UNIVERSITI TEKNIKAL MALAYSIA MELAKA

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

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

by

AMIR AHLAMI BIN ZAHARI

B071310597

940210115763

FACULTY OF ENGINEERING TECHNOLOGY

2016

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **Effect of Branch Pipe Diameter Ratio of Fluid Flow in Y-Junction Pipe for Dividing Flow**

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

Saya **AMIR AHLAMI BIN ZAHARI**

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:

B11/4/4 Perumahan PDRM

Padang Hiliran, 21100, K.Terengganu

Terengganu Darul Iman.

Cop Rasmi:

Tarikh: 9 DESEMBER 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.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

CERTIFIED TO ISO 9001:2008 CERT. NO. : AR 3543

FAKULTI TEKNOLOGI KEJURUTERAAN
Tel : +606 234 6623 | Faks : +606 23406526

Rujukan Kami (Our Ref) :
Rujukan Tuan (Your Ref) :

01 JAN 2015

Pustakawan
Perpustakaan UTeM
Universiti Teknikal Malaysia Melaka
Hang Tuah Jaya,
76100 Durian Tunggal,
Melaka.

Tuan/Puan,

PENGKELASAN LAPORAN PSM SEBAGAI SULIT/TERHAD LAPORAN PROJEK SARJANA MUDA TEKNOLOGI KEJURUTERAAN PEMBUATAN (COURSE NAME): AAA BIN BBB

Sukacita dimaklumkan bahawa Laporan PSM yang tersebut di atas bertajuk ***"Development of Integrated Failure Mode And Effect Analysis (I-FMEA) For Automotive Industry"*** mohon dikelaskan sebagai *SULIT / TERHAD untuk tempoh LIMA (5) tahun dari tarikh surat ini.

2. Hal ini adalah kerana IANYA MERUPAKAN PROJEK YANG DITAJA OLEH SYARIKAT LUAR DAN HASIL KAJIANNYA ADALAH SULIT.

Sekian dimaklumkan. Terima kasih.

Yang benar,

____
Tandatangan dan Cop Penyelia

* Potong yang tidak berkenaan

NOTA: BORANG INI HANYA DIISI JIKA DIKLASIFIKASIKAN SEBAGAI SULIT DAN TERHAD. JIKA LAPORAN DIKELASKAN SEBAGAI **TIDAK TERHAD**, MAKA BORANG INI **TIDAK PERLU DISERTAKAN** DALAM LAPORAN PSM.

DECLARATION

I hereby, declared this report entitled “Effect of Branch Pipe Diameter Ratio on Fluid Flow in Y-Junction Pipe for Dividing Flow” is the results of my own research except as cited in references.

Signature	:	………………………………………….
Author’s Name	:	AMIR AHLAMI BIN ZAHARI
Date	:	9 DESEMBER 2016

APPROVAL

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

………………………………

ISKANDAR BIN WAINI

(Project Supervisor)

ABSTRACT

This research is about the study of the effects of branch pipe diameter ratio of fluid flow for Y-junction pipe for dividing flow. One of the simplest devices use for this purpose is the Y-junction connected with the main channel and branch pipe. The understanding of fluid properties likes velocity, pressure loss and viscosity is the main study in this project due to different diameter of branch pipes. In order to achieve the objective of the study, the Y-junction pipe models are designed using Catia software. After that, the pipe models are simulated in AcuSolve of HyperWorks software. In order to observe the behaviour of fluid flow, generate mesh and apply boundary conditions has been done in AcuSolve for all different diameter ratios of branch pipes. One of the main findings is the highest pressure loss occurs at the 26.6 mm which is the smallest diameter of branch pipe among other two different branch pipes. The pressure losses will decreases when the diameter ratio of branch pipe increases for all inlet conditions. Besides that, the velocity shocks occur at the edge of junction pipes for all branch pipe diameters. This is due to change of pipe direction (junction) during the flow of fluid. It also observed that the higher the diameter ratio of branch pipe, the velocity magnitude at branch outlet will decrease. Then, viscosity of liquids are the main factor that affect the the pressure and velocity during fluid flow in pipe. At the end of the task, the conclusion is making and several recommendations are suggested to advance this field further for future study.

ABSTRAK

Kajian ini adalah mengenai kajian kesan nisbah diameter paip bercabang bagi aliran bendalir untuk paip berbentuk Y-persimpangan dalam aliran pemisah. Salah satu peranti mudah untuk tujuan ini ialah paip cabang yang bersambung dengan saluran utama dan cabang paip. Pemahaman sifat bendalir seperti halaju, kehilangan tekanan dan kelikatan adalah kajian utama dalam projek ini kerana penggunaan saiz diameter paip yang berbeza. Dalam usaha untuk mencapai objektif kajian, model paip Y-persimpangan direka menggunakan perisian CATIA. Selepas itu, model paip telah disimulasikan dalam AcuSolve iaitu salah satu perisian dalam HyperWorks. Dalam usaha untuk memerhatikan ciri-ciri aliran bendalir, menjana mesh dan keadaan awal bagi aliran masuk telah dilakukan di AcuSolve bagi kesemua nisbah diameter paip yang berbeza. Salah satu penemuan utama adalah kehilangan tekanan yang paling tinggi berlaku pada diameter 26.6 mm yang merupakan diameter terkecil cabang paip berbanding dua cabang paip yang berbeza. Kehilangan tekanan akan berkurangan apabila nisbah diameter cawangan paip bertambah bagi kesemua parameter keadaan awal aliran masuk. Selain itu, kejutan halaju berlaku di bucu paip bercabang untuk semua diameter berbeza bagi paip cawangan. Ini disebabkan oleh perubahan arah paip (persimpangan) semasa aliran cecair. Ia juga diperhatikan bahawa nisbah diameter yang lebih tinggi daripada paip cawangan, nilai halaju pada cawangan keluar akan berkurangan. Kemudian, kelikatan cecair adalah faktor utama yang memberi kesan kepada tekanan dan halaju semasa aliran bendalir di dalam paip. Pada tugasan akhir, kesimpulan telah dibuat dan beberapa cadangan dicadangkan untuk memajukan bidang ini dengan lebih lanjut untuk kajian masa depan.

DEDICATION

I want to thanks to my dear family, lecturers and friends who have given a lot of help and encouragement to me to complete this project.

ACKNOWLEDGEMENT

In the name of Allah S.W.T, the most gracious and merciful, praise to Allah the lord of universe and may blessing and peace of Allah be upon his messenger Muhammad S.A.W. First, and foremost thank to Allah for giving me wellness and ideas to do this project. Without any of it, I surely cannot complete this project in the time given.

I would like to express my deepest gratitude towards to my project supervisor Mr. Iskandar bin Waini and every lecturer who has help and guide me by giving advices encouragement and patience during the time period to completing this project.

Last but not least, I like to express my very thankful and send our grateful to my entire friends and my family for the moral and financial support. Their views and this are useful definitely. Without all these people encouragement, support and advices this thesis project might not be successfully carried out. To those that I forget to mention, please forgive me. I do appreciate all the things you have done for me.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS, SYMBOLS AND NOMENCLATURE

A	Pipe cross sectional area
l	Length of Pipe
D	Pipe Diameter
R	Radius
λ	Friction Factor
ρ	Pressure
q	Flow Rate Ratio
h_{major}	Major Head Loss
P_{major}	Minor Loss
H_L	Loss Coefficients
g	Gravity Acceleration
v	Velocity
η	Dynamic Viscosity
α	Angle in Y- Junction
θ	Angle Between Main Pipe and Branch
B	Edge of Branch Pipe
Re	Reynold Numbers
LES	Large Eddy Simulation
CFD	Computer Fluid Dynamics
CAD	Computer Aided Design
SST	Shear-Stress Transport Model
k–e	Kappa Epsilon Model
SA	Splart-Allmaras model

CHAPTER 1

INTRODUCTION

1.0 Introduction

In this chapter, it will discuss about the framework of the project that including brief introduction about background of the study, objective, problem statement, scope of the project and purpose of the study.

1.1 Background of Study

In fluid dynamics, pipe network analysis is the analysis of the fluid flow through a hydraulics network, containing several or many interconnected branches. The aim is to determine the properties of fluid flow and pressure drops in the individual sections of the network. Pipe network is usually used for transport and supply of liquid and gas. Pipe networks vary from little pipe to many pipes. There are many fittings in network pipes such as T-piece, Y-junction, bends, valves, elbows and many other components. All of these components result in pressure loss due to change in flow momentum due to friction and pipe components. This means conversion of flow energy in to heat due to friction or energy lost due to turbulence. Municipal water supplies always use the water supply network for supplying water direct to the public. A major part of this network will consist of interconnected and junction pipes. Y-shaped branch pipeline with one fixed and one adjustable branch angle was built up. In pneumatic engineering, problem frequently encountered is the branch pipes in dividing flow. One of the simplest devices used for this purpose was the Y-junction connected with the main channel and two branch pipes. Y-junction also is common device in pipeline system, as used in nuclear engineering, chemical engineering and oil and gas engineering. In this study, the effect of fluid flow with different branch pipe diameter ratio at Y-junction pipe for dividing flows will be studied. The analysis and the simulation will be conducted by using CFD software which is HyperWorks.

1.2 Problem Statement

Pipe network are mainly used for transportation and supply of fluids or gasses from one location to the other location. During the process it may be cause loss in pressure due to change in momentum of the flow caused to friction and pipe component. Therefore, the understanding of the behaviour of fluid like velocity and pressure loss occur in pipe is required. In order to get clear view of the problem the following research questions are considered. Firstly, how to design and simulate the Y-junction model using CAD software and simulation software. Then, what the effects of different diameter ratio on the behaviour of fluid flow and what the type of fluid used for the flow.

1.3 Objectives

a) To design Y-junction pipe by using CATIA software
b) To simulate fluids flow through Y-junction pipe using (AcuSolve) in Hyper Work software
c) To investigate flow properties and pressure loss at Y-junction pipe with different diameter ratio and fluid used in pipe for dividing flow

1.4 Scope

There is some work scopes considered in this project. First, the flows in the pipe are considered to be turbulent. The turbulent flow in in Y-junction is significant research area where K-Epsilon model are used to describe the behaviour of turbulent flow. Second, the angle of Y-junction is fixed which at 45° degree. Next, the fluid use is water with normal properties at room temperature. Besides, there are another two type of fluids are used such as ethylene glycol and ammonia. The different type of fluid used give the different properties of fluid flow and pressure loss in the pipe and this fluid will be compared to water. Lastly, the fluid flow in Y-junction pipe is dividing flow.

1.5 Purpose of Study

In this project, the purpose are to investigate the flow properties at Y-junction pipe and pressure loss suffered by the flow after passing through Y-junction pipe with different branches pipe diameter for dividing flow. Catia software is used to design Y-junction pipe and HyperWorks software will be used to run the simulation of fluid flow in Y-junction pipe. In this study, water is considered as type of fluid use in Y-junction pipe. The data of pressure loss at outlet pipes with given parameters can be observed from simulation. Overall of this study is to determine and identify the suitable diameter ratio in minimizing the pressure loss and its flow properties in Y-junction pipe for dividing flow will be investigated.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

In this chapter, reviews and summarization of previous studies will be discussed. This literature review highlights the related arguments, theories, explanations, findings, and methodologies from previous research done by researchers in order to understanding the concept about the particular field. In point view of engineering, pipe network analysis is very important where many engineering problems dealt by it. Due to rigorous engineering application and implications the analysis is important. Diameters of branch pipe are considered with three different values which are 0.5 inch, 1 inch and 2 inch. There are two type directions in fluid flow which are combining and dividing flows. In this project, only dividing flow is considered. Behaviour of fluid flows discussed in this topic is pressure loss, friction loss and confluence loss. In order to determine the properties of fluid flow and head loss, the Y-junction pipe is designed by using Catia software. CFD simulation used is Hyperwork software to analyse the fluid behaviour.

2.1 Previous Study of Fluid Flow in Pipe

Espinosa *et al.* (2000) studied the turbulent flow in a 90° tee pipe junction for the reverse flow at the branch exit. The aim of this paper is to produce the numerical results of turbulent flow from 3 turbulence models at 90° pipe junction in the branch exist. Method used was conducted by numerical solution which utilizing three models for turbulence, k±e, renormalization group theory (RNG) and Reynolds stretch model, utilizing a refined network to display the smooth chamfer of radius, r = 0:25D as a major aspect of the physical tee branch. The result shows that pressure estimations demonstrate no symmetry in the branch exit regarding the horizontal centreline as opposed to the generally assumed symmetry.

Vasava (2007) studied the Fluid Flow in T-Junction of Pipe. In this paper, the author studied the behaviour of liquid at pipe of T-junction, head loss created by T-junction and change in pressure loss with change in angle of the intersection and with various territory of cross-segment of the fundamental pipe and branch pipe. Method used in this work were Comsol Multiphysics and Fluent software to simulate the stream and analysis flow properties and head loss at T-conjugation of after fluid flow through the junction. There are different loss at the junction of the pipe is observed through the calculation of the software Comsol and Fluent. Then, it was observed that when the diameter of main pipe is reduced, the head loss will be reduced and also found the increasing in angle also affect the increasing of head loss for combining and dividing flow.

Gomez *et al*. (2008) investigated the solute mixing phenomena at different flow rates inside a cross branch, which is normally found in city drinking water dissemination system. Recreations utilizing CFD were utilized by the author to comprehend mixing phenomena at the branch and to look at the general pattern of per cent solute split. Then, experimental result was used to evaluate representative Sct numbers for different flow conditions. The authors found that among the effects of various water distribution network analysis on the branch pipe for solute mixing behaviour are the prediction residual disinfectants, the best location for water quality sensors, predictive model for early warning systems, numerical scheme to identify the source of retrograde, and quantitative risk assessment.

Singh (2009) studied of air flow in a network of pipe used in Aspirated Smoke Detectors. The author investigated systematically the impacts of the jet disturbances on friction factor and to tentatively decide the local loss coefficients of different fittings, for example, twists, joints, and junction. The finding shows that as the quantity of fittings (elbows, T-intersections, Y intersections, and so forth.)

expands, the exactness of the ASPIRE®model show drops relatively and branched pipe arrangements are past the capacity of the ASPIRE® display.

Kaji and Azzopardi (2009) studied the effect of pipe diameter on the structure of gas or liquid flow in vertical pipes. In this paper, the authors analysed the effect of pipe diameter on the parameters that indicate the flow structure by combining the effects of cross-sectional estimates found the average vacancy fraction and the pressure gradient of the experimental work using 19 mm pipe diameter and information bank from 5-70 mm internal diameter pipes. Time arrangements of cross-sectional normal void fraction were acquired by ring-type conductance tests mounted on a 19 mm pipe. The velocity of the periodic structure obtained by cross correlation using the two probes in a row. At the same time, the pressure gradient was obtained. As a result, the authors found that the pressure gradient decrease systematically with increase of pipe diameter.

Hager (2010) studied Losses in Flow. In his paper, the friction and local losses in flow studied by using equation Colebrook and White and by the solution of the equation of Gauckler-Manning-Strickler which methods to find friction loss. For local loss, it was learned for various conductor geometries and channel arrangements and ideal dimensional proportions were indicated. Then, flows in pressure conduits and free-surface channels are compared. To combine the flow at the intersection of the flow channel has a structure similar to the contraction of the channel and the intersection of the flow channel similar to channel development. Losses in both of these elements are governed by flow separation from the walls, a phenomenon that becomes more pronounced with decreasing velocity. Loss coefficient is reduced by entering part of the wall and then amounted to a loss coefficient of the other branch is determined by the ratio of emissions to match.

Guangbin *et al.* (2010) studied the experimental investigation of gas–solid two-phase flow in Y-shaped pipeline. The authors analysed the flow pattern, the mass ratio flowing in the changeable branch and pressure drop on each branch. Experiment of Y-junction pipe was established and the experimental two-phase flow gas-solid was conducted with deliver micro-glass and conveying millet. In the finding of study, the solids distribution was essentially chosen by the angle between the junction and the main pipe. Conversely the solids distribution bends, to the chosen solids materials, the pattern was comparable. But on the other hand, a micro-glass bead with a greater density solid has a more stable trend in the curve property. It was found that the distribution of solids flow and pressure drop both materials have the same trend and has been significantly affected by the angle and velocity of the gas outlets.

Anand and Sandeep (2010) studied the Effect of Angle of Turn on Flow Characteristics of Y- Shaped Diffusing Duct Using CFD. The aim of this paper is to reveal the fluid flow characteristics in the Y-shaped diffusing ducts; two inlets and one outlet. The whole investigation is carried out by the authors in two phases; in the first phase of commercial CFD codes are confirmed for the ability to analyze fluid flow in diffusers S-shaped and in the second phase of the investigation were conducted to study the effect of rotation angle in the Y-shaped channel. Intensive investigation was carried out using various turbulence models available in commercial CFD code, FLUENT to solve the conservation equations that govern fluid flow with the formulation of the finite number. The result showed that the static pressure recovery coefficient decreases with increase in the angle of rotation and pressure loss coefficient increases with the angle of the bend.

Anand *et al*. (2010) studied the Flow and performance characteristics of a Y-shaped diffusing duct using CFD. The aim of this paper is to understand the flow and performance characteristics of the Y-shaped vessel sink has a circular inlet and outlet round with a total area ratio of 2 for uniform flow at the inlet. They carried out experiment using a variety of turbulence models available in commercial CFD code, FLUENT to solve the conservation equations that govern fluid flow with the formulation of the finite number. It was reported that the performance parameters, static pressure recovery coefficient is much lower than the ideal pressure recovery. However, the analysis revealed the facts that the inlet flow diffuser is uniform, secondary flow are reduced due to the effects of curvature and changes in cross-sectional area.

Liu *et al*. (2011) studied the Pressure Drop of Y-Shaped Branch Pipe in Gas-Solid Flow. The authors aim to analyze the flow pattern, the pressure drop in each branch under different solid loading ratio respectively. This paper established Y-shaped branch pipe experimental system, gas-solid two-phase flow experiments were conducted with sorghum and deliver micro-glass beads with a diameter of solid particles of similar and different density. As a result, the trend of the pressure drop in each branch pipe has been disclosed to carry out resistance properties. It can be concluded that the pressure drops of the two substances have the same trend and has been influenced by the angle and velocity of the gas outlets.

Geun-jong *et al*. (2012) studied the Characteristics of turbulent flow distribution in branch piping system. Three-dimensional turbulent flow analyses were performed to evaluate the performance and impact of the turbulence model and the pressure upstream branch pipe geometry. Three different models of turbulence k-ε standard model, realizable k-ε model and the standard k-ω were compared to identify proper one for the branch pipe flow analysis. Then, detailed numerical analysis was performed using ANSYS-FLUENT and experiments were performed with two different flow directions. Lastly, computational results were compared with experimental results. The standard k-ε model is selected since it is known as well-proven model and simple to apply. Next, the authors observed that the large area ratio and lateral resistance tend to increase total loss resulting in high pressure drop and rounded connection of main and branch pipes leads to reduced total loss and pressure drop.

Xin and Shaoping (2013) studied the flow field and pressure loss analysis of junction and its structure optimization for aircraft hydraulic pipe system. The authors investigated the flow field and pressure loss in T-junction, Y-junction and arc junction of pipeline system and also to observe the turbulent flow in junction mixing. In their work, the flow fields of T-junction and Y-junction were analysed using shear stress transport (SST) model in ANSYS/CFX software. To describe turbulence in CFD simulation, there are two model can be used which are k–e model and SST model. Besides that, SST model from k–e model also can be used. In Y-junction, there were three loss to determine the pressure loss occur which are friction, curved and confluence losses. The finding shows that loss coefficient meeting at T-junction is greater than Y-junction. Main factor that affects the velocity shock and pressure loss in junction pipe is the curvature radius *p*. Then, the head loss of Y-junction is lower compared to that T-junction, and minimum value occurs at angle 45° among others three angles of Y-junction. When α is 75°, the value is larger than that of 45° due to the high confluence loss. If the values of α are too small or too large, the pressure loss would increase due to the curved loss and confluence loss. Finally, they stated that the loss of head (pressure) curve intersection is the lowest among the three types of junctions.

Sochi (2013) studied the Fluid Flow at Branching Junctions. The aim of this paper is to observe the flow of fluid at the junctions with given angle, radius and effect of branching in biological flow network. There are two prototypes of the famous and widely used in the investigation of the network and the hemodynamic especially, for modelling and simulation of flow in transportation networks fluids: model Poiseuille flow in rigid tubes, and one-dimensional Navier-Stokes model for flow in tubes distensible. The result is carried out when the increase in the total cross-sectional area is, the blood flow is not compressible will delay direction from large to small vessels in the vascular network. This has a direct impact on some of the phenomena that depend on the speed of flow and deformation rate such as pressure and non-Newtonian rheology. There is one thing that seems to set about branching radius in the main parts of the circulatory system which is the increase diameter of branched junction in the transition from large to small vessels with clear consequences that flow in a direction generally slow down the branch direction.

Hirani *et al*. (2013) carried out CFD Simulation and Analysis of Fluid Flow Parameters within a Y-Shaped Branched Pipe. In this paper, the effect of bend angle, pipe diameter, pipe -length, Reynolds number on the resistance coefficient in a wye branch (Y-shape) were studied by the authors. In the present work, effect of angle of turn/bend for a Y-shape pipe were studied computationally using ANSYS CFX software and inlet pipe considered horizontal and outlet pipes are at angle of 45°, 60°, 90° and 180° were modelled. The results showed that as the bend angle increases from 45 to 90 the resistance coefficient also increases, but at angle of 180