UNIVERSITI TEKNIKAL MALAYSIA MELAKA

NUMERICAL SIMULATION OF INCOMPRESSIBLE FLOW OF DIFFERENT TYPES OF FLUID IN A T-JUNCTION

This report submitted in accordance with requirement of the UniversitiTeknikal
Malaysia Melaka (UTeM) for the Bachelor Degree of Mechanical Engineering
Technology
(Automotive Technology) (Hons.)

by

MUHAMMAD MUIZZUDIN BIN ADENAN
B071310364
910914-04-5233

FACULTY OF ENGINEERING TECHNOLOGY
2016

DECLARATION

I hereby, declared this report entitled “Numerical Simulation of Incompressible Flow of Different Types of Fluid in a T-junction” is the results of my own research except as cited in references.

Signature : …………………………..
Author’s Name : Muhammad Muizzudin Bin Adenan
Date : 9/12/2016

APPROVAL

This report is submitted to the Faculty of Engineering Technology of UTeM as a partial fulfilment of the requirements for the degree of Bachelor Degree of Mechanical Engineering Technology (Automotive Technology) with Honours. The member of the supervisory is as follow:

…………………………….

(Mdm. NURUL AMIRA BINTI ZAINAL)

ABSTRACT

The aim of this study is to investigate of fluid flow behaviour at T-junction of pipe. In this study, the result has compare with different flow such as combining and dividing flow and also compare with different fluid substance in the T-junction. The T-junction pipe have design using CATIA V5 software and simulated with SolidWorks. By using SolidWorks software the T-junction pipe can have better view of fluid flow inside the junction and study velocity vector, pressure distribution. By using the software to simulations T-junction, it will be save a lot of time and can be performed without actually doing the experiment. There were no real life experiments made, the results obtained completely rely on accuracy of software and numerical methods used. The study of behaviour of fluid flow was successfully done by using SolidWorks software. The fundamental system properties and flow pattern in T-junction pipe is analysed. Finally, the behavior of two different fluids in T-junction pipe has been compared in this study.

ABSTRAK

Tujuan kajian ini adalah untuk mengkaji tindak balas aliran bendalir pada paip sambungan ‘T’. Kajian ini telah membandingkan dua aliran yang berbeza seperti menggabungkan dan membahagikan pada aliran bendalir dan juga membandingkan dengan bendalir yang berbeza pada paip sambungan ‘T’. Mereka bentuk paip sambungan ‘T’ menggunakan perisian CATIA V5 dan simulasi dengan SolidWorks. Dengan menggunakan perisian SolidWorks pada paip sambungan ‘T’, dapat memberikan pandangan yang lebih baik pada aliran bendalir dalam persimpangan dengan kajian halaju vektor dan taburan tekanan. simulasi seperti ini menjimatkan banyak masa dan boleh dilakukan tanpa melakukan eksperimen fizikal. Keputusan yang diperolehi benar-benar bergantung kepada ketepatan perisian dan kaedah berangka digunakan. Kajian mengenai sifat aliran bendalir telah berjaya dilakukan dengan menggunakan perisian SolidWorks. Asas sistem dan corak aliran dalam paip sambungan 'T' dianalisis. Akhir sekali, tingkah laku dua cecair yang berbeza di dalam paip sambungan ‘T’ telah dibandingkan dalam kajian ini.

DEDICATION

To my father & my mother,

Mr. Adenan Bin Asari & Mdm. Ha’inah Binti Buang

All my friends and relatives

All the lecturers especially Mdm. Nurul Amira Binti Zainal Thousand thank and appreciation for your support, encouragement and understanding

ACKNOWLEDGEMENT

Bismillah ir-Rahman ir-Rahim,

Alhamdulilah thank to Allah the Most Gracious and Most Merciful for His grace and permission to finish my Final Year Project with flying colours as requirement to fulfill my Bachelor’s in Engineering Technology (Automotive) from Universiti Teknikal Malaysia Melaka (UTeM).

I would like to express my thanks to Mdm. Nurul Amira Binti Zainal for her sincere advice and guidance provided throughout my studies. Her patience, knowledge and trust personality have always been an inspiration for me and deeply influence my career and future life.

Also, thank you also to my fellow classmate, lecture and staff of Faculty Technology Engineering through the continued support and opinions for me thus enable me to complete my Final Year Project. Last but not least, I also thanks to my beloved family for support and advise throughout this project. Thank you very much.

TABLE OF CONTENT

LIST OF TABLE

LIST OF FIGURE

CHAPTER 1

INTRODUCTION

1.0 Introduction

Chapter 1 is the framework of this study that includes brief introduction about the T-junction pipes, problem statement, objective, scope and finally the outline of the report is presented.

1.1 Background of the Study

Pipe networks are used for supply fluid mainly liquids and gases. The easiest example of liquid supply pipe networking is the water supply network. These water supply networks provide the water supply to the whole city. In addition to pipes, the network also consists of elbows, T-junctions, bends, expansions, valves, pumps and many other components. All these components cause loss in pressure due to change in momentum of the flow caused due to friction and pipe components. Under these conditions there is a dire necessity to find out the pressure loss that take place during the flow process at micro level (Abdulwahhab et al. 2013).

Pipe networks are mainly used in industries, where liquid or gases are to be supply from one location to other. Disturbances from a condition of straight fully developed flow, such as those created by fittings and accessories, dissipate extra energy given by (White et al. 1999), which is directly proportional to the intensity of the disturbance. The head loss (pressure loss) may different depending on the type of components occurring in the network, material of the pipe and type of fluid supply through the network. In industries the networks are usually large and require very precise pressure at certain points of network.

Thus, this study aims to investigate these phenomena to enhance the performance of the network. Tee junctions are frequently present in industrial systems and water distribution networks where there is the need to separate or merge flows. This study concentrates on common component of pipe network which is called a T-junction or ’Tee’. T-junction is a very common component in pipe networks, mainly used to divide the flow from main pipe to several branching pipes and to combine 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 first series of systematic experimental work in this geometry was carried out in Munich by Vogel in 1932. He investigated the division and merging of flows in no symmetric bifurcations, analyzing the effects of branch pipe diameter and edge radius.

In fluid dynamics, head is the difference in elevation between two points in a column of fluid, and the resulting pressure of the fluid at the lower point. It is possible to express head in either units of height such as meters or in units of pressure such as Pascal. When considering a flow, one says that head is lost if energy is dissipated, usually through turbulence equations such as the Darcy-Weisbach equation. This equation have been used to calculate the head loss due to friction. A head loss consists of two types, major head losses and minor head losses. Major head losses (also called frictional losses) are due to rough internal surface of pipe and occur over length of pipe. Meanwhile, minor losses are losses due to the change in fluid momentum. They are mainly due to pipe components due to bends, valves, sudden changes in pipe diameter and etc. Minor losses are usually negligible compared to friction losses in larger pipe systems.

This study will employ CATIA and SolidWorks software in order to design and to simulate T-junction pipe to investigate the flow behaviour occurred in T-junction pipe. Both dividing and combining flow will be considered.

1.2 Problem Statement

Flows in T-junction pipe are naturally complex and most of the pipes were design in three dimensional. This obviously need high-cost and require huge man power. The amount of available information on 90° T-junction flows is rather limited and consequently less accurate, (Costa et al. 2006). Therefore, in this study numerical treatments is employed in order to predict and investigate the behaviour of the flow in T-junction pipes. This will reduce time, cost, man power and risk rather than doing a real experiment work.

1.3 Objective of the Study

The objectives of this study are:-

1. To simulate numerically the fluid flow interaction in T-junction pipe by using SolidWorks.
2. To analyze the fundamental system properties and flow patterns in T-junction pipe.
3. To compare the behaviour of two different fluids in T-junction pipe.

1.4 Scope of the Study

The scope of this study is an investigation of fluid flow behaviour in a T-junction pipe, considering both dividing and combining flow. The diameter of the pipe is set to 1 inch. Two different types of fluids is considered, which are water and olive oil. The temperature is set to be uniform. It is assumed that no-slip boundary condition at the wall of pipes. The geometry used in this study is actually implemented from previous research (Abdulwahhab et al. 2013) with some improvision. Finally, for the simulation process, SolidWorks software is used in order to achieve the objectives of this study.

1.5 Outlines of the Report

This report is divided into five chapter including this introductory chapter. Chapter 1 briefly discussed some general introduction and highlight the objective of this study also the problem statement and scope of the study.

Chapter 2 makes review about the literature review of this study, meanwhile some mathematical formulation which describes on model descriptions is discussed in Chapter 3.

Followed by Chapter 4, complete pre-processing results of the problem is obtained in details and is discussed briefly.

Chapter 5 finally provides the conclusions of this study as well as some suggestion and recommendations for future study.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

This chapter provides the summary of literature reviews on topics related to the T-junction pipes. It focuses on the related fields and knowledge pertaining to the accomplishment of this study. Reading includes such as reference books, papers, journal articles, websites, conference articles and any documentation concerning the related applications and research works.

2.1 Study on T-junction Pipes

Junctions between pipes can involve the mixing or splitting of fluids. In single phase flow, the flows about the junction are very complex with recirculation possible in the outlet pipes, features first observed by Leonardo da Vinci (see Figure 2.1).

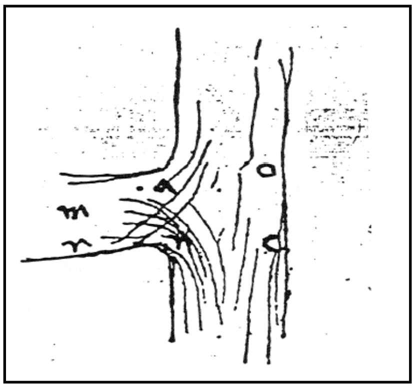


Figure 2.1: Recirculation During Single Phase Flow Split at a T-junction.

In dividing junctions, the actual division of the flow will depend on the pressure drops in the two downstream legs. Apart from the usual losses in the pipes and other downstream equipment, there are specific losses at the junction itself. These are best illustrated in Figure 2.2. As the velocity in pipe 2 is lower than in pipe 1, the pressure rises. The junction pressure drops are defined from the extrapolation of pressure profiles from the undisturbed regions far from the junction back to the junction. The equations describing these pressure losses and the effect of parameters such as flow split, diameter ratio, angle between the pipes and degree of rounding of the corner. Obviously, the junction pressure drops are most important when the pressure drops in the downstream lines are small.

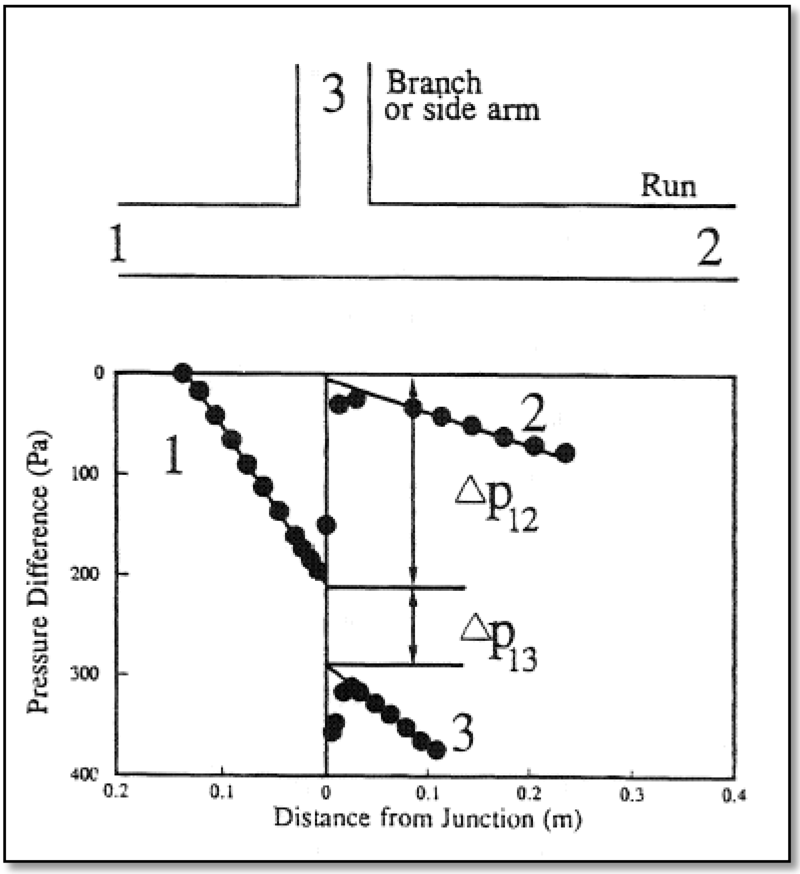


Figure 2.2: Pressure Profiles of Dividing Junction.

For combining flow, there are equivalent pressure drops for which proposed equations. In addition, there is considerable interest in the distance along the outlet pipe required to achieve mixing of the two streams.

When the split at a junctions involves more than one phase, the process has the added complication that the ratio of the phases in the outlet pipes is almost inevitably different to that at inlet. This process is controlled by the momentum fluxes of the phases. For example, for gas/liquid annular flow, the portion of the liquid travelling as a film on the wall has a much lower velocity and hence momentum flux than that travelling as drops. Consequently, it is not surprising that the liquid film and the gas nearest the junction are taken off through the side arm while the drops carry on along the main pipe. A review of the behaviour of gas/liquid split at junctions is given by Azzopardi and Hervieu (1994). They discussed other mechanisms together with available data and models for the prediction of the phase split.

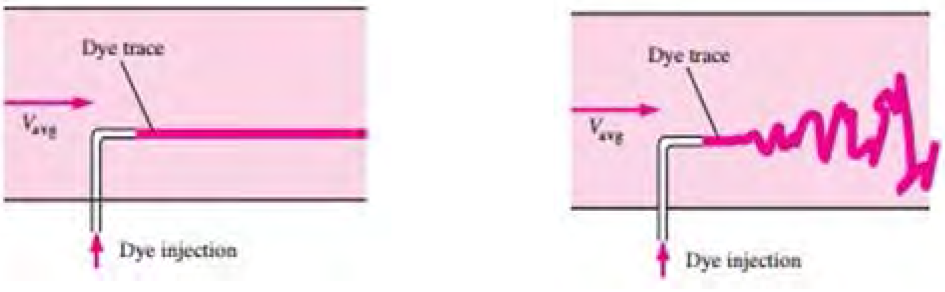
As in single phase flow, combination of two-phase flows at junctions involves extra pressure changes at the junction itself. However, there is much less information about other aspects such as mixing length and etc. (Azzopardi, 1988).

2.2 Study on Laminar and Turbulent Flow

Flow in a pipe reveals that the fluid flow is streamlined at low velocities but turns chaotic as the velocity is increased above a critical value, as shown in Figure 2.3 and Figure 2.4. The flow regime is said to be laminar, characterized by smooth streamlines and highly ordered motion, and for turbulent flow, where it is characterized by velocity fluctuations and highly disordered motion. The transition from laminar to turbulent flow does not occur suddenly rather, it occurs over some region in which the flow fluctuates between laminar and turbulent flows before it becomes fully turbulent. Most flows encountered in practice are turbulent. Laminar flow is encountered when highly viscous fluids such as oils flow in small pipes or narrow passages.

From the flow, the existence of these laminar, transitional, and turbulent flow regimes by injecting some dye streaks into the flow in a glass pipe. (Reynolds, 1883). The dye streak forms a straight and smooth line at low velocities when the flow is laminar (see Figure 2.3), has bursts of fluctuations in the transitional regime, and zigzags rapidly and randomly when the flow becomes fully turbulent. These zigzags and the dispersion of the dye are indicative of the fluctuations in the main flow and the rapid mixing of fluid particles from adjacent layers.

The intense mixing of the fluid in turbulent flow as a result of rapid fluctuations enhances momentum transfer between fluid particles, which increases the friction force on the surface and thus the required pumping power. The friction factor reaches a maximum when the flow becomes fully turbulent.



a) Laminar Flow

b) Turbulent

Figure 2.3: a) The Behaviour Coloured Fluid Injected into the Flow in Laminar and b) Turbulent Flow in a Pipes.

2.2.1 Types of Turbulent Model

It is understood that turbulent flow are fundamentally assigned by the variances of the velocity fields. Different transported amounts such as force, momentum and so forth likewise vacillate for this variance of speed field and these changes can be of high recurrence and little scale are extremely difficult and computationally essential to analyse directly in industrial engineering calculations. The turbulence model needs to be selected based on some considerations, in example the physics of the flow, the insight into the capabilities and limitations of turbulence models, the attempt for the specific problem by other researchers, the accuracy needed, the available computational resources, and time. (Dutta et al. 2016).

2.2.1.1 K-epsilon Model

The baseline two transport equation model solving for kinetic energy, *k* and turbulent dissipation ε. Turbulent dissipation is the rate at which velocity fluctuations dissipate. This is the default k- ε model. Coefficient are empirically derived which is valid for fully turbulent flow only. In the standard k- ε model, the eddy viscosity is determined from a single turbulent length scale, so the calculated turbulent diffusion is that which occurs only at the specified scale, where as in reality all scales of motion will contribute to the turbulent diffusion. The k- ε model uses the gradient diffusion hypothesis to relate the Reynolds stresses to the mean velocity gradient and the turbulent viscosity. The most disturbing weakness is lack of sensitivity to adverse pressure gradient which is another shortcoming is numerical stiffness when equation are integrated through the viscous sublayer which are treated with damping functions that have stability issues. (Menter, 1994).

2.2.1.2 K-omega Model

A two-transport-equation model solving for kinetic energy k and turbulent frequency ω. This is the default k–ω model. This model allows for a more accurate near wall treatment with an automatic switch from a wall function to a low-Reynolds number formulation based on grid spacing. The k-ε model uses the gradient diffusion hypothesis to relate the Reynolds stresses to the mean velocity gradients and the turbulent viscosity. Solves one equation for turbulent kinetic energy, *k* and a second equation for the specific turbulent dissipation rate (or turbulent frequency) ω. This model performs significantly better under adverse pressure gradient conditions. The model does not employ damping functions and has straight forward Dirichlet boundary conditions, which leads to significant advantages in numerical stability. This model under predicts the amount of separation for severe adverse pressure gradient flows. (Menter, 1994).

2.2.1.3 Spalart-Allmaras Model

Spalart-Allmaras (SA) is a turbulence model for modeling different type of turbulent flows, specifically aerodynamic flows. Different terms of the governing equation of this model were modified after 1992, when it was published by Spalart and Allmaras. (Javaherchi, 2010). Spalart-Allmaras model is a one-equation model that solves a modelled transport equation for the kinematic eddy turbulent viscosity. Spalart-Allmaras has proven to be numerically well behaved in cost cases.