# CFD SIMULATION OF A SHELL AND TUBE HEAT EXCHANGER; EFFECT OF PHYSICAL FEATURES



UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## CFD SIMULATION OF A SHELL AND TUBE HEAT EXCHANGER; EFFECT OF PHYSICAL FEATURES

# NUR FARAHIN BINTI MOHD AHADLIN



# UNIVERSITI TEKNIKAL MALAYSIA MELAKA

**JUNE 2017** 

## DECLARATION

I declare that this project report entitled "CFD Simulation of a Shell and Tube Heat Exchanger; Effect Of Physical Features" is the result of my own work except as cited in the references.



### **APPROVAL**

I hereby declare that I have read this project report and in my opinion this report is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering (Thermal-Fluids).



# DEDICATION

To my beloved mama and abah

Also to my lovely husband and cute son



### ABSTRACT

This project is mainly focusing on designing one type of a heat exchanger which is shell and tube heat exchanger with single segmental baffle. The process in solving simulation consists of modelling and meshing the basic geometry of shell and tube heat exchanger using CFD package ANSYS 16.0. The objective of the project is to perform the CFD analysis on behaviour of flow, investigate the effect of several number of baffle configuration and determine the heat transfer performance inside the shell using ANSYS software tools. Many considerations were taken to design this heat exchanger. The heat exchanger contains 7 tubes of 184 mm long and a shell with diameter of 50 mm. The baffle segment of segmental baffle varies from 2 to 4. Results show how the pressure, temperature and velocity variations depending on number of baffle segment. The flow behaviour in the shell side of the heat exchanger with single segmental baffles was forced to be zigzag manner due to the geometry of the segmental baffles, which results in a significant increase in heat transfer coefficient per unit pressure drop in the heat exchanger. The results obtained can be used to analyse which baffle number is better. Based on the overall calculated value in shell-side of the shell and tube heat exchanger in simulation, 4 baffle segmental number has the highest value of heat transfer rate at 294.24kW, pressure drop at 60.61kPa and overall heat transfer coefficient 2994.35W/ $m^2$ . K. As a result, shell and tube exchanger with 4 baffle segment is chosen as the best single segmental baffle compared to the other two baffles.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

#### ABSTRAK

Projek ini berkisar tentang salah satu rekabentuk penukar haba iaitu penukar haba cangkerang dan tiub dengan sesekat segmen tunggal. Proses menyelesaikan simulasi terdiri daripada pemodelan dan meshing geometri asas penukar haba cangkerang dan tiub menggunakan pakej CFD ANSYS 16.0. Objektif bagi projek ini adalah untuk melaksanakan analisis CFD mengenai tingkah laku aliran, mengkaji kesan berdasarkan bilangan konfigurasi sesekat dan menentukan prestasi pemindahan haba di dalam cangkerang menggunakan alat perisian CFD. Terdapat banyak pertimbangan yang telah diambil untuk merekabentuk penukar haba ini. Penukar haba ini mengandungi 7 tiub dengan 184mm panjang dan cangkerang dengan diameter 50mm. Jumlah segmen sesekat diubah antara 2 hingga 4. Hasil simulasi menunjukkan bagaimana variasi tekanan, suhu dan halaju bergantung kepada bilangan nombor segmen sesekat. Kelakuan aliran di sisi cangkerang penukar haba dengan sesekat segmen tunggal menghasilkan corak zigzag kerana geometri sesekat segmen, yang menyebabkan peningkatan ketara dalam pekali pemindahan haba per unit penurunan tekanan. Hasil yang diperoleh dari simulasi boleh digunakan untuk menganalisis jumlah sesekat berapa yang lebih baik. Berdasarkan nilai kiraan keseluruhan di bahagian sisi cangkerang dalam penukar haba cangkerang dan tiub, 4 segmen sesekat mempunyai nilai tertinggi kadar pemindahan haba 294.24kW, penurunan tekanan 60.61kPa dan pekali pemindahan haba keseluruhan 2994.35W /m<sup>2</sup>.K. Hasilnya, penukar haba cangkerang dan tiub dengan 4 segmen sesekat dipilih sebagai yang terbaik berbanding dua sesekat yang lain.

#### ACKNOWLEDGEMENT

#### Bismillahirrahmanirrahim,

Gratefully to Allah S.W.T because for His blessing, I am successful to complete my final year project. All of challenge and obstacle during develop this project were overcome properly. I would like to express my warmest gratitude to my supervisor, Dr. Fatimah Al-Zahrah bt Mohd Sa'at for her guidance, support, continuous encouragement and the confidence she has shown in me over the years in making this research possible. Her constant encouragement in my dissertation work that helped to look forward to the future with enthusiasm and confidence in my abilities. I would like also to express my sincere thanks to all my coursemates and members of the staff of the Mechanical Engineering Department, UTeM, who helped me in many ways and made my stay at UTeM pleasant and unforgettable.

I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream, prayer and sacrifice throughout my life. I cannot find the appropriate words that could properly describe my appreciation for their devotion, support and faith in my ability to attain my goals. To my brothers and sister who are really helpful in supporting me mentally.

Finally, I would like to thank my husband, for his support both emotionally and financially, in completing this endeavour. I sincerely appreciate his patience and understanding while waiting for me to complete my project. I would like to acknowledge his comments and suggestions, which was crucial for the successful completion of this study. Without his overwhelming positive influence on my project, I would not have been able to achieve my goals.

**TABLE OF CONTENT** 

CHAPTER	CON	ITENT	PAGE
	DEC	LARATION	ii
	APP	ROVAL	iii
	DED	DICATION	iv
	ABS	TRACT	v
	ABS	TRAK	vi
	ACK	KNOWLEDGEMENT	vii
	TAB	LE OF CONTENT	viii
	LIST	Γ OF TABLES	X
	LIST	Γ OF FIGURES	xii
	LIST	<b>FOF ABBREVIATIONS</b>	xiv
5	LIST	r of symbols	XV
CHAPTER 1	INTRODUCTION		1
00	1.1	Background	1
	1.2	Problem Statement	4
رك	1.3	Objective under the objective	5
	1.4	Scope Of Project	6
UNI	VER	SITI TEKNIKAL MALAYSIA MELAKA	
CHAPTER 2	LITI	ERATURE REVIEW	7
	2.1	What is Heat Exchanger?	7
		2.1.1 Classification of Heat Exchanger	9
		2.1.2 The Overall Heat Transfer Coefficient	10
		2.1.3 Analysis of Heat Exchangers	13
		2.1.4 The Log Mean Temperature Difference	13
		Method	
		2.1.5 The Effectiveness	14
	2.2	Shell and Tube Heat Exchanger	16
	2.3	Governing Equations	17

	2.4	CFD studies on Shell and Tube Heat	18
		Exchanger	
CHAPTER 3	МЕТ	THODOLOGY	27
	3.1	Introduction	27
	3.2	Research Method	29
	3.3	Geometric Configuration	31
	3.4	Mesh Generation	33
	3.5	Boundary Conditions	37
	3.6	Solver Settings	39
	3.7	Result Analysis	42
CHAPTER 4	RES	ULT AND DISCUSSION	43
2	4.1	Analysis of CFD Simulation	43
and the second se		4.1.1 Pressure Variations	45
TEX		4.1.2 Temperature Variations	51
E		4.1.3 Velocity Variations	57
	4.2	Calculation	60
sh	1 (	4.2.1 Calculation on Overall Heat Transfer	60
(		Coefficient	
LIN	IVER	4.2.2 Calculation on Heat Transfer Rate	62
ON		4.2.3 Calculation on Effectiveness-NTU	63
		Method	
		4.2.4 Calculation on Pressure Drop	63
		4.2.5 Calculation on Operating Cost	64
	4.3	Heat Transfer Performance	65
CHAPTER 5	CON	CLUSION AND RECOMMENDATIONS	70
	REF	ERENCE	72

# LIST OF TABLES

# TITLE

# PAGE

2.1	Comparison of simulation and experiment.	19
3.1	Structural parameters of the STHX with segmental	31
	baffles.	
3.2	Mesh details and metrics.	39
3.3	Data for design of shell and tube heat exchanger.	40
4.0	Graph of scaled residuals with different baffle number.	44
4.1	Average of surface vertex values static pressure.	46
4.1(a)	Single segmental baffle pressure contours; (a) 2 baffle	47
UNI	number. (b) 3 baffle number and (c) 4 baffle number VERSITI TEKNIKAL MALAYSIA MELAKA	
4.1(b)	Single segmental baffle pressure contours in shell side;	48
	(a) 2 baffle number. (b) 3 baffle number and (c) 4	
	baffle number	
4.1(c)	Single segmental baffle pressure contours in tube side;	50
	(a) 2 baffle number. (b) 3 baffle number and (c) 4	
	baffle number	
4.2	Average of surface vertex values static temperature.	52

4.2(a)	Single segmental baffle temperature contours; (a) 2 baffle number. (b) 3 baffle number and (c) 4 baffle number	53
4.2(b)	Single segmental baffle temperature contours in tube side; (a) 2 baffle number. (b) 3 baffle number and (c) 4 baffle number	54
4.2(c)	Single segmental baffle temperature contours in shell side; (a) 2 baffle number. (b) 3 baffle number and (c) 4 baffle number	56
4.3	Average of surface vertex values static velocity magnitude.	58
4.3(a)	Single segmental baffle velocity contours; (a) 2 baffle number. (b) 3 baffle number and (c) 4 baffle number	59
4.4	Overall calculated value in shell-side of shell and tube heat exchanger in simulation.	65
	UNIVERSITI TEKNIKAL MALAYSIA MELAKA	

## LIST OF FIGURES

FIGURE	TITLE	PAGE

1.1	Cross section of shell and tube heat exchanger.	1
1.2	U-tube heat exchanger.	2
1.3	Fixed-tubesheet heat exchanger.	2
1.4	Floating-head heat exchanger.	3
2.1	Classification according to process function.	9
2.2	Direction of flow.	11
2.3	Effectiveness for heat exchangers.	15
2.4	Particle velocity path lines.	21
2.5	Output for thermal analysis calculation by using C	22
	Program.	
2.6	The effect of sealers on the exergy coefficient.	23
2.7	The effect of sealers on shell-side pressure drops.	24
2.8	Residuals versus the predicted costs obtained by the	25
	selected ANN configuration.	
3.1	Flowchart of the Methodology.	30
3.2	Isometric view of shell and tube heat exchanger.	33

3.3(a)	Front view of shell side with four baffles segmental.	34
3.3(b)	Front view of shell side with three baffles segmental.	34
3.3(c)	Front view of shell side with two baffles segmental.	35
3.4	Isometric view of tube side with seven tubes.	35
3.5	Front view of triangular baffles arrangement.	36
3.6	Isometric view of shell and tube heat exchanger with label.	36
3.7	Isometric view of unstructured tetrahedral grid.	37
3.8	Baffle cut.	38
4.1	Pressure drop versus Reynolds number.	66
4.2	Overall heat transfer coefficient versus Reynolds number.	68
4.3	Heat transfer rate versus Reynolds number.	69
	UNIVERSITI TEKNIKAL MALAYSIA MELAKA	

## LIST OF ABBEREVATIONS

CFD Computational Fluid Domain Shell and Tube Heat Exchanger STHX Finite Element Analysis FEA ANN Artificial Neural Networks ORC Organic Rankine Cycles American Society of Heating, Refrigeration and Air-Conditioning ASHRAE Engineers TEMA Tubular Exchanger Manufacturers Association UNIVERSITI TEKNIKAL MALAYSIA MELAKA

## LIST OF SYMBOL

Е	=	Effectiveness, Turbulent dissipation energy
Т	=	Temperature
Δ	=	Increment
U	=	Overall Heat Transfer Coefficient
h	=	Enthalpy
R	=	Total Thermal Resistance
Α	=	Surface area
q	= (0)	Heat flux
Ż	=	Heat Transfer Rate
NTU	=	Number of Transfer Units
ρ	= L	DensityRSITI TEKNIKAL MALAYSIA MELAKA
LMTD	) =	Log mean temperature difference
k	=	Thermal conductivity, Turbulent kinetic energy
С	=	Specific heat
F	=	Fouling factor
L	=	Length
'n	=	Mass flow rate
n	=	Number of tubes

и	=	Velocity	
Re	=	Reynold number	
μ	=	Dynamic viscosity	
d	=	Diameter	
В	=	Baffle spacing	
С	=	Heat capacity rate	
Р	=	Pressure	
Nu	=	Nusselt number	



## **CHAPTER 1**

## **INTRODUCTION**

## 1.1 Background

Shell and tube heat exchanger (STHX) is one of the versatile type of heat exchanger design which transfer heat between fluids in a small space. This device is built to prevent the fluids from mixing by using a solid tube walls as shown in Figure 1.1. Shell and tube heat exchanger consists of a bundle of tubes placed inside a shell which is a normally made of large pressure vessel. The tube bundle is known as the series of tubes and commonly operated at high-pressure application. The process of transferring heat can happen in two ways; either absorb the heat or provide the heat between the two fluids. This is done by running the fluids through the tubes and another fluid flows through the shell.



UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Figure 1.1: Cross section of shell and tube heat exchanger (Cengel, 2015)

In addition, this device is further divided into three different categories; U-tube, Floating Head and Fixed Tube Sheet as shown in Figure 1.2 to 1.4. As the name implies, the tube of the U-tubes heat exchanger bent in the shape of U. Shell and tube heat exchanger with single pass tube side is free to 'float' within the shell is called as Floating Head. A Fixed-Tube Sheet is known as straight-tube passes that are secured at both ends to tube sheet welded to the shell.



Figure 1.3: Fixed-tubesheet heat exchanger (Mukherjee, 1998)



Figure 1.4: Floating-head heat exchanger (Mukherjee, 1998)

In order to achieve optimum heat transfer operation, the correct consideration can help when choosing the suitable shell and tube heat exchanger. Tube should be made of material with good thermal conductivity to be able to transfer heat well. In order to minimize corrosion, the tube material should be suitable with the fluids for long periods during heat transfer operation. The good calculation for flow distribution, heat transfer coefficient, pressure drop, temperature difference and surface area must be made to understand the performance of the system.

Shell and tube heat exchanger is widely used including in oil and gas industry application for cooling and heating fluids. The fluids can be gases or liquids, which hot fluids flow over the outside of the tubes and cool airs flow through the tubes to exchange the heat during the application process. This device is designed to improve the quality of serviceability to transfer large amount of heat and to provide an ideal heat exchanger solution for industrial needs. Shell and tube heat exchangers are available in many sizes to suit industrial operations. Shell and tube heat exchangers are commonly used in heat engineering tasks due to its efficiency, cost, construction material, available utilities, operating application and consideration for future expansions. The heat transfer performance of a shell and tube heat exchanger is very important because it is the yardstick in determining the ability of a heat exchanger to transfer heat from a fluid to pass to another fluid. There are various aspects to be assessed when determining the performance of shell and tube heat exchanger.

The shell and tube heat exchanger is separated by two pressure chambers (shell chamber and tube chamber) which is also known as non-fired pressure system. They will mutually exchange heat without mixing the fluids during the heat transfer process between a temperature differences. The case of single pass or multiple pass heat exchanger is depending on the requirement of effectiveness, pressure loss and speed.

#### **1.2 Problem Statement**

The main problems affecting the performance of the shell and tube heat exchanger are normally due to dead zones, fouling, leakage and tube vibrations. Areas that have minimum flow or non-existent flow is usually producing lower heat transfer and is called as 'dead zones'. Basically shell and tube heat exchanger use baffles to retain the required heat transfer. Besides that, fouling factor that affect heat exchanger performance when the surfaces of the heat exchanger experience corrosion. Corrosion occurs after long use due to interaction between the fluids and the materials used in the construction of the heat exchanger. The tube must be cleaned periodically to get the best performance in heat transfer process. In addition, overstressing of the rolled joints is caused by leakage at the tube to tube sheet joints of Fixed Tube Sheet exchangers. It can also cause differential thermal expansion between the tubes and the shell. Another problem that often arises in connection with the use of heat exchangers is tube vibration damage. Tube vibration damage can also occur in noncross flow implementations in the case of very high fluid velocities. Reducing the velocity can eliminate the tube vibration.

All problems listed above are very much depending on segmentation of baffles. There is various type of baffles segmentation we can found in the market. These arrangements will have different dimensions in example the thickness, length, diameter, layout and etc. Since there are different number of baffles segmentation, it's hard to know which one is able to produce high performance and case problems. These different segment will produce difference effect of flow. In order to find baffles segments with higher performance during operation, proper analysis need to be done.

## 1.3 Objectives

The objectives of this project are as follows:

# UNIVERSITI TEKNIKAL MALAYSIA MELAKA

- To perform the Computational Fluid Dynamics (CFD) analysis to simulate the behaviour of flow on shell and tube heat exchanger.
- 2. To investigate the effect of several number of baffles configurations of shell and tube heat exchanger
- 3. To determine the heat transfer performance of the heat exchanger

# 1.4 Scope of Project

The scopes of this project are:

- 1. The study of effect of the physical features on heat transfer performance of the heat exchanger.
- 2. CFD Simulation of the shell and tube heat exchanger is simulated only for several number of baffle segmentation.



#### CHAPTER 2

#### LITERATURE REVIEW

#### 2.1 What is Heat Exchanger?

According to Bagi (2012), a heat exchanger is a device that is used to transfer thermal energy (enthalpy) between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact. It is responsible in transferring heat typically from one medium to another. In many heat exchangers, the fluids are separated by a heat transfer surface, and ideally they do not mix or leak. They are widely used in space heating, refrigeration, air conditioning, and power plants. Heat exchangers play an important role in product quality, energy utilization, and systemic economy efficiency (Wang, 2015). Many parameters and several factors need to be considered when selecting or designing the heat exchangers, including thermal analysis, heat transfer rate, construction type, weight, size, pressure drop, materials, operating environment and cost. Economics play a key role in the design and selection of heat exchangers equipment. The weight and size of heat exchangers are significant parameters in the overall application and thus may still be considered as variables involved in economic evaluation.

There are three basic modes of heat transfer; conduction, convection and radiation. All three modes may occur simultaneously in problems of practical importance. Conduction is nearly involved in all operations in which heat transfer is taking place. Transfer of heat via conduction occurs through a solid surface that separates fluids having different temperatures. For transferring heat by the process of conduction, heat exchangers are the most common equipment used in process industries (Salahuddin, 2015). Different types of heat exchangers are used worldwide that differ from each other because of their specific requirements, such as double pipe, shell and tube, plate fin, plate and shell, pillow plate, etc. are a few types of heat exchangers used on an industrial scale.

In a counter-flow type, the hot and cold fluids enter the heat exchanger at opposite ends and flow in opposite directions, whereas in a parallel-flow type, both the hot and cold fluids enter the heat exchanger at the same end and move in the same direction. In compact heat exchangers, the two fluids move perpendicular to each other, and such a flow configuration is called cross flow. Other common type of heat exchangers in industrial application are the plate and the shell and tube heat exchangers.

UNIVERSITI TEKNIKAL MALAYSIA MELAKA



**UNIVERSITI TEKNIKAL MALAYSIA MELAKA** Figure 2.1: Classification according to process function (Karthikeyan, 2016)

## 2.1.1 Classification of Heat Exchanger

There are many differences of the heat exchanger according to construction features, heat transfer mechanisms, flow arrangements, transfer processes, surface compactness and number of fluids (Bhatt & Javhar, 2014). Each type of heat exchanger is specialized only for large scale industries. According to Maraie et. al (2016), this high degree of acceptance is due to the comparatively large ratio of heat transfer area to volume and weight, easy cleaning methods, easily replaceable parts etc. If heat exchanger is not applied in the right condition, the heat exchanger would not provide necessary beneficial when running the operation.

### 2.1.2 The Overall Heat Transfer Coefficient

Heat exchangers are used in the majority of chemical processes heat is either given out or absorbed, and in a very wide range of chemical plants, chemical engineers are involved in heating and cooling fluids. Thus, one of the major problems in furnaces, evaporators, distillation units, dryers, and reaction vessels is that of transferring heat at the desired rate. Alternatively, it may be necessary to prevent the loss of heat from a hot vessel or steam pipe. The key in chemical engineering practice is to control the flow of heat in the desired manner.

According to Cengel (2015), heat transfer in heat exchanger usually involves convection in each fluid and conduction through the wall separating the two fluids. In the analysis of heat exchangers, it is convenient to work with an overall heat transfer coefficient, U or a total thermal resistance, R expressed as:

$$\frac{1}{UA_s} = \frac{1}{U_i A_i} = \frac{1}{U_o A_o} = R = \frac{1}{h_i A_i} + R_{wall} + \frac{1}{h_o A_o}$$
(2.0)

Where subscripts i and o stand for the inner and outer surfaces of the wall that separates the two fluids, respectively. When the wall thickness of the tube is small and the thermal conductivity of the tube material is high, the relation simplifies to

$$\frac{1}{U} = \frac{1}{h_i} + \frac{1}{h_o} \tag{2.1}$$

12 11

(- - **)** 

Where  $U \approx U_i \approx U_o$ . The effect of fouling on both the inner and the outer surfaces of the tubes of a heat exchanger can be accounted for by

$$\frac{1}{UA_s} = \frac{1}{U_i A_i} = \frac{1}{U_o A_o} = R$$
(2.2)

$$= \frac{1}{h_i A_i} + \frac{R_{f,i}}{A_i} + \frac{\ln(D_o/D_i)}{2\pi kL} + \frac{R_{f,o}}{A_o} + \frac{1}{h_o A_o}$$

Where  $A_i = \pi D_i L$  and  $A_o = \pi D_o L$  are the areas of the inner and outer surfaces and outer surfaces and  $R_{f,i}$  and  $R_{f,o}$  are the fouling factors at those surfaces.



Tho = Hot fluid outlet temperature

Figure 2.2: Direction of flow.

It can be expected that the heat flux may be proportional to a driving force. In heat flow, the driving force is taken as  $T_h - T_c$  where  $T_h$  is the average temperature of the hot fluid and  $T_c$  is that of the cold fluid. The quantity  $T_h - T_c$  is the over-all temperature difference. It is denoted by $\Delta T$ . It is clear from Figure 2.2 that T can vary considerably from point to point along the tube, and therefore since the heat flux is proportional to  $\Delta T$ , the flux also varies with tube length. It is necessary to start with a differential equation, by focusing attention on a differential area, dA through which a differential heat flow, dQ occurs under the driving force of a local value of  $\Delta T$ . The local flux is then  $\frac{dq}{dA}$  and is related to the local value of  $\Delta T$  by the equation:

$$\frac{dq}{dA} = U\Delta T = U(T_h - T_c)$$
(2.3)

The overall heat transfer coefficient, U is defined by Equation (2.4) as a proportionality between  $\frac{dq}{dA}$  and  $\Delta T$ . Then, the theoretical overall heat transfer coefficient, U also can be calculated by the following equation:

$$\frac{UA}{UNIVERSITI TEKNIKACminIALAYSIA MELAKA}$$
(2.4)

$$U = \frac{NTU \times C_{min}}{A}$$

The assumptions for the calculation of overall heat transfer coefficient are

- 1. The overall heat transfer coefficient, U is constant.
- 2. The specific heats of the hot and cold fluids are constant.
- 3. Heat exchange with the surroundings is negligible.
- 4. The flow is steady.

#### 2.1.3 Analysis of Heat Exchangers

In a well-insulated heat exchanger, the rate of heat transfer from the hot fluid is equal to the rate of heat transfer to the cold one. That is,

$$\dot{Q} = \dot{m}_c c_{pc} (T_{c,out} - T_{c,in}) = C_c (T_{c,out} - T_{c,in})$$
(2.5)

And

$$\dot{Q} = \dot{m}_h c_{ph} (T_{h,out} - T_{h,in}) = C_h (T_{h,out} - T_{h,in})$$
 (2.6)

where the subscript c and h stand for the cold and hot fluids, respectively, and the product of the mass flow rate and the specific heat of a fluid  $\dot{m}c_p$  is called the heat capacity rate.

### 2.1.4 The Log Mean Temperature Difference Method

According to ASHRAE Handbook (2009), with heat transfer from one fluid to another (separated by a solid surface) flowing through a heat exchanger, the local temperature difference  $\Delta T$  varies along the flow path. Of the two methods used in the analysis of heat exchangers, the log mean temperature difference (LMTD) method is the best suited for determining the size of a heat exchanger when all the inlet and outlet temperatures are known. The effectiveness-NTU method is best suited to predict the outlet temperatures of the hot and cold fluid streams in a specified heat exchanger. In the LMTD method, heat transfer rate may be calculated using

$$\dot{Q} = UA_s \Delta T_{lm} \tag{2.7}$$

where *U* is the overall uniform heat transfer coefficient, *A* is the area associated with the coefficient, *U* and  $\Delta T_{lm}$  is the appropriate mean temperature difference. For a parallel or counter flow heat exchanger, the mean temperature difference is given by

$$\Delta T_{lm} = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1 / \Delta T_2)}$$
(2.8)

where  $\Delta T_1$  and  $\Delta T_2$  are temperature differences between the fluids at each end (inlet and outlet) of the heat exchanger. For the special case like multipass and cross flow shell and tube heat exchangers, the logarithmic mean temperature difference is related to the counter flow one  $\Delta T_{lm,CF}$  referred as:

$$\Delta T_{lm} = F \Delta T_{lm,CF} \tag{2.9}$$

where F is the correction factor, which depending on the heat exchanger geometry and the inlet and the outlet temperatures of the hot and cold fluid streams.

#### 2.1.5 The Effectiveness

Effectiveness is independent of exchanger inlet temperatures. For any exchanger in which  $c = c_{min}/c_{max}$  is zero (where one fluid undergoing a phase change, as in a condenser or evaporator, has an effective  $c_p = \infty$ ). The effectiveness of a heat exchanger is defined as

$$\varepsilon = \frac{\dot{Q}}{\dot{Q}_{max}} = \frac{Actual \ heat \ transfer \ rate}{Maximum \ possible \ heat \ transfer \ rate}$$
(2.10)

$$\varepsilon = C_h (T_{h,in} - T_{h,out}) / C_{min} (T_{h,in} - T_{c,in})$$
(2.11)

$$\varepsilon = C_c (T_{c,out} - T_{c,in}) / C_{min} (T_{h,in} - T_{c,in})$$
(2.12)

and

$$\dot{Q}_{max} = C_{min} (T_{h,in} - T_{c,in})$$
 (2.13)

where  $C_{min}$  is the smaller of the hot  $[C_h = \dot{m}_h c_{ph}]$  and cold  $[C_c = \dot{m}_c c_{pc}]$  fluid capacity rates, W/K;  $C_{max}$  is the larger. The effectiveness of heat exchangers can be determined from effectiveness relation or charts. Effectiveness chart is very useful when calculating the overall heat transfer coefficient, U.



Figure 2.3: Effectiveness for heat exchangers. (Cengel, 2011)

## 2.2 Shell and Tube Heat Exchanger

According to Chunangad (2003), more than 35–40% of heat exchangers are of the shell-and-tube heat exchangers (STHXs) type mainly due to their wide range of allowable design pressures and temperatures, their rugged mechanical construction, and ease of maintenance. Tubular Exchanger Manufacturers Association (TEMA) regularly publishes standards and design recommendations in 9th edition is published in 2007 (Standards of the Tubular Exchanger Manufacturers Association, 2007). STHX have been very successfully designed according to TEMA standards and using recommended correlation based on analytical approaches. These approaches have constantly been improved since the early days due to accumulating industrial experience and operational data, and improving instrumentation. The correlation based approaches can be used for sizing and can also be used iteratively to obtain general performance parameters (rating) of a heat exchanger (Ozden, 2009).

There are many advantages of shell and tube heat exchanger; Condensation or boiling heat transfer can be accommodated in either the tubes or the shell, and the orientation can be horizontal or vertical. Besides that, the pressures and pressure drops can be varied over a wide range and thermal stresses can be accommodated inexpensively. There are substantial flexibility regarding materials of construction to accommodate corrosion and other concerns. The shell and the tubes can be made of different materials. Next, the extended heat transfer surfaces (fins) can be used to enhance heat transfer. Cleaning and repair of shell and tube heat exchangers are relatively straightforward, because the equipment can be dismantled for this purpose. The robustness and medium weighted shape of shell and tube heat exchangers make them well suited for high pressure operations (Haran, Reddy and Sreeharihas, 2013).

# 2.3 Governing Equations

According to Yang, Zeng & Wang (2015), this turbulent flow model was based on the numerical solution of continuity, momentum, energy, k and  $\varepsilon$  equations in the computational domain. The equations are given in equations (2.14) to (2.21); Continuity equation:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{2.14}$$

where  $\rho$  is a fluid density and  $u_i$  is the flow velocity vector field at point x

Momentum equation:  

$$\frac{\partial}{\partial x_i}(\rho u_i u_k) = \frac{\partial}{\partial x_i}\left(\mu \frac{\partial u_k}{\partial x_i}\right) - \frac{\partial P}{\partial x_i}$$
(2.15)  
Energy equation:  

$$\frac{\partial}{\partial x_i}(\rho u_i t) = \frac{\partial}{\partial x_i}\left(\frac{\partial t}{\partial x_i} \frac{k}{c_p}\right)$$
(2.16)

Turbulent kinetic energy (k) equation: IKAL MALAYSIA MELAKA

$$\frac{\partial}{\partial x_i}(\rho u_i k) + \frac{\partial \rho k}{\partial t} = \frac{\partial}{\partial x_i} \left( \left( \mu + \frac{u_i}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right) + S_k$$
(2.17)

Turbulent dissipation energy ( $\varepsilon$ ) equation:

$$\frac{\partial}{\partial x_i}(\rho u_i \varepsilon) + \frac{\partial \rho \varepsilon}{\partial t} = \frac{\partial}{\partial x_i} \left( \left( \mu + \frac{u_i}{\sigma_{\varepsilon}} \right) \frac{\partial \varepsilon}{\partial x_i} \right) + S_{\varepsilon}$$
(2.18)

where the source terms  $S_k$  and  $S_{\varepsilon}$  are expressed as

$$S_k = \tau_{ij}^R \frac{\partial u_i}{\partial x_i} - \rho \varepsilon + u_t P_B \tag{2.19}$$

$$S_{\varepsilon} = C_{\varepsilon 1} \frac{\varepsilon}{k} \left( f_1 \tau_{ij}^R \frac{\partial u_i}{\partial x_i} + u_t C_B P_B \right) - C_{\varepsilon 2} f_2 \frac{\rho \varepsilon^2}{k}$$
(2.20)

where  $P_B$  represents the turbulence generation due to buoyancy forces and can be defined as

$$P_B = -\frac{g_i}{\sigma_B} \frac{1}{\rho} \frac{\partial \rho}{x_i}$$
(2.21)

where the subscript  $g_i$  is the component of gravitational acceleration in direction  $x_i$ .

## 2.4 CFD studies on Shell and Tube Heat Exchanger

Recent work by Bahirat (2014) uses ANSYS Fluent software for three dimensional CFD simulations to investigate heat transfer and fluid flow characteristics of two different baffle segment geometries with shell and tube heat exchangers. The numerical simulation of the shell and tube heat exchanger was performed by using a three dimensional (3D) numerical computation technique. Geometry of the model was created and meshed by using ANSYS Workbench software. To solve the equation for the fluid flow and heat transfer analysis ANSYS FLUENT was used in the shell and tube heat exchanger. After run using the calculation, fluent software calculates the different properties of interest and result was shown in the form of contour. Filled contour of velocity, pressure, temperature and kinetic energy are saved in the form of an image. Prandtl number, Nusselt number and Stanton number are also observed for calculating the heat transfer rate. Result can also be saved in form of Graphs and animations.

There are three methods in study of Fluid which are theory analysis, experiment and simulation (CFD). The computational fluid dynamic (CFD) is accurate, but slow and long time consuming compared to experiment are not informative. The experiments were very short time consuming than the CFD simulations. Although the accuracy of the experiment results is not as good as those of CFD. (Zuo & Chen,2009). The comparison between CFD and experiment are shown in Table 2.1

	Simulation (CFD)	Experiment	
Cost WALAYS	Cheap	Expensive	
Time	Short	Long	
Scale	Any	Small/Middle	
Information	تيڪنيڪل ملي	Measured Point	
Repeatable	TEKNIKAL MALAY	SIA MELAKA	
Safety	Yes	Some Dangerous	

Table 2.1: Comparison of Simulation and Experiment (Zuo, 2005).

According to Zhi et al (2009), the inlet and outlet duct geometry in an air to air compact heat exchanger is always irregular. Such duct placements usually lead to a nonuniform flow distribution on core surface. The author predicts the flow distribution and next calculated the heat exchange effectiveness and the thermal performance deterioration factor with finite difference scheme by using a CFD model. Experiments were performed to
validate the flow distribution and heat transfer model. The results indicated that the flow distribution is quite homogeneous when the channel pitch was below 2.0 mm, and the thermal deterioration due to flow maldistribution can be neglected. However, the maldistribution was quite large when the channel pitch was larger than 2 mm, and a 10–20% thermal deterioration factor could be found.

From recent study Ozden & Tari (2010), has worked on the design of shell and tube heat exchanger by numerically modelling it in particular the baffle spacing, baffle cut and shell diameter dependencies of heat transfer coefficient and pressure drop. The flow and temperature fields are resolved by using a commercial CFD package and it is performed for a single shell and single tube pass heat exchanger with a variable number of baffles and turbulent flow. The best turbulent model was selected and compared for heat transfer coefficient, outlet temperature and pressure drop with the Bell-Delaware method result. The effect of the baffle spacing to shell diameter ratio on the heat exchanger performance for two baffle cut values investigated by varying the flow rate. For the first and second order discretization to mesh density, the author takes three turbulence models. By comparison with the Bell-Delaware results, the  $k - \varepsilon$  realizable turbulence model was selected as the best simulation approach. The simulation results are compared with the results from the kern and Bell-Delaware methods by varying baffle spacing between 6 to 12, and the baffle cut values of 36% and 25% for 0.5 and 2 kg/s flow rate. The results showed that the CFD simulation results were in very good agreement with the Bell-Delaware methods. The differences between Bell-Delaware method and CFD simulations results of total heat transfer rate were below 2% for most of the cases.



Figure 2.4: Particle velocity path lines (Ender Ozden & Ilker Tari, 2010)

Haran, Reddy and Sreeharihas (2013) compared result of C Program and ANSYS and getting an error of 0.0274 in effectiveness. ANSYS CFD simulation was used to calculate the thermal analysis in less time and the analysis report were also almost accurate. In this paper the authors showed how to do the thermal analysis by using theoretical formula. The authors choose a practical problem of counter flow shell and tube heat exchanger of water and oil as working media. They used Pro-e to create the design model of shell and tube heat exchanger and do the thermal analysis. The authors also compared the result that obtained from ANSYS software and theoretical formula. The authors done a C code which is useful for calculating the thermal analysis of a counter flow of water-oil type shell and tube heat exchanger using simplified theoretical calculations.



Figure 2.5: Output for thermal analysis calculation by using C Program language. (Haran,

Reddy and Sreeharihas, 2013)

According to Wang, Wen & Li (2009), improvements of shell and tube heat exchanger can be made through the installation of sealers in the shell-side to improve heat transfer enhancement. This method is cheap, firm and convenient to install. Sealers effectively decreases the short-circuit flow in the shell-side and decrease the circular leakage flow. The heat transfer performance inside the heat exchanger intensifies the original short-circuit flow which participates in heat transfer. The results of heat transfer experiments showed that heat transfer coefficient of the improved heat exchanger increased by 18.2–25.5%, the overall coefficient of heat transfer increased by 15.6–19.7%, and the energy efficiency increased by 12.9–14.1% on the shell-side of the shell and tube heat exchanger. Increment of pressure losses by 44.6–48.8% with the sealer installation, the energy utilization improves, which follow the significance of the optimum design to the shell-and-tube heat exchanger. The effect of baffle-shell leakage flow in tube-and-shell heat exchanger can be settled by putting the sealers on shell and tube heat exchanger. Improvement of this convectional shell and tube heat exchanger on heat transfer performance is increased, which is a benefit for optimizing of heat exchanger design.



Figure 2.6: The effect of sealers on the exergy coefficient. (Wang, Wen & Li, 2009)



Figure 2.7: The effect of sealers on shell-side pressure drops. (Wang, Wen & Li, 2009)

Duran et al (2009) presented in their technical paper to develop and test a model of cost estimation for the shell and tube heat exchangers in the early design phase via the application of Artificial Neural Networks (ANN). An ANN model can help the designers to make decisions at the early phases of the design process of shell and tube heat exchanger. It is possible to obtain a fairly accurate prediction with an ANN model, even when enough and adequate information are not available in the early stages of the design process. This ANN model proved that neural networks are capable of reducing uncertainties related to the cost estimation of a shell and tube heat exchangers.



Figure 2.8: Residuals versus the predicted costs obtained by the selected ANN

configuration (Duran et al, 2009).

Gaddis and Gnielinski (1997) presented the procedure for evaluating the shell side pressure drop in shell-and-tube heat exchangers with segmental baffles. The implementation is in view of correlations for figuring the pressure drop in a perfect gas tube bank coupled for correction factors, which consider the impact of spillage and bypass streams, and on equations to ascertaining those pressure drop in a window segment from the Delaware system.

In previous research Walvaren, Laenen & D'haeseleer (2014) compared the result between shell and tube with plate heat exchangers for the use in low-temperature organic Rankine cycles (ORC). The plate heat exchangers are generally known that it could reach higher heat-transfer coefficients and therefore lower pinch-point-temperature differences. The results showed that heat-exchanger surface increases with increasing the efficiency of an ORC and a heat-source inlet temperature for every fluid in maximal level.



## **CHAPTER 3**

#### METHODOLOGY

#### 3.1 Introduction

This chapter describes the methodology used in this project to obtain the analysis data that effect physical features of different segmentation of baffles using CFD. Due to the advances in computational hardware and available numerical methods, CFD is a powerful tool for the prediction of the fluid motion in various situations, thus, enabling a proper design. CFD is a sophisticated way to analyse not only the fluid flow behaviour but also the processes of heat and mass transfer. There is a variety number of baffles segment used in shell and tube heat exchanger. Each baffle is design with distinctive capabilities in producing a good thermo-hydraulic performance during operation. Income inequality in performance is due to several factors such as the size of the tubes, surface area of shell, tube material and so on. When designing a shell and tube heat exchanger, baffle segment selection is very important. This is to ensure that the resulting shell-side flow distribution, heat transfer coefficient and the pressure drop of shell and tube heat exchanger must be operated at optimal conditions as possible.

This chapter shows how the process of study to determine which tube and baffles arrangement have the best performance if use in heat exchanger by using CFD The analysis can be carried out by doing ANSYS CFD Fluent simulation in two dimensional drawing on the selected segmental baffles with a fixed tube length and constant velocity of fluids. Results obtained from the analysis such as velocity and pressure distribution, heat transfer coefficient and mean logarithmic temperature difference is going to be used to determine which shell and tube heat exchanger have better performance.

To assist in the selection of the correct baffles segmentation, information on the performance level of the shell and tube heat exchanger must be known. The steps illustrated in the Figure 3.1. The computational techniques for CFD simulation involve pre-processing (geometry creation, geometry modification, meshing and solver), solution and post-processing are carried out in an ANSYS 16.0, Workbench environment with an ANSYS Fluent system. The computer run based on number of iterations set by user choose. This number needs to be large enough to allow the solution to essentially stop changing between iterations. Generally, in the CFD simulation, the required steps as shown in Figure 3.1 in performing analysis involves:

- 1) Develop or import geometry configuration
- 2) Generate a mesh generation
- 3) Set up the boundary conditions
- 4) Execute the solver TEKNIKAL MALAYSIA MELAKA
- 5) Model validation and verification
- 6) Visualize the result in a post processor panel

The simple calculation of the best number of segmental baffles effectiveness is calculated after run the simulation by using ANSYS FLUENT.

## **3.2** Research Method

The information has been gathered from journal articles, books, manual, lecture notes, internet and lecturer to build a flow chart of project implementation. The information that has been collected is gains from terms in the title. Set of equipment proposed in this analysis consists a computer/laptop (high processor ability if possible) which has completed installed ANSYS 16.0 software has been used for the purpose of model generation and its further analysis.

The steps are expressed in Figure 3.1 and the work started by gathering related information. The information gained then used as the literature review and in the same time the type of baffle segment is selected. The structural parameter of the shell and tube heat exchanger with segmental baffles is collected. This information is very important during construction the shell and tube heat exchanger in the ANSYS CFD. The simulation then run according to the parameters that has been set. The results obtained needed to be evaluated in order to determine whether the result are acceptable or not. The evaluation is based on the information related to this analysis that have been obtained before. If acceptable, analysis and discussion will be done to determine the best number of baffle segmentation and if not acceptable the simulation need to be run again until acceptable results are obtained.



Figure 3.1: Flowchart of the Methodology

## **3.3** Geometric Configuration

Geometry was created using ANSYS Design Modeler software specifically designed for the creation and preparation of a geometry for simulation. A domain has to be built in shell and tube heat exchanger to study heat transfer coefficient, mass flow rate, pressure drop and flow which baffle segment have lower the operating cost of heat exchanger. The geometry modelling in this study is done by using Design Modeler 16.0. Two different number of baffles are drawn according to the parameter shown in Table 3.1.

2	20		
and the second s	Material	Stainless steel	
X	A.		
F	Tube internal diameter	4 mm	
E			
age .	Tube external diameter	6 mm	
AIND			
del (	Tube arrangement	Triangular	
2) John	and Sine	au, in	ويتومر
4	Tube number		1
	NTI TEKNIKAL M	AL AMOLA N	
UNIVER	Tube effective length	184 mm	ELAN
	Shell internal diameter	44 mm	
	Shell external diameter	50 mm	
	Baffle number	4	
	<b>D</b> 001	2.2.6	
	Baffle spacing	35.6 mm	
		1	
	Battle thickness	1 mm	
	Deffle aut	220/	
	Barne cut	~22%0	

Table 3.1: Structural parameters of the STHX with segmental baffles, (Maakoul, Laknizi et al, 2016)

Before the model is drawn, the information on the shell and tube heat exchanger need to be obtained first. This information is consisting of collecting parameter data on the baffle segment such as tube length, shell diameter, tube diameter and etc. As can be seen from Figure 3.2, the simulated STHX has four baffles number in the shell side direction with total seven number of tubes with a spacing of 35.6 mm and the baffles thickness is 1 mm. The whole computation domain is bounded by the wall of the shell in a counter current configuration contains cool water flow and the warm water in the tube side. The inlet and outlet of the domain are connected with the corresponding tubes. The shell has an internal diameter of 44 mm, an external diameter of 50 mm, a thickness of 3 mm, and a length of 200 mm. The tube is completed with an external diameter of 6 mm, an internal diameter is 4 mm, an effective length of 184 mm and the tubes are installed inside the shell with triangular arrangement as shown in Figure 3.5. Process collecting the baffle segment coordinate by sketching the baffle segment on the graph paper. This coordinate is then used to plot the construction point in the CFD software.

Some basic characteristics of the process following assumptions are made to simplify numerical simulation:

- 1. The shell side fluid was constant thermal properties.
- 2. The fluid flow and heat transfer processes are turbulent and in steady state.

3. The leak flows between tube and baffle and that between baffles and shell are neglected.

- 4. The natural convection induced by the fluid density variation is neglected.
- 5. The tube wall temperature is kept constant in the whole shell side.

6. The heat exchanger is well insulated hence the heat loss to the environment is totally neglected.

The shell is drawn from XY plane, which the X-Y-Z axis (as origin). The shell side, then drawn on left (Hot inlet), right (Hot outlet), bottom (Cold outlet) and top (Cold inlet) of the shell. After that, the tube then drawn again inside the shell by adding the plane offset Z from XY plane. During extrude process, the cylinder undergoes 'Add Frozen' process to make sure that the shell and tube are suppressed from the water to create a single body domain for the purpose of the CFD simulation. The suppressed body represents the wall of shell and tube heat exchanger around the surrounding water.

As mentioned above, shell and tube heat exchanger with three number of baffle segments are modelled. Using the above derived dimensions of shell, tubes and baffles a software model using ANSYS 16.0 was developed. The parts individually as well as in assembly are as shown in Figure 3.2 to 3.5



Figure 3.2: Isometric view of shell and tube heat exchanger.



Figure 3.3(a): Front view of shell side with four baffles segmental.



Figure 3.3(b): Front view of shell side with three baffles segmental.



Figure 3.3(c): Front view of shell side with two baffles segmental.



Figure 3.4: Isometric view of tube side with seven tubes.



Figure 3.6: Isometric view of shell and tube heat exchanger with label. 36

## 3.4 Mesh Generation

Meshing is a process where the geometry is divided into a thousand cells. It is called sub-domain. All sub-domains are actually non-overlapping sub-divisions. Objective of doing this is because the solution of the problems is defined in the nodes inside each cell. Therefore, the grid will affect the accuracy of the results. Meshing is the most crucial part in CFD simulations because the mesh quality will affect the computational results. Smoother the grid means more accurate the result will be. The type of meshing is set to 'AUTO'. This to ensure that all number of baffles segments will have the same meshing type. The next mesh setting is to set up the sizing. This setting function is to smoothing the grid cell. Smoother grid cell will produce better results, but it requires higher computer processing ability.



Figure 3.7: Isometric view of unstructured tetrahedral grid. 37



As mentioned above, for each of two studied shell and tube heat exchangers, two domains are defined, two fluid domains which are water in the tube and shell sides and the solid domain are tubes bundle and baffles. The meshing approach used in this computational domain is unstructured tetrahedral grid which are generated by using the ANSYS MESHING tool. The quality of the mesh for each shell and tube heat exchanger including skewness, orthogonal quality and elements quality was evaluated using the built-in Mesh Metrics in ANSYS MESHING as tabulated in Table 3.2. One of the most important features that determines the quality of mesh is mesh skewness.

Segmental	Nodes	Elements	Average	Average	Average
Baffles Type			Skewness	Element	Orthogonal
				Quality	Quality
Single with 4 baffles	744248	3708741	0.21371	0.84553	0.86369
Single with 3 baffles	265947	1350572	0.22513	0.83937	0.86040
Single with 2 baffles	264968	1349701	0.22544	0.83914	0.86016

Table 3.2: Mesh details and metrics

#### **3.5 Boundary Conditions**

The boundary conditions used in this research are standard conditions. The material for the shell and tube are stainless steel. The momentum boundary condition of no penetration and no slip is set for all the solid walls. The thermal boundary condition of zero heat flux is set for the shell wall, inlet and outlet nozzle walls, while the walls of tubes, baffles and tube bundle which also point out the solid-fluid interfaces between the two fluid domains which are water in the shell and the tube side and the solid domain which is tube bundle and baffles, have the thermal boundary condition of coupling heat transfer. The outlet for the tube and shell side are set as boundary conditions of pressure-outlet, the inlets are set as velocity-inlet. The inlet pressure is equal to the pressure drop on both shell and tube sides because the outlets are assumed to have no pressure of zero.

	PROPERTY	VALUE	
	Shell side fluid		
	T <sub>hot,inlet</sub>	90°C	
	T <sub>hot,outlet</sub>	70°C	
	Density	971.8kg/m <sup>3</sup>	
	Specific heat capacity	4.1963 <i>kJ/kg</i> . <i>K</i>	
	Viscosity	0.354mPa.s	
	Conductivity	0.67W/m.K	
MAL	Fouling factor	$0.0002m^2K/W$	
KIIIK	Flow rate	0.3kg/s	
TT II			$\mathbf{V}$
PAINER	Tube side fluid-	cold water	
J ملاك	T <sub>cold,outlet</sub>	30°C	ونبؤم
	T <sub>cold,inlet</sub>		
UNIVER	Density	984kg/m <sup>3</sup>	
	Specific heat capacity	4.178kJ/kg.K	
	Viscosity	0.725mPa.s	
	Conductivity	0.623W/m.K	
	Fouling factor	$0.0002m^2K/W$	
	Flow rate	0.7533kg/s	

Table 3.3: Data for design of shell and tube heat exchanger

## 3.6 Solver Settings

Analysis type 'Steady State' is selected under the set-up of Fluent of the ANSYS Workbench, which is the heat transfer and fluid flow are turbulence. Activate the 'Energy' model and the viscous setting is changed to 'Realizable  $k - \varepsilon$ '. Realizable  $k - \varepsilon$  turbulence model can provide higher in rank performance for flows involving rotation. The 'scalable wall function' will be utilized to numerical computational close the wall will evade issues about progressive refinements to standard wall function meshes. The appropriate boundary conditions were applied to the domains. Define the type of fluid use after settings the solver. Materials 'Steel' and 'Water' were assigned under the 'Cell Zone Conditions'. Water is considered as incompressible fluid and Newtonian with constant thermo-physical properties (Maakoul et al, 2016).

ANSYS is a general-purpose finite element analysis (FEA) software package. Finitevolume formulation with simple algorithm is a solution for the governing equations. The software implements equations that govern the behaviour of these elements and solves them all. The Second Order Upwind were chosen for this analysis, which is very useful for momentum, energy, turbulence and its dissipation rate. Second order is more difficult to converge but it is more accurate than first order. These results can be presented in tabulated or graphical forms. After that, the number of iterations is set with a large value range from 0 to 7000 to enable the solution to essentially stop changing between iterations.

## 3.7 Result Analysis

The results are going to be examined which is the post processing step when solution converged. Temperature distribution and heat flux as well as parameters like Nusselt Number, heat transfer coefficient and changes in other parameters can also be predicted by computational analysis. Result analysis will be presented on various number of baffles segment in term of its effect of physical features on performance. Solutions will be proposed based on the analysis of the shell and tube heat exchanger performance. CFD simulations of pressure, temperature and velocity profile in STHX with different number of baffle configurations have been discussed. Overall heat transfer coefficient and heat transfer rate has been compared to conclude the best design parameters. For the solution method, the hybrid initialization is going to be a suitable approach with respect to the FLUENT for initializing the velocity and temperature (Haolin Ma,2012). Once the solution has been initialized, the project is ready to be simulated. The pressure and velocity contour of each segment need to be analyse and related to the theoretical conditions. The result from the analysis is going to be use to make recommendation on which number of baffles segment has the highest performance.

## **CHAPTER 4**

#### **RESULTS AND ANALYSIS**

#### 4.1 Analysis On CFD Simulation

After complete the simulation for all design, the result on CFD Simulations of pressure, temperature and velocity profile across STHX with segmental baffle configurations have been presented. Fluid flow distribution in the shell and tube heat exchanger is analysed with single segmental baffle as discussed. As the main heat exchanging units of a shell and tube heat exchanger, baffles play a key role in heat transfer. Distribution of fluid flow is determined by analysis of heat transfer performance of the STHX in the flow direction. In addition, the baffles segmentation with different baffles number in the flow direction will affect the distribution of pressure, temperature and velocity inside the shell and tube heat exchanger. The solution of flow equations is converged after approximately 2752 iterations for 2 baffle number, 1016 iterations for 3 baffle number and 6541 iterations for 4 baffle number when the maximum residual value of 1.0e-4 was assigned as shown in Table 4.0.



Table 4.0: Graph of Scaled Residuals with Different Baffle Number.



## 4.1.1 Pressure Variations with Segmental Baffle Configuration in Shell and Tube Heat Exchanger (STHX)

Table 4.1 shows the average of surface vertex values static pressure inside shell and tube heat exchanger with different baffle number. Based on the data, with every increase in baffle number, the pressure decline inside the shell from inlet to outlet. The difference between the lowest and the highest pressure for 2 baffle number is 98.82% followed by 3 and 4 by 99.83% and 99.94% respectively. Static pressure of the three baffle numbers seemed to be identical.

Baffle Number	Surface	Pressure (Pa)
2	Cold-inlet	0.7956
	Cold-outlet	0.0049
	Hot-inlet	0.4152
	Hot-outlet	0.0590
3	Cold-inlet	3.0149
	Cold-outlet	0.0051
	Hot-inlet	2.0880
	Hot-outlet	0.0494
4	Cold-inlet	1.6525
	Cold-outlet	0.0012
MALAYSI	Hot-inlet	2.3390
and and a second se	Hot-outlet	0.0503
8		

Table 4.1: Average of Surface Vertex Values Static Pressure.

The pressure variation on the baffle segments was obtained at different conditions are shown. Table 4.1(a), 4.1(b) and 4.1(c) depicts the pressure distribution across STHX on single segmental baffles at different number of baffle segment. It can be seen that flow distribution on the shell side of STHX with baffle segment is zigzag pattern between the tube bundles because it has high pressure drop. High pressure state at the baffles opening and large fluid recirculation. The pumping cost are particularly linked in pressure drop. The lower the pressure drop, the lower the operating costs. The pressure drop is highly related in designing the shell and tube heat exchanger.

Table 4.1(a): Single segmental baffle pressure contours; (a) 2 baffle number. (b) 3 baffle number and (c) 4 baffle number.





WALAYSIA

Table 4.1(b): Single segmental baffle pressure contours in shell side; (a) 2 baffle number.



(b) 3 baffle number and (c) 4 baffle number.



Table 4.1(c): Single segmental baffle pressure contours in tube side; (a) 2 baffle number.



(b) 3 baffle number and (c) 4 baffle number.



# 4.1.2 Temperature Variations with Segmental Baffle Configuration in Shell and Tube Heat Exchanger (STHX)

Three different baffle segment of the average of surface vertex values static temperature are tabulated in comparison with the baffle number at 2, 3 and 4. Comparing baffle number 2 with 3 and 4 shows that, the baffle segment will affect the temperature distribution of the fluid. Based on the Table 4.2, with every increase in baffle number, the temperature incline inside the tube from inlet to outlet for cold water while the temperature decline inside the shell from inlet to outlet for hot water. Temperature of the hot water at inlet between 351.82K to 363.15K with decreasing at outlet.

Baffle Number	Surface	Temperature (K)
2	Cold-inlet	303.15
	Cold-outlet	340.86
	Hot-inlet	351.82
	Hot-outlet	304.86
3	Cold-inlet	303.15
	Cold-outlet	312.51
	Hot-inlet	351.82
	Hot-outlet	327.78
4	Cold-inlet	303.15
	Cold-outlet	320.58
MALAYSIA 4	Hot-inlet	363.15
N. N	Hot-outlet	327.78

Table 4.2: Average of Surface Vertex Values Static Temperature.

Table 4.2(a), 4.2(b), and 4.2(c) represent the contour temperature distribution of fluid (water-liquid) produced within STHX on single segmental baffles for three various baffle number. The baffles arrangement will affect the temperature distribution of the fluid which result in zigzag pattern in the shell side of heat exchanger. This proven that the temperature variation is shown on the shell and tube heat exchanger using a colour scale. For this case study, the red colour on the contour represent the hottest temperature whereas the blue colour represent the coolest temperature. In conclusion, the temperature contour influenced by the baffle segmentation and the result clearly shows that the 4 baffle number is the best baffle segmentation that dissipate heat mostly.

Table 4.1(a): Single segmental baffle temperature contours; (a) 2 baffle number. (b) 3



baffle number and (c) 4 baffle number.



MALAYSIA

Table 4.1(b): Single segmental baffle temperature contours in tube side; (a) 2 baffle






ANSYS Temperature Contour 1 3.631e+002 3.571e+002 3.511e+002 3.451e+002 3.391e+002 3.331e+002 3.271e+002 3.211e+002 3.151e+002 3.091e+002 (a) 3.031e+002 [K] 0.045 0.090 (m) WALAYSIA 0.0225 0.067 Temperature Contour 1 ANSYS 3.631e+002 3.571e+002 3.511e+002 3.451e+002 3.391e+002 3.331e+002 3.271e+002 3.211e+002 (b) 3.151e+002 3.091e+002 LAKA 3.031e+002 [K] 0.050 0.100 (m) 0.075 0.025

Table 4.1(c): Single segmental baffle temperature contours in shell side; (a) 2 baffle

number. (b) 3 baffle number and (c) 4 baffle number.



# 4.1.3 Velocity Variations with Segmental Baffle Configuration in Shell and Tube Heat Exchanger (STHX)

AALAYS/A

Table 4.1 shows the average of surface vertex values static velocity magnitude inside shell and tube heat exchanger with different baffle number. Based on the data, with every increase in baffle number, the velocity increase inside the shell from inlet to outlet for cold water. The difference between the lowest and the highest velocity for 2 baffle number is 14.09% followed by 3 and 4 by 8.31% and 8.22% respectively. Static velocity of the three baffle numbers seemed to be similar.

Baffle Number	Surface	Velocity (m/s)	
2	Cold-inlet	0.5122	
	Cold-outlet	0.5505	
	Hot-inlet	0.5581	
	Hot-outlet	0.5962	
3	Cold-inlet	0.5122	
	Cold-outlet	0.5586	
	Hot-inlet	0.5581	
	Hot-outlet	0.5394	
4	Cold-inlet	0.5122	
	Cold-outlet	0.5512	
	Hot-inlet	0.5581	
	Hot-outlet	0.5400	
×	>		

Table 4.3: Average of Surface Vertex Values Static Velocity Magnitude.

From Table 4.3(a) demonstrate the flow distribution of the velocity profile of shell and tube heat exchanger at different number of baffle. The single segmental baffle heat exchanger is modeled considering the plane symmetry. From this velocity contour, it is inferred that zigzag manner between tube bundles in the shell side enhances the local mixing and turbulent intensity. The blue colour of the vector on cold water at inlet displays the lowest state of velocity.

Table 4.3(a): Single segmental baffle velocity contours; (a) 2 baffle number. (b) 3 baffle number and (c) 4 baffle number.





4.1.4 Calculation on Overall Heat Transfer Coefficient of STHX The properties of water at 81°C through tubes are (Table A-9)  $\rho = 971.06 \, kg/m^3$  Pr = 2.192 $k = 0.6706 \, W/m. K$   $v = \mu/\rho = 3.6105 \times 10^{-7} \, m^2/s$ 

The properties of water at 39°C through shell are (Table A-9)

$$\rho = 992.48 \, kg/m^3$$
  $Pr = 4.422$   
 $k = 0.6294 \, W/m.K$   $v = \mu/\rho = 6.7145 \, m^2/s$ 

The average velocity of water in the tube and the Reynold number are

$$V = \frac{\dot{m}}{\rho n A_c} = \frac{\dot{m}}{\rho (7) (\frac{1}{4} \pi D_i^2)} = \frac{1 \, kg/s}{(971.06 \, kg/m^3) (7) \left[\frac{1}{4} \pi (0.004m)^2\right]} = 11.707 \, m/s$$

And

$$Re = \frac{VD_i}{v} = \frac{(11.707 \ m/s)(0.004m)}{3.6105 \times 10^{-7} \ m^2/s} = 129,699$$

Which is greater than 10,000. Therefore, the flow of water is turbulent. Assuming the flow to be fully developed, the Nusselt number can be determined from

$$Nu = \frac{hD}{k} = 0.023Re^{0.8}Pr^{0.4} = 0.023(129,699)^{0.8}(2.192)^{0.4} = 387.63$$

Then,

$$h = \frac{k}{D_i} Nu = \frac{(0.6706 \ W/m.\ K)(387.63)}{0.004m} = 64,985.76 \ W/m^2.\ K$$

Now, we repeat the analysis. The hydraulic diameter for annular space is

$$D_h = D_o - D_i = 0.044 - 0.004 = 0.04m$$

The average velocity and the Reynold number in this case are

$$V = \frac{\dot{m}}{\rho A_c} = \frac{\dot{m}}{\rho \left[\frac{1}{4}\pi (D_o^2 - D_i^2)\right]} = \frac{1 \, kg/s}{(992.48 \, kg/m^3) \left[\frac{1}{4}\pi (0.044^2 - 0.004^2)\right]}$$
$$= 0.6682 \, m/s$$

And

$$Re = \frac{VD}{v} = \frac{(0.6682 \, m/s)(0.04m)}{6.7145 \times 10^{-7} \, m^2/s} = 39,805$$

Which is greater than 10,000. Therefore, the flow of water is turbulent. Assuming the flow to be fully developed, the Nusselt number can be determined from

$$Nu = \frac{hD}{k} = 0.023Re^{0.8}Pr^{0.4} = 0.023(39,805)^{0.8}(4.422)^{0.4} = 199.49$$

Then,

$$h = \frac{k}{D}Nu = \frac{(0.6294 \ W/m.\ K)(199.49)}{0.04m} = 3138.98 \ W/m^2.\ K$$

Then, the overall heat transfer coefficient for this STHX becomes

$$U = \frac{1}{\frac{1}{h_i} + \frac{1}{h_o}} = \frac{1}{\frac{1}{\frac{1}{64,985.76W/m^2.K} + \frac{1}{3138.98W/m^2.K}}} = 2994.35W/m^2.K$$

## 4.1.5 Calculation on Heat Transfer Rate of STHX

The temperature difference between hot and cold water at the two ends of the shell and tube heat exchanger is

$$\Delta T_1 = T_{h,in} - T_{c,out} = 90^{\circ}C - 47^{\circ}C = 43^{\circ}C$$
$$\Delta T_2 = T_{h,out} - T_{c,in} = 72^{\circ}C - 30^{\circ}C = 42^{\circ}C$$

and

$$\Delta T_{lm} = \frac{\Delta T_1 - \Delta T_2}{\ln(\Delta T_1 / \Delta T_2)} = \frac{43^{\circ}C - 42^{\circ}C}{\ln(43^{\circ}C / 42^{\circ}C)} = 42.5^{\circ}C$$

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

Then, the rate of heat transfer in the STHX can be determined

$$\dot{Q} = UA_s \,\Delta T_{lm} = U(\pi D_i L) \Delta T_{lm} = (2994.35 \,W/m^2.K)(\pi)(0.004m)(0.184m)(42.5^{\circ}C)$$
$$= 294.24kW$$

## 4.1.6 Calculation on Effectiveness-NTU Method of STHX

In the effectiveness-NTU Method, first of all determine the heat capacity rates of the hot and cold fluids and identify the smaller one.

$$C_{h} = \dot{m}_{c}c_{ph} = (1 kg/s)(4.1978 kJ/kg.K) = 4.1978 kW/K$$
$$C_{c} = \dot{m}_{c}c_{pc} = (1 kg/s)(4.1788 kJ/kg.K) = 4.1788 kW/K$$

Therefore,

$$C_{min} = C_c = 4.1788 kW/k$$

And

 $c = C_{min}/C_{max} = 0.9955$ 

Then the maximum heat transfer rate is determined to be

$$\dot{Q}_{max} = C_{min} (T_{h,in} - T_{c,in}) = (4.1788 kW/K)(90^{\circ}C - 30^{\circ}C) = 250.73 kW$$

Thus, the effectiveness of the heat exchanger is

UNIVERSITI TEKN
$$\dot{Q}$$
KA 294.24kW/SIA MELAKA  
 $\varepsilon = \frac{\dot{Q}}{\dot{Q}_{max}} = \frac{294.24kW}{250.73kW} = 1.1735$ 

## 4.1.6 Calculation on Pressure Drop

The equivalent roughness values for Stainless steel

$$\varepsilon = 0.002mm$$

Since Re is greater than 10,000, the flow is turbulent. The relative roughness of the tube is

$$\varepsilon/D = \frac{0.002mm}{4mm} = 0.0005$$

The friction factor corresponding to this relative roughness and Reynolds number can simply be determined from Moody chart.

$$\frac{1}{\sqrt{f}} = -2.0 \log\left(\frac{\varepsilon/D}{3.7} + \frac{2.51}{Re\sqrt{f}}\right)$$
$$\frac{1}{\sqrt{f}} = -2.0 \log\left(\frac{0.0005}{3.7} + \frac{2.51}{129,699\sqrt{f}}\right)$$
$$f = 0.0198$$

Using an equation solver or an iterative scheme, the friction factor is determined to be

f = 0.0198. Then the pressure drop becomes



The heat exchanger will operate 24 hours a day and 365 days a year. Therefore, the annual operating hours are

*Operating hours* = 
$$(24 h/day)(365 day/year) = 8760 h/year$$

Noting that this heat exchanger saves 294.24kW of energy, the energy saved during entire year will be

 $= (294.24kW)(8760 h/year) = 2.578 \times 10^{6} kWh/year$ 

Noting that the price of the electricity tariff is RM0.218/kWh, the amount of money saved becomes

Money saved =  $(2.578 \times 10^6 \, kWh/year)(RM0.218/kWh) = RM592.00/year$ 

### 4.2 Heat Transfer Performance

Before the line graph is plotted, the information on overall calculated value of shell and tube heat exchanger in simulation need to obtained first. This information is consisting of calculating parameter data on different baffle number such as Reynold number, pressure drop, overall heat transfer coefficient and heat transfer rate. The computational model of an experimental testing of 2, 3 and 4 baffle number and the calculated data are listed in Table 4.4.

Table 4.4: Overall Calculated Value in Shell-Side of Shell and Tube Heat Exchanger in

اونوم سيخ تتصiation کے ملبسیا ملاك

	1.4		a gav	
Baffle	Reynold	Pressure	Overall Heat Transfer	Heat Transfer
Number	Number,	Drop, $\Delta P$	Coefficient, U	Rate, Q
	Re	(kPa)	$(W/m^2.K)$	(kW)
2	33903	59.93	2778.22	274.94
3	36841	60.33	2879.82	282.48
4	39805	60.61	2994.35	294.24

65



Figure 4.1: Pressure drop versus Reynolds number.

From the graph on Figure 4.1 represents the comparisons of pressure drop in the shell side versus Reynolds number of the shell and tube heat exchanger with single segmental baffle for the three kinds number of baffles. It can be seen that the Reynolds number obtained by CFD model increases proportional to the number of baffle segment from 2 until 4 single segmental baffles which is 4 baffle produce largest Reynolds number and highest pressure drop than STHX with 2 and 3 baffle. The comparison shows a good agreement between 3 models with 4 number of baffle segment is highest which 39805 compared with 36841 for 3 baffle number and 33903 for 2 baffle number. Pressure drop is a major constraint in thermal design of shell and tube heat exchangers. A thermal design of a shell and tube heat exchanger very important when it is optimum and the extent of the optimality is constrained by the pressure drop. It has been found that the increasing baffle number from 2 to 4 affect the outlet temperature of shell side significantly. Pressure drop inside Shell with respect to baffle segmental number is provided. The pressure drop for the STHX with 2 and 3 number of

baffle segment is about 59%. The highest pressure drop is located at 60.61kPa is 4 baffle number in shell and tube heat exchanger compared with 60.33kPa for 3 baffle number and 59.93kPa for 2 baffle number. This indicates a large local pressure drop could be generated when the fluid flow through a large number of single segmental baffle. So, it is generalized that with increase in baffle number, Reynolds number increases, so that it affects in pressure drop, which is increased.





Figure 4.2: Overall heat transfer coefficient versus Reynolds number.

Variation in the overall heat transfer coefficient versus Reynolds number for three different number of baffle segment configurations in the shell and tube heat exchanger as shown in Figure 4.2. It can be observed that slope of the curves of overall heat transfer coefficient in the shell side is generally found to increase with the increases Reynolds number. The increment in baffle number was mainly accounts for overall resistance from convection and conduction. Overall heat transfer coefficient of 2 and 3 number of segmental baffle is increases with 47%. STHX with 4 number of baffle segment is the highest by about 2994.35  $W/m^2$ . *K* compared with 2879.82  $W/m^2$ . *K* for 3 baffle number and 2778.22  $W/m^2$ . *K* for 2 baffle number. From the above c4 baffle number show the largest overall heat transfer coefficient, it is necessary to increase in frictional pressure drop.



Figure 4.3: Heat transfer rate versus Reynolds number.

The line graph on Figure 4.3 represents the variations in heat transfer rate versus Reynolds number for the three single segmental baffles with different baffle number. It can be observed that the heat transfer rate increases proportional in the Reynolds number for STHX 2, 3 and 4 baffle. As expected previously, single segmental baffles with 4 baffle produce highest heat transfer rate than STHX with 2 and 3 baffle. STHX with 2 number of baffle segment is lowest by about 274.94*kW* compared with 282.48*kW* for 3 baffle number and 294.24*kW* for 4 baffle number. The rate of heat transfer initially is low, but the rate continues to increase until the outlet of the shell. The rate of heat transfer decrease along the STHX. 2 number of baffle show a sign of poor heat transfer rate in shell and tube heat exchanger. Based on the results discussed above, it is clear that STHX with 4 baffle number performs better for the heat transfer rate studied.

### **CHAPTER 5**

#### **CONCLUSION AND RECOMMENDATIONS**

The experimental analysis as well as the CFD simulation on three different number of baffle segmentation in shell and tube heat exchanger is carried out. Based on the analysis, it shows that the baffle number can be one of the biggest cause in affecting the shell and tube heat exchanger thermal performances.

The heat transfers and flow distribution is discussed in detail and proposed model is compared with increasing baffle segmental number. The result showed pressure, temperature and velocity distributions due to the number of baffle The results indicate that, compared to the 2 and 3 baffle number, the use of the 4 baffle segmentation provide a good balance between heat transfer rate and pressure drop characteristics which at 294.24 kW and 60.51kPa The 4 segmental baffle provide good overall heat transfer coefficient thus consume large pumping power which at 2994.35W/m<sup>2</sup>.K.

For the recommendation, the shell and tube heat exchanger should be analysed to obtain different manipulated variable, such as baffle inclination angle. Other than that, the grid independence test should be done at least twice or more to increase the accuracy of the simulation result. Furthermore, the CFD simulation can be compared with the experimental result to verify the method. The heat transfer rate is poor because most of the fluid passes without the interaction with baffles. Thus the design can be modified for better heat transfer in two ways either decreasing the baffle spacing, so that it will be a proper contact with the segmental baffle or by adding the baffle number so that baffles will be proper contact with the shell. Lastly, the study of the effect on physical features can be analysed by using different Workbench environment such as thermal analysis or CFX to compare the result.



#### REFERENCES

Ender Ozden & Ilker Tari. (2010). Shell Side CFD Analysis of a Small Shell-And-Tube Heat Exchanger. *Energy Conversion and Management*, *51*, 1004–1014.

Usman Salahuddin & Muhammad Bilal, Haider Ejaz. (2015). A Review of the Advancements Made in Helical Baffles Used in Shell and Tube Heat Exchangers. *International Communications in Heat and Mass Transfer*, 67, 104–108.

Balaram Kundu. (2015). Beneficial Design of Unbaffled Shell-And-Tube Heat Exchangers for Attachment of Longitudinal Fins with Trapezoidal Profile. *Case Studies in Thermal Engineering*, *5*, 104–112.

Kai Wanga, Xin-Cheng Tu, Chan-Hyo Bae, & Hyoung-Bum Kim. (2014). Optimal Design of Porous Baffle to Improve the Flow Distribution in The Tube-Side Inlet of a Shell and UNIVERSITIEN AND ADDITIONAL AND A

Dillip Kumar Mohanty & Pravin M. Singru. (2014). Fouling Analysis of a Shell and Tube Heat Exchanger Using Local Linear Wavelet Neural Network. *International Journal of Heat and Mass Transfer*, 77, 946–955. Yongqing Wang, Xin Gu, Zunlong Jin, & Ke Wang. (2015). Characteristics of Heat Transfer for Tube Banks in Crossflow and Its Relation with That in Shell-And-Tube Heat Exchangers. *International Journal of Heat and Mass Transfer, 93*, 584–594.

Jian Wena, Huizhu Yang, Guanping Jian, Xin Tong, Ke Li, & Simin Wangb. (2016). Energy and Cost Optimization of Shell and Tube Heat Exchanger with Helical Baffles Using Kriging Metamodel Based On MOGA. *International Journal of Heat and Mass Transfer*, *98*, 29–39.

Fornarelli F., Camporeale S.M., Fortunato B., Torresi M., Oresta P., Magliocchetti L., Miliozzi A., & Santo G. (2016). CFD Analysis of Melting Process in A Shell-And-Tube Latent Heat Storage for Concentrated Solar Power Plants. *Applied Energy*, *164*, 711–722.

Walraven D., Laenen B., & D'haeseleer W. (2014). Comparison of Shell-And-Tube with Plate Heat Exchangers for The Use in Low-Temperature Organic Rankine Cycles. *Applied Mechanics and Energy Conversion.* 87, 227-237.

## **UNIVERSITI TEKNIKAL MALAYSIA MELAKA**

Bin Gao, Qincheng Bi, Zesen Nie, Jiangbo Wu. (2015). Experimental Study of Effects of Baffle Helix Angle on Shell-Side Performance of Shell-And-Tube Heat Exchangers with Discontinuous Helical Baffles. *Experimental Thermal and Fluid Science*, 68, 48–57.

Bagi J.S., Mane V.N. & Arvind B. Datye. (2012). Improvement in Quality Dimensions of Shell and Tube Type Heat Exchanger by Using Design Software. *International Journal of Advanced Engineering Technology*, 183-188.

Edward S. Gaddis & Volker Gnielinski. (1997). Pressure Drop On the Shell Side of Shell and-Tube Heat Exchangers with Segmental Baffles, *Chemical Engineering and Processing*. *36*, 149–159.

Aniket Shrikant Ambekar, Sivakumar R., Anantharaman N. & Vivekenandan M. (2016). CFD Simulation Study of Shell and Tube Heat Exchangers with Different Baffle Segment Configurations. *Applied Thermal Engineering*, *108*, 999–1007.

Anas El Maakoul, Azzedine Laknizi, Said Saadeddine, Mustapha El Metoui, Abdelkabir Zaite, Mohamed Meziane & Abdelatif Ben Abdellah. (2016). Numerical Comparison of Shell-Side Performance for Shell and Tube Heat Exchangers with Trefoil-Hole, Helical and Segmental Baffles. *Applied Thermal Engineering, 109*, 175–185.

Zuo, W. & Chen, Q. (2009). Real Time or Faster-Than-Real-Time Simulation of Airflow in Buildings. *Indoor Air*, 19(1), 33-44.

## **UNIVERSITI TEKNIKAL MALAYSIA MELAKA**

Zhang, L.-Z. (2009). Flow Maldistribution and Thermal Performance Deterioration in A Cross-Flow Air to Air Heat Exchanger with Plate-Fin Cores. *International Journal of Heat and Mass Transfer*, *52*, 4500-4509.

Hari Haran, Bg. Ravindra Reddy and Cb. Sreehari. (2013). Thermal Analysis of Shell and Tube Heat Ex-Changer Using C and ANSYS. *International Journal of Computer Trends and Technology (IJCTT)*, 4(7), 2340-2345.

Simin Wang, Jian Wen & Yanzhong Li. (2009). An Experimental Investigation of Heat Transfer Enhancement for a Shell-And-Tube Heat Exchanger. *Applied Thermal Engineering*, 29, 2433-2438.

Orlando Duran, Nibaldo Rodriguez & Luiz Airton Consalter. (2009). Neural Networks for Cost Estimation of Shell and Tube Heat Exchangers. *Expert Systems with Application, 36*, 7435–7440.

Haolin Ma. (2012). Computational Fluid Dynamics and Heat Transfer Analysis for a Novel Heat Exchanger. *Theses and Dissertations*. 1154.

ې تېكنىكا ملىسيا ملاك

Arjun K.S. and Gopu K.B. (2014). Design of Shell and Tube Heat Exchanger Using Computational Fluid Dynamics Tools. *Research Journal of Engineering Sciences*. 3(7), 8-16.

Bendekar A. & Sawant B., (2016). Heat Transfer Optimization of Shell and Tube Heat Exchanger Through CFD Analysis. *International Journal of Advanced Technology in Engineering and Science*. 4(4), 159-172.

Manu P.S. 2016). Comparison of Heat Transfer in Shell and Tube Heat Exchanger with Modified Baffles using ANSYS Fluent. *International Journal of Engineering Research & Technology (IJERT)*. *5*(2), 157-161.