

**ANALYSIS OF COOLING PERFORMANCE FOR AIR COOLING  
CHANNELS OF A PEM CELL USING CFD: INLET GEOMETRY DESIGN**

**NUR HAMIZA BINTIRAMLI**

(2007270922)

A thesis submitted in partial fulfillment of the requirements for the award of  
Bachelor Engineering (Hons) Mechanical

**Faculty of Mechanical Engineering  
Universiti Teknologi MARA (UiTM)**

**MAY 2010**

## ACKNOWLEDGEMENT

I would like to express my sincere gratitude to my project supervisor Prof Madya Dr Rahim Atan, and co - supervisor En Wan Ahmad Najmi bin Wan Mohamed for their continuous support, generous guidance, help, patience, advices, and encouragement in the process of making this final year project until its completion.

Not to forget, a special thanks to my family, my friends, and for the people who were involved directly or indirectly in completing this project.

With all the support, I have finally managed to complete this project and I hope it will fulfill the requirements for the award of Bachelor of Engineering (Hons) Mechanical. Last but not least, I hope that this project will be utilized for the sake of mankind.

## ABSTRACT

This study focused on designing and simulating alternative air inlet geometry of air cooling on a single channel (SSI) and multichannel (MS60) channel types for PEM fuel cell bipolar plate with a power rating of 3kW. Two air inlet geometry designs generated are nozzle type with three different angles and quarter circle type with three different radiuses. The designs were generated using CATIA software and simulation work using STAR-CCM; a CFD (Computational Fluid Dynamics) software type. The simulations were conducted to observe and determine whether these new inlet geometry designs give some improvement and better results in heat transfer rate of fuel cell compare to conventional design. The higher heat transfer rate of the fuel cell results in higher heat transfer efficiency of the fuel cell. Each design was simulated in four different velocities from 0.5 m/s to 3.0 m/s with similar boundary conditions such as laminar flow, air inlet temperature, and constant heat flux. The simulation results show that each design generates different flow field that gives higher heat transfer rate than conventional design. The simulated air inlet geometry shows 0.04% to 24.6% improvement on heat transfer rate of a bipolar plate fuel cell. The best design of air inlet geometry is 15 degree nozzle type.

## TABLE OF CONTENTS

	<b>CONTENTS</b>	<b>PAGE</b>
	PAGE TITLE	1
	ACKNOWLEDGEMENT	ii
	ABSTRACT	Hi
	TABLE OF CONTENTS	iv
	LIST OF TABLES	viii
	LIST OF FIGURES	ix
<b>CHAPTER I</b>	<b>INTRODUCTION</b>	
	1.0 Background of The Project	1
	1.1 Problem Statement	2
	1.2 Objectives	2
	1.3 Scope of Study	3
	1.4 Significance of Study	3
<b>CHAPTER H</b>	<b>LITERATURE REVIEW</b>	
	2.0 Introduction	4
	2.1 Heat Transfer Fundamental	5
	2.2 Nusselt Number	6
	2.3 Prandtl Number	7

2.4	Reynolds Number	7
2.5	Computational Fluid Dynamics	8
2.6	Flow Inlet Boundaries	9
2.7	Thermal Management	10

## **CHAPTER III METHODOLOGY**

3.0	Introduction	11
3.1	Work Sequences	12
3.2	Project Methodology	12
3.2.1	Geometry Generation	12
3.2.2	Comparative Method	13
3.2.3	Numerical Method	13
3.3	Simulation	15

## **CHAPTER IV INLET GEOMETRY DESIGN AND BOUNDARY CONDITION**

4.0	Introduction	17
4.1	Air Cooling Channel Design	18
4.1.1	LI Standard/Conventional Inlet Geometry Design	19
4.1.2	Nozzle Type Inlet Geometry	21
4.1.3	Quarter Circle Inlet Geometry	22
4.2	Boundary Conditions	24
4.2.1	Air Boundary Conditions	24
4.2.2	Bipolar Plate Boundary Conditions	24
4.3	Simulation Parameters	25
4.3.1	Physics Parameter	25
4.3.2	Mesh Parameter	26
4.4	Assumption Calculations	27