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

NUR FARAHIN BINTI MOHD AHADLIN

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

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

NUR FARAHIN BINTI MOHD AHADLIN

This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering (Thermal-Fluids)

Faculty of Mechanical Engineering

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.

Signature	:	....
Author	:	**NUR FARAHIN BINTI MOHD AHADLIN**
Date	:	....

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).

Signature :. ....

Name of Supervisor : **DR. FATIMAH AL-ZAHRAH BT MOHD SAAT**

Date : ....

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 2994.35$W/m^2.K$. 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.

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 2994.35*$W/m^2.K$*. 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

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBEREVATIONS

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

LIST OF SYMBOL

ε	=	Effectiveness, Turbulent dissipation energy
T	=	Temperature
Δ	=	Increment
U	=	Overall Heat Transfer Coefficient
h	=	Enthalpy
R	=	Total Thermal Resistance
A	=	Surface area
q	=	Heat flux
$\dot{Q}$	=	Heat Transfer Rate
NTU	=	Number of Transfer Units
ρ	=	Density
$LMTD$	=	Log mean temperature difference
k	=	Thermal conductivity, Turbulent kinetic energy
c	=	Specific heat
F	=	Fouling factor
L	=	Length
$\dot{m}$	=	Mass flow rate
n	=	Number of tubes

u = Velocity

Re = Reynold number

μ = Dynamic viscosity

d = Diameter

B = Baffle spacing

C = Heat capacity rate

P = 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.

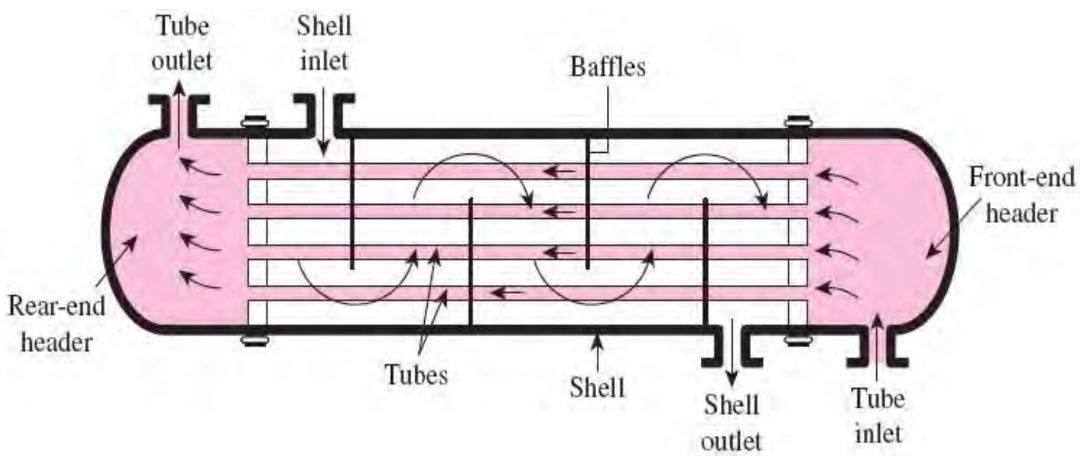

Figure 1.1: Cross section of shell and tube heat exchanger (Cengel, 2015)

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.

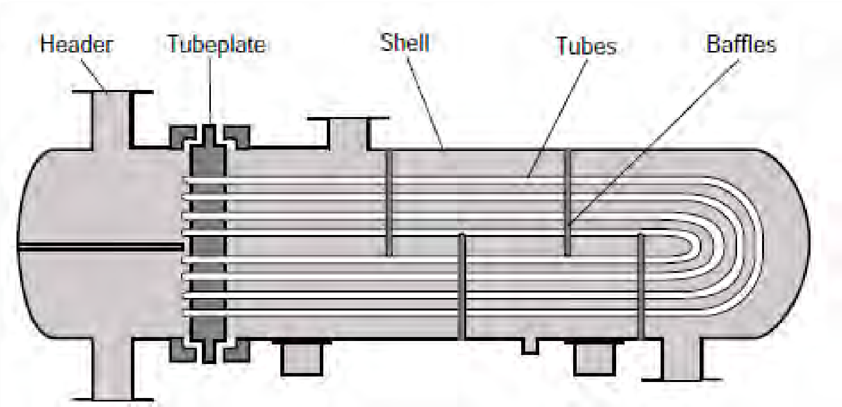

Figure 1.2: U-tube heat exchanger (Mukherjee, 1998)

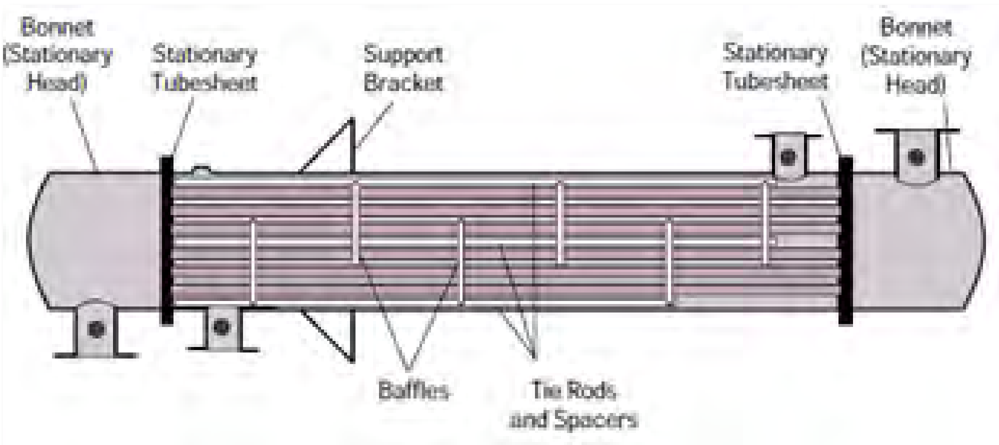

Figure 1.3: Fixed-tubesheet heat exchanger (Mukherjee, 1998)

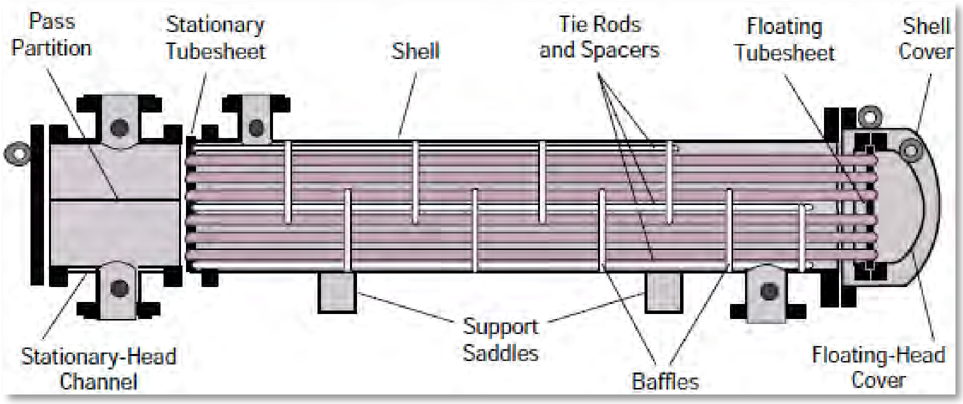

Figure 1.4: Floating-head heat exchanger (Mukherjee, 1998)

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 non-cross 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:

1. 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.