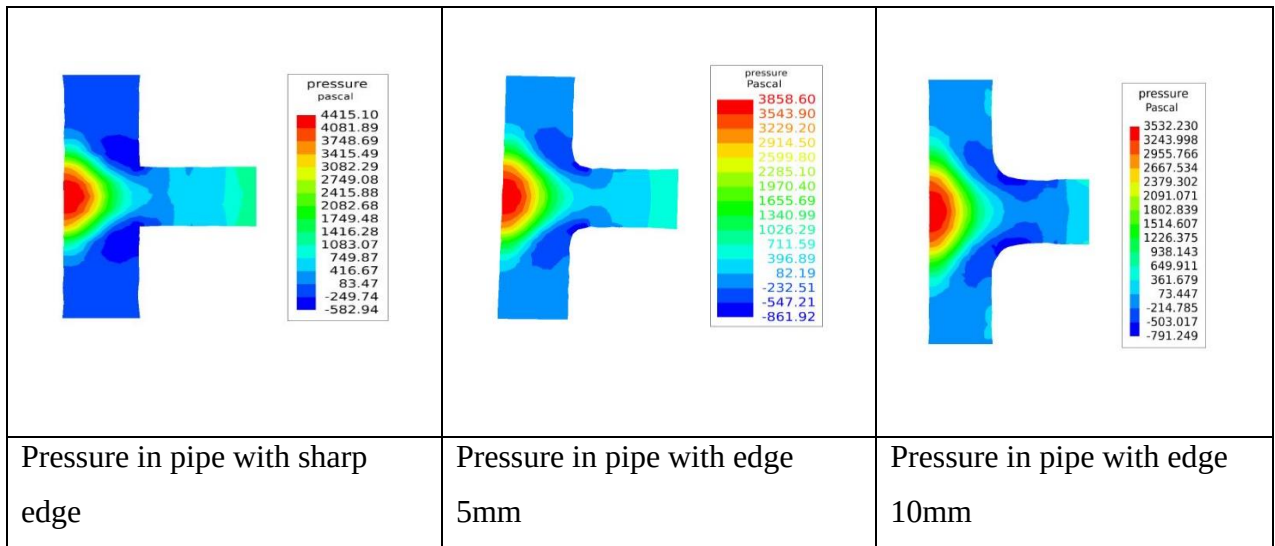


Figure 4.4.: Effect of pressure on T-junction Pipe with different edge

Pipe with sharp edge shows the highest value of pressure at the middle of the straight pipe also at the edge of the pipe. Based on the Figure 4.4.1, the value of pressure of each pipe is slightly different due to the fluid flow through the edge of the pipe. Pipe with edge 10mm gives the minimal value of pressure in both at the middle of the straight pipe and at the edge of the pipe.

اوپنورسیتی ٹیکنیکل ملیسیا ملاک
UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Table 4.3: Pressure value for each pipe

	Sharp Edge	Edge 5mm	Edge 10mm
	Pressure (Pascal)		
Middle of straight line	4415.10	3858.60	3532.23
Corner of the pipe	83.47	82.19	73.45

Meanwhile, the value of velocity is set in range ± 0.5 m/s to ensure the result is being more able to define. Results show the reading of the pressure for each type of pipes with different inlet velocity. Based on the figure shown below, the increasing of

pressure value influenced by the increasing of the velocity. The edge of pipes also give the influences on the pressure in the pipe. Table 4.4 clearly shows the pressure reading for each various velocity in which the readings also explained how the increasing pressure influenced by the velocity.

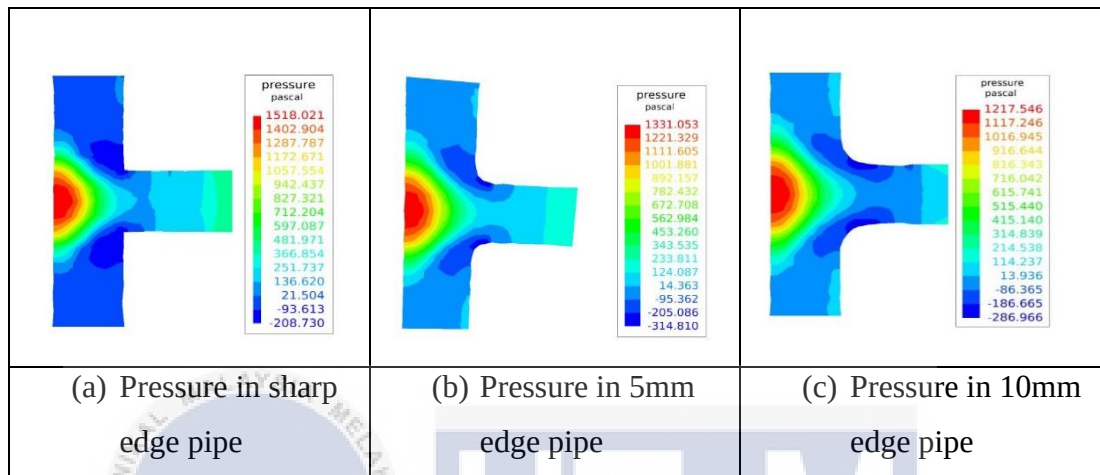


Figure 4.5: Pressure effect with inlet velocity 1.5 m/s

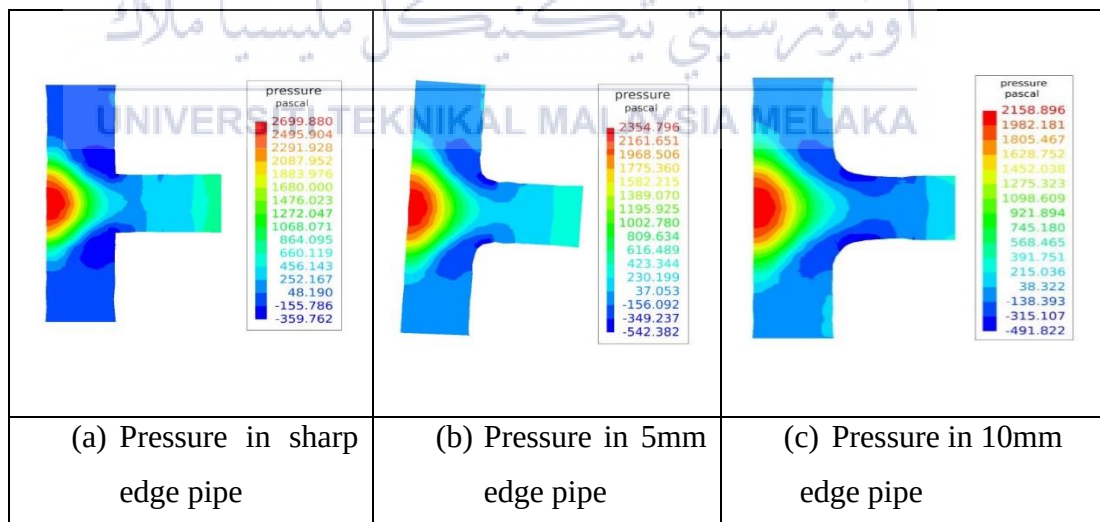


Figure 4.6: Pressure effect with inlet velocity 2.0 m/s

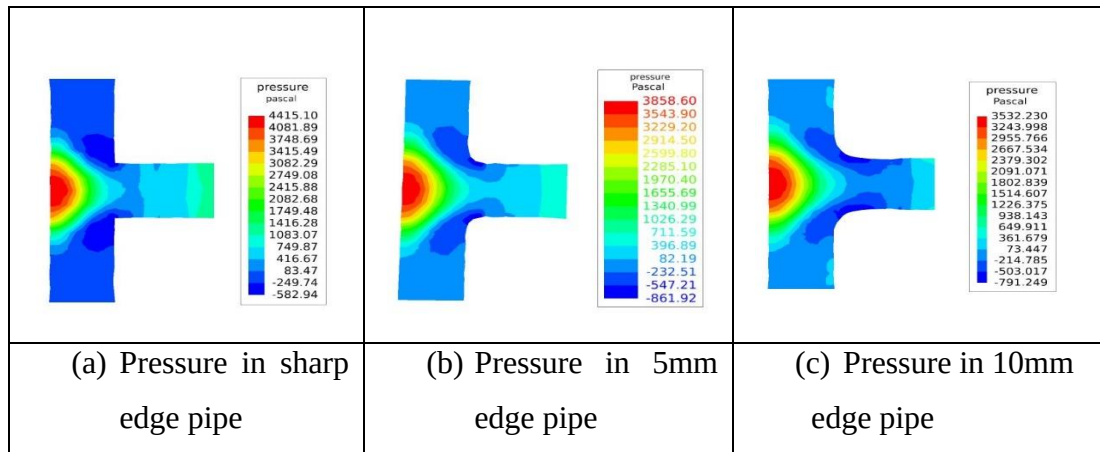


Figure 4.7: Pressure effect with inlet velocity 2.56 m/s

Table 4.4: Pressure value in various velocity

Pressure in pipe (Pascal)/ Velocity(m/s)	Sharp Edge	5mm Edge	10mm Edge
1.50	4415.10	3858.60	3532.23
2.0	2699.88	2354.80	2158.60
2.56	1518	1331	1217.50

4.5 Effect on wall shear stress with various inlet velocity and edges

Hyperwork simulation also shows the effect of wall shear stress depends on the selected axis of the pipe. Wall shear stress is define as the shear stress in the layer of fluid next to the wall of a pipe. As a wavy flow of fluid in a straight vessel, at each point of the vessel fluid does not move at the same velocity. Instead, fluid flow is fastest at the centre and slowest close to the wall. The fluid velocities assume a parabolic profile referred to as the "laminar flow" profile. Laminar flow (or streamline flow) occurs when a fluid flows in parallel layers, with no disruption between the layers. At low velocities, the fluid tends to flow without lateral

mixing, and adjacent layers slide past one another. This pattern of flow is the result of friction within the fluid and between the fluid and the vessel wall and is related to the fluid viscosity. This friction creates a tangential force exerted by the flowing fluid and is referred to as the "wall shear stress". How fast the fluid velocity increases when moving from the vessel wall toward the centre of the vessel is define as the magnitude of wall shear stress. Shear stress resulted on the surface of the pipe wall with the presence of fluid flow. The result may slightly different among the three pipes because of the various edges for all pipes. Figures (4.8, 4.9, and 4.10) shows the result of wall shear stress obtained from the simulation while Table 4.5 shows the higher reading of wall shear stress for each axis in each pipes.

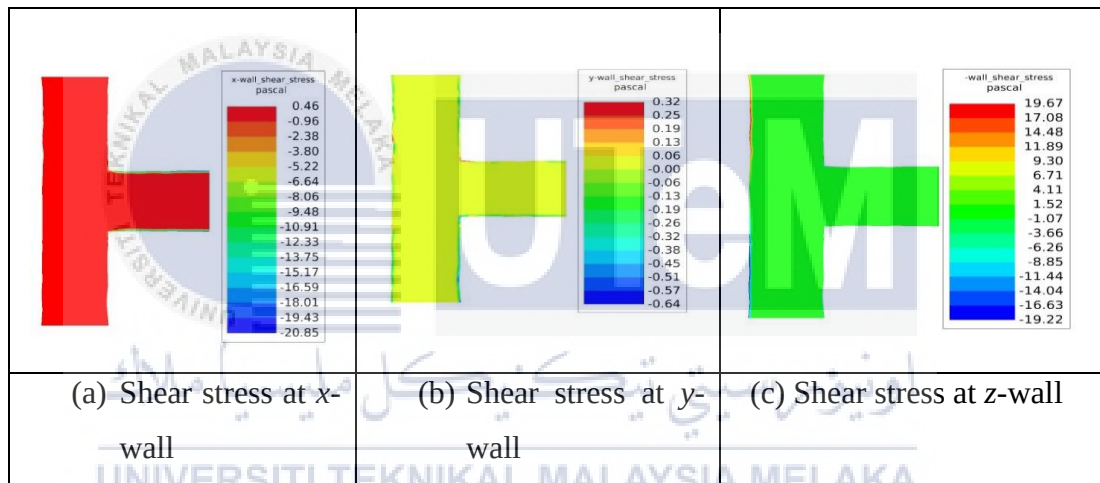


Figure 4.8: Shear stress at wall of pipe with sharp edge

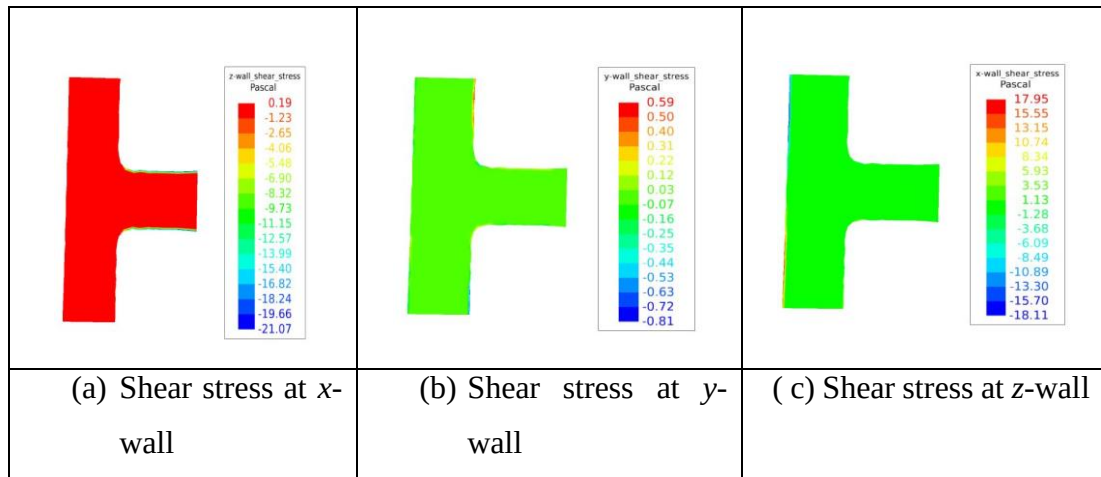


Figure 4.9: Shear stress at wall of pipe with 5mm edge

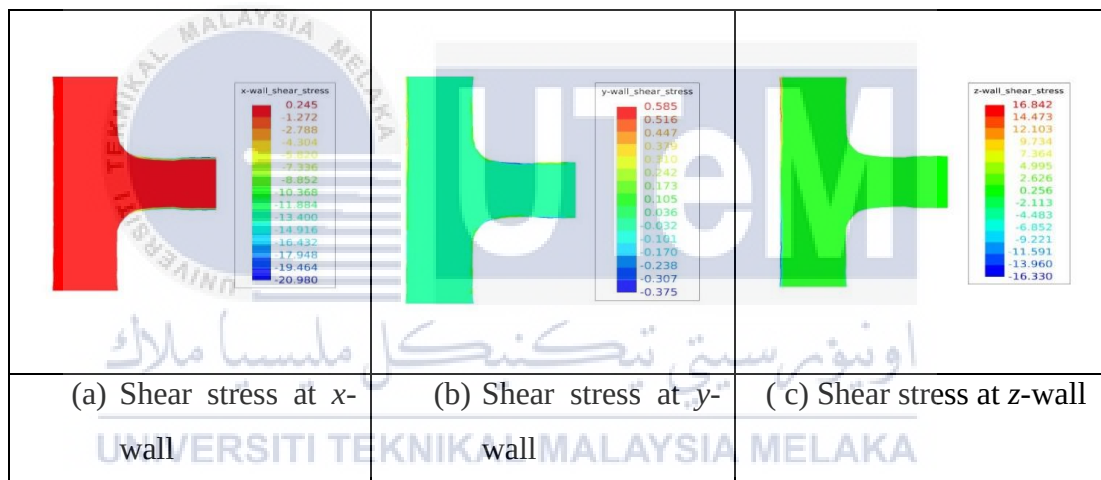


Figure 4.10: Shear stress at wall of pipe with 10mm edge

Table 4.5: Wall shear stress on the axis of each type

Wall Shear Stress (Pascal)			
Pipe/ axis	x-axis	y-axis	z-axis
Sharp Edge	0.46	0.32	19.67
5mm Edge	0.19	0.59	17.95
10mm Edge	0.25	0.58	16.84

4.6 Effect on velocity magnitude with various velocity and edge of pipe

Velocity is a physical vector quantity. Thus, both magnitude and direction are needed to define the value of velocity. A simulation that has been conducted for velocity magnitude resulted in giving highest reading of velocity of the fluid flow in the specific area of the pipe.

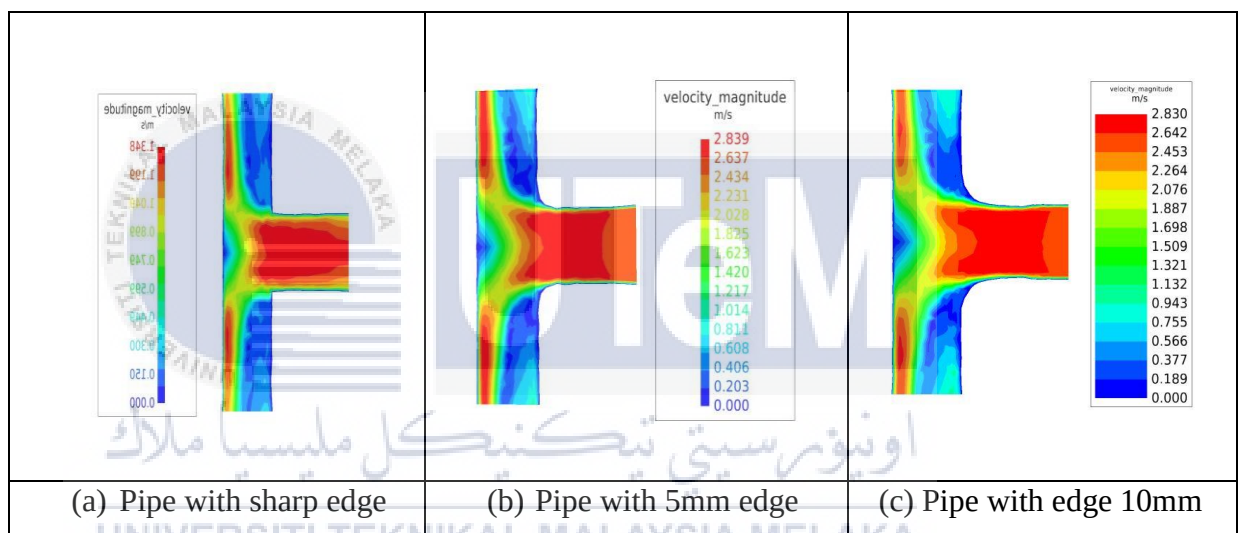


Figure 4.11: Velocity magnitude for T-junction pipes

Table 4.5: Value of velocity magnitude

Pipe	Velocity Magnitude (m/s)
Sharp edge	1.348
5mm	1.320
10mm	2.830

Based on the result shown, the value of velocity magnitude for pipe with edge 10 mm has the highest reading of velocity which is 2.830 m/s. Reading of the velocity magnitude is high due to the large edge of the pipe and give smooth flow of the fluid. Compared with the other two type of pipe which has smaller edge, the value of velocity magnitude is 1.348 and 1.320 respectively lower than the velocity magnitude of pipe with 10mm edge. Meanwhile, velocity magnitude also give an effect on the various velocity instead of various edge of pipes.

Figure 4.12 shows the result of velocity magnitude with various velocity for each type of pipes. The observation is then recorded as shown in Table 4.7.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

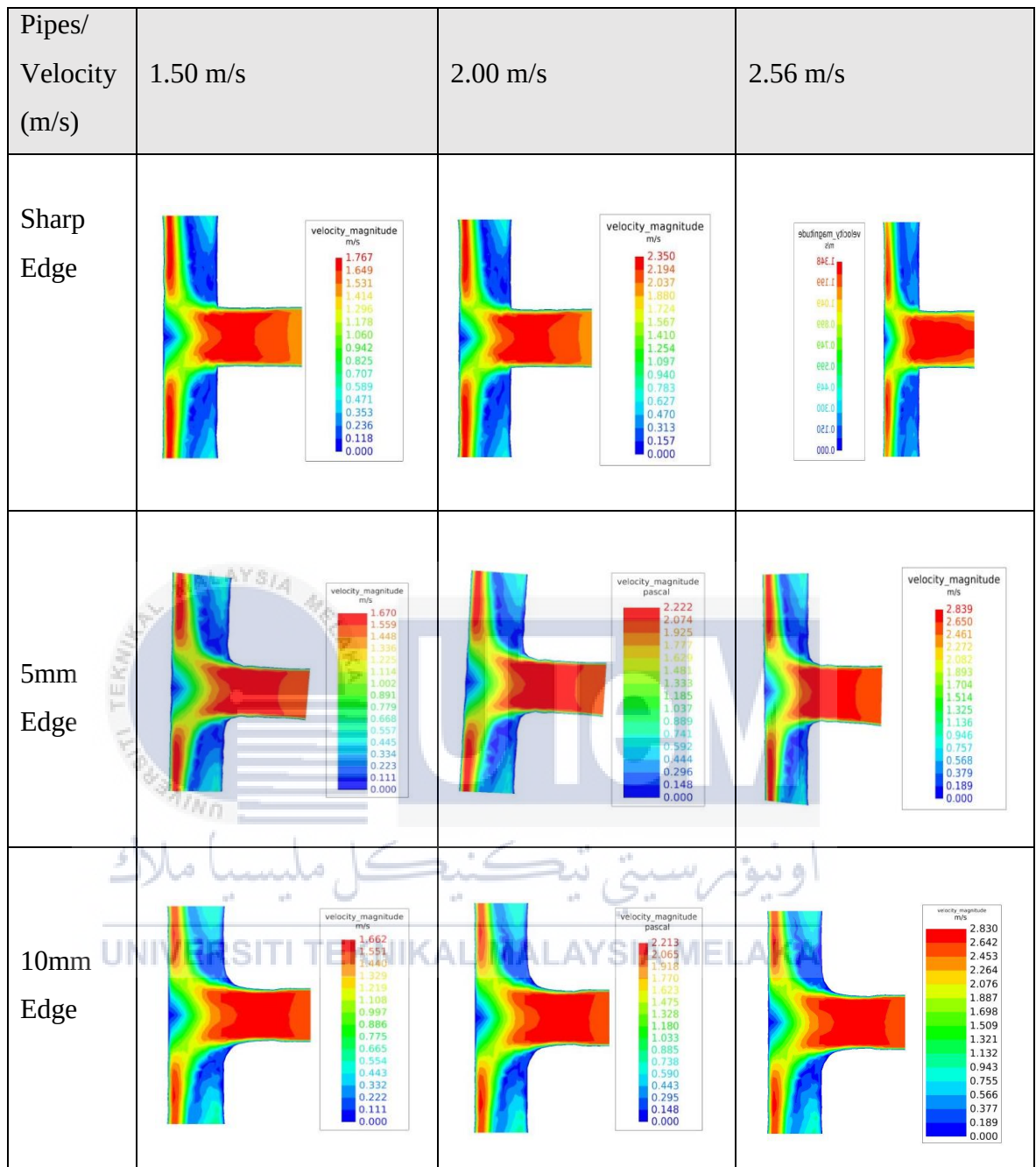
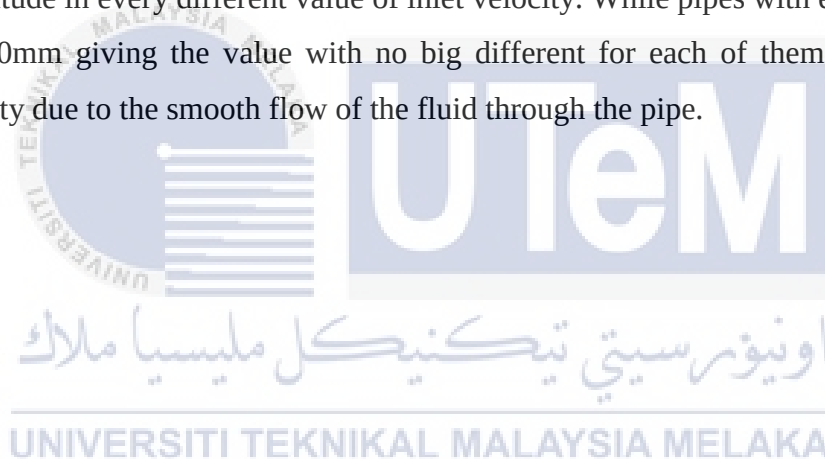


Figure 4.12: Velocity Magnitude in various inlet velocity

Table 4.7: Value of velocity magnitude with various inlet velocity

Pipes/ Velocity (m/s)	1.50 m/s	2.00 m/s	2.56 m/s
Sharp Edge	1.76	2.35	2.86
5mm Edge	1.67	2.22	2.84
10mm Edge	1.66	2.21	2.83

Based on Table 4.7 shows that the sharp edge pipe give the highest reading of velocity magnitude in every different value of inlet velocity. While pipes with edge with 5mm and 10mm giving the value with no big different for each of them in every inlet velocity due to the smooth flow of the fluid through the pipe.



CHAPTER 5

CONCLUSION & FUTURE WORK

5.0 Conclusion

The project primary challenge was to set up the parameter of the pipe in which studies must be held by referring on previous studies. T-junction pipe with dividing flow is set with constant parameter (diameter, velocity). 3D modelling pipe is then designed by using suitable drawing software (CATIA). In order to determine the effect of inlet velocity of fluid flow in the pipe, simulation (Hyperwork software) has been conducted on the pipe modelling. Investigation on the effect of the inlet velocity, pressure, wall shear stress and defining velocity magnitude is held on the T-junction pipe model with three (3) different edge. The result shown depends on the parameter of the pipe and the selected element for investigation.

According to the simulation on the studies of the effect on the velocity with various inlet velocity and edge towards the T-junction pipe, the result shows the biggest value of edge have the minimum reading of velocity of the fluid flow. The result of the velocity with different axis is almost similar with the result of velocity magnitude obtained from the simulation, in which the sharp edge of pipe giving the highest reading of the velocity of the fluid flow. Both values of velocity magnitude and velocity with different axis are influenced by the size of pipe's edge.

Meanwhile, based on the simulation that has been held on the effect of pressure and wall shear stress towards the T-junction pipe, results that are obtained from both simulations shows the highest inlet velocity will influence the increasing value of pressure and wall shear stress. Also, the size of the edge is indirectly affected by the resulted value of pressure and wall shear stress.

5.1 Future Studies

Simulation on the pipe can be improved by using other simulation software. For example by using FLUENT software, which can give more advance on the simulation's studies. The element and parameter provided in FLUENT can give better result in pipe modelling. Other than that, FLUENT not only can give the best result of simulation on fluid but also can conduct the simulation on gas and oil. Thus, more studies can be handled in order to enhance an acknowledgement on the pipe's behavior.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Abdulwahhab, M., 2013. Numerical prediction of pressure loss of fluid in a T-junction. *Journal homepage: www. ...*, 4(2), pp.253–264. Available at: http://www.ieefoundation.org/ijee/vol4/issue2/IJEE_09_v4n2.pdf.

Azzopardi, B.J. & Whalley, P.B., 1982. The effect of flow patterns on two-phase flow in a T junction. *International Journal of Multiphase Flow*, 8(5), pp.491–507.

Bertani C., Malandrone M., Panella B., Quaranta A., Squillari P., 2007. Pressure drop and phase separation in a horizontal T-junction, *Procs. XXIV Congresso Nazionale sulla Trasmissione del Calore (National Heat Transfer Conference)*, pp. 267-274 Napoli(Italy)

Costa, N.P. *et al.*, 2006. Edge effects on the flow characteristics in a 90 deg tee junction. *JOURNAL OF FLUIDS ENGINEERING-TRANSACTIONS OF THE ASME*, 128(6), pp.1204–1217.

Li, X. & Wang, S., 2013. Flow field and pressure loss analysis of junction and its structure optimization of aircraft hydraulic pipe system. *Chinese Journal of Aeronautics*, 26(4), pp.1080–1092

Paál, G., Maia, R. and Pinho, F. T., 2003. Numerical predictions of turbulent flow in a 90 tee junction. *Proceedings of 12th International Conference on Modelling Fluid Flow*, Budapest, Hungary, 3rd to 6th September, paper IF0-31, pp. 573-580

Paritosh R. Vasava, 2007. Fluid Flow in T-Junction of Pipes. LAPPEENRANTA UNIVERSITY OF TECHNOLOGY

Peng, F., Shoukri, M. & Ballyk, J.D., 1998. AN EXPERIMENTAL INVESTIGATION OF STRATIFIED STEAM-WATER FLOW IN T-JUNCTIONS Phase Redistribution Results. In *Third International Conference on Multiphase Flow*. pp. 1–8.

Saffari, H. & Dalir, N., 2013. Calculation of pressure drop inside condensing vertical pipes in new inlet pressures using a new modified three-fluid model. *Scientia Iranica*, 20(3), pp.477–482.

Yang, L. & Azzopardi, B.J., 2007. Phase split of liquid-liquid two-phase flow at a horizontal T-junction. *International Journal of Multiphase Flow*, 33(2), pp.207–216.