UNIVERSITI TEKNIKAL MALAYSIA MELAKA

EFFECTS OF BRANCH PIPE DIAMETER RATIO ON FLUID FLOW IN Y-JUNCTION PIPE FOR COMBINING FLOW

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

by

MOHAMAD ZUHAILMIE BIN MOHD ZAKARIA

B071310068

900226-06-5039

FACULTY OF ENGINEERING TECHNOLOGY

2016

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **Effects Of Branch Pipe Diameter Ratio On Fluid Flow In Y-Junction Pipe For Combining Flow**

SESI PENGAJIAN: **2016/17 Semester 1**

Saya **MOHAMAD ZUHAILMIE BIN MOHD ZAKARIA**

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	

(TANDATANGAN PENULIS) Disahkan oleh:

__________________________ __________________________

Alamat Tetap:
NO. 335, Felda Tersang 03,
27600, Raub, Pahang
Pahang Darul Makmur

Cop Rasmi:

Tarikh: 09 DECEMBER 2016

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 "Effects Of Branch Pipe Diameter Ratio On Fluid Flow In Y-Junction Pipe For Combining Flow" is the results of my own research except as cited in references.

Signature	:	………………………………………….
Author's Name	:	Mohamad Zuhailmie Bin Mohd Zakaria
Date	:	09/12/2016

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 Manufacturing Engineering Technology (Process and Technology) with Honours. The member of the supervisory is as follow:

………………………………

(Mr Iskandar Bin Waini)

ABSTRAK

Paip cabang "Y" digunakan secara meluas dalam sektor industri, sistem rumah, automotif dan lain-lain lagi, untuk membawa atau memindahkan gas dan cecair seperti air, minyak, cecair kimia, oksigen dan sebagainya. Paip cabang ini akan membantu paip lurus untuk melaksanakan atau meningkatkan proses pengangkutan seperti dari satu salur masuk menjadi dua salur keluar dan dari dua salur keluar menjadi satu salur masuk. Setiap sistem aliran paip adalah berbeza mengikut kegunaan dan parameter paip tersebut. Dalam kajian ini, siasatan tertumpu kepada nisbah kesan diameter paip cabang terhadap ciri-ciri tekanan dan halaju yang berlaku dalam paip cabang "Y". Perisian CATIA V5 digunakan untuk menghasilkan reka bentuk paip cabang "Y". Bagi menentukan kesan pelbagai nisbah diameter terhadap aliran gabungan, simulasi terhadap model paip dijalankan dengan menggunakan perisian HyperWorks. Dalam usaha untuk memerhati ciri-ciri aliran bendalir, "mesh" dibina dan digunakan pada syarat sempadan untuk semua nisbah diameter paip cabang "Y" dalam AcuSolve. Jenis-jenis aliran masuk yang digunakan dalam sempadan permukaan adalah fluks jisim, halaju, tekanan dan kadar aliran. Jenis-jenis sempadan isipadu yang digunakan dalam simulasi ini adalah air, diesel dan cecair ammonia. Keputusan halaju terhadap tekanan menunjukkan, nilai tertinggi magnitud halaju dan tekanan berlaku pada paip cabang 2 inci bagi keputusan simulasi kadar aliran, halaju dan tekanan. Kemudian, keputusan simulasi bagi fluks jisim menunjukkan, halaju dan tekanan yang paling tinggi berlaku pada 1.5 inci dan 1 inci paip cabang. Kemudian, kelikatan sesuatu cecair memberi kesan ke arah ciri-ciri aliran cecair dalam sistem paip. Kelikatan yang tinggi membawa kepada rintangan aliran yang tinggi di dalam paip dan menyebabkan halaju dalam paip ini juga berkurangan. Pada akhir tugas, kesimpulan dibuat dan beberapa cadangan diusulkan untuk memajukan lagi kajian ini untuk kajian di masa depan.

ABSTRACT

Y-Junction pipes are widely used in industries sector, home systems, automotive and others. In industries, pipe networks widely used to transport fluid and gases such as water, oil, chemical liquid, oxygen and so on. This junction will help the straight pipe to perform or improve the process of transportation such as from one inlet to be two outlets and from two outlets to be one inlet. Each pipe flow systems are different according to their usability and parameter. In this study, the investigations focus on the effect diameter ratio of branch pipe towards the pressure and velocity behaviour that occurs in the Y-junction pipe. CATIA V5 software is used in order to create a design of pipe. As for determining the effect of various diameter ratios for combining flow, simulation upon the pipe model is conducted by using HyperWorks software. In order to observe the behaviour of fluid flow, generate the mesh and apply boundary conditions in AcuSolve for all diameter ratios of the branch pipe. The inflow types used in surface boundary condition are mass flux, velocity, pressure and flow rate. The types of volume boundary condition used in this simulation are water, diesel and ammonia liquid. The results of velocity against pressure shows, the highest value of velocity magnitude and pressure occurs at the 2 inch branch pipe for result simulation of flow rate, velocity and pressure. Then, the result simulation of mass flux condition shows, the highest velocity and pressure occurs at the 1.5 inch and 1 inch pipe branch respectively. Then, the viscosity of the fluid gives the effect towards the fluid flow behaviour in the pipe system. The high viscosity leads to the high flow resistance in the pipe and causes the velocity in this pipe also decrease. At the end of the task, the conclusion is making and several recommendations are suggested to further advance in this field for future study.

.

DEDICATION

This Projek Sarjana Muda (PSM) is dedicated to my beloved parents, Mohd Zakaria Bin Ismayatin and Farsiah Binti Haji Abas. Also, not forget to express my appreciation to Supervisor, Mr. Iskandar Bin Waini and Co-Supervisor, Mr. Mohamad Ridzuan Bin Mohamad Kamal, who kindly assisted, support, encouragement and guided me during the PSM process.

ACKNOWLEDGEMENT

By taking this opportunity, praises and thanks to Allah, Lord of the world for His guidance and will, for the revelation of some of His knowledge for me in the successful of completing my Projek Sarjana Muda (PSM).

First of all, I would like to extend my gratitude to my supervisor, Mr Iskandar Bin Waini for his guidance and advice during this PSM. He has taught me with passion and patience that lead me in completing this PSM. He also provides me with useful sources and information that really helps me in finishing my PSM. I will never forget his assisting, support, encouragement, and advice given to me.

Last but not least, I want to express my appreciation to Co. Supervisor Mr Mohamad Ridzuan Bin Mohamad Kamal for his guidance and advice that was given to me. Also, to my beloved parents, who kindly assisted and guided me during the evaluation progress. Finally, thank you to all those involved directly and indirectly helping me out during my PSM which I cannot state out every one of them.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

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.

1.1 Background of Study

Normally, the purpose of pipe network is to transport and supply the gases and fluid from one location to the other location. In industries, pipe networks widely used to transport fluid and gases such as water, oil, chemical liquid, oxygen and so on. Nowadays, the uses of pipe increasing demand in the industry. The principle types of pipes that are available in the market are Cast Iron pipes and fitting, Plastic or Pvc pipes, Galvanized Steel pipes, Asbestos Cement (AC) pipes and Concrete pipe.

In the pipe networks, there are several types of junction or branch is commonly used in pipe networks such as Y-junction pipe, T-junction pipe and Arc junction pipe. These junction devices will help the straight pipe to perform or improve the process of transportation such as from one inlet to be two outlets and from two outlets to be one inlet. These devices can use as a dividing or combining flow in the pipe network. The difference of diameter ratio of junction also can affect the fluid flow properties and pressure in the pipe network. Normally in the pipe network have a two types of fluid flow caused by the junction device and each type of fluid flow have its own characteristic which able to affect the fluid properties and the pressure. There are laminar flow and turbulent flow. At the certain point of a network especially in industries sector, requires a large and very precise pressure. The pressure loss is divided into three parts. There is friction loss between the fluid and pipe wall. Then, the curved loss due to flow direction alteration and the last one is the confluence loss due to the flow cross section area alteration in the junction. The main problem occurred when using the junction device are the performance or properties of the flow field is not consistent such as fluctuating pressure, velocity increase or decrease and others factor will affect the flow field in the pipe system.

In this existing find out about focused on a small and frequent component of pipe community which is Y-junction (some additionally refer as ’Wyes’). The Y-junction gives the highest efficiency and highest power output. It also gives the smallest mixing volume and shorter bubble. It is one of the junction devices which would convey angular tube due to its structure. A standard Y-junction allow splitting a branch equally in two directions to reduce friction in pipe flow.

1.2 Problem Statement

Pipe system or pipe networks are in generally used for transfer or transport and supply of fluids or gases from one location to other location. The loss pressure during the transfer and supply process it may happen due to trade in the momentum of the flow triggered to friction and pipe component. Therefore, this project requires the understanding of the behaviour of fluid flow that occurs in the pipe. In order to get a clear view of the problem, the simulation of fluid flow in a pipe is conducted using software. In the industries field, the pipe networks are commonly large and require very particular pressure and the problem will become up when some fluid flow cannot achieve the flow target due to losses in pipe. In this study, the investigations focus on the effect diameter ratio of branch pipe towards the pressure and velocity behaviour that occurs in the Y-junction pipe.

1.3 Objective of Project

In this project, the following objectives are considered:

1) To design Y-junction pipe using CATIA software.
2) To simulate fluids flow through Y-junction pipe for combining flow by using HyperWorks software.
3) To investigate velocity and pressure behaviour in Y-junction pipe with difference branch pipe diameter ratio for combining flow.

1.4 Scope of Project

In this present study, the investigations focus on the effect diameter ratio of branch pipe towards the pressure and velocity behaviour. Therefore, some limitation or scope must be considered to accomplish this investigation. Overall scopes of this project are stated as follows;

1) The fluid flow is considered to be turbulent by using Spallart Allmaras Equation.
2) The type of fluid is water.
3) The angle of Y-junction pipe is 45º.
4) The standard diameter of the main pipe is considered as 2 inch.
5) The branch diameters of pipe are 1 inch, 1.5 inch and 2 inch.
6) CATIA modelling and HyperWorks simulation are used to conduct and collect the data of this study before it merged into the end of the result.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

This section will show the overview of the fundamentals, ideas, theories, design, simulation and significant literature related to the objective of this project in detailed. This section provides summarizing for previous researches to be a guide in this project. Besides that, this section also discusses the previous researches as references in order to gain the knowledge about the particular field. Therefore, this section is very important to achieve the goal of the study. The researches for this section has been prepared or sorted ascending by year on a journal that had been published. This section will explain the investigation of flow properties (velocity) at Y-junction pipe and pressure loss after passing through Y-junction pipe with different of branch pipe diameters.

2.1 Previous Researches of Fluid Flow in Pipes

Analysis of the fluid flow in two intake pipes with a junction is investigated by Moraes *et al*. (2004). The authors aim to investigate the wave pressure phenomena at the presence of junctions and its influence on the mass flow rate through the intake valves. The experiment is conducted by using the flow bench is an apparatus used for gas flow studies in the intake and exhaust systems of inner combustion engines, underneath regular or unsteady conditions. The result shows, the position of the junction can influence the mass flow. The minimal point appears in different rotations for each configuration. Besides that, the junction is proven to be an efficient device to increase the mass of air admitted into the cylinder for high rotations. Thus, this phenomenon occurs by increasing the mass flow.

Muntean *et al.* (2005) studied the numerical investigation of the 3D swirling in a pipe with a constant diameter. The authors aim to assess the numerical set-up for swirling flows. Also, the authors aim to estimate the accuracy of the numerical data compared with available experimental data. The investigation is conducted by using Reynolds numbers and Euler model. The result shows, increasing Reynolds number, the viscous core region will decrease the constant swirl number. This statement is proved by viscous core radii computed at two Reynolds values. The discrepancy result between numerical and experimental is due to the Euler model. This model includes only numerical dissipation. Therefore, the vortex decay computed with Euler model is smaller than the experimental data.

The investigate pressure drop for the flow of a Newtonian fluid in 90 deg tee junctions with sharp and round corners is conducted by Pinho *et al.* (2006). The experimental were carried out in the closed loop rig schematically. The Reynolds number is conducted in this experiment. The end result indicates that rounding the corners reduced the energy losses relying on the flow rate ratio and rounded corners also increase in turbulence in the branch pipe. Moreover, the increase in dissipation in the branched flow coefficient and in phrases of the total energy loss is adequate to justify that the rounded "Tee" is more efficient for all the investigated Reynolds numbers.

The energy losses and the size and strength of the recirculation region in the branch pipe are conducted by Paal *et al.* (2006). Also, the authors aim to investigate the turbulent flow in 90° T-junctions with sharp and rounded corners. An extensive numerical parametric (LDA-measurements and flow simulations) is conducted. The results show that 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. The influence of rounding the edge has been confirmed both by the simulations and the experiments which result shows the branch pipe loss coefficient significantly decreased.

The flow properties at T-junction of pipe, pressure loss suffered through the flow after passing through T-junction with the various region of the cross-section of the primary pipe and junction pipe is investigated by Vasava (2007). Also, the author aims to determine the accuracy of classical formulas compared to CFD software package. The simulation is conducted by using FLUENT, Comsol Multiphysic and CFD software. The classical formula by A. Vazsonyi and Andrew Gardel (1916-2003) is considered. Observation in the case of combining flow by using the tool of Comsol (3D experiment) and classical, result indicates the distinction were in the range of 3.2 to 5.1 %. Observation in the case of dividing flow, the distinction was in the range of 4.5 to 5.5 %. The flow situations with turbulence cannot deal with Comsol Multiphysics (version 3.2a). The difficult of inner surface for pipes also no longer have the ability to model by this version of Comsol Multiphysics. The fluid that is incompressible and inviscid able used in the classical formulas. Fluent is a best solution or method for heavy and unique simulations and has the capability to model turbulence with a verity of Kappa-Epsilon models.

Dobler *et al.* (2010) studied the comparison the head losses and the corresponding local loss coefficient with a linear or polynomial extrapolation in a Y-bifurcator. The installation of the hydraulic model test based on Reynold law is conducted by using linear polynomial and linear extrapolation The result shows the coefficient of 1.08 is chosen upstream of the Y-bifurcator. Meanwhile downstream of the Y-bifurcator the measured coefficient is 1.01 due to the smoothing effect of the confuser. Besides, the linear polynomial extrapolation causes less deviation as the quadratic polynomial extrapolation. The linear extrapolation is more suitable for the determination of the loss coefficient. The linear extrapolation yields more reasonable results than does the polynomial extrapolation of Klasnic et al. (1992).

The comparison of the flow in pressure conduit with free-surface channels using the equation of Colebrook and White, Transition regime and Rough turbulent regime are conducted by Hanger (2010). The result shows that pressurized flow represents a special case of free-surface flow and that the loss coefficients (obtained for pressurized flows) can be carried over to the corresponding open channel flow situation provided the Froude number remains small. The absolute higher limit is F=1, so-called essential flow, for which the pressure totally vanished. In practice, the transformation concept of the loss coefficient to free-surface channels can be applied up to about F = 0.7.

The flow field of T-junction, Y-junction and arc junction using the shear stress transport (SST) model in ANSYS/CFX software is investigated by Xin and Shoaping (2013). To describe the turbulence in CFD simulation, the k-Ɛ model and SST have been used in this experiment. The simulation from three structures of Y-junction, T-junction and Arc-junction under the same space condition is conducted. Arc-junction is the minimum of the pressure drop compared to Y-junction and T-junction. Through the CFD simulation, the velocity peak decreases when the radius of curvature increasing with specific curvature radii of the arc-junction and after 35 mm the value of velocity peak changes smoothly. Arc-junction is the lowest one amongst three kinds of the junction in term of the head (pressure) loss. It is considered as, the lowest pressure loss and weak velocity shock is the most advantageous structure of arc-junction. The curved loss occurs due to the changing of the pipe direction will elevate speed shock exists in the corner. Besides that, the pressure loss would extend due to the curved loss and confluence loss when if the values of α (confluence angle) are too small or too large.

The pressure loss of fluid with turbulent incompressible flow through a 90° tee junction of the sharp edge is studied analytically and experimentally by Abdulwahhab *et al.* (2013). Also, the authors investigate the relation of flow rate ratio with the pressure and total energy losses. Numerical simulation is conducted for two different values of area ratio between the main pipe and the branch pipe and flow rate ratios by using CFX 5 code ANSYS FLUENT13. The result suggests that increasing the flow rate ratio, the pressure and total energy losses also enlarge due to the fact of the presence of recirculation and the robust streamline curvature. Besides, the pressure loss coefficient given with the aid of the numerical results is higher than those obtained from theoretical and experimental results.

The CFD analysis of fluid flow parameter within a Y-junction pipe is studied by Singh *et al*. (2013). The authors aim to investigate the effect of bend angle, pipe diameter, pipe length and Reynolds number on the resistance coefficient for a Y-shaped pipe. The CFD evaluation for the Y-shape pipe joint is carried out with the aid of the use of SolidWorks Flow Simulation. The end result shows that secondary flow induced by means of a totally developed straight pipe profile at the begin of the bend increases to a maximum at a bend attitude 60°. In the long bend, a mechanism for preventing the secondary go with the flow increasing indefinitely, and for permitting the flow to become totally developed, is supplied by using the formation of total pressure gradients, opposite in signal to these at the start of the bend and the consequent production of vorticity of opposite rotational sense. When the bend is increased beyond 90° so that this terrible vorticity turns into advantageous and the secondary flow tends to increase. The CFD analysis which validates the practical application of Y-pipe at a bend angle of 45° was the resistance coefficient which comes out to be zero.

The characteristic of the pipe junction of 2.4 cross-section area ratio with the five different angles of the adjacent branch is investigated by Stigler and Sperka (2013). The characteristic of the pipe is conducted by using a mathematical model, experiment model and a polynomial function. The result shows, for low values of flow rate ratio for angle 30° have the lowest losses. The middle values of flow rate ratio for angle 45° have the lowest losses. The high values of flow rate ratio for angle 60° have lowest losses. The fascinating end result is that the losses are no longer lowest for a complete range of flow rate ratio for pipe junction with angle 30°.In the case of momentum coefficient it is interesting, that the coefficients for the angles 30° and 150°, 45° and 135°, 60° and 120°, 75° and 105° are rather close to every other and the maximal values are for the angle 90º.

Stigler *et al*. (2014) studied the characteristic of the T-junction with the equal diameters for the variable perspective of the adjacent branch. The authors goal to evaluate the numerical calculation with experiment for the attribute of the T-junctions with five different angles of the adjacent branch. The assessment between characteristics is performed by the numerical calculation and through the experimental. The result shows, this mathematical model can be used for the answer of fluid flow in pipe structures and additionally for assessment of the different shapes of the T-junctions.

The simulation and flow evaluation through specific pipe geometry is conducted by Kumar (2014). The author aims to inspect the minor losses and find the coefficient of loss for their geometry. Also, the author aims to evaluate simulation effects with the experimental result. The simulation for the parameter (velocity, pressure, turbulence) is performed with the aid of using ANSYS software. The end result indicates the loss coefficient of bends is slightly greater with experimental outcomes than the end result obtained from ANSYS. Besides that, the result of loss coefficient for sudden expansion from ANSYS is more accurate than the experimental.

The frictional losses in a standard pipe the use of different pipe substances and a fluid flowing through the pipe is studied by Sambit (2014). Various models of pipes were design and ANSYS Software for the analysis is conducted. The consequences show that the most viscous the fluid more is the frictional coefficient and accordingly greater the frictional loss. The crude oil offers the experienced the most loss, accompanied via diesel, accompanied with the aid of water and then liquid ammonia. The pipe material and the viscosity of the fluid considers as a two factor affected the frictional loss. G.I pipe offers the maximum loss, followed by Brass pipe and minimum loss is Stainless Steel pipe. Increasing the velocity will decrease the frictional loss between the wall and the fluid. The increase in diameter of pipe will minimize the frictional coefficient.

The head losses in junction manholes for free surface go with the flow in circular conduits are investigated by Pfister and Gisonni (2014). The authors purpose to check out the local head losses of combining flows at 45° and 90°junction manholes on circular conduits with various diameters. The experiment is performed through the use of physical models to enable version of the influx aspects and the junction geometry. The results show the absolute values of the coefficients for the 45° junction manhole are systematically large than the 90° junction manhole.

2.2 Y-junction Pipe with an Angle of 45º

Y-junction is any another type of joint and has different structures. This junction used to combine and divide flow such as from one inlet for two outlets and two inlets for one outlet. Y-junction, T-junction and Arc junction are three common devices in the pipeline system. Y-junction has a number of elements that must be considered to keep away from a loss in the pipe system. The confluence angle (α) is a shape of two inlets. Meanwhile, there exists an angle β which could bring curved loss due to the inlet from the Y-direction. Following previous results confirmed that the highest power output is Y-junction device and then the perfect efficiency occurs on 45-degree junction. The smallest mixing volume and shorter bubbles occur on Y-junction pipe. The comparison between the loss coefficient of the sharp and rounded junction was done by Xin and Shoaping (2013). The results show the rounded corners reduced the pressure losses but lead to higher turbulence in the branch pipe compared to the sharp corners.

The Y-junction pipe also called as a "Wye" branch a. The function of Wye branch is allowing splitting a branch line equally in two directions. It is a fitting with three openings and is used to create branch lines. A standard 45 degree angle for wye allows one pipe to be joined to another pipe. Wyes are comparable to tees except the branch line is angled to reduce friction that should hamper the flow. Thus, the connection is commonly at a 45-degree angle rather than a 90-degree angle (Singh *et al*., 2013).

Figure 2.1: Y-junction pipe with an angle of 45º