



**COMPUTATIONAL SIMULATION OF HEAT TRANSFER IN A
FLOW AROUND A TUBE BANKS USING FLUENT**

**IRFAH BINTI ISMAIL
(2002334908)**

**BACHELOR ENGINEERING (HON) (MECHANICAL)
UNIVERSITI TEKNOLOGI MARA (UiTM)
OCTOBER 2005**

ACKNOWLEDGEMENT

In the name of ALLAH, the Almighty and the Most Merciful. I'm very thankful to ALLAH who gave the strength and blessing in order to complete this project report.

Firstly, I would like to extend my gratitude and appreciation to my project advisor, Mr. Azli bin Abd Razak for his guidance, advice and give an opportunity to do the project with him. I'm so thankful for his motivation and encouragement that he gives to me.

I'm also want to thanks Ir. Khalid, Ms. Nurlin and Mr. Hazleen for their helps, information and advices in completing our study.

To my parents, thanks for their spiritual supported in completing my study in UiTM and not forget to all my friends for their support in collecting information and ideas.

Lastly, I would like to thank to all individual whether directly or indirectly involve during completing this project.

Thank you.

ABSTRACT

Two-dimensional of simulation modelling of heat transfer in an incompressible fluid flow around the tube banks had been conducted. This study is about computational simulation of heat transfer around the tube banks by using FLUENT. The result for in-line and staggered cases for the maximum velocity, temperature distribution and the flow pattern are discuss.

TABLE OF CONTENTS

	CONTENTS	PAGE
	TITLE	i
	ACKNOWLEDGEMENT	ii
	ABSTRACT	iii
	TABLE OF CONTENTS	iv
	LIST OF TABLES	vi
	LIST OF FIGURE	vii
	LIST OF NOMENCLATURE	ix
CHAPTER I	INTRODUCTION	
	1.1 Introduction	1
	1.2 Project Background	2
	1.3 Objectives	2
	1.4 Methodology	3
	1.5 Literature Review	5
CHAPTER II	THEORY	
	2.1 Introduction to Heat Transfer	6
	2.2 Governing Equation	10

CHAPTER I

INTRODUCTION

1.1 Introduction

An overview of major theoretical, numerical and computational study advancement in heat transfer is presented in this project. The used of Computational Fluid Dynamics (CFD), FLUENT software is used to simulate the heat transfer around the tube banks. This will serve as a general motivation for this project. This thesis is a research concerning about the temperature and heat distributions around the tube banks. The study of analytical solution for the heat transfer in tube banks in crossflow is presented by A.Zukauskas and R. Ulinskas [1].

The thesis consists of simulation of heat transfer using CFD software, and finally compare with the existing results. The simulation of heat transfer in a flow around a tube banks is very important in order to study the phenomena of heat transfer and it will help to design the heat exchanger and boiler.