EVALUATION PERFORMANCE OF LOUVERED-FIN COMPACT HEAT EXCHANGER

HASZIL ADHA BIN HASLAN

Laporan ini dikemukakan sebagai
memenuhi sebahagian daripada syarat penganugerahan
Ijazah Sarjana Muda Kejuruteraan Mekanikal (Termal-Bendalir)

Fakulti Kejuruteraan Mekanikal
Universiti Teknikal Malaysia Melaka

Mac 2007

‘Saya/Kami* akui bahawa telah membaca
karya ini dan pada pandangan saya/kami* karya ini
adalah memadai dari segi skop dan kualiti untuk tujuan penganugerahan
Ijazah Sarjana Muda Kejuruteraan Mekanikal (Termal-Bendalir)’

Tandatangan :…………………………………..
Nama Penyelia I:………………………………..
Tarikh :………………………………………….

Tandatangan:……………………………………
Nama Penyelia II:………………………………
Tarikh:………………………………………….

"Saya akui laporan ini adalah hasil kerja saya sendiri kecuali ringkasan dan petikan yang tiap-tiap satunya saya telah jelaskan sumbernya"

Tandatangan :……………………………..
Nama Penulis :……………………………
Tarikh :……………………………………

ACKNOWLEDGEMENT

Alhamdulillah with His Mercy and Blessings, my greatest gratitude to Allah the Almighty for the chances given to me to complete this project. My first acknowledgement goes to Mr Shamsul Bahari Bin Azraai, my project supervisor. Thanks for his tremendous help, advice, inspiration and unending guidance to me until completing this research. Without him, this project will not exist. The fundamental idea behind this project is his and he never stop from giving me support and encouragement in completing this project. Without his valuable guidance, I would not have been able to achieve the objectivity of this project.

Not forgotten, to my beloved parents, Mr Haslan Bin Hassan and Mrs Nur Aziah Binti Abdul Latif. Your love and caring really touched me. My sister and brother, Haszianaliza Binti Haslan and Hasziq Hafify Bin Haslan. Thanks so much for giving me lifelong encouragement, inspiration and support. May Allah bless all of you.

Lastly, my acknowledgement go to the technician, Mr. Razmi who have lend to me some help. My friends who really understand and willing to cooperate with me in the process of completing this project. Without your supports, I will never go this far.

ABSTRACT

The research of performance of louvered-fin Compact Heat Exchanger is done by simulating the problem using Ansys CFX Software. This report will represent the result from the simulation done by CFX Software along with the discussion and conclusion. The fluid used in the simulation is air at 25°C and the speed of the air enter through the inlet is 15m/s. At the end of this study the optimum geometry that is suitable for mass use will be discussed. This study have found that the optimum geometry to enhance the heat transfer performance for louvered fin compact heat exchanger is 29° louver angle and 1.30mm louver pitch. The results of the optimum geometry of the louvered fin have been validated by study done by previous academician. The performance of this louvered fin type compact heat exchanger has been evaluated by monitoring the temperature drop at the fin by the observing of the temperature contour. The conclusion of this study is that the louvered fin will enhance the heat transfer performance because of the increasing in the heat transfer surface area. And when the air is flowing through the louvered fin array, the flow is louver directed pattern and will cause great temperature drop at the fin plat. Louver directed flow and axial directed flow will be the main reason in enhancing the heat transfer performance of the louvered fin compact heat exchanger.

ABSTRAK

Kajian tentang kecekapan penukar haba padat bersirip tetingkap dilakukan dengan mensimulasikan masalah tersebut dengan menggunakan perisian Ansys CFX. Laporan ini akan memaparkan keputusan daripada simulasi yang dilakukan bersama-sama dengan perbincangan tentang keputusan yang diperolehi dan juga kesimpulan yang dapat dibuat. Bendalir yang digunakan dalam simulasi ini adalah udara pada 25°C dan kelajuan udara tersebut melalui masukan adalah 15m/s. Diakhir kajian ini, rekabentuk optimum yang sesuai untuk penggunaan secara besar-besaran akan dibincangkan. Kajian ini telah menemui bahawa geometri optimum untuk meningkatkan kecekapan penukaran haba untuk penukar haba padat bersirip tetingkap adalah 29° sudut tetingkap dan 1.30mm jarak tetingkap. Keputusan yang diperolehi tentang rekabentuk geometri optimum telah di sahkan dengan merujuk kepada kajian yang telah dilakukan oleh pengkaji sebelum ini. Kecekapan penukar haba padat bersirip tetingkap telah dinilai dengan melihat kepada pengurangan suhu yang berlaku pada sirip tersebut dengan melihat kepada kontor suhu. Kesimpulan yang dapat dibuat daripada kajian ini adalah sirip jenis tetingkap akan dapat menambahkan kecekapan penukaran haba kerana pertambahan luas permukaan penukaran haba. Dan apabila udara melalui susunan sirip tetingkap tersebut, aliran udara tersebut akan menjadi aliran berarahkan tetingkap dan ini akan menyebabkan kejatuhan suhu yang banyak pada plat sirip tersebut. Aliran berarahkan tetingkap dan aliran berarahkan paksi menjadi penyebab utama dalam menambahbaikkan kecekapan penukaran haba pada penukar haba padat bersirip tetingkap.

TABLE OF CONTENT

CHAPTER	TITLE	PAGE NO

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

q	=	Total Heat Transfer, W
U	=	Overall Heat Transfer Coefficient, W/m².K
ΔT_m	=	Mean Temperature Different, K
f	=	Friction Factor
Δp	=	Pressure Different, N/m²
ρ	=	Density, kg/m³
A	=	Total Heat Transfer Area, m²
A_c	=	Minimum Flow Area, m²
St	=	Stanton Number
h_c	=	Heat Transfer Coefficient, W/m².K
c_p	=	Specific Heat at Constant Pressure, kJ/kG.˚C
Q	=	Heat Transfer Rate, W
$LMTD$	=	Log Mean Temperature Different
T_o	=	Outlet Temperature, K
T_i	=	Inlet Temperature, K
T_a	=	Air Temperature, K
T_f	=	Fin Temperature
Re	=	Reynolds Number
υ_s	=	Mean Fluid Velocity, m/s
L	=	Characteristic Length, m
μ	=	Dynamic Fluid Viscosity, kg/m.s
ν	=	Kinematic Fluid Viscosity, m²/s
L_p	=	Louver Pitch, m
d_h	=	Hydraulic Diameter, m
L_s	=	Fin Length, m
Re_d	=	Reynolds Number Based On Hydraulic Diameter
θ	=	Louvered Angle
F_p	=	Fin Pitch, m

T_p = Transverse Pitch, m

LIST OF ABBREVIATIONS

CHE	=	Compact Heat Exchanger
CFD	=	Computational Fluid Dynamics
etc	=	Etcetera
LMTD	=	Log Mean Temperature Different

LIST OF APPENDIXES

CHAPTER I

INTRODUCTION

This chapter is to introduce the definition of CHE. The purpose, scope and problem statement for this research is also stated here.

1.1 Introduction

Compact Heat Exchanger is commonly used to enhance the heat transfer rate in application that involves with high temperature. Generally there are two types of CHE which are plate-fin type and tube-fin type. CHE is only used for gas-gas or gas-liquid application [1]. Plate Fin Compact Heat Exchanger is widely used in gas-gas application [1]. In this study we will consider plate-fin type with louvered fin.

Nowadays CHE is widely used in various type of industries such as automotive, aeronautics and astronautics, electric and electronic equipment, refrigeration and else. CHE is preferred for application that needs heat exchanger because of its compactness, high-effectiveness, low cost needed, small volume and low in weight [1].

Because of it is widely used in industries, so the optimization of CHE design is important. This optimization can be done by using various type of design and selection of good material to reduce cost but give high effectiveness. There are several parameters to consider CHE performance such as the outlet temperature, heat duty involves and pressure drop in the CHE.

There are some parameters that could be change in order to evaluate the performance of CHE. The geometry parameters such as louver pitch, louver angle, fin pitch and tube pitch. Figure 1.1 shows the louvered fin type of CHE.

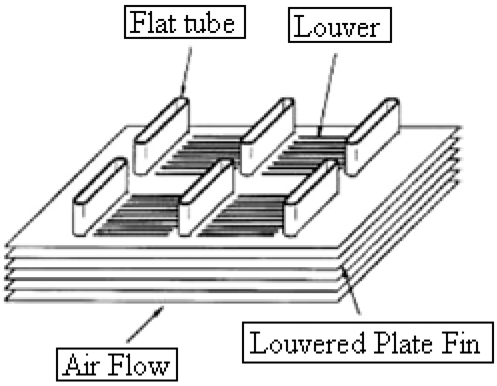

Figure 1.1: Louvered fin type CHE [2]

1.2 Objectives

The objectives of this study are stated as below:

- To study the effect of louver directed flow and axial directed flow around the louvered fin array in the heat transfer enhancement
- To visualize the pressure, velocity and temperatures profiles around louvered fin array
- To determine the optimum geometry design of louvered fin array for better heat transfer

1.3 Scopes

The scopes of this study are detailed as below:

- Study the various geometry design in louvered fin array by varying the value of louver angle and louver pitch
- Design CFD Louvered fin geometry using CFX software (Design Modeler)
- Simulate air flow through louvered fin array
- Study the effect of flowing air through the louvered fin array

1.4 Problem Statement

The problem that we have to consider here is by comparing the performance of this type of CHE by varying geometrical parameter such as louvered angle and louver pitch.

Figure 1.2 shows an array of louvered fin with geometrical parameters in CHE. This study will only focus the effects of varying the value of louver angle and louver pitch.

The flow phenomena of the air will be either axial directed flow or louver directed flow or both. The effects of the flow phenomena will be discussed to determine the heat transfer efficiency between the flowing air and the louvered fin array. The value of fin pitch will remain constant at 2.11mm.

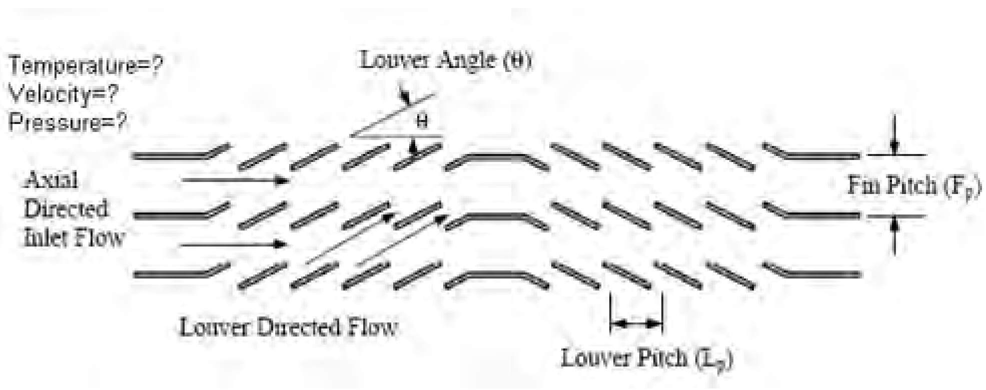

Figure 1.2: An Array of louvered fin with geometrical parameters [3]

Figure 1.3 shows the schematic of louvered-fin design. The studied air will be flowing into the louvered fin array from the inlet and will be drawn out through the outlet. The symmetry part will be transform later in CFX Post to view the real simulation of the fin array.

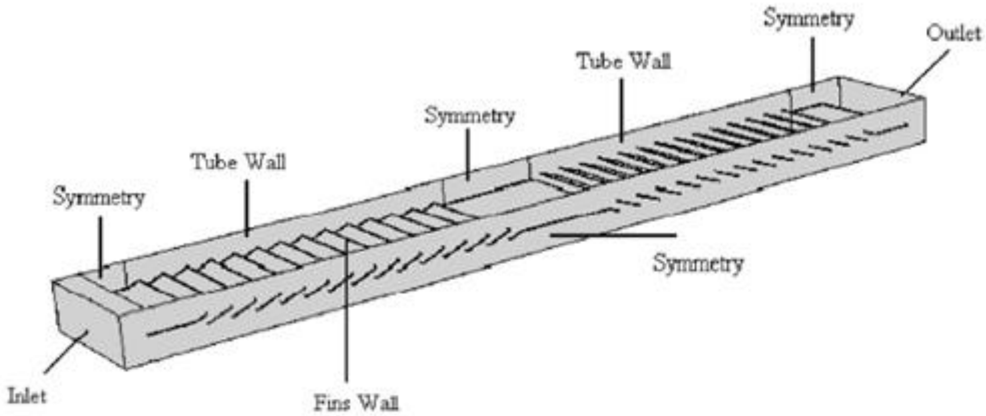

Figure 1.3: Schematic of louvered-fin design to be considered [2]

Air flows through a louvered array at air velocity which ranging from 10m/s to 30m/s. The temperature of the fin is maintained constant at specific temperature. Ansys CFX Software is used to simulate air flow through a fin array. Finally, the heat transfer rate by means of temperature drop of the louvered fin will be determined from observation of velocity, temperature and pressure profile.

CHAPTER II

LITERATURE REVIEW

This chapter will discuss about all the equations, theory, past studies and parameters used in this research.

2.1 Introduction

Heat exchanger is a device that efficiently transfers or exchange heats from a fluid to another fluid without any contact between those two [4]. Figure 2.1 shows the simplest heat exchanger.

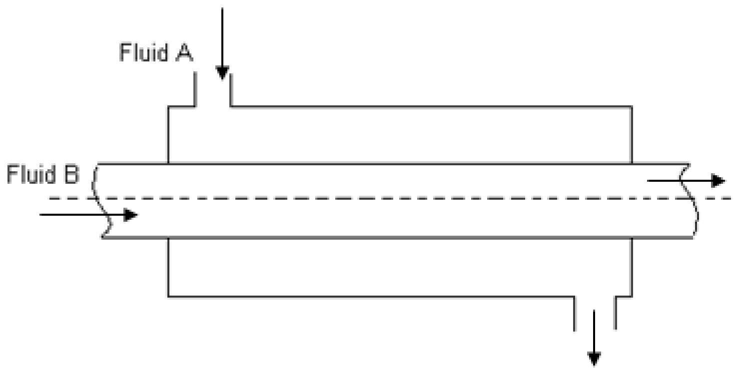

Figure 2.1: Schematic of Heat Exchanger

Fluid A will cool down Fluid B as both of the fluid moves. The cooling process is by convection as both of the fluid does not contact with each other because of the barrier in the middle. All heat exchanger concepts are base on this. What differs is only the design of each heat exchanger.

2.2 Compact Heat Exchanger

The definition of CHE is based on heat exchanger definition but in CHE, compact is added. And so, based on heat exchanger definition the definition of CHE will be; a heat exchanger that is compact which means it has small volume, small size and also low in weight. CHE are typically used when a large heat transfer surface area per unit volume is desired or at least one of the fluids is a gas [5]. A CHE can reach an area density higher than $500 m^2/m^3$ with respect to $100\ m^2/m^3$ - $200\ m^2/m^3$ of the shell and tube heat exchanger [6]. There are generally two types of CHE which are plate-fin type or tube-fin type [1]. Figure 2.2 shows the example of plate-fin type CHE.

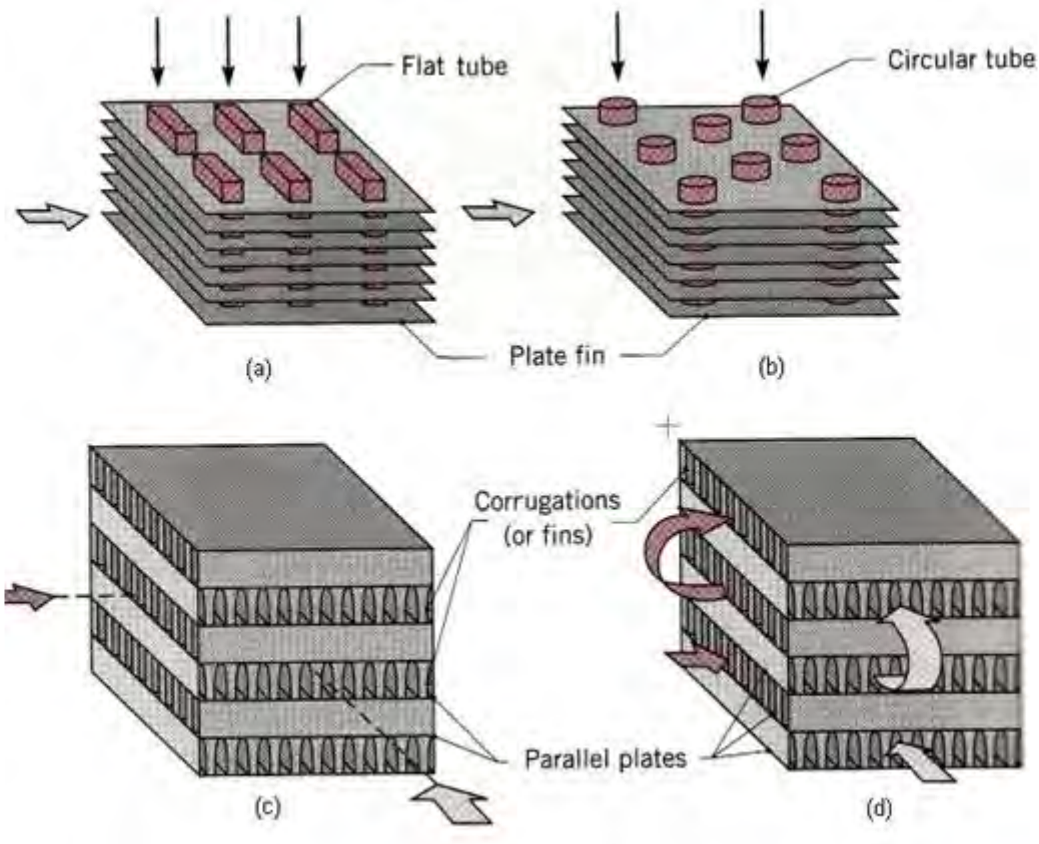

Figure 2.2: Compact Heat Exchanger cores. (a) Fin-tube(flat tubes, continuous plate fins). (b) Fin-tube (circular tubes, continuous plate fins). (c) Plate fin (single pass). (d) Plate fin (multipass) [5]

CHE appliances that are commonly used are car radiator or automotive radiator. In this case of CHE, the heat exchanger is used to cool the engine as the engine is heated because of the combustion process. Because of the combustion process of the engine, the engine's temperature will later increased and to avoid overheating of the engine, CHE is used where in this case, fluid in the CHE is commonly called as coolant and this coolant will move around the engine and cooled it down. The coolant is cooled because of the air flowing through the geometry of the CHE act as a cooling fluid to the fin array. Figure 2.3 shows the car radiator that use louvered fin CHE.

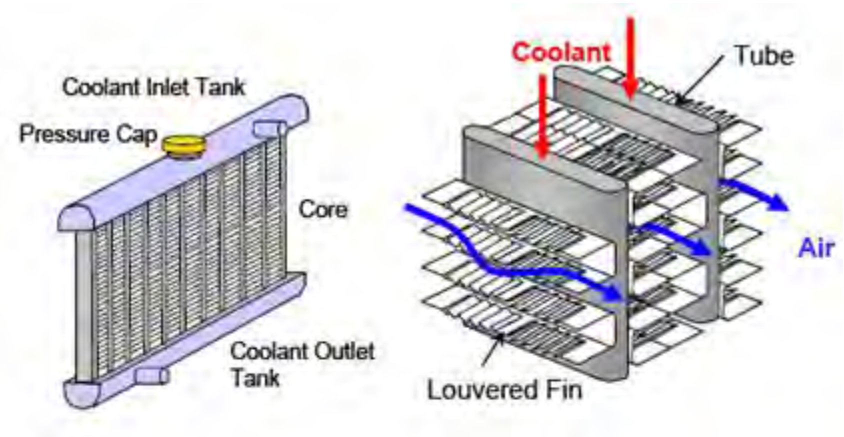

Figure 2.3: Car radiator that use louvered fin CHE [7]

Coolant will enter the tube from the coolant inlet tank and as the coolant moves through the tube, heat transfer happened between the coolant and air. As a result of this heat transfer process, the coolant will later being cooled down and exit to the coolant outlet tank.

2.3 Louvered Fin Compact Heat Exchanger

Louvered is defined as an open window liked geometry. So in this type of CHE, louvered fin CHE means a CHE that have fin geometry like an open window. To make this matter clearer, Figure 2.4 shows the schematic of louvered fin CHE.

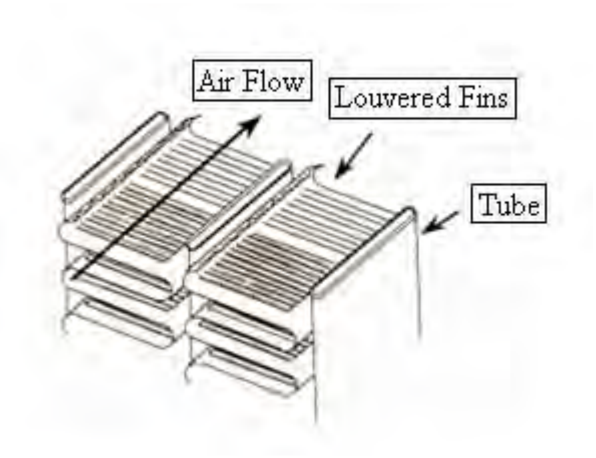

Figure 2.4: Example of louvered fin CHE [3]

When this louvered fin was firstly introduced, it was believed that increased heat transfer was due to added turbulence initiated by flow through the louvered array [3]. But this theory was later proved wrong by the visualization studies on scaled up louver design [3]. It was observed that as air passes through the louver, the flow resulted in two distinct flow directions to be classified: axial (or duct) directed flow and louver flow. Axial flow occurs when the flow maintains the directions of the inlet flow. Louver directed flow occurs when the flow was aligned parallel to the louver [3]. Figure 2.5 shows clearly the two directions indicated in the louvered fin cross section.

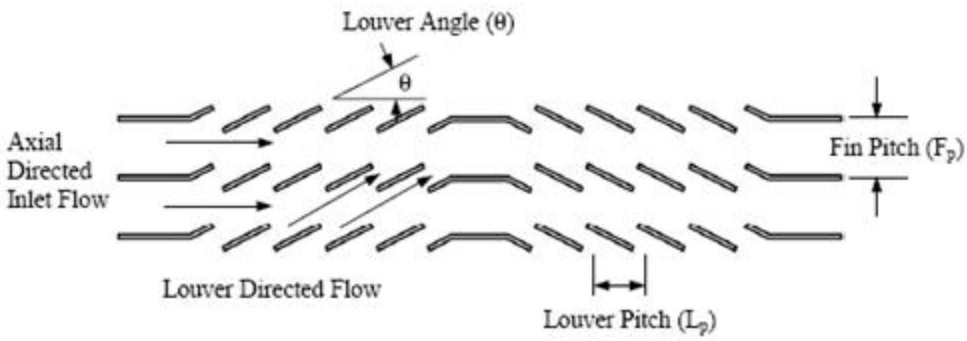

Figure 2.5: Louver directed flow and axial directed flow [3].

The explanation for increased heat transfer came from the realization that along with the new boundary layer that formed on each of these louver arrays, came a corresponding high heat transfer coefficient.

2.4 Evaluation Performance of Compact Heat Exchanger

There are generally two ways in evaluating the performance of CHE which are by numerical predictions using Computational Fluid Dynamic or by experimental measurement where direct experiment was done in order to obtain data needed for the performance calculation. For this research, CFD method is used in evaluating the performance of CHE. Software used for this simulation is Ansys CFX.

2.4.1 Computational Fluid Dynamic

CFD is a branch of fluid dynamics where it is use widely to solve and analyze problem that involve with fluid flow and heat transfer using numerical method and algorithm [8]. Supercomputer or computer that works together in parallel is used to solve millions of equation required to simulate interaction of fluid and gasses. However, whatever value we get from CFD is only approximate solutions until the results have been validated by experimental results. The basic of solving any CFD equations is by using the Navier-Stokes equations, which defines any single-phase fluid flow. These equations will be simplified later by removing terms describing viscosity to yield the Euler equation. Later, by removing terms describing vorticity it will yields full potential equations. Finally these equations can be linearized to yield the linearized potential equation [8].

2.4.2 Log Mean Temperature Different

To design or to predict the performance of a heat exchanger, it is essential to relate the total heat transfer rate to quantities such as the inlet and outlet fluid temperatures, the overall heat transfer coefficient, and the total surface area for heat transfer [5]. Two such relations may readily be obtained by applying overall energy