‘I admit that I have read the whole of this report and in my opinion this report is fully in scope and high quality for conferment of Bachelor Degree in Mechanical Engineering (Thermal-Fluid)’

Signature :……

Supervisor’s name :……

Date :……

© Universiti Teknikal Malaysia Melaka

PARAMETER STUDY ON HEAT EXCHANGER

MASITA BINTI MOHD BARI

This report is to fulfill a partial of requirement conferment of Bachelor Degree in Mechanical Engineering (Thermal-Fluid)

Faculty of Mechanical Engineering
Universiti Teknikal Malaysia Melaka

MAY 2008

© Universiti Teknikal Malaysia Melaka

"I admit that I have made this report on my own except the summary and sentences whish is explained from the references"

Signature :……

Author's Name :……

Date :……

© Universiti Teknikal Malaysia Melaka

ACKNOWLEDGEMENT

I would like to express a warm gratitude to everyone who involved directly or indirectly during accomplishment this report. Heartfelt thankfulness goes to Mdm. Fatimah Az-Zahrah bt Mohd Sa'at for guiding in order to complete this report. Because of her guiding I have accomplish this report from beginning and complete promptly. Also, I would like to thankful to personnel who lecturer from Department of Mechanical Engineering because have continuously, encouraging and guiding me and also gave me supports and advices.

Deepest appreciation goes to my parents and all of my family members. Besides, I would like to thank my entire friend that had give support and help. Finally I would like to thankful to all individual that have involve direct or indirectly to accomplish this report. Thank you.

© Universiti Teknikal Malaysia Melaka

ABSTRACT

This study presents a set of parametric study on heat exchanger which specificily to automotive radiator. Numerical simulation was done in this study to investigate the influences of parameter to the radiator. Heat transfer performance prediction has been done using CFD codes. Three-dimensional simulation of flat tube and fin has been conducted. Simulation has been performed with different value of air inlet temperature, air and coolant mass flow. The experimental data bank had been used to verify and validate this numerical data. In this study, CFD calculation has been compared to previous published paper. The results have show the utility of this numerical simulation model as a rating and design tool for radiator manufacturers.

© Universiti Teknikal Malaysia Melaka

ABSTRAK

Kajian ini menampilkan satu set kajian parameter terhadap penukar haba yang menjurus kepada radiator automotif. Simulasi berangka telah digunakan dalam kajian ini untuk mengkaji pengaruh parameter terhadap radiator. Ramalan prestasi pemindahan haba menggunakan kod CFD. Simulasi tiga-dimensi tiub leper dan sirip telah dilaksanakan. Simulasi telah dijalankan pada nilai yang berbeza dengan kepelbagaian suhu udara masukan, jisim aliran air dan udara. Simulasi berangka ini telah menggunakan data daripada experimen untuk pengesahan data. Dalam kajian ini, pengiraan menggunakan CFD telah dibandingkan dengan jurnal kajian yang sebelumnya. Keputusan kajian ini akan menunjukkan kegunaan model simulasi berangka sebagai perkadaran dan bahan rekaan untuk pengilang radiator.

© Universiti Teknikal Malaysia Melaka

TABLE OF CONTENTS

CHAPTER	CONTENT	PAGE
	TESTIMONIAL	ii
	AKCKNOWLEDGMENT	iii
	ABSTRACT	iv
	ABSTRAK	v
	TABLE OF CONTENTS	vi
	LIST OF TABLE	ix
	LIST OF FIGURE	x
	LIST OF SYMBOL	xiii
	APPENDICES	xiv
CHAPTER 1	**INTRODUCTION**	
	1.1 Background	1
	1.2 Problem Definition	3
	1.3 Objective	4
	1.4 Scope	4
CHAPTER 2	**LITERATURE REVIEW**	
	2.1 Radiator	5
	2.2 Parameter Commonly Involved in Heat Exchanger Analysis	11
	2.2.1 Air and Coolant Mass Flow	11
	2.2.2 Air Inlet Temperature	13

© Universiti Teknikal Malaysia Melaka

2.2.3 Coolant Fluid
15
2.2.4 Fin Pitch
15
2.2.5 Louver Angle
18

CHAPTER **CONTENT** **PAGE**

2.2.6 Louver Pitch
19
2.2.7 Coolant Flow Lay-out 19
2.3 Numerical Analysis of Radiator Heat Exchanger 21

CHAPTER 3 METHODOLOGY

3.1 Introduction to Numerical Software 30
3.2 Flow Chart for Project 32
3.3 Mathematical Equation 33
3.4 Physical Domain 36
3.4.1 3-Dimensional Computational Domain 36
3.4.2 Air Flow 37
3.4.3 Fins 38
3.4.4 Tubes 38
3.4.5 Liquid/ Coolant 39
3.5 Boundary Condition 39
3.6 External CAD Drawing 40
3.6.1 Basic Concept of SolidWork 40
3.6.2 Sketching Concepts 40
3.6.3 Feature- based Models 41
3.6.4 Cutting the Features 44
3.7 The ANSYS Workbench 44
3.7.1 Design Modeler 45
3.7.2 Procedure in ANSYS Workbench 45
3.8 CFX-Mesh 47
3.8.1 Procedure in CFX meshes 47

© Universiti Teknikal Malaysia Melaka

3.9	CFX-Pre	51
3.10	Obtaining a Solution Using CFX Solver Manager	55
3.11	Viewing Result in CFX Post	56
	3.11.1 Creating a Streamline	56
	3.11.2 Adding Contour	56

CHAPTER	CONTENT	PAGE
CHAPTER 4	**RESULTS AND DISCUSSIONS**	58
	4.1 Air mass flow influence	63
	4.2 Air inlet temperature influence	65
CHAPTER 5	**CONCLUSION AND RECOMMENDATIONS**	69
	REFERENCES	71
	1) Journal	71
	2) Books	72
	BIBLIOGRAPHY	73
	APPENDICES	74

© Universiti Teknikal Malaysia Melaka

LIST OF TABLE

NO:	TITLE	PAGE
1	Baseline working conditions for automobile radiator (Source: Oliet et al. (2007))	39
2	The setting domain in CFX-Pre	52
3	Setting the boundary condition and boundary details	53

© Universiti Teknikal Malaysia Melaka

LIST OF FIGURE

NO:	TITLE	PAGE
2.1	Radiator and fin pitch (Source: Carluccio *et al*. (2005))	7
2.2	Flat-sided tube and louvered plate fin heat exchanger (Source: Malapure *et al*. (2007))	8
2.3	Air and coolant mass flow influence on the thermal and fluid-dynamic performance of the automotive radiator (Source: Oliet *et al*. (2007))	12
2.4	Air inlet temperature influence on the thermal and fluid-dynamic performance of studied radiator (Source: Oliet *et al*. (2007))	14
2.5	Geometry or air-side fin (Source: Carluccio *et al*. (2005))	16

© Universiti Teknikal Malaysia Melaka

2.6	Fin spacing influence on thermal and fluid-dynamic behaviour of heat exchanger (Source: Oliet *et al*. (2007))	17
2.7	Performance maps obtained for a parametric study fin pitch, F_p, in this case. On the left, heat transfer depends on air and coolant flow rates. At the right, overall enhancement versus air and coolant flow regime (Source: Oliet *et al*. (2007))	22
2.8	Schematization for the first run (Source: Carluccio *et al*. (2005))	23
2.9	Shell side surface mesh (near inlet plane). (Source: Witry *et al*. (2005))	24
2.10	Tube side surface mesh (near inlet). (Source: Witry *et al*. (2005))	24
2.11	The simulated set up (Source: Witry *et al.* (2005))	25
2.12	Dimensional detail with thickness 0.05mm,, tube size 16 x 2mm, longitudinal tube pitch 20mm and length of fin in airflow direction 21.6mm for single row and 41.6mm for double row (Source: Malapure *et al.* (2007))	27
2.13	3D computational domain and boundary condition (Source: Perrotin and Clodic (2004)).	29
3.1	3D computational domain (Source: Oliet *et al.* (2007))	37
3.2	The 3D geometry of louvered fin and half tube	43
3.3	Front-view of the geometry	43

© Universiti Teknikal Malaysia Melaka

3.4	Side-view of the geometry	43
3.5	After importing the geometry to the ANSYS Workbench and be frozen	46
3.6	Add material at tube and fin for the coolant and air flowing through it	46
3.7	After generating meshing	51
3.8	The meshing structure of the surface	51
3.9	The direction inlet and outlet of flow for coolant and air	54
3.10	The example of the solver result after run is completed	55
4.1	Pressure drop along the air flow direction for Air inlet temperature 40°C at air flow rate 0.4 kg/s	59
4.2	Temperature drop for air inlet at 40 ° C at air flow rate 0.4 kg/s	59
4.3	Velocity along fin for air inlet at 40 ° C at air flow rate 0.4 kg/s	60
4.4	Temperature drop for air mass flow rate at 0.08 kg/s and coolant mass flow rate at 0.69 kg/s	60
4.5	Pressure drop for air mass flow rate at 0.08 kg/s and coolant mass flow rate 0.69 kg/s	61
4.6	Velocity along fin for air mass flow rate at 0.08 kg/s and coolant mass flow rate 0.69 kg/s	61
4.7	Graph Air pressure drop versus air mass flow for air mass	

© Universiti Teknikal Malaysia Melaka

flow influence 63

4.8 Graph difference temperature versus air flow rate for air mass flow influence 64

4.9 Graph difference temperature versus air inlet temperature for air inlet influence 66

4.10 Graph air pressure drop versus air inlet temperature at air mass flow rate 0.08 kg/s 67

4.11 Graph air pressure drop versus air inlet temperature for air mass flow rate 0.4 kg/s 67

LIST OF SYMBOL

A	=	Area, m^2
C_p	=	constant pressure specific heat, (kJ/kg.K)
α	=	Louver angle (˚)
f	=	friction factor
F_p	=	fin pitch (mm)
h	=	convective coefficient (W/m^2K)
L_p	=	Louver pitch (mm)

© Universiti Teknikal Malaysia Melaka

$\dot{m}$	=	mass flow rate (kg/s)
Nu	=	Nusselt number
ρ	=	density (kg/s)
ΔP	=	pressure drop (Pa)
$\dot{Q}$	=	heat transfer rate or cooling capacity (kW)
R	=	radius (m)
Re	=	Reynolds number
T_p	=	Transverse pitch (mm)
S	=	strength (N/m^2)
U & V	=	velocity (m/s)
μ	=	viscosity (ms/kg)
$\dot{V}$	=	volume flow rate (m^3/s)

APPENDICES

NO:	TITLE	PAGE
A.	Part of Geometry Radiator and Geometry Description under Study	75
B.	Part of The Geometry that Involved in This Study	76
C.	The Pressure Contour as The Result of This Study	78

© Universiti Teknikal Malaysia Melaka

D. Project Planning for PSM 1 83

E. Project Planning for PSM 2 84

© Universiti Teknikal Malaysia Melaka

CHAPTER 1

Introduction

1.1 Background

A heat exchanger is a specialized device that assists in the transfer of heat from one fluid to the other. When a fluid is used to transfer heat, the fluid could be a liquid, such as water or oil, or could be moving air. This heat will transfer through the metal surface, while convection is used to transfer heat through the liquid. In some cases, a solid wall may separate the fluids and prevent them from mixing. In other designs, the fluids may be in direct contact with each other. In the most efficient heat exchangers, the surface area of the wall between the fluids is maximized while at once minimizing the fluid flow resistance. Fins or corrugations are sometimes used with the wall in order to increase the surface area and to induce turbulence.

There are various types of heat exchangers which are air-to-liquid cooling, liquid-to-air cooling, liquid-to-liquid cooling, or air-to-air cooling. With air-to-liquid cooling, heat is transferred from the air to a liquid. One example of air-to-liquid cooling is cabinet cooling. With liquid-to-air cooling, the heat is transferred from the liquid to the air. This type of cooling is generally used to cool process fluids. Liquid-to-liquid cooling is also used to cool process fluids, but the heat is removed by another liquid instead of by air. Lastly, with air-to-air cooling, heat is transferred from one air or gas stream to another.

© Universiti Teknikal Malaysia Melaka

For oil cooler flat tube heat exchangers have tubes and fin; however, the tubes are flat instead of round. This helps to minimize pressure drop when oil or ethylene glycol is used as the coolant. The surface area of the flat tubes is also much greater than the surface area of the tubes in a tube and fin heat exchanger. The additional surface area of the tubes in an oil cooler flat tube heat exchanger maximizes heat transfer when poor heat transfer fluids like oil or ethylene glycol are used. These oil cooler heat exchangers consist of fin, flat tubes, a welded header with inlets and outlets, and plates, including an optional fan plate.

Another type of heat exchanger is a plate-fin heat exchanger, which can provide air-to-air, air-to-liquid, liquid-to-air, or liquid-to-liquid cooling. Plate-fin heat exchangers consist of finned chambers separated by flat plates and are circuited in alternating hot and cold fluid passages. Heat is transferred via fins in the passageways, through the separator plate, and into the cold fluid via the separator plate, and into the cold fluid via fin once again. The heat exchanger also has manifold ducting, mounting brackets, and a frame. Liquid-to-liquid brazed plate heat exchangers also have plates, but with a herringbone pattern of groves, stacked in alternating directions. This forms separate flow channels for two liquid streams so that the two fluids are never in direct contact. The heat exchanger plates are brazed together at the edges and at a matrix of contact points between sheets. The liquid-to-liquid heat exchanger can be compared to a shell and tube heat exchanger, which is used in similar applications.

The thermal performance of heat exchanger technologies can vary quite a bit, so when selecting a heat exchanger it's important to understand what performance is needed as well as what fluids are available for removal of heat. It's also important to evaluate the entire system when deciding on a heat exchanger, as there are a number of considerations including flow rate, pressure drop, materials compatibility, and more.

Common appliances containing a heat exchanger include air conditioners, refrigerators, and space heaters. Heat exchangers are also used in chemical processing and power production. The most well known type of heat exchanger is a car radiator, which cools the hot radiator fluid by taking advantage of airflow over the surface of the

© Universiti Teknikal Malaysia Melaka

radiator. In radiator, a solution of water and ethylene glycol, will transfer heat from the engine to the radiator and then from the radiator to the ambient air flowing through it. The solution of water and ethylene glycol known as antifreeze. This process helps to keep a car's engine from overheating. Similarly, heat exchangers are designed to remove excess heat from aircraft engines, optics, x-ray tubes, lasers, power supplies, military equipment, and many other types of equipment that require cooling beyond what air-cooled heat sinks can provide.

1.2 Problem Definition

The automotive manufacturing is endlessly involved in a well-built competitive career to attain the best automobile design in multiple aspects (performance, fuel consumption, aesthetic, safety, etc.). The air-cooled heat exchanger creates in an automotive (radiator, AC-alternative current, condenser and evaporator, charge air cooler, etc.) cover a significant function in its weight and also in the design of its front-end module, which also has a strong impact on an automobile aerodynamic behavior. For these challenges, an optimization process is compulsory to obtain the best design negotiation between performance, size/shape, and weight. The approach of this study is oriented to computationally characterize the full heat exchanger including air and liquid side influences on overall performance which are cooling capacity and pressure drop. For the case of automobile radiator, the dominant thermal resistance generally occurs on the airside which is the reduction in the air-side thermal resistance will improve the heat transfer performance of the heat exchanger. The louvered fin are expensive when the manufactured by low-speed production techniques.

© Universiti Teknikal Malaysia Melaka

1.3 Objective:

The aim of this study is to characterize the heat exchanger influences on overall performance in radiator.

1.4 Scope:

The approach of this study is oriented to computationally characterize the full heat exchanger including air and liquid side influences on overall performance which are cooling capacity and pressure drop. This study is structured in a first part where the modeling technique and formulation are briefly summarized. The numerical analysis with complex mathematical model becomes the great solution to give detailed predictions on the effects that the heat transfer enhancing techniques and design which can have on the devices performance.

This study is limited:

i) To identify parameter that influences heat exchanger performance. This study is focusing on the influence of working condition on both fluids which are air and coolant mass flows air inlet temperature.
ii) To computationally simulate the effect of those parameter to heat exchanger performance. In this study, the CFX. Codes will be used.

© Universiti Teknikal Malaysia Melaka

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction to Radiator

In recent years a growing and intense attention has been turned to the study of concept compact heat exchanger which is application in automotive radiator. Since it represent a good solution in terms of dimension and efficiency for industrial application compared to the traditional approach. In automotive applications the use of compact heat exchanger helps in reaching high levels of temperature and pressure within severe constraints of dimensions. Manufacturing compact heat exchanger is worth as heat can be transferred using devices of limited dimensions (Carluccio *et al*.2005). According to Dittus *et al*. (1985), heat to be dissipated from water-cooled internal combustion engines is usually transferred to the atmosphere by means of devices commonly called radiator. Radiator is a component that control engine temperature in automobile where liquid to be cooled by air. Air is a medium to transfer heat to the surrounding. In radiator, cooling is made by means of air pushed to the heat exchanger by an external fan. Cihat and Feridum-Ozguc.A (2006) found that radiator is hotter than surrounding, where heat will transfer to the air and water exits at lower temperature. It was equipped with convection fins to improve heat output which is mostly by natural convection. The contribution of radiative heat transfer to the total heat transfer is smaller than the natural convection heat transfer. As found by Olsson and Sunden (1996), the medium conveying heat to radiator is water where heat is flowing from tube to liquid and medium conveying heat away is

© Universiti Teknikal Malaysia Melaka

air. Liquid will be flown in tubes while the air flow in channel set up by fin surfaces. The thermal resistance on the air is larger than on the liquid side. Besides, the previous study by Dittus and Boelter (1985) found that heat carried from warm water to colder tube by two processes which are convection and conduction. Inside the tube, fluid flow mostly turbulent. In the region of turbulent flow, most of the heat is transferred from the liquid to the tube wall by forced convection. As a consequence of the low thermal conductivity of fluids, very little heat is transferred from the center of the stream to the tube wall by conduction. In the viscous flow region, heat will transfer from interior stream to tube wall by conduction.

In order to improve the heat transfer surfaces applied the turbulence flow (Olsson and Sunden (1996)). The fluid flow is turbulent, unless the tube diameter is very small (Dittus and Boelter (1985)). The tubes dimension length can not be used very long because temperature different between bulk of fluid and wall at outlet measure with sufficient accuracy. The inlet tubes suspended with no nearby surface and at atmospheric pressure and no effect of distortion due to low pressure in tubes. Water flowing around the tube in cross flow with speed 1m/s and two-dimensional contraction placed upstream of tube to make water flow uniform. The heat transfer coefficient at the outer tube is larger than the inside wall. For the higher velocities, the maximum heat transfer rate appears at higher radius of tube. The variations of maximum heat transfer rate and optimum dimension as a function of radiator volume fraction for three different environment temperatures. The increase of radiator volume fraction increases the maximum heat transfer rate and optimum tube diameter (Olsson and Sunden, (1996)). For this study, the flow inside the tube was assumed turbulent flow. The variations of maximum heat transfer rate and optimum tube radius as a function of radiator volume fraction for three different ambient fluid temperatures. Cihat and Feridum-Ozguc (2006) found that the maximum heat transfer rate depends on ambient temperature but the corresponding optimum tube radius does not depend on this parameter.

© Universiti Teknikal Malaysia Melaka

Figure 2.1: Radiator and fin pitch (Source: Carluccio *et al*. (2005))

In addition, for the smooth tube exhibits the lowest friction factor. All the other tubes which are rib-roughened, dimpled and offset strip fin tubes have larger friction factors, *f*, at both laminar and turbulent. All the enhanced tubes have better heat transfer performance than the smooth tube. The dimpled tubes show a slightly better performance than the offset strip fin tube, while the smooth tube takes the lowest position. For the offset strip fin tubes, boundary layers are developed periodically I-short channel formed by the fins (Olsson and Sunden (1996)).

The rate heat from water to its air retarded by resistance on water side of tube surface, thermal resistance and resistance on air side of tube (Dittus and Boelter (1985)). Besides, the thermal resistance outside the tubes is less than the inside of tube so assume it negligible. The thermal resistance on the air is larger than on the liquid side. Flat tubes on liquid have small dimension to enhance heat transfer processes. For higher effectiveness the tubes made of brass and the fin made of aluminum (Olsson and Sunden

© Universiti Teknikal Malaysia Melaka

(1996)). Besides, the study by Malapure *et al*. (2007) found that the temperature of the fin decreases with the distance from the tube surface because of the thermal resistance of fin and carried away by the cooling air. Analysis by Dittus and Boelter (1985) on heat transfer in tubular radiator indicates that a large part of the total resistance to heat flow because the relatively low film transfer factors on the air side of the tubes. So that, any attempt to increase the overall heat transfer factor in radiator should begin by improving the film transfer factor on the air side of the tube. Some improvements might be also made by decreasing the film resistance on the liquid side of the tubes, although the total gain will be slight unless the air film transfer factor increases materially at the same time.

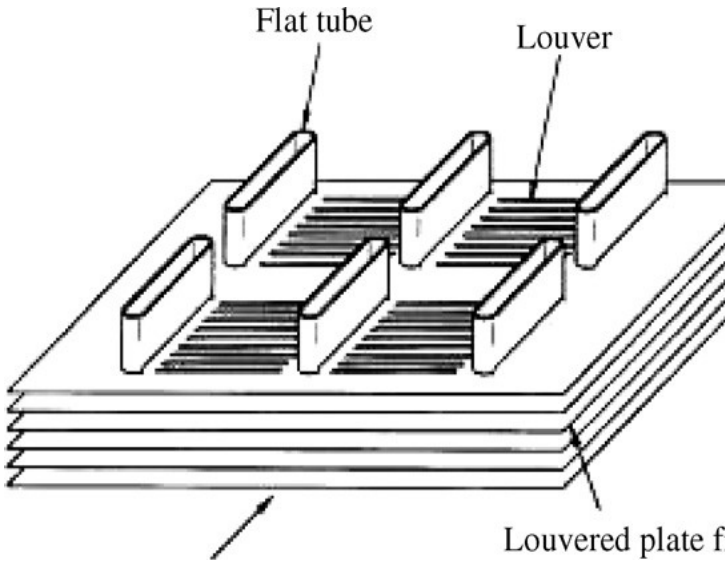

Figure 2.2: Flat-sided tube and louvered plate fin heat exchanger
(Source: Malapure *et al*. (2007))

While, the tube length influence on the Nusselt number, *Nu* is more pronounced in laminar flow than in turbulent flow. The difference in performance between the enhanced tubes will be decreased (Olsson and Sunden (1996)). Besides, the decreasing values of Nusselt number, *Nu,* with the distance from the entrance together with the decrease of the temperature difference between wall and air. The Nusselt number, *Nu,* with reference to the flow direction will be decreased of local heat transfer convective coefficient, *h,* going from the entrance to the exit and makes possible to understand the heat transfer coefficient behaviour at the direction changes (Carluccio *et.al.* (2005)).

© Universiti Teknikal Malaysia Melaka