



UNIVERSITI TEKNIKAL MALAYSIA MELAKA

**EFFECTS OF INLET VELOCITY ON FLUID FLOW IN
T-JUNCTION PIPE FOR COMBINING FLOW**

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

by

MUHAMMAD IRSYAD HAZIEM BIN ZULKIPLI

B071310899

940920-06-5975

FACULTY OF ENGINEERING TECHNOLOGY

2016

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **Effects of Inlet Velocity of Fluid Flow in T-Junction Pipe for Combining Flow**

SESI PENGAJIAN: **2015/16 Semester 2**

Saya **MUHAMMAD IRSYAD HAZIEM BIN ZULKIPLI**

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

TERHAD

(Mengandungi maklumat yang berdarjah keselamatan atau

TIDAK TERHAD

kepentingan Malaysia sebagaimana yang termaktub dalam
(Mengandungi maklumat TERHAD yang telah ditentukan oleh

organisasi/badan di mana penvelidikan dijalankan)

Disahkan oleh:

Alamat Tetap: No 25, JLN MEGAH
5, TTAMAN MEGAH, 82000,
PONTIAN, JOHOR

Cop Rasmi:

Tarikh:

DECLARATION

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

Signature :
Author's Name : MUHAMMAD IRSYAD HAZIEM BIN ZULKIPLI
Date : 9 December 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

Objektif tesis sarjana muda yang telah dijalankan ialah untuk mendedahkan kaedah dan pandangan bagaimana teknologi Sistem Perpaipan boleh digunakan untuk mencapai penggunaan optimum mereka dalam matlamat industri. Dalam kajian ini, medan aliran simpang perpaipan telah dianalisis dengan menggunakan AcuField View simulasi dalam perisian pengaturcaraan HyperWorks. Keputusan simulasi dikenal pasti sebagai variasi puncak halaju di simpang dengan frekuensi yang berbeza dan perbezaan fasa arah masuk, sementara itu, eddy dan halaju kejutan wujud di sudut simpang, dan had kerja telah diabaikan. Walaupun eddy hilang dalam T-simpang, perbezaan halaju dan kehilangan tekanan masih terlalu besar. Bagi menangani masalah ini, jenis cecair telah ditukarkan kepada tiga jenis sahaja iaitu air sebagai jmedium utama aliran yang digunakan manakala dua lain jenis cecair ialah Diesel dan Ammonia. Selain itu, jenis bahan yang digunakan untuk membuat paip seperti besi bergalvani, besi tahan karat, dan tembaga juga diabaikan kerana hanya memberi tumpuan kepada analisis cecair yang bermakna geseran tidak dipertimbangkan dalam kajian ini. Melalui pengiraan kehilangan tekanan dari paip, ia telah mendapati bahawa halaju masuk yang lebih tinggi menyebabkan kehilangan tekanan yang lebih rendah seperti yang telah dinyatakan.

ABSTRACT

The objective of bachelor thesis is to reveal the methods and insights how Piping System technology can be use of significant for attaining their optimal use in industry goals. In this study, the flow fields of T-junction were analyzed by using AcuField View simulation benchmarks in HyperWorks programming software. The simulation results identified the variation rule of velocity peak in T-junction with different frequencies and inlet direction phase differences, meanwhile, the eddy and velocity shock existed in the corner of the T-junction, and limit working state was neglected. Although the eddy disappeared in T-junction, the velocity differences and pressure loss were still too big. To address this problem, type of fluids were change into three type only which is Water as the main type of flow used while other two type of fluids are Diesel and Ammonia. Additionally, type of pipe materials such as Galvanized Iron, Stainless Steel, and Brass were also neglected due to only focusing on fluid analysis which mean friction were not considered in this study. Through the computation of the pressure loss from pipe, it has been found that the higher the inlet velocity the lower pressure loss indicated.

DEDICATION

This thesis is dedicated to my beloved father, for earning an honest living for us and supporting and encouraging me to believe in myself and become who I am right now

It is dedicated also to my beloved mother, a strong and gentle soul who taught me to keep trust in Allah S.W.T, believe in hard work which later make me success in the future.

I would like to give my deepest gratitude to my kindest supervisor, Mr. Iskandar Bin Waini for his endless support, mentorship and kindness to me throughout in completion of this project

This is also dedicated to my friends who keep accompany me in the endless day and night to complete this project.

ACKNOWLEDGEMENT

Firstly, I owe a great many thanks to my research advisor, Mr. Iskandar Bin Waini who give continuous support, knowledge, his motivation and guiding and correcting various document of mine with full attention and care. With his guidance, this thesis is complete with success. I astounding with his vast knowledge in area expertise such as HyperWorks, AcuField View and etc. I would state that, Mr. Iskandar is the best advisor and teacher to me.

Secondly, toward my co-supervisor, Mr. Mohd Hidayat Bin Ab Rahman, who give explanations and assisting me to complete this project. I would like to give my gratitude to Mr. Sufian for helping me understand and teaching me the outcome of my thesis and its result.

Many thanks to my fellow friends and classmates for keep push me to complete this projects and fighting until to its fullest together with me. Last but not least, I would like to give my thankfulness to my family which is my parents and my sisters for keep trusting and supporting me in completing this project. Without all people I mentioned above, I would not complete this thesis.

TABLE OF CONTENT

DECLARATION	iii
APPROVAL	iv
ABSTRAK	v
ABSTRACT	vi
DEDICATION	vii
ACKNOWLEDGEMENT	viii
TABLE OF CONTENT	ix
LIST OF TABLE	xii
LIST OF FIGURE	xii
CHAPTER 1	1
INTRODUCTION	1
1.1 Background of Study	1
1.2 Problem Statement	2

1.3 Objective of Project	3
1.4 Project Scope	3
1.5 Layout of Project	4
1.6 Conclusion	5
CHAPTER 2	6
LITERATURE REVIEW	6
2.1 Introduction	6
2.2 Previous Research of Fluid Flow in Pipe	6
2.3 T-junction 90° Pipe	11
2.4 Turbulence Flow	12
2.5 Combining Flow in T-Junction	13
2.6 Newtonian and non-Newtonian Fluids	14
CHAPTER 3	16
METHODOLOGY	16
3.0 Introduction	16
3.1 CATIA Modelling of T-Junction Pipe.	18
3.2 Simulation by Using HyperWorks	27
CHAPTER 4	33
RESULTS AND DISCUSSION	33
4.0. Introduction	33
4.1. Flow Parameters	33
4.2. Inlet Velocity with Streamline Projection	35
4.3 Velocity Flow Fields Analysis in T-Junction Pipes	37
4.4. Pressure Analysis in T-Junction Pipes	42
4.5 Analysis of Flow Rate	45
4.6 Various Viscosity and Density of Fluid	47

CHAPTER 5	51
CONCLUSION	51
5.0 Introduction	51
5.1 Summary of Project	51
5.2 Achievement of Project Objectives	51
5.3 Significant of Research	52
5.4 Problem Occurred during Research	52
5.5 Suggestion for Future work	53
REFERENCES	54

LIST OF TABLE

TABLE 1.1 CHAPTER ORGANIZATION	4
TABLE 3.1 METHOD USE IN SIMULATION	29
TABLE 4.1 VISCOSITY AND DENSITY FOR DIFFERENT FLUIDS	50

LIST OF FIGURE

FIGURE 2.1 EXAMPLE OF T-JUNCTION WITH 90° ANGLE	11
FIGURE 2.2 (A) CASE 1	13
FIGURE 2.3 (B) CASE 2	13
FIGURE 2.4 GRAPH OF NEWTONIAN AND NON-NEWTONIAN FLUIDS	15
FIGURE 3.1 FLOW CHART OF THE PROJECT	17
FIGURE 3.2 CATIA V5 SOFTWARE	19
FIGURE 3.3 FIRST STEP IN DESIGNING PIPE	19
FIGURE 3.4 T-JUNCTION PIPE RENAME	20
FIGURE 3.5 YZ PLANE	20
FIGURE 3.6 DRAW CIRCLE WITH CONSTRAINT	21
FIGURE 3.7 PAD THE CIRCLE SHAPE.	21
FIGURE 3.8 MIDPOINT EXTENDED	22
FIGURE 3.9 FIRST COMPLETE PART	22
FIGURE 3.10 ZX WORKBENCH	23
FIGURE 3.11 CREATE CIRCLE	23

FIGURE 3.12 PAD SECOND CIRCLE	24
FIGURE 3.13 SOLID 3D PART	24
FIGURE 3.14 CREATE SHELL AT THREE FACES	25
FIGURE 3.15 HOLLOW INNER PART	25
FIGURE 3.16 CREATE EDGE FILLET	26
FIGURE 3.17 COMPLETED PIPES DESIGN	26
FIGURE 3.18 MAIN STAGE OF HYPERWORKS SIMULATION	27
FIGURE 3.19 CFD GENERAL PROCESSES	28
FIGURE 4.1 SCHEMATIC DIAGRAM OF T-JUNCTION	34
FIGURE 4.2 STREAMLINE FLOW FOR CASE 1	35
FIGURE 4.3 STREAMLINE FLOW FOR CASE 2	36
FIGURE 4.4 VELOCITY PROFILES AT 25.4 MM BRANCH PIPE WITH VARIOUS INLET	38
FIGURE 4.5 VELOCITY PROFILES AT 25.4 MM BRANCH PIPE WITH VARIOUS INLET VELOCITY FOR CASE 2	40
FIGURE 4.6 PRESSURE FIELDS WITH VARIOUS INLET VELOCITIES FOR CASE 1	42
FIGURE 4.7 PRESSURE FIELDS WITH VARIOUS VELOCITIES FOR CASE 2	43
FIGURE 4.8 VELOCITY CONTOUR FOR FLOW RATE OF DIFFERENT INLET BRANCHES	45
FIGURE 4.9 PRESSURE CONTOUR FOR FLOW RATE OF DIFFERENT INLET BRANCHES	45
FIGURE 4.10 VELOCITY CONTOUR FOR VARIOUS TYPE OF FLUIDS	47
FIGURE 4.11 PRESSURE CONTOUR FOR VARIOUS TYPE OF FLUIDS	48

Chapter 1

INTRODUCTION

1.1 Background of Study

In this chapter pipe networks are usually used to transport and supply the fluid and gases from one location to the other location. All this transportation networks vary from fewer pipes to the other pipes which number might be considering thousands. So many type of components will be involved such as bends, elbows, meters, pumps, turbines, T-junctions and many more components. All of these components will causes loss such as in pressure due to changes in momentum of flow which mainly friction then converted into heat or any energy loss due to turbulence.

In engineering, piping design were studies especially in design to get efficient transport of gases and liquid such as oil, water, chemical liquid, oxygen, and etc. Industrial process piping can be manufacturing from many types of materials such as Cast iron pipes, fiberglass, wood, steel, aluminum, glass, plastic or pvc pipes, copper, concrete, Asbestos Cement (AC) pipes. The variety types of material is due to its demand which is increasing proportional with the rapid growth of industries.

The head loss which also known as pressure loss may different with each components happen in network, material used for pipe and type of fluid transport among network branches. Based on Vasava, (2007) head losses are of two types major and minor. Major Head losses (also called Frictional losses) are due to rough internal surface of pipe and occur over length of pipe. They are mainly due to friction.

In pipe networks, there are several types of junction or branch which is commonly used in pipe network such as Y-junction pipe, T-junction pipe and Arc junction pipe. For this research, studies about T-junction will be focus mainly. T-junction is a very familiar component in pipe networks, mainly used to dispense flow from main pipe to several branches of pipes and converge from many pipes to a single main pipe. These type of junction pipes have two types which is combining flow and dividing flow in pipe network. Combining flow happen when pipe from other branches transportation into two inlet and flow to one inlet vice versa with dividing flow which transportation from other branches into one inlet to two outlets. Depending on directions of inlet and outlet both flow has their own properties which in this study, combining flow will be investigated further.

1.2 Problem Statement

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 causes 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 by using software. The following research questions are considered. Firstly, what is the best direction of various inlet in pipe with minimal pressure loss and best inlet velocity? Then, how to simulate the part model of fluid flow through T-junction by using HyperWorks simulation software according to parameters and condition selected in CATIA V5. After that, what are the effects of inlet velocity of fluid flow in T-junction pipe for combining flow. Lastly, what is the material used for piping.

1.3 Objective of Project

In this project, the following objectives considered;

1. To design T-junction pipes by using CATIA V5.
2. To simulate fluids flowing through T-junction pipe by using Hyperworks.
3. To investigate effect of inlet velocity towards the fluid flow properties and pressure loss at T-junction with equal diameter ratio for combining flow.

1.4 Project Scope

In this research, the following scope has been set to be notified a certain boundaries or limitation including flow of fluid are considered to be turbulent flow which is Spalart-Allmaras model. Fluid have been divided into two part which is laminar and turbulent flow. Type of fluid used for the piping is water with room temperature with neglected friction of walls. The focus of the project fluid mainly for water type rather than gases or oil. The pipeline has the same measurement diameter of 25.4 mm eventually with fixed angle at T-junction of 90°. The eddy shock existed in the corner neglected and flow is restricted to combining flow.

1.5 Layout of Project

Table 1.1 Chapter Organization

Chapter	Content	Division
1	Introduction	PSM 1
	<ul style="list-style-type: none"> 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	
	<ul style="list-style-type: none"> 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	

	<ul style="list-style-type: none"> 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. 	PSM 2
4	Results and discussion	
	<ul style="list-style-type: none"> This chapter is the results and discussion obtained based on the finding in the simulation that were base on design of pipe. The results has been analysed in order to find inlet velocity and pressure loss that occur. 	
5	Conclusion	
	<ul style="list-style-type: none"> We can conclude that the higher velocity inlet will causes the pressure loss to drop based on Bernoulli's Principle. 	

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

2.1 Introduction

This chapter will discuss and explain about the fundamentals, theories and concepts of this project. In this chapter also explains about the perspective, components and method that will be used in this project. The concept of the Effects of Inlet Velocity of Fluid Flow in T-Junction Pipe for combining flow will be discussed and explain in detail.

2.2 Previous Research of Fluid Flow in Pipe

Pinho (2006) studied the edge effects on the flow characteristics in a 90 degree Tee Junction. The objective of the study is 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) investigated the flow properties at T-junction of pipe, pressure loss suffered by the flow after passing through T-junction and to study reliability one of classical engineering formulas used to find head loss for T-junction of pipes. CFD software had been used to determine accuracy of classical formula such as in Finite Volume Method (FVM), Navier-Stokes (NS) equations, momentum change, flux, variation of forces such as external and gravitational and K-epsilon model. While, for simulation of the fluid in pipe FLUENT and

Comsol Multiphysics software were adopted. Result shows that the differences between observations by 2D simulations of software and classical formula were considerably larger than the difference between observations by 3D simulation of software and classical formula. FLUENT software is the best software so far in creating simulation among others.

Li-yang *et al.* (2008) studied the oil/water two-phase flow inside T-junctions by simulating with a 3-D two-fluid model and to find the turbulence inside the model. Kappa-epsilon ($k-\epsilon$) model was used to find turbulence in the pipe. The result shows that separating performance of T-junction was massively depends on the inlet volumetric fraction and flow patterns, thus $k-\epsilon$ turbulence model can be used to simulate the oil/water two-phase flow in T-junction.

Nematollahi and Khonsa (2012) studied the comparison of T-junction flow pattern of water and sodium for different geometries of power plant piping systems. CAST3M code developed by CEA and Kappa-epsilon ($k-\epsilon$) model is used in this study. Both the water and sodium shows very good agreement with slightly changes in flow pattern.

Xin and Shaoping (2012) studied the flow field and pressure loss analysis of junction and its structure optimization of aircraft hydraulic pipe system. The flow fields of T-junction and Y-junction is investigated using shear stress transport (SST) model. ANSYS/CFX has been used for the simulation in this process. 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 results shows that, the variation rule of the velocity peak is obtained and the eddy current does exist in the corner of T-junction.

Abdulwahhab *et al.* (2013) investigated the pressure loss of fluid with turbulent incompressible flow through 90 degree tee junction numerically. Kappa-epsilon model with 2 different values of area ratio between the main and branch pipe were 1.0 and 4.0 of flow rate ratios. Reconsidered simple algorithm were used to find the relationship between the pressure and velocity fields inside the pipe. CFX 5 code ANSYS FLUENT 13 software has been used to simulate three dimensional steady state flow. Result shows the pressure loss coefficient by the

numerical results is higher than theoretical and experimental results. The higher the flow rate ratio the higher difference between them.

Chen *et al.* (2014) studied the thermal mixing characteristic in T-junction piping. Objective of this study is to investigate the effects of flow velocity ratio on thermal mixing phenomenon are the major parameters. Reverse flow which might produce cracks in pipe mostly happen in branch flow with higher velocity than the main flow.

Selvam *et al.* (2014) investigated on high cycle thermal fatigue failures in a nuclear power plant T-junction piping system. It may be caused by near-wall temperature fluctuations due to the thermal mixing of hot and cold fluid streams with temperatures different of 117 Kelvin and 123 Kelvin. Large Eddy Simulation (LES) method and also ANSYS CFX 14.0 software has been used. LES result obtained will be validated with the Fluid-Structure Interaction (FSI). Mixing of fluids occurs in the lower cross sectional region of the pipe with lower Reynolds number of 3670 in the cold fluid coming from the branch pipe along with buoyancy effects in the flow due to higher inflow temperature in the main pipe.

Štigler *et al.* (2014) studied the characteristics of the T-junction with the equal diameters of all branches for the variable angle of the adjacent branch. The author 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 have been 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.

Poole *et al.* (2014) investigated the effect of fluid viscoelasticity on the symmetry-breaking bifurcations in T-channel flows. In aspect-ratio-dependent Reynolds number, flow in three-dimensional T-channel junction which is two-opposing planar channel streams joining and turning through 90 degrees angle can break symmetry which bifurcation has been proposed as a method in enhance mixing in microfluidic channels. Instability according to Maxwell and

Oldroyd-B models occur at lower Reynolds numbers for viscoelastic fluids in comparison to the Newtonian base case while Deborah higher numbers lead direct to an unsteady flow.

Holmberg *et al.* (2014) studied the interaction between the hydrodynamic and the acoustics field in certain duct or pipe configurations such as T-junctions which known to origins of harsh acoustics resonance in pipe networks. Method were used is CFD prediction tools and also Large Eddy Simulation (LES). For Reynolds numbers increases grazing flow speed yields more severe interaction but based on the present data the trend is not a simple proportionally.

Lam *et al.* (2014) studied the aeroacoustics of a combining flow with all same width but different merging angles. Direct aeroacoustics simulation (DAS) has been used as one of the method with conservation and solution element (CE/SE). Acoustic efficiency of the merging flows increases with merging scale as expected. The data obtained also show that acoustic efficiency in the upstream branches by follow the dipole scaling law.

Matos and Oliveira (2014) investigated the influence of constitutive model and fluid elasticity begin with the main characteristics of recirculating flow pattern at the junction and also shear stress fields. Objective of this study is to examine viscosity ratio in order to find Deborah number. Some method have been used such as FENE-CR OR FENE-MCR model which are based on the non-linear finite extensibility dumbbell model but with different derivation of model equations. Other than that, numerical method with geometry and computational mesh were practiced as methodology stated above. The higher the gradient of FENE-MCR model due to re-entrant corners where shear stress gain extreme values between both models.

Mohammad *et al.* (2015) investigated the scour, sedimentation and the flow pattern in T-shaped spurs dike field in a 90 degree channel bend using numerical model. Method used such as Governing equations, numerical model and Kappa-epsilon ($k-\epsilon$) model. In the submerged mode, the maximum scour depth decreased to 22% compared to non-submerged mode.

Kharoua *et al.* (2015) investigated an important phenomenon when populations of particles are split within junctions of gas piping networks. Computational fluid dynamics (CFD),

Kappa-epsilon ($k-\epsilon$) model are used with Discrete Phase Model (PDM). The combination of gravitation and presence of primary vortices, with each side of junction play a major influence solid-phase split.

Chen *et al.* (2015) investigated the micro-droplet breakup in micro-channel T-junction. 3D Volume of Fluid (VOF) method by visual experiment is conducted to validate the numerical simulation. Flow arrangement maps under different viscosity ratio between two phases with new correlation been proposed to predict the transformation between breakup and non-breakup of droplet. Methodology used is Volume of Fluid (VOF) which to reveal the breakup mechanism of micro-droplets in a micro-channel T-Junction. The droplet breakup and non-breakup dynamics of micro-droplets in a symmetric micro-channel T-junction was numerically investigated using VOF model depend mainly on the relative dominance of surface tension represented in the Ca number and relative droplet length.

Lin *et.al.* (2016) investigated on thermal mixing and flow in a T-junction with different flow rate in main pipe or branch pipe which cold water in the branch and hot water in the main pipe. CFD simulation had been used in order to validate useful data in T-junction simulation. Besides that, the particle image velocimetry (PIV) is used in order to perform temperature distribution and velocity field in a T-junction. Based on these steady-state measured temperature distributions, the thermal mixing of a T-junction can be improved by increasing the flowrate in the branch, or decreasing the flowrate in the main pipe

Lin and Ferng (2016) studied on how to resemble the thermal mixing and reverse flow characteristics in a T-junction. These process is conducted in order to find different in steady-state of turbulence model. Method that included in this process is 3-D computational fluid dynamics (CFD) and also Kappa-epsilon ($k-\epsilon$) model which is used to find turbulence. Variety in turbulence model is due to temperature changes and reverse flow from the branch injection back upstream in T-junction.

2.3 T-junction 90° Pipe

T-junction pipe is a small, but very important part of the pipeline system and the angle of this pipeline is 90°. There are several shape of the pipe, and it is primarily used whether in dividing or combining of flow in pipeline system. T-junction is good in order for mixing two different types of liquids, gaseous, or water. Besides that, it can also act as a simple pump which is called ejector, but in this research will be list further detailed especially on how to minimize pressure loss and analyze the best diameter ratio. While the velocity during inlet and outlet of branch pipe will be investigated further in order to find most suitable T-junction. The results showed that 90° junction gave the highest efficiency. According to Xin & Shaoping, (2012) due to particular structure of the aircraft hydraulic pipelines, the traditional configuration of two flows converging is 90°.

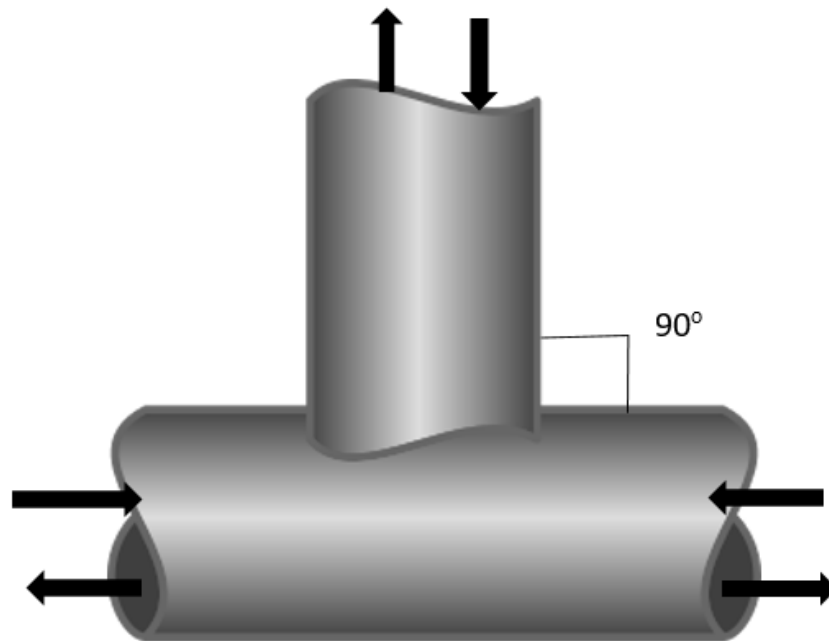


Figure 2.1 Example of T-Junction with 90° angle