

**STUDYING THERMAL PERFORMANCE OF SPLIT-TYPE AIR CONDITIONERS AT  
BUILDING RE-ENTRANT VIA COMPUTER SIMULATION**



**INSTITUT PENGURUSAN PENYELIDIKAN  
UNIVERSITI TEKNOLOGI MARA  
40450 SHAH ALAM, SELANGOR  
MALAYSIA**

**BY :**

**FAUZIAH JERAI  
OW CHEE SHENG**

**FEBRUARI 2011**

Tarikh : 28 Febuari 2011  
No. Fail Projek : Dst 248/2009

Encik Mustafar Kamal Hamzah  
Ketua Penyelidikan(Sains dan Teknologi)  
Institut Pengurusan Penyelidikan (RMI)  
Universiti Teknologi MARA  
40450 Shah Alam  
Selangor Darul Ehsan

Tuan,

**LAPORAN AKHIR PENYELIDIKAN ‘STUDYING THERMAL PERFORMANCE OF SPLIT-TYPE AIR CONDITIONERS AT BUILDING RE-ENTRANT VIA COMPUTER SIMULATION’**

Merujuk kepada perkara di atas, bersama-sama ini disertakan 3 (tiga) naskah Laporan Akhir Penyelidikan bertajuk “STUDYING THERMAL PERFORMANCE OF SPLIT-TYPE AIR CONDITIONERS AT BUILDING RE-ENTRANT VIA COMPUTER SIMULATION”.

Sekian, terima kasih.

Yang benar,

**FAUZIAH JERAI**

Ketua  
Projek Penyelidikan

## **ABSTRACT**

Split-type air conditioners used in residential or office buildings often have the outdoor condensing units install at the sidewalls. In condominium or apartment buildings, the building re-entrants are the most frequent choice of the project team for placing the outdoor air-cooled condensing units. To avoid wastage in electrical energy and unnecessarily heat rejection to the environment, the HVAC engineer and the team should predict the thermal effect on the air-conditioners usage from the layout design stage. Additionally, the layout design is also continuously changing before the final design stage and this enhances the difficulty for the team to estimate the actual effect of placing over a hundred outdoor units inside one single re-entrant of a high rise building. In order to tackle this problem, a simulation approach is applied using Computational Fluid Dynamics (CFD), (Phoenics v2008). The energy performances as well as the levels of thermal comfort for the proposed option were evaluated.

## TABLE OF CONTENTS

<b>ABSTRACT</b>	ii
<b>ACKNOWLEDGEMENT</b>	iii
<b>TABLE OF CONTENTS</b>	iv
<b>LIST OF FIGURES</b>	vii
<b>LIST OF TABLE</b>	ix
<b>NOMENCLATURE</b>	xvii
<b>ABBREVIATIONS</b>	xviii

### **CHAPTER 1: INTRODUCTION**

1.1	Background Of Study	1
1.2	Scope Of Study	2
1.3	Objective Of Study	2
1.4	Layout Of Report	3

### **CHAPTER 2: LITERATURE REVIEW**

2.1	Condensing Units	4
2.1.1	Vapor-compression Refrigeration Cycle	5
2.2	COMPUTATIONAL FLUID DYANAMICS (CFD)	7
2.2.1	History of CFD	8
2.2.2	Turbulences model in CFD	11
2.2.2.1	Transport equations for the RNG $k-\epsilon$ model	13
2.2.2.1	Mixing length model	14
2.2.2.2	K- $\epsilon$ model	15
2.2.2.3	Calculating the inverse effective Prandtl numbers	17
2.2.2.4	The $R_\epsilon$ term in $\epsilon$ equation	18
2.2.2.5	Large Eddy Simulation (LES)	19
2.3	Bouyancy Effect	20
2.4	Conjugate Heat Transfer Problems	20

2.5	CFD Simulations and experimental results	21
-----	--	----

### **CHAPTER 3: COMPUTATIONAL SETUPS**

3.1	Introduction to CFD	24
3.2	Basic Components of CFD Code	25
3.2.1	Pre-Processor	25
3.2.2	Solver	26
3.2.3	Post Processor	26
3.3	Conservation Laws of Fluid Motion	27
3.3.1	Fluids Flow and Properties	27
3.3.2	Basic Conservation Equations Describing Fluids In Motion	29
3.4	Simulation Methodology	30
3.4.1	Description of Physical Problems	30
3.4.2	Defining The Problems In CFD Codes	31
3.4.3	Domain – Geometry and Time Definition	
3.4.3.1	Time setting	32
3.4.4	Process Definition	
3.4.5	Objects Definition	35
3.4.5.1	Blockages	35
3.4.5.2	Plates	37
3.4.5.3	Fans	38
3.4.5.4	Opening	39
3.4.6	Heat Sources	40
3.4.7	Numerical Settings	40
3.4.7.1	Number of Iterations	40
3.4.7.2	Relaxation Control	41
3.4.8	Monitored Location and Field Dumping	42