NUMERICAL STUDY OF FLUID FLOW AND HEAT TRANSFER IN MIXING ELBOW

MOHD IDHAM KHALID BIN ABU HASAN

This Report Is Submitted In Partial Fulfillment Of Requirements For The Bachelor Degree of Mechanical Engineering (Thermal-Fluid)

Faculty of Mechanical Engineering

University Technical Malaysia Melaka

JUNE 2013

SUPERVISOR DECLARATION

"I hereby declare that I have read this thesis and in my opinion this report is sufficient in terms of scope and quantity for the award of the degree of Bachelor of Mechanical Engineering (Thermal-Fluids)"

Signature: ……………………………………….

Supervisor: NUR HAZWANI BT MOKHTAR

Date: 4/7/2013

DECLARATION

"I hereby declare that the work in this report is my own except the summaries and quotations which have been duty acknowledge."

Signature: ……………………………..…………………..

Author: MOHD IDHAM KHALID B. ABU HASAN

Date: 04th July 2013

Dedicated to my beloved Mother,

Father and Brothers...

ACKNOWLEDGEMENT

All praise to Allah Most Gracious, Most Merciful, Who, Alone, brings forgiveness and light and new life to those who call upon Him.

First of all, thanks to Allah s.w.t for all His guidance and blessing through all the hardship encountered whilst completing this thesis. In preparing this thesis, I was in contact with many people, researchers, academicians and practitioners where they have contributed towards my understanding and thoughts. In particular, I wish to acknowledge with much appreciation the crucial role of my supervisor, Miss Nur Hazwani Mokhtar for guidance, encouragement, critics and friendship.

I would like to express my deepest appreciation to my family especially my beloved mother, Mimunah Muhamad and all of my siblings for all of their loves and supports. I am also indebted to FKM lecturers for their guidance to complete this thesis, especially to my final year project panels Mr. Mohd Hafidzal Bin Mohd Hanafi and Mr. MD Isa Bin Ali. Without ther continued support and interest, this thesis would not have been presented here.

Lastly I would like to recognized my fellow undergraduate colleagues for their support and assistant. All of their helped will be remembered.

ABSTRACT

Study of heat transfer and fluid flow in piping system often involves of temperature, heat flow, velocity and dynamic viscosity from the entrance the fluid enter until the fluid flow out of the pipe outlet. This study focus on the behaviour of heat transfer and fluid flow in mixing elbow. Mixing elbow is a region where two types of fluid flow with different temperature and high Reynolds number is intensively mixed together and is among typical geometries exactly where temperature fluctuation happens. Temperature fluctuation in liquid causes high cycle thermal fatigue within structure materials based on temperature distributions as well as time variations. The main objective of this study is to simulate and analyze the flow and temperature field in the mixing elbow by simulate with several models of different inlet diameters. The simulation process is done by using Computational Fluid Dynamic (CFD) software. From the simulation, it is proven that the mixed flow appeared as turbulent and caused the temperature changes which eventually generate a cyclic stress that contributes to thermal fatigue failure on the welded joint and Flow accelerated corrosion (FAC) also founds to be the contributor for thermal fatigue failure at the elbow. Besides that, it is found that from the simulation test of three models with different diameters, it would longer the service life of mixing elbow and improved it safety by increased the diameter of the inlets.

ABSTRAK

Kajian tentang pemindahan haba dan aliran bendalir dalam sistem perpaipan sering melibatkan suhu, aliran haba halaju, dan kelikatan dinamik dari awal kemasukan bendalir sehingga aliran bendalir keluar dari salur keluar paip. Kajian ini memberi tumpuan kepada tingkah laku pemindahan haba dan aliran bendalir dalam siku pencampuran. Siku pencampurkan merupakan tempat di mana dua jenis aliran cecair dengan suhu yang berbeza dan dengan nombor Reynolds yang tiggi dicampurkan bersama secara intensif dan adalah antara geometri yang kebiasaannya menjadi tempat di mana turun naik suhu berlaku. Turun naik suhu dalam cecair menyebabkan kitaran keletihan haba yang tinggi dalam struktur bahan berdasarkan taburan suhu serta variasi masa. Objektif utama kajian ini adalah untuk mensimulasikan dan menganalisis aliran dan medan suhu dalam siku pencampuran dengan menjalankan simulasi terhadap beberapa model yang mempunyai saluran diameter masuk yang berbeza. Proses simulasi dilakukan dengan menggunakan perisian Dinamik Bendalir Komputeran (CFD). Daripada simulasi yang telah dijalankan, ianya terbukti bahawa aliran bercampur menghasillkan arus bergelora dan menyebabkan perubahan suhu yang akhirnya menjana tekanan kitaran yang menyumbang kepada kegagalan lesu haba pada sendi kimpalan dan Aliran Pecutan Hakisan (FAC) juga terbukti menjadi penyumbang untuk kegagalan lesu haba di sesiku. Selain daripada itu, didapati bahawa daripada ujian simulasi tiga model dengan diameter yang berbeza, hayat perkhidmatan siku campuran akan jadi lebih lama dan keselematan siku campuran boleh ditingkatkan dengan membesarkan saiz diameter paip kemasukan.

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURE

LIST OF SYMBOL

ρ	=	Density (kg/m^3)
U	=	Velocity (m/s)
D	=	Diameter of pipe (m)
μ	=	Dynamic viscosity (kg/ms)

LIST OF APPENDIX

CHAPTER 1

INTRODUCTION

1.0 INTRODUCTION

In mixing elbow, heat transfer through convective fluid flow occurs due to differences in density of the fluid at the surface. In addition, there is also a flow of changes that cause the phenomenon of laminar and turbulent flow depends on the flow of phase change. The mixing between two types of fluid with different temperature will caused a temperature fluctuation which will generate a cyclic stress and contributes to thermal fatigue failure (Aulery et al,2012). Besides that, the changes in flow direction in the mixing elbow will produce a secondary or flow separation due to biased velocity distribution . The parameter taken into consideration are the pipe diameter or length of pipe required, velocity of fluid flow and fluid flow rate.

In general, heat transfer process can be seen all around us. Basically the heat transfer mechanism can flowing through three processes of conduction, radiation and convection (Holman, J.P., 1986). In recent times, a lot of research in the field of heat transfer are focused on the process of convection. Heat transfer is a condition in which the energy transfer occurs between systems that communicate with each other due to differences in temperature. It is also a phenomenon that occurs at the boundary of the system is a redistribution of internal energy in the system.

Convective heat transfer process is mainly a surface phenomenon. Surface may be a surface in, such as the wall of the pipe or channel, or the outer surface of a particular body (Holman, J.P., 1986). In the process of convection, mass and momentum transfer and heat transfer occur simultaneously.

The design and analysis of engineering systems involving fluid flow is either by experiment or calculation. Previous method involves building models and tested in wind tunnels or other equipment. Latest methods involving differential equations, or computing solutions.

CFD code is used to calculate the properties related to heat transfer and fluid flow. Practical problems in engineering mostly involving fluid flow and heat transfer. CFD is also used to shorten the design cycle by controlling the parameters relevant. In addition, it can reduce the amount of equipment to make the experiment thus reduced the cost of making the experiment.

CFD focuses on fluid flow equations using a computer software. Modern engineers have now been applied to both the experimental and CFD analysis to obtain the details of the velocity profile of the fluid flow, pressure and temperature. Meanwhile, experimental data are often used for validation of the CFD solution with computing systems adapt and experiment to find the overall quantity.

Therefore, the process of simulating and analyzing the fluid flow and heat transfer for this study can be done by using the CFD software. The simulation can be carried by simulating with different inlet diameters for the mixing elbow.

1.1 IMPORTANCE OF STUDY

This research focus on the heat transfer and fluid flow inside a mixing elbow, starting from the flow entering the inlet until it flow out through the pipe outlet. The importance of this study are as follow:

i. To equip students with the knowledge and skills related to disentangle the various methods in heat transfer and fluid flow.
ii. To apply the application of CFD simulation and knowledge up on the run.
iii. To be able to analyze the results of the simulation data and make recommendations regarding the improvement of the research.
iv. To be able to implemented ANSYS FLUENT 14.0 software in a simulation of the fluid flow and heat transfer.

1.2 OBJECTIVES

This research is focussing on the heat transfer and the fluid flow inside a mixing elbow, starting from the flow entering the inlet until it flow out through the pipe outlet. The objectives are as follow:

i. To simulate and analyze the flow and temperature field in the mixing elbow.
ii. To simulate with several models of different inlet diameters.

1.3 RESEARCH SCOPE

This research is focussing on the heat transfer and the fluid flow inside a mixing elbow, starting from the flow entering the inlet until it flow out through the pipe outlet. The scope of this research is as follow:

i. Design a 3-dimensional (3-D) model of mixing elbow by using CFD application.

ii. Simulate the 3D mixing elbow model with various inlet diameter by using CFD

iii. Visualize and analyze the results of the fluid flow and heat transfer in mixing elbow

1.4 PROBLEM STATEMENT

Mixing elbow is a region where a cold and hot flow is intensively mixed together, where the flow appears as turbulent flow. Geometry of mixing elbow is consists of two inlets and one outlet. The main inlet is where the entrance of the hot flow and the branch inlet is where the cold flow entering the piping system. The mixed flow is then will flow out through the outlet of the mixing elbow. At the mixing region, a heat transfer process takes place which caused a temperature changes. In another words, once the two types of fluid mixed together, temperature fluctuation will happens. This temperature fluctuation will generate a cyclic stresses in the mixing region. The cyclic stresses will be transmitted to the adjoining structure, which is the weakest part of the mixing elbow with fairly low attenuation. In addition, cyclic stress is found to be a potential cause of thermal fatigue failure in any mixing elbow piping system. Mostly, the properties of fluids that entering the mixing elbow is a highly reactive, for example a flammable substances. Once there is a crack, the fluid will leaked to outside and will caused a fatal incidence. Therefore, by simulating and analyzing the flow and temperature field in the mixing elbow will provides a solution to prevent the thermal fatigue failure. Only then, the safety requirement for the mixing elbow can be acquired. Based on that reason, this study is being conducted (Aulery et al, 2012).

CHAPTER 2

LITERATURE REVIEW

2.0 INTRODUCTION

Mixing elbow is a region where two types of fluid flow with different temperature and high Reynolds number is intensively mixed together. Aulery et al, (2012) states that, heat transfer process takes place in the mixing elbow from high temperature substances to lower temperature substances. Mixing elbow is among typical geometries exactly where temperature fluctuation happens. The principle of mixing elbow is as illustrated in Figure 2.1 below.

As seen from Figure 2.1, mixing elbow is consists of two inlet and one outlet. The two inlets can be classified as main inlet and branch inlet. A hot properties fluid will enter the main inlet and flow through the pipe. Meanwhile, a cold fluid will enter the pipe through the branch inlet. Until it reached a certain point, the two types of flow will mixed together at the mixing region and appear as turbulent flow.

During the mixing of the two fluids, there will be a temperature changes which known as temperature fluctuation. Aulery et al, (2012) explained that, this temperature fluctuation will generate a cyclic stresses in the mixing region. The cyclic stresses that been generated will be transmitted to the adjoining structure, which is the weakest part of the mixing elbow with fairly low attenuation. As seen from Figure 2.2 below, the cyclic stress is acted on the location of the weld, where later on it will initiate a crack on the adjoining structure. This cyclic stress is a potential cause of thermal fatigue failure (Aulery et al, 2012).

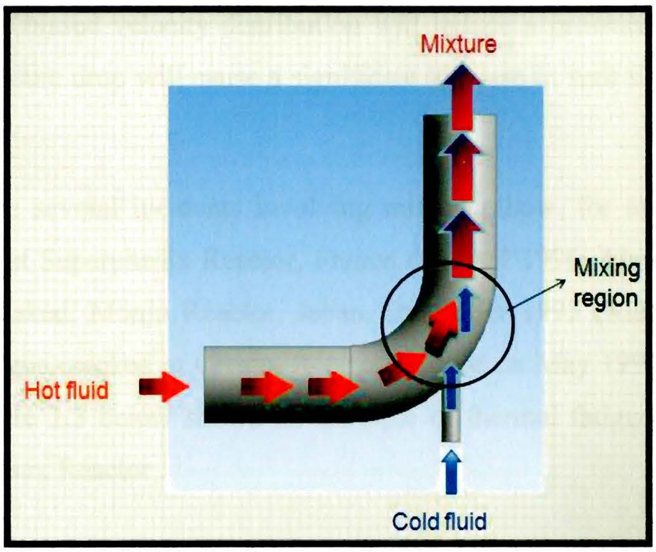

Figure 2.1: Mixing elbow principle.

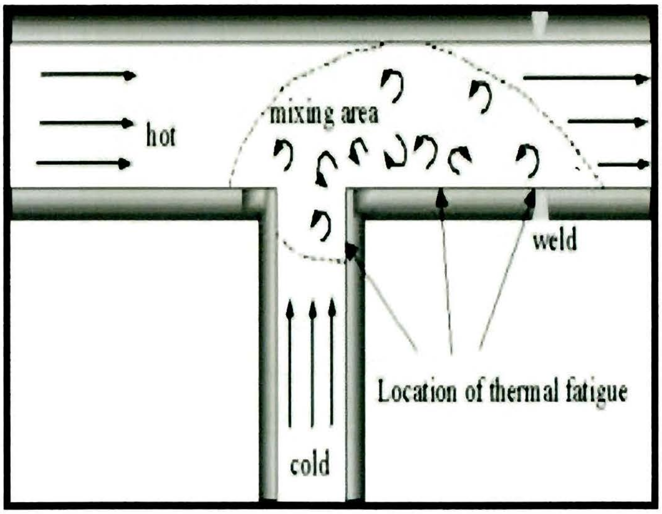

Figure 2.2: Location of thermal fatigue (Paffumi et al, 2012).

On the other hand, El-Gammal et al, (2010) states in the finding that the changes in flow direction due to the geometry of the mixing elbow will produce a secondary flow or flow separation due to biased velocity distribution. This biased velocity is a factor that responsible for Flow Accelerated Corrosion (FAC) at the elbow. FAC is a slow degradable process where the flow will damaging or thinning the protective layer of the piping component and soon will initiate a crack on the pipe surface. Meanwhile, Crawford et al, (2007) also states that the secondary flow that is produced by the biased velocity distribution will induce a pressure drop along the elbow. This pressure drop will cause a significant increase in wall shear stress along the mixing elbow.

There are several incidents involving mixing elbow, for examples thermal fatigue cracked at Superphenix Reactor, France on April 1996, Almeria solar plant, 1996, sodium leaked, Monju Reactor, Japan, December 1995 (Aulery et al, 2012) and thermal fatigue cracked at Civaux Reactor, France on May 1998 (Passuto et al, 2007). The Figure 2.3 below shown an example of thermal fatigue cracked which happened at Civaux Reactor .

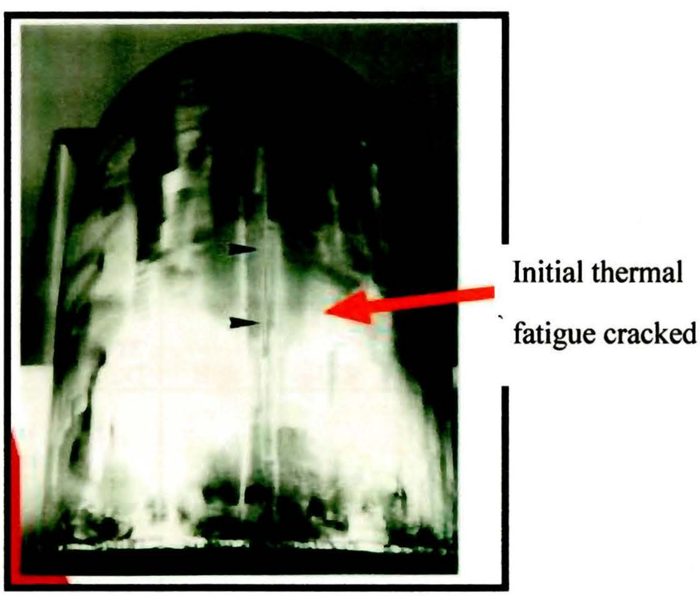

Figure 2.3: Cracked on Civaux Reactor mixing elbow (Passuto et al, 2007).

2.1 CORRELATION MATRIX

Table 2.1 below shows the lists journal that is found to be related to this study. The journals have been extracted in terms of CFD solver, Reynolds number, simulation method, grid system, and results and analysis. As seen from the matrix below, most of previous researchers used ANSYS FLUENT as their solver which ideal on solving internal flow with high Reynolds number. The common method that have been used in their research is Large Eddy Simulation (LES). Meanwhile, the results of their simulation were validated by compare with the experimental data or previous researcher's data.

Table 2.1: List of journals.

Journals	Solver	Reynolds Number	Simulation Method	Grid System	Result & Analysis
Numerical Simulation of Sodium Mixing in a T-junction (Aulery et al, 2012)	ANSYS FLUENT	High	RANS & LES	Coarse/ Fine	• Wall temperature fluctuation. • Evaluation of Thermal Fatigue. • Compare with experimental data.
Large-Eddy Simulation Study of Turbulent Mixing in T-junction (Kuczaj et al, 2010)	ANSYS FLUENT	High	k-ε RANS & LES	Coarse/ Medium Fine	• Flow prediction. • Comparison with various resolution.
Large Eddy Simulation of a thermal mixing tee in order to assess the thermal fatigue (Galpin,J. Simoneau, J.P., 2011)	ANSYS FLUENT	High	LES	Fine	• Prediction on mean and fluctuating temperature. • Compare with experimental data.
Numerical analysis of thermal striping induced high cycle thermal fatigue in a mixing tee (Jeong et al, 2009	ANSYS FLUENT	High	LES	Coarse/ Fine	• Analyze temperature fluctuation. • Structural response of coolant piping at mixing tee. • Compare with experimental data.
Suitability of wall-functions in LES for thermal fatigue in a T-junction (Jayaraju et al, 2010)	ANSYS FLUENT	High	LES	Coarse/ Fine	• Analyze the stability of wall functions. • Compare with existing experimental data.
Thermal stress analysis for fatigue damage evaluation at mixing tee (Kamaya et al, 2011)	ANSYS FLUENT	High	k-ω & LES	Fine	• Thermal stress analysis. • Compare with existing experimental data.

Table 2.1: List of journals (cont.).

Journals	Solver	Reynolds Number	Simulation Method	Grid System	Result & Analysis
Experimental study on fluid mixing phenomena in T-pipe junction with upstream elbow (Ogawa et al,2005)	Not available	High	Water experiment	Not available	• Flow pattern in the tee. • Flow velocity in the elbow. • Temperature field in the tee. • Influence of elbow in the wall jet case.
Three-dimensional numerical investigation of flow at 90° open channel junction (Al-Mussawi et al, 2009)	ANSYS FLUENT	High	Finite Volume Analysis	Coarse/ Medium/ Fine	• Velocities profile with different discharge ratios. • Compare with experimental data.
A hybrid RANS-LES model for combining flows in open-channel T-junctions (Cheng et al, 2010)	ANSYS FLUENT	High	RANS & LES	Fine	• Velocity and water surface profile. • Compare with experimental data.
Large eddy simulation (LES) of temperature fluctuations in a mixing tee with/without porous medium (Jiang et al, 2010)	ANSYS FLUENT	High	LES	Coarse/ Fine	• Temperature fluctuations with/without porous media. • Temperature and velocity fields. • Compare with experimental data.
The hydrodynamic effect of single-phase flow on flow accelerated corrosion in 90° elbow (El-Gammal et al, 2010)	ANSYS FLUENT	High	RSM & Experiment	Medium	• Hydrodynamic effects of single phase flow on FAC. • Surface wear patterns.

2.2 SIMULATION METHOD

The simulation method that have previously used in the study of fluid mixing is based on these two methods:

i. Experiment
ii. Computational Fluid Dynamic (CFD)

2.2.1 Experiment

Experiment on the fluid mixing phenomena have been done by Ogawa et al, (2005). Their experiment being conducted in order to study the mixing phenomena in the T-pipe junction with upstream elbow. The experiment have various procedures where the method that being used for this experiment is Water Experiment. The experiment set up is as illustrated in Figure 2.4 below.

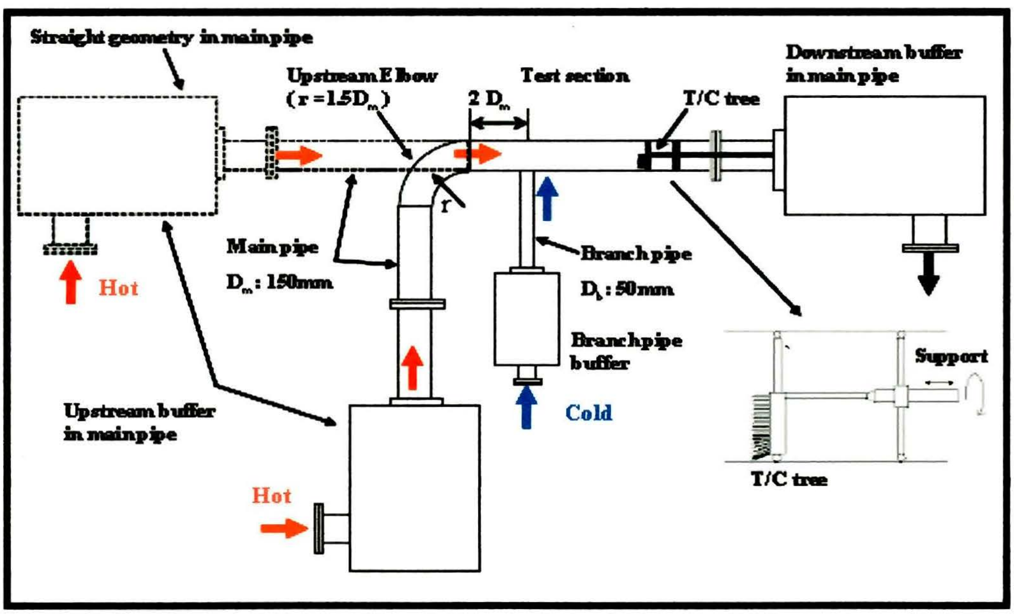

Figure 2.4: Schematic of test section (Source: Ogawa et al, 2005).