UNIVERSITI TEKNIKAL MALAYSIA MELAKA

FLUID-SOLID INTERACTION MODELLING AND ANALYSIS OF 45 AND 90 DEGREE PIPE ELBOW BY USING ANSYS

WAHADATUL NUR FATI'HAH BINTI SA'AUDI
B091810358

BACHELOR OF MECHANICAL ENGINEERING TECHNOLOGY (MAINTENANCE TECHNOLOGY) WITH HONOURS

2022

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Faculty of Mechanical and Manufacturing Engineering Technology

FLUID-SOLID INTERACTION MODELLING AND ANALYSIS OF 45 AND 90 DEGREE PIPE ELBOW BY USING ANSYS

WAHADATUL NUR FATI'HAH BINTI SA'AUDI

Bachelor of Mechanical Engineering Technology (Maintenance Technology) with Honours

2022

FLUID-SOLID INTERACTION MODELLING AND ANALYSIS OF 45 AND 90 DEGREE PIPE ELBOW BY USING ANSYS

WAHADATUL NUR FATI'HAH BINTI SA'AUDI

A thesis submitted
in fulfillment of the requirements for the degree of
Bachelor of Mechanical Engineering Technology (Maintenance Technology) with Honours

Faculty of Mechanical and Manufacturing Engineering Technology

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2022

DECLARATION

I declare that this Choose an item. entitled “Fluid-Solid Interaction Modelling And Analysis Of 45 And 90 Degree Pipe Elbow By Using Ansys” is the result of my own research except as cited in the references. The Choose an item. has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : 

Name : Wahadatul Nur Fati’hah Binti Sa’audi

Date : 14/01/2022

APPROVAL

I hereby declare that I have checked this thesis and in my opinion, this thesis is adequate in terms of scope and quality for the award of the Bachelor of Mechanical Engineering Technology (Maintenance Technology) with Honours.

Signature : .....

Dr Setyamartana Parman
Senior Lecturer
Faculty of Mechanical & Manufacturing Engineering Technology
Universiti Teknikal Malaysia Melaka

Supervisor Name : Dr. Setyamartana Parman

Date : 14/01/2022

DEDICATION

This report is dedicated to my cherished parents and family members, who have educated and supported me throughout the entire process of completing this project. I'd also like to express my gratitude to my supervisor, Dr. Abdul Munir Hidayat Syah Lubis and Dr. Setyamartana Parman, as well as my coursemates, who have all encouraged, led, and motivated me to complete this research.

ABSTRACT

Pipes are key components in maintaining a progressive pipe system in industries since they serve as a medium for fluid or gas passage from one place to another. When a flowing fluid travels through a bend or turn in a pipe, it causes a flow inside the pipe known as "secondary flow," which results in a stress distribution inside the bend. Because the pipe bend portion is fixed at both ends, this stress causes the pipe to deform; if the deformation forces are significant enough, the pipe can break. Even if the imposed load does not immediately create a rupture, vibrations inside the flow may occur. More serious risks, including as explosions, will arise if the situation is not appropriately controlled and resolved. The major goal of this research is to analyse the influence of pressure and velocity on pipe deformation at 90° and 45° bending, as well as the effect of a flow separator on pipe deformation at 90° and 45° bending. One of the key reasons we selected this bending pipe specification is the relationship between flow and structure. An ANSYS software simulation is a tool for determining the permissible flow through a pipe. This tool will aid in the resolution of piping system difficulties caused by flow vibration and fluid solid interation between the flow and the pipe structure. Furthermore, performing this simulation first will save money and time. As a result, it will depict the fluid movement and heat transfer that is possible in pipe modelling. We will learn about the influence of velocity and pressure on the pipes in this project.

ABSTRAK

Paip adalah komponen utama dalam mengekalkan sistem paip progresif dalam industri kerana ia berfungsi sebagai medium untuk laluan cecair atau gas dari satu tempat ke tempat lain. Apabila bendalir yang mengalir bergerak melalui selekoh atau membelok dalam paip, ia menyebabkan aliran di dalam paip yang dikenali sebagai "aliran sekunder", yang mengakibatkan pengagihan tegasan di dalam selekoh. Oleh kerana bahagian lentur paip ditetapkan pada kedua-dua hujung, tegasan ini menyebabkan paip berubah bentuk; jika daya ubah bentuk cukup ketara, paip boleh pecah. Walaupun beban yang dikenakan tidak segera membuat pecah, getaran di dalam aliran mungkin berlaku. Risiko yang lebih serius, termasuk sebagai letupan, akan timbul jika keadaan tidak dikawal dan diselesaikan dengan sewajarnya. Matlamat utama penyelidikan ini adalah untuk menganalisis pengaruh tekanan dan halaju pada ubah bentuk paip pada lenturan 90° dan 45°, serta kesan pemisah aliran pada ubah bentuk paip pada lenturan 90° dan 45°. Salah satu sebab utama kami memilih spesifikasi paip lentur ini ialah hubungan antara aliran dan struktur. Simulasi perisian ANSYS ialah alat untuk menentukan aliran yang dibenarkan melalui paip. Alat ini akan membantu dalam penyelesaian kesukaran sistem paip yang disebabkan oleh getaran aliran dan interasi pepejal cecair antara aliran dan struktur paip. Tambahan pula, melakukan simulasi ini terlebih dahulu akan menjimatkan wang dan masa. Akibatnya, ia akan menggambarkan pergerakan bendalir dan pemindahan haba yang mungkin dalam pemodelan paip. Kita akan belajar tentang pengaruh halaju dan tekanan pada paip dalam projek ini.

ACKNOWLEDGEMENTS

In the Name of Allah, the Most Gracious, the Most Merciful

First and foremost, I would like to express my gratitude and appreciation to Universiti Teknikal Malaysia Melaka (UTeM) for providing me with this opportunity to continue my bachelor's degree in Mechanical Engineering Technology (Maintenance Technology) in the Faculty of Mechanical and Manufacturing Engineering Technology. My supervisor, Dr. Abdul Munir Hidayat Syah Lubis and Dr. Setyamartana Parman, has my heartfelt gratitude for his guidance, advice, and inspiration during my final year. His unwavering patience in guiding and imparting priceless insights will be remembered for the rest of my life. My sincere appreciation goes to my precious parents, Mr. Sa'audi bin Mahidin and Mdm. Zuraidah binti Abu Bakar, and extended family members, Wahadatul Artai'ah and Wahadatul Zur'ain, who have always encouraged and supported me in my efforts to reach the goal of my project. A special thanks to my course mates and friends for their unwavering support throughout the years. Finally, I would like to express my gratitude to all of the individuals who have assisted me, supported me, and inspired me to pursue this course of study.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS AND ABBREVIATIONS

D,d	-	Diameter
ρ	-	Fluid density
η	-	Viscosity
ν	-	velocity
N_R	-	Reynold's number
P	-	Static Pressure head
h	-	height of the container or the pipe here the fluid is flowing
g	-	gravitational acceleration
F,f	-	net force acting on the body
a	-	acceleration of the body
m	-	mass
$m\vec{V}$	-	linear momentum of the system
μ	-	dynamic viscosity
$\nabla \cdot \sigma$	-	shear stress
u	-	velocity

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Background

Pipes is a hollow cylinder that has made from metal, concrete, plastic and others. There are many variety types of pipes with their own specification, sizes, material, length and more. The main function of the pipes is to convey any fluid or chemical substances through it either short or long distances from one point to another for the operation on industries, powerplant and any sector related. Piping system is a combination network that build consists pipes, fitting, valve and other for specific work such as transmitted liquid such as gas or liquid or transfer one equipment to others (zeeshan,2012). The basic example for piping system is the piping system in home that transfer water from the tank to the entire house such as sink, bathroom and washing machine, while steam piping in powerplant is an example for piping system in industry.

Fluid flow on the pipes has their own flow rate, velocity and pressure. Flow rate is defined as how the speed for the fluid or substances through the pipes from particular point. Meanwhile, velocity define as rate and direction of motion (andrew,2019). pressure in fluid is force over unit area in a closed container or in fluid (ron,2017). Gravity, acceleration, forces by the surrounding of the closed container are the factor to the pressure in fluid as well. These parameters affect the pipes structure and leads to fluid-structure interaction (FSI) problems. Common issues, problems, failures that occurs in piping system are related to the interaction between fluid flow and the structure of pipes due to internal and external fluid flows (almasi, 2020). In addition, external and internal fluid flow exposed the pipe

component or piping system to vibration in industrial plants and piping system (Udoetok, 2018)

Fluid Solid Interaction (FSI) defined as interaction between fluid and solid structure that cause deformation towards the structure. It is related to, force that occurs when the fluid flow through the structure that cause the structural deformation. The deformation occurs on the object is depending how the pressure, velocity, flow rate and material properties of the structure interaction. This fluid structure interaction phenomenon arises in many form of natural system and human-made object. In nature system, typical example phenomenon applies between a tree and wind as well as groundwater interaction with the soil. in human-made object, example of fluid structure interaction is between offshore platform with the ocean current and dam with water pressure.

Furthermore, phenomenon that typically occurs in fluid solid interaction coupling is flow induced vibration. The hydrodynamic load caused by the pulsation of the fluid flow cause of the structural vibration. any types of flow instabilities that occurs in piping system, valves and other devices in the piping system can lead to strong vibration. Pumps, valves, strainers, fans, pressure fluctuations, weak foundation and support structures, external cross-flows and internal fluid flow combination is the sources of deformation and vibration of a piping system. the vibration of piping can be done by listening humming sound or by placing a hand on a pipe for the vibration feel. The sign of failure of pipes could be seen from unusual high vibration of piping system. If the vibration level in piping system left unchecked, it will affect the life span of piping system and at once, affect the entire the operation and the life span of the system.

The Computational Fluid Dynamic (CFD) method is used to analyze fluid flow and fluid structure interaction, as well as to aid in the solution of fluid dynamic problems. Today, CFD visualized gas or liquid flow by using applied mathematics, physics and computational

software. It is process of mathematically modeling a physical phenomenon by using computational process, the problem can solving numerically when it involves the fluid flow. Computational Fluid Dynamic (CFD) simulation is believed to be one of the best tools to deal and overcome the fluid dynamic effect. ANSYS simulation software is one of tools in analyze the fluid flow and Fluid Solid Interaction (FSI). The impact of using this simulation tool are aiding in fluid dynamic problem, analyzed fluid flow and fluid solid interaction which provided and obtained the data accurately compared to other alternatives and capabilities for solving complex fluid flow and heat transfer issues.

1.2 Problem Statement

In industries, pipes are vital components in ensuring the progressive piping system since its functioning as medium for fluid/gas flow from one point to another. The pipe failure or damaged that take place will disturb system and operation system in related field and reduced the life span of the pipes. The pressure of fluid flow velocity affected the pipes structure when fluids travelling via the pipeline which leads to deformation and vibration on the pipe structure.

Pipe deformation can be happened because of stress and vibration in the flow through the pipes. when a flowing fluid passes through a bend or turn in a pipe, it creates a flow known as 'secondary flow' inside the pipe, resulting in a stress distribution inside the pipe body at the bend. This stress causes the pipe to deform, because the pipe bend section is fixed at both ends, if the deformation forces are large enough, the pipe can rupture. Even if the applied stress does not directly cause a rupture, it may cause vibrations within the flow. The most common cause of pipe vibration is a sudden change in fluid flow direction, such as in a pipe bend, resulting in a sudden direction change in fluid flow velocity, which leads to an inverse pressure gradient caused by the turn, increasing fluid particle velocity towards the outside of the curvature (Jafari, 2017). In this phenomenon, the fluid particle velocity is reduced

towards the inside of the turn, causing a stress and pressure gradient to form inside the pipeline. because fluids have a low vibration damping capability. in general, a vibration cause in one section can travel long distances downstream, resulting in a possible vibration resonance wherever it matches the natural frequency of the pipeline. A resonance within a pipe section causes great vibrations, which build tension and strain within the section, and if the deformations are large enough, the section may fail. Without proper management and solution in overcoming the situation, more serious risks like explosion will occur.

1.3 Research Objective

The objectives of this project are :

1. To observe the effect of pressure to the bending pipe.
2. To observe the effect of velocity to the bending pipe.

1.4 Scope of study

1. The analysis for this project is only focused on fluid flow in 90° and 45° bending component in the piping system. other components are not considered in this field.
2. To perform the task of modelling part, ANSYS Simulation Software is the applied as analysis tool
3. The project is employ incompressible fluid flow under turbulent condition.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter covers the fundamentals of fluid mechanics, flow induce vibration and as well as other related studies on modelling and analysis. .

2.2 Closed Channel Flow

Based on (Burgi, 1984), closed-conduit flow is defined as flow in completely filled conduits with no free water surface, or flow (relative motion) deep enough in a body of water that surface effects are minimal. Flow in pumping plant discharge lines and municipal water supply systems are examples of the first portion of the definition, while deeply submerged submarines are examples of the second.

According to (CP Kothandaraman, R Rudramoorthy , 1999), in many industrial processes, fluids are conveyed (transported) through closed conduits. It is discovered that the pipe system must be designed to transport a specified amount of fluid between specified locations with the least amount of pressure loss. The initial cost of the piping system must also be considered. The flow could be laminar, with fluid flowing in an orderly fashion and layers not mixing macroscopically. Pure diffusion is used to induce momentum transfer and shear at the molecular level. This type of flow occurs with highly viscous fluids. The flow of blood through the arteries and veins is typically laminar. In fluids flowing through pipes, the laminar condition prevails up to a certain velocity. Under certain conditions, the flow becomes turbulent due to macroscopic mixing of fluid layers in the flow. The velocity varies around a mean value at any given location. In general, air and water flow in pipes are turbulent. The flow is controlled by the following factors:

(i) pressure gradient,

(ii) pipe diameter or hydraulic mean diameter,

(iii) fluid properties such as viscosity and density, and

(iv) pipe roughness.

The design is also influenced by the velocity distribution in the flow and the flow state, which is either laminar or turbulent. The main quantity to be calculated is the pressure drop for a given flow rate through a duct for a specified fluid. On occasion, the inverse, namely the quantity flow for a given pressure drop, must also be calculated. The fundamental laws of incompressible flow are :

(i) the law of conservation of mass and;

(ii) Newton's laws of motion.

In addition to these laws, the modified Bernoulli equation applies in these flows.

2.2.1 Features Of Laminar And Turbulent Flow

According to (*Flow in Closed Conduits.*, 1989), There is energy loss in a closed conduit system due to friction that occurs within the flowing fluid, which is affected by the type of fluid, velocity of flow, and the nature of the surface of the stationary pipe wall.

Friction losses can be substantial. Pipe fittings, bends, changes in flow area, valves, and other factors contribute to friction loss. Based on (Maatooq, 1883), The flow in a closed conduit (flow in a pipe) differs from the flow in an open channel in that the flow in a pipe is at a pressure (it has no a free surface). The flow in a pipe can be demonstrated using examples such as:

1. Laminar flow
2. Transitional flow
3. Turbulent flow

Laminar flow can be defined as smooth and steady flow with little or no evidence of mixing of the various parts of the stream. while turbulence flow has seemed chaotic and rough with a lot of fluid intermixing.

To identify between the characteristics, the well-known “Reynold’s Number” can be used, based on experiments conducted by “Osborn Reynold” in the 19th century. Osborne Reynolds demonstrated the existence of two distinct types of flow in 1883 by injecting a very thin stream of color fluid with the same density as water into a large transparent tube through which water is flowing.

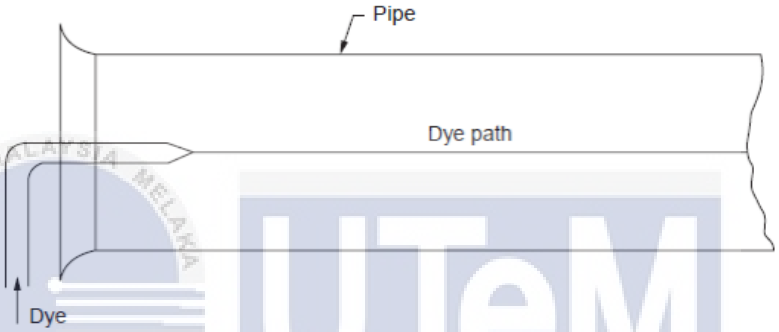

Figure 2.1 Reynold Experiment (CP Kothandaraman, R Rudramoorthy , 1999)

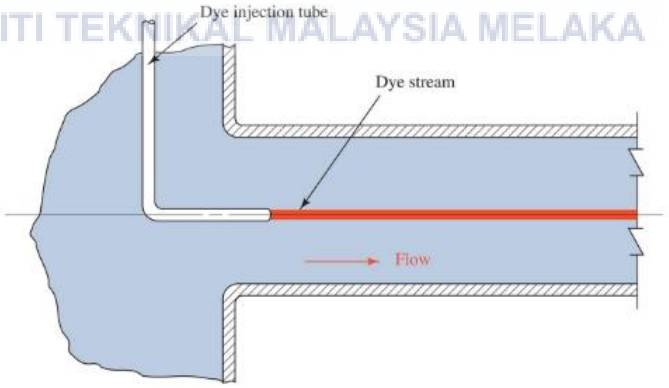

Figure 2.2 Dye stream in laminar flow (Flow in Closed Conduits., 1989)

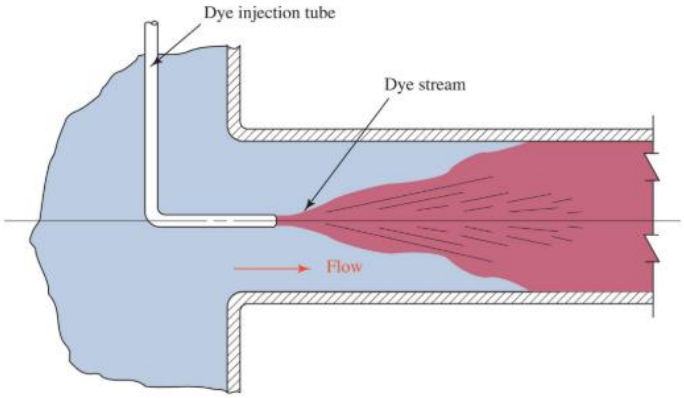

Figure 2.3 Dye stream mixing with turbulent flow (Flow in Closed Conduits., 1989)

The fluid's behaviour is determined by whether the flow is laminar or turbulent. It can be demonstrated experimentally and analytically that the flow character of a round pipe is determined by four variables :

1. Fluid density, ρ
2. Viscosity, η
3. Pipe diameter, D
4. Average velocity of fluid, v

$$N_R = \frac{vD\rho}{\eta}$$

Figure 2.4 The formula for Reynolds number (Flow in Closed Conduits., 1989)

Flows with high Reynolds numbers, typically due to high velocity and/or low viscosity, are turbulent. Fluids with high viscosity and/or moving at low speeds have low Reynolds numbers and tend to be laminar. For noncircular cross sections, open