SIMULATION STUDY OF HEAT EXCHANGER DESIGN

By:

WAN MOHD ALIMIE BIN WAN AHMAD

(Matrix no.: 120427)

Supervisor:

Dr. Mohd Azmi Bin Ismail

June 2017

This dissertation is submitted to
Universiti Sains Malaysia
As partial fulfillment of the requirement to graduate with honors degree in
BACHELOR OF ENGINEERING (MECHANICAL ENGINEERING)

School of Mechanical Engineering
Engineering Campus
Universiti Sains Malaysia

DECLARATION

This work has not previously been accepted in substance for any degree and is not being concurrently submitted in candidature for any degree.

Signed

…………………………………………………………………………………………

(Wan Mohd Alimie Bin Wan Ahmad)

Date: 7^{TH} JUNE 2017

STATEMENT 1

This thesis is the results of my own investigations, except where otherwise stated. Other sources are acknowledged by giving explicit references. Bibliography/references are appended.

Signed

……………………………………………………………………………………......................

(Wan Mohd Alimie Bin Wan Ahmad)

Date: 7^{TH} JUNE 2017

STATEMENT 2

I hereby give consent for my thesis, if accepted, to be available for photocopying and for interlibrary load, and for the title and summary to be made available to outside organization.

Signed

……………………………………………………………………………………......................

(Wan Mohd Alimie Bin Wan Ahmad)

Date: 7^{TH} JUNE 2017

ACKNOWLEDGEMENT

In the name of Allah S.W.T the Almighty God, the Most Gracious, the Most Merciful and peace and blessings be upon the Prophet Muhammad S.A.W, with this my final year project was completed.

Adequate efforts had been taken to complete this project where a lot of time had been spent. It would not have been possible for me to finish this project without help and support from many individuals and organization.

I am eminently indebted to my supervisor Dr Mohd Azmi Ismail for his kindness in delivering valuable guidance and continual supervision as well as providing necessary information regarding this project and also for the moral support in completing this project.

Furthermore, I would like to express my gratitude towards my parent that always pray for me to successfully accomplish this task and not to forget all the acquaintances from Universiti Sains Malaysia especially School of Mechanical Engineering for their kindness towards me. Plus their willing to give me a great cooperation and encouragement that really help me in the completion of the assigned task. Special thanks to engineers and staff from Petronas Carigali Sdn Bhd (Mechanical Static Department) for their willingness to share some information related to my project.

Thank you to people stated above and also for anybody that I might unintentionally forget to acknowledge their help and support. Grateful and thankful to the Almighty God that gave me vast amount of strength and energy so this project, Simulation Study of Heat Exchanger Design accomplished and so this thesis manuscript is composed.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF EQUATIONS

LIST OF ABBREVIATIONS

HX	Heat Exchanger
PV	Pressure Vessel
ASME	American Society of Mechanical Engineer
DOSH	Department of Occupational Safety and Health
PTI	Permission To Install
PTO	Permission To Operate
TEMA	Tubular Exchanger Manufacturers Association
API	American Petroleum Institute
BS	British Standard
GA	General Arrangement Drawing
FEA	Finite Element Analysis
CFD	Computational Fluid Dynamic
GUI	Graphical User Interface
LMTD	Log Mean Temperature Difference

ABSTRAK

Penukar haba yang digunakan untuk memindahkan haba dari cecair pada suhu yang tinggi untuk cecair pada suhu yang lebih rendah. Penukar haba digunakan dalam keperluan industri dalam industri kimia, loji kuasa nuklear, kilang penapis, pemprosesan makanan, dan lain-lain. Saiz penukar haba memainkan peranan yang sangat penting untuk pengiraan kos. Selain itu, kecekapan dan keberkesanan penukar haba adalah satu parameter penting semasa pilihan penukar haba industri. Kaedah untuk penambahbaikan terhadap pemindahan haba telah dikerjakan selama bertahun-tahun untuk mendapatkan kecekapan yang tinggi dengan kos yang optimum. Dalam projek ini, kami menganalisis penukar haba shell dan tiub dengan plat sesekat dengan menukar bahan tiub. Proses dalam menyelesaikan simulasi terdiri daripada pemodelan dan bersirat geometri asas shell dan haba tiub penukar menggunakan pakej ANSYS FLUENT 14.0. Dimensi untuk model yang diambil oleh penukar haba yang sedia ada digunakan oleh PETRONAS Carigali SDN BHD. Dalam projek ini, pemodelan shell dan haba tiub penukar dengan sesekat tiub dan sokongan dicipta dengan menggunakan perisian GAMBIT. Kemudian, kondisi akan ditetapkan dan simulasi dalam FLUENT berdasarkan data industri. Skop kajian ini adalah untuk mengkaji kadar pemindahan haba penukar haba menggunakan alat perisian ANSYS FLUENT 14.0. Medan aliran dan suhu di dalam shell dan tiub diselesaikan menggunakan pakej FLUENT komersial. Penukar haba kami terdiri daripada dua cecair (minyak mentah dan Danol-XHT) suhu bermula berbeza mengalir melalui penukar haba. Kadar pemindahan haba penukar haba telah dikira dengan persamaan pemodelan matematik. Tiub telah diubah dengan bahan-bahan yang berbeza seperti tembaga, aluminium, keluli karbon dan graphene. Akhirnya analisis yang telah dilakukan dengan mengubah bahan-bahan tiub dan oleh itu ia adalah diperhatikan bahawa bahan tembaga memberikan kadar pemindahan haba lebih baik daripada bahan aluminium, keluli karbon dan graphene.

ABSTRACT

Heat exchangers are used to transfer heat from fluid at high temperature to fluid at lower temperature. Heat exchangers are used in industrial purposes in chemical industries, nuclear power plants, refineries, food processing, etc. Sizing of heat exchangers plays very significant role for cost optimization. Also, efficiency and effectiveness of heat exchangers is an important parameter while selection of industrial heat exchangers. Methods for improvement on heat transfer have been worked upon for many years in order to obtain high efficiency with optimum cost. In this project, we are analyzing shell and tube heat exchanger with baffle plates by changing tube material. The process in solving simulation consists of modeling and meshing the basic geometry of shell and tube heat exchanger using FLUENT package ANSYS 14.0. Dimensions for the model are taken by existing heat exchanger used by PETRONAS Carigali SDN BHD. In this project, modeling of shell and tube heat exchanger with baffles tubes and supports created by using GAMBIT software. Then, the boundary condition will be set and simulate in FLUENT based on the industrial data. The scope of this paper is to study the heat transfer rate of heat exchanger using ANSYS FLUENT 14.0 software tool. The flow and temperature fields inside the shell and tube are resolved using a commercial FLUENT package. Our heat exchanger consists of two fluids (crude oil and Danol-XHT) of different starting temperatures flow through the heat exchanger. Heat transfer rate of the heat exchanger were calculated by mathematical modeling equations. Tubes were varied with different materials likes copper, aluminum, carbon steel and graphene. Finally analysis has been done by varying the tube materials and hence it is observed that copper material gives the better heat transfer rates than aluminum, carbon steel and graphene.

CHAPTER 1

INTRODUCTION

1.1 Brief Introduction

A heat exchanger is a gadget worked for proficient heat exchange starting with one medium then onto the next, regardless of whether the media are isolated by a strong divider so that they never blend, or the media are in direct contact. In other word, heat exchanger is a gadget that exchanges heat starting with one liquid then onto the next through a progression of tubes/plates from one side to the opposite side. For this venture, we are concentrating on heat exchanger for oil and gas enterprises which is Floating Head Shell-and-Tube Heat Exchanger. There are different sorts of heat exchanger, for example,

1. Tubular
    - 1.1 Shells-and-Tube Heat Exchanger
        - i. Fixed Tube Sheet
        - ii. U-Tube
        - iii. Floating Head
        - iv. Kettle Reboiler
    - 1.2 Air-Cooled Heat Exchanger
    - 1.3 Double-Pipe Heat Exchanger
2. Non-Tubular
    - 2.1 Plate Heat Exchanger
    - 2.2 Printed Circuit Heat Exchanger

For this project, we are using heat exchanger from PETRONAS. The design of heat exchanger is based on few codes and standards as below;

1. PETRONAS Technical Standard (PTS)

    31.22.10.31 Pressure Vessels (Based on ASME Section VIII)

    31.21.01.30 Shell and Tube heat exchanger (Amm./Supp. to ISO 16812)

    31.22.10.32 Pressure Vessels (Amm./Supp. to PD 5500)

30.10.02.11 Metallic Materials

31.21.01.31 Selected Construction material for Shell and Tube Heat Exchanger

2. Factory and Machinery Act, 1967 (Act 139)
   - Submission of PV design (document by fabricator and owner) to DOSH
     - a Design Calculations and drawings
     - b PTI
     - c PTO
     - d CF renewal
     - e Repair Procedure
     - f Hydrotest (witness)

3. ASME (American Society of Mechanical Engineers)
   - Section VIII
     - a Division I - Rules for Construction of Pressure Vessel
     - b Division II - Alternative Rules
   - Section IX - Welding and Brazing Qualifications
   - Section V - Nondestructive Examination
   - Section II – Materials

4. TEMA (Tubular Exchanger Manufacturers Association)

5. API (American Petroleum Institute)
   - API 660 Shell-and-Tube Heat Exchangers for General Refinery Services (ISO 16812)
   - API 661 Air-Cooled Heat Exchangers for General Refinery Service (ISO 13706)
   - API 662 Plate Exchangers for General Refinery Services (ISO 15547)

6. BS (British Standard)
   - BS PD5500 Specification of Unfired Fusion Welded Pressure Vessel

A shell and tube heat exchanger is a class of heat exchanger plans. It is the most well-known kind of heat exchanger in oil refineries and other extensive concoction forms and is suited for higher-weight applications. It comprises of a tube bundle encased in a tube shaped packaging called a shell. One liquid goes through the tubes, and another liquid stream over the tubes (through the shell) to exchange heat between the two liquids.

Two liquids, of various initial temperatures, move through the heat exchanger. One moves through the tubes (the tube side) and alternate streams outside the tubes however inside the shell (the shell side). Heat is exchanged from one liquid to the next through the tube dividers, either from tube side to shell side or the other way around. The liquids can be either fluids or gasses on either the shell or the tube side. With a specific end goal to exchange heat proficiently, a substantial heat exchange range ought to be utilized, so there are many tubes. Along these lines, squander heat can be put to utilize. This is an extraordinary approach to monitoring vitality.

Regularly, the finishes of each tube are associated with plenums through gaps in tube sheets. The tubes might be straight or twisted in the state of a U, called U-tubes. Most shell-and-tube heat exchangers are 1, 2, or 4 pass designs on the tube side. This refers to the number of times the fluid in the tubes passes through the fluid in the shell. In a single pass heat exchanger, the liquid goes in one end of each tube and out the other.

There are two basic types of shell-and-tube exchangers. The first is the fixed tube sheet unit, in which both tube sheets are fastened to the shell and the tube bundle is not removable. The second type of shell-and-tube unit has one restrained tube sheet, called the stationary tube sheet, located at the channel end. Differential expansion problems are avoided by use of a freely riding floating tube sheet at the other end or the use of U tubes. This design may be used for single or multiple pass exchangers. The tube bundle is removable from the channel end, for maintenance and mechanical cleaning. There are often baffles directing flow through the shell side so the fluid does not take a shortcut through the shell side leaving ineffective low flow volumes.

Counter current heat exchangers are most efficient because they allow the highest log mean temperature difference between the hot and cold streams. Many companies however do not use single pass heat exchangers because they can break easily in addition to being more expensive to build. Often multiple heat exchangers can be used to simulate the countercurrent flow of a single large exchanger. Shell-and-tube exchangers are designed and fabricated according to the standards of the Tubular Exchanger Manufacturers Association (TEMA).

1.2 Project Background

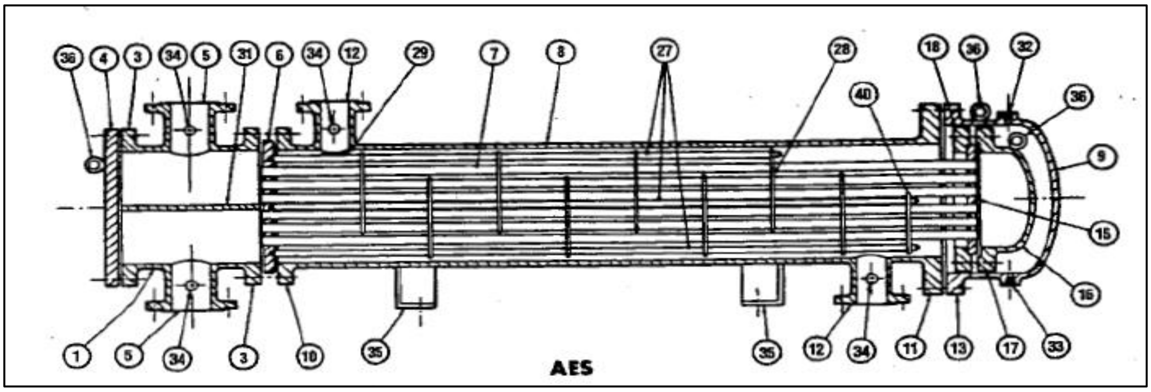
Previously, for Industrial Training EML 451 subject, I have undergone my internship programme at PMA-PETRONAS Carigali Sdn Bhd. From there, I have learnt and aware about shell and tube heat exchanger used in Terengganu Crude Oil Terminal (TCOT) in Paka, Terengganu. There are two types of heat exchanger in TCOT which are HX210 (hot oil – hot oil) and HX220 (crude – hot oil). Furthermore, I have few data and information about HX220 that may be used for this simulation study of heat exchanger design project.

Thus for this project, we are considering crude as fluid flow in the tube and hot oil as counter-flow fluid in shell. All the data related to heat exchanger is referred to existing PETRONAS Carigali Sdn Bhd heat exchanger which is HX220 (Hot Oil – Crude).

This project is more related to application of heat exchanger design. It is full simulation and programming project. For this project, I need to re-draw the existing HX220 and improve better. For example, change the material of tube and compare the heat transfer rate of the crude. I am referring the dimension and design from General Arrangement Drawing of HX220. Design of heat exchanger will be draw in Gambit 2.3.16 and simulation will be run using ANSYS Fluent 14.0.

ANSYS Mechanical software is a comprehensive finite element analysis (FEA) tool for structural analysis, including linear, nonlinear, dynamic, hydrodynamic and explicit studies. It provides a complete set of elements behaviour, material models and equation solvers for a wide range of mechanical design problems. ANSYS Fluent is the most-powerful computational fluid dynamics (CFD) software tool available, empowering us to go further and faster. Fluent includes well-validated physical modelling capabilities to deliver fast, accurate results across the widest range of CFD and Multiphysics applications.

PETRONAS used Tubular Exchanger Manufacturers Association (TEMA) as design reference for their heat exchanger. The components for floating head heat exchanger as shown in Figure 1.1 below.



1. Stationary Head-Channel
2. Stationary Head-Bonnet
3. Stationary Head Flange-Channel or Bonnet
4. Channel Cover
5. Stationary Head Nozzle
6. Stationary Tubesheet
7. Tubes
8. Shell
9. Shell Cover
10. Shell Flange-Stationary Head End
11. Shell Flange-Rear Head End
12. Shell Nozzle
13. Shell Cover Flange
14. Expansion Joint
15. Floating Tubesheet
16. Floating Head Cover
17. Floating Head Cover Flange
18. Floating Head Backing Device
19. Split Shear Ring
20. Slip-on Backing Flange
21. Floating Head Cover-External
22. Floating Tubesheet Skirt
23. Packing Box
24. Packing
25. Packing Gland
26. Lantern Ring
27. Tierods and Spacers
28. Transverse Baffles or Support Plates
29. Impingement Plate
30. Longitudinal Baffle
31. Pass Partition
32. Vent Connection
33. Drain Connection
34. Instrument Connection
35. Support Saddle
36. Lifting Lug
37. Support Bracket
38. Weir
39. Liquid Level Connection
40. Floating Head Support

Figure 1.1 Components of Shell and Tube Heat Exchanger

1.3 Problem Statement

From existing crude-hot oil heat exchanger HX220, the heat transfer rate of the tube is 9113553.3W with tube material of SA210A1 (Seamless Carbon steel). The heat exchanger HX220 is used to heat up crude oil in the process. The crude oil is pumped directly from offshore terminal to onshore terminal for other process. Thus, in order to ensure the flow of the crude is smooth, the crude must be heat up. Once it heated up, it will prevent from the crude oil from clogging in the pipeline system. Moreover, that is the purpose of HX220 to heat up crude oil. Besides, if the heat exchanger itself does not have high effectiveness with good heat transfer rate, the process of heating up crude oil will be poor.

Furthermore, design of heat exchanger with low thermal conductivity material also will affect the effectiveness and heat transfer rate of the heat exchanger. Thus, this project will study on how different materials of tube will affect the heat transfer rate of crude and hot oil through the tubes.

1.4 Objective

There are few objectives that need to be achieving at the end of this project;

1. To study the design, and performance of existing shell and tube heat exchanger used by Petronas Sdn Bhd.
2. To study the effect on heat transfer rate of crude and hot oil by changing the materials of tubes using simulation in ANSYS FLUENT and theoretical calculation.

1.5 Scope of Work

In completing this project, a lot of works needed to be done. It is started from studying the research area by looking back on what has been done by other people. A finding from literature review session help to focus more on research are and broadening the idea of doing this project.

The design of heat exchanger has being done by using Gambit 2.3.16. The model was drawn in 3D. Most of dimensions have been referred from existing General Arrangement drawing of Crude/Hot Oil Heat Exchanger HX220 from Petronas Carigali Sdn Bhd. Meshing of the model also being done by using Gambit 2.3.16. Simulation and analysis of the geometry has been done by using ANSYS Workbench (FLUENT).

1.6 Thesis Outline

This thesis consist of five (5) chapters that fully describe the details of the Final Year Project from the beginning until end of the project where the publishing of this thesis manuscript signify the completion of the Final Year Project with title Simulation Study of Heat Exchanger Design.

Briefly, Chapter One is the introduction chapter to introduce the project. This chapter provided a whole picture of the project. Next, Chapter Two is the fundamental component of this project structure. This chapter contains literature review that is used in completing this project where a lot of theories, concept and idea were collected and learnt in completing this project.

Chapter Three is the outline of the project starting from the planning until the project finished. This chapter explains the overall project flow step-by-step. Chapter Four continue with the results and analysis of the project. The finding from this project was the discussed with the help of figure, pictures, graphs and table.

Chapter Five is the final chapter in this manuscript. It's finalized the project analysis into focused and concrete evidence of the accomplished objective. This chapter concludes the project with some recommendation that could be done if the project is further develops. The flow chart of this project is shown in Figure 1.2 below.

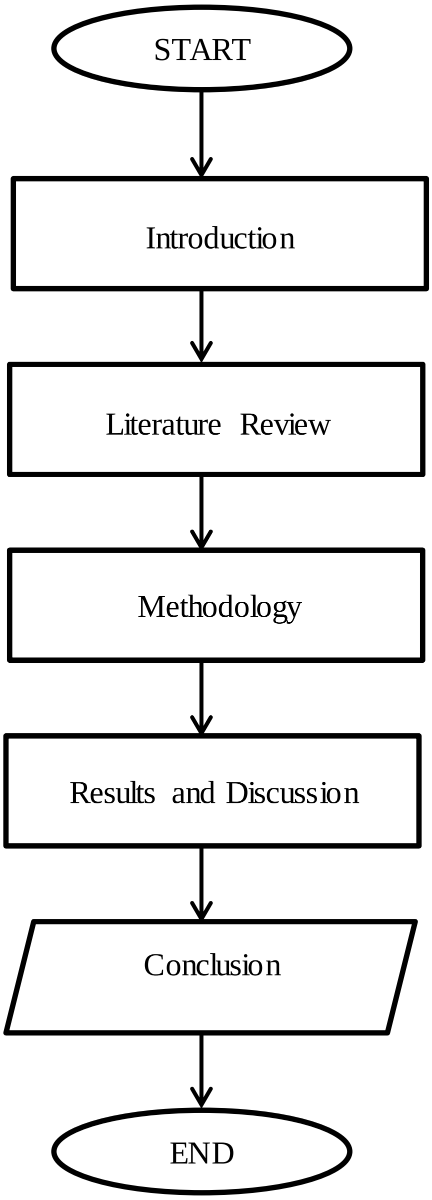


Figure 1.2 Flow chart of thesis outline

CHAPTER 2

LITERATURE REVIEW

2.1 Heat Exchanger

Heat exchanger is a device that exchange heat between two fluids that are different temperatures and separated by solid wall. By this case, the heat transfer occurs between fluid in tube and fluid in shell. Heat exchangers are typically classified according to flow arrangement and type of construction. The simplest heat exchanger is one for which the hot and cold fluids move in the same or opposite directions in a concentric tube (or double-pipe) construction. In the parallel-flow arrangement of Figure 2.1(a), the hot and cold fluids enter at the same end, flow in the same direction, and leave at the same end. In the counter flow arrangement of Figure 2.1(b), the fluids enter at opposite ends, flow in opposite directions, and leave at opposite ends [1].

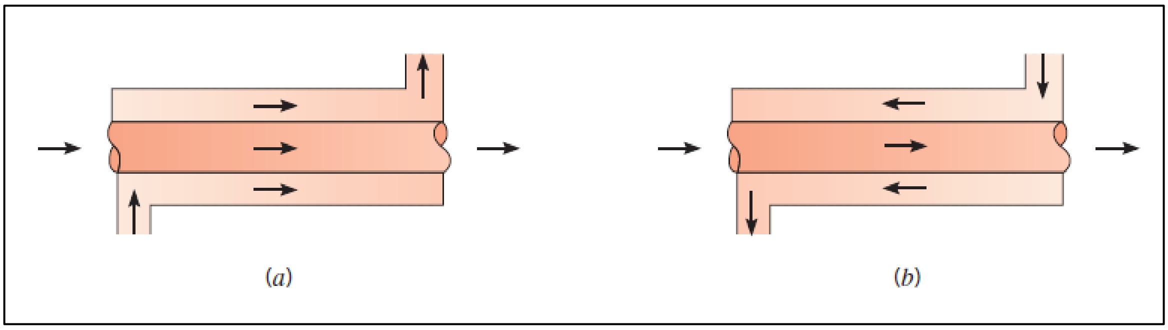


Figure 2.1 Concentric tube heat exchanger. (a) Parallel flow. (b) Counterflow

During normal heat exchanger operation, surfaces are often subject to fouling by fluid impurities, rust formation, or other reactions between the fluid and the wall material. The subsequent deposition of a film or scale on the surface can greatly increase the resistance to heat transfer between the fluids [1].

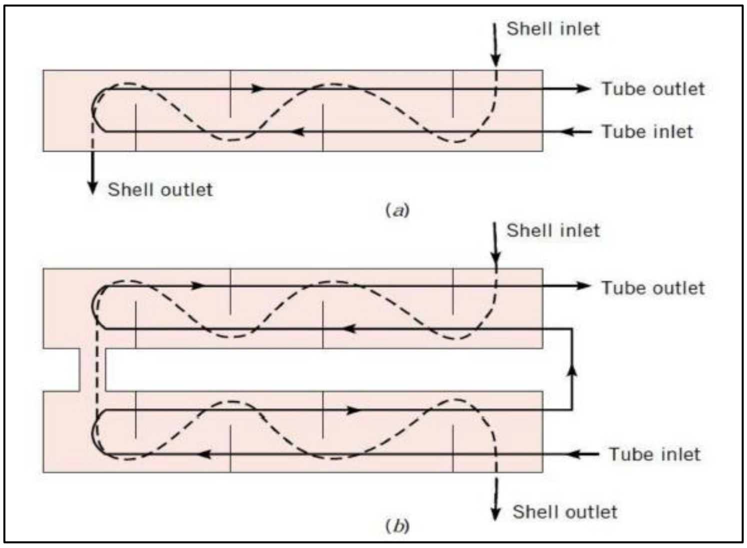


Figure 2.2 Shell and Tube Heat Exchanger (a) One Shell Pass and two tube passes (b) Two Shell Pass and four tubes passes

The design of heat exchanger must consider few parameters such as heat transfer. The heat-transfer requirements must be met in the selection or design of any heat exchanger. By forcing the fluids through the heat exchanger at higher velocities the overall heat transfer coefficient may be increased, but this higher velocity results in a larger pressure drop through the exchanger and correspondingly larger pumping costs. If the surface area of the exchanger is increased, the overall heat-transfer coefficient, and hence the pressure drop, need not be so large; however, there may be limitations on the physical size that can be accommodated, and a larger physical size results in a higher cost for the heat exchanger [2].

In order to have more efficient shell and tube heat exchanger, the design must consist of baffles. Baffles play a significant role in shell and tube heat exchanger assembly. They provide support for tubes, enable a desirable velocity to be maintained for the shell-side fluid flow, and prevent the tubes from vibrating. Baffles also guide the shell- side flow to move forward across the tube bundle, increasing fluid velocity and heat transfer coefficient [3]. For this project, the existing PETRONAS HX220 used single segmented baffles. Single segmental baffles provide good overall heat transfer coefficient but with large pressure drop and thus consume large pumping power [3].

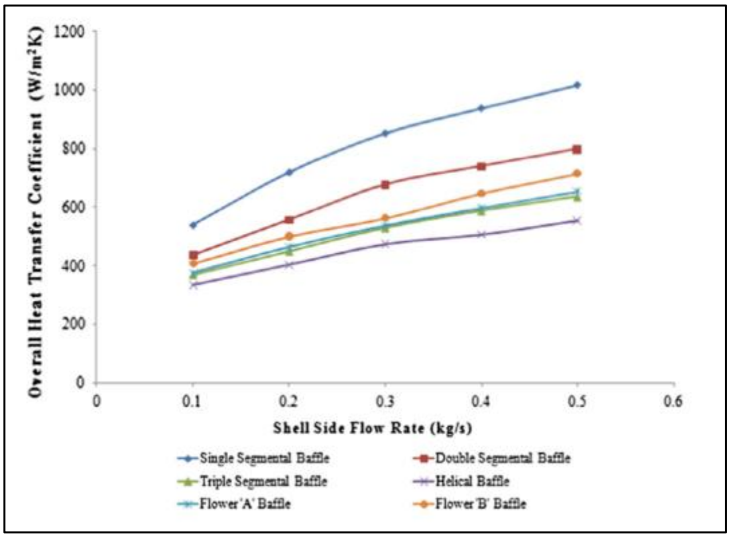


Figure 2.3 Overall heat transfer coefficient vs. shell side flow rate

Most of heat exchanger has poor heat transfer rate 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 the decreasing the shell diameter, so that it will be a proper contact with the helical baffle or by increasing the baffle so that baffles will be proper contact with the shell. It is because the heat transfer area is not utilized efficiently. Thus the design can further be improved by creating cross-flow regions in such a way that flow doesn't remain parallel to the tubes. It will allow the outer shell fluid to have contact with the inner shell fluid, thus heat transfer rate will increase [4].

2.2 Materials

For selecting materials of heat exchanger, we first must consider few parameters such as nature of the fluids on both the tube and shell side, flow rate, temperature and pressure, start-up and shutdown conditions, upset conditions, special conditions like product purity requirements, hazardous effects of intermixing of shell and tube side fluids, radioactivity and associated maintenance [5]. In order to improve the efficiency of the heat exchanger, once must have material with high heat transfer coefficient (requiring high thermal conductivity for tube material). Plus, thermal expansion coefficient to be low and as compatible as possible with those of the materials used for tube sheet, tube support and shell to provide resistance to thermal cycling [5].

A number of factors need to be considered when selecting a tube material. They include potential for corrosion and erosion, consideration of vibration and mechanical property requirements. In some applications, thermal conductivity and potential for fouling may be a big factor. One needs to be cognizant of the potential for mechanical damage and choose an alloy tolerant for high energy locations. If an application has a history with certain metals, it is important to use that knowledge in selection of the future replacement material. A full root cause analysis on the failure mode is critical component of the selection. Additionally, one should always consider specifying an appropriate corrosion test for the material and application [6].

Most of existing heat exchanger used common materials for the tubes. For example, copper, brass, titanium and aluminum. In order to have high efficiency of heat exchanger, manufacturer needs to consider high thermal conductivity materials for tubes. Thermal conductivity measures the ability of a metal to conduct heat. This property changes crosswise over various sorts of metal and is essential to consider in applications where high working temperatures are normal.

Copper is a prevalent decision for heat exchangers in modern offices, cooling, refrigeration, boiling point water tanks and under-floor heating frameworks. Its high thermal conductivity enables heat to go through it rapidly. Copper has extra properties attractive in heat exchangers including imperviousness to consumption, biofouling, stress and thermal extension. Aluminum can likewise be utilized as a part of some heat exchanger applications as a more financially savvy elective [7].

Liquid used in the heat exchanger as hot oil is Danol XHT32. Petronas Danol XHT is finest quality heat transfer oil formulated for use in most of direct and indirect fired heat transfer systems with forced circulation where the fluid is subjected to the most severe operating conditions. This fluid is used in the shell. It is high thermal conductivity - heats quickly for fuel economy and even heat distribution [8]. It is also non-corrosive to aluminum, brass, bronze, copper, and steel. Typical physical characteristics of Danol XHT are shown in Table A1. The cold fluid in the tubes is crude oil [9].

CHAPTER 3

METHODOLOGY

3.1 Introduction

This simulation is done to find the possible data of the shell and tube heat exchanger by analyze the effect on heat transfer rate and flow rate of crude and hot oil. The manipulation variable for this study is the type of materials of the tubes. In order to start the study, research plan has been done. Few literature reviews has been done to get some knowledge about shell and tube heat exchanger. Then the simulation started with 3D drawing by using Gambit 2.3.16. The geometry being creates and edit by using Gambit software. Next, the geometry is imported to ANSYS Workbench 14.0. All the data and solution on heat transfer rate was generated in the Fluent. Finally, manual calculation has been done to compare with the simulation data and results. The flow of this project can be observed as shown in the Figure 3.2 below.

(a) (b)

Figure 3.1 (a) GAMBIT (b) ANSYS Fluent

3.2 Research Planning

Literature review is the main component for this application and study. Literature review gives researcher the idea and knowledge as a guide to develop their research. It also gives an overall picture to the research field being conducted as well as fundamental concept of conducting a proper study.

For this case study, few literature reviews related to shell and tube heat exchanger has been done. This review was classified into three components which are the heat transfer rate, materials used for the tube and specification and parameter of the fluid flow in the tube and shell.

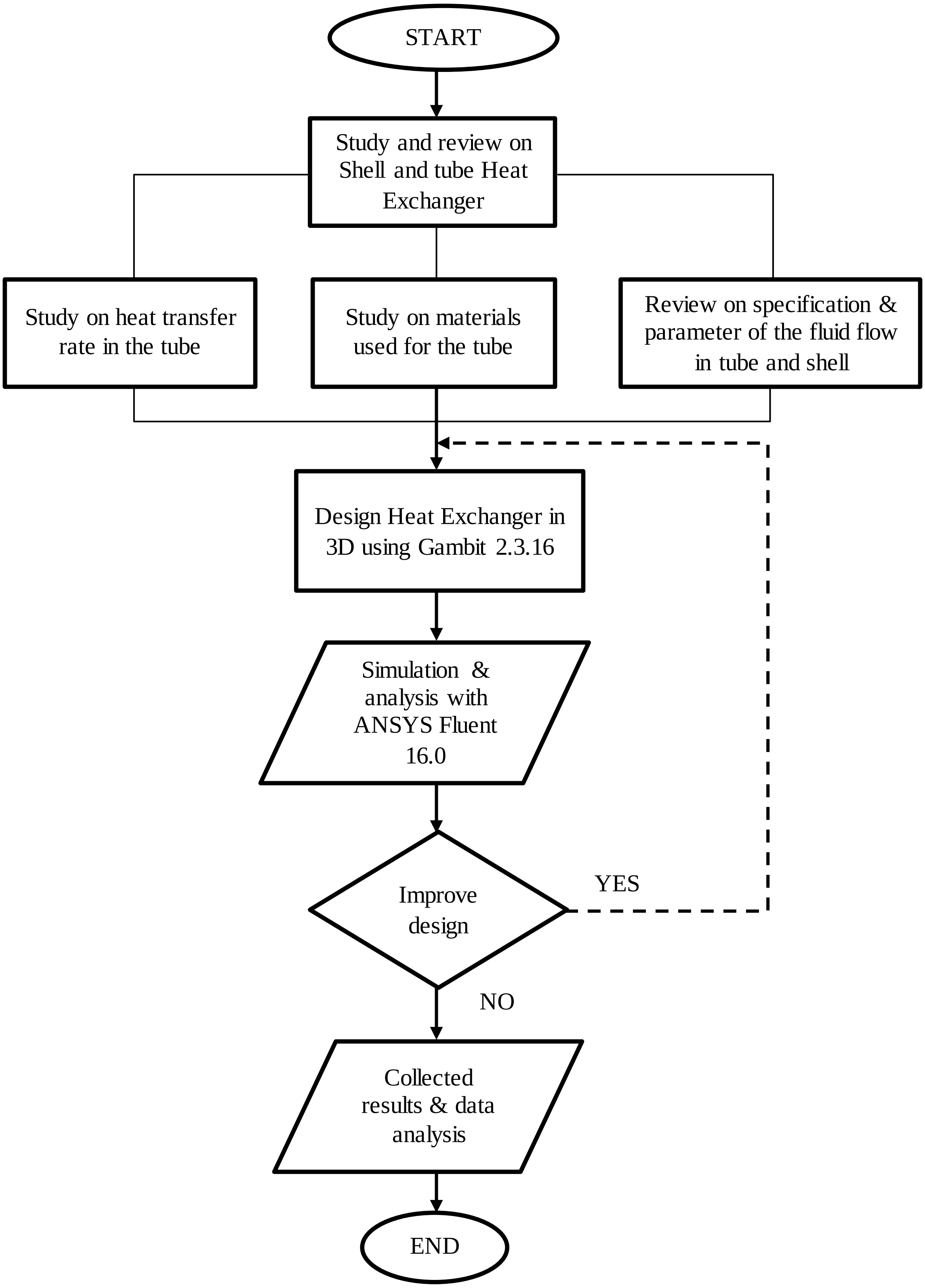


Figure 3.2 Final year project flow chart

3.3 Geometry Design

3.3.1 Overview

For this study, the design of shell and tube heat exchanger is drawn by using GAMBIT 2.3.16. GAMBIT is a software package designed to help analysts and designers build and mesh models for computational fluid dynamics (CFD) and other scientific applications. GAMBIT receives user input by means of its graphical user interface (GUI). The GAMBIT GUI makes the basic steps of building, meshing, and assigning zone types to a model simple and intuitive, yet it is versatile enough to accommodate a wide range of modeling applications [10].

GAMBIT is geometry and mesh generation software for computational fluid dynamics (CFD) analysis. GAMBIT has a single interface for geometry creation and meshing that brings together several preprocessing technologies in one environment. Advanced tools for journaling let you edit and conveniently replay model building sessions for parametric studies. GAMBIT can import geometry from virtually any CAD/CAE software in Parasolid, ACIS, STEP, IGES, or native CATIA V4/V5 formats [11].

In this study, all the dimensions for the shell and tube heat exchanger is taken from the existing heat exchanger drawing by Petronas. The General Arrangement Drawing for the heat exchanger is shown in Figure A1 in appendices. The figure had shown full technical drawing for HX220 Hot Oil – Crude Heat Exchanger.

For this case study, the dimensions taken were only for the shell and tube size [12]. Other parts like distributor, floating head and baffle were not using the actual dimensions as it will be difficult to mesh. The actual HX220 has 870 numbers of tubes, but for this study, only 4 tubes been used. There are 8 baffles used in the heat exchanger.

3.3.2 Design Specification of Heat Exchanger

The design parameters were set as follow:

Hot fluid (In shell)	=	Danol XHT
Cold Fluid (In tube)	=	Crude Oil

The dimension for the heat exchanger is shown as follow:

No of tubes	=	4**
Length of tubes	=	5507mm
Diameter of tube	=	200mm**
Thickness of tube	=	5mm**
Length of shell	=	5507mm
Diameter of shell	=	1150mm
Number of baffle	=	8
Radius of baffle	=	575mm (50% cut off) **
Height of baffle	=	575mm **

**Edited dimension from actual drawing of HX220 Shell and Tube Heat Exchanger.

The boundary condition was set as

Temperature hot inlet, T_{hi}	=	189ºC
Temperature hot outlet, T_{ho}	=	94ºC
Temperature cold inlet, T_{ci}	=	46ºC
Temperature cold outlet, T_{co}	=	75ºC
Mass flow rate of crude	=	137.2 kg/s
Mass flow rate of Danol	=	49.02 kg/s

Properties of each material are shown in Table 3.1 below

.

Table 3.1 Thermal properties of materials used in the simulation

Material	**Thermal Conductivity, K (**W/m.K)	**Density, ρ (**kg/m³)	**Specific Heat, Cp (**J/kg.K)	**Viscosity μ (**kg/m.s)	**Prandtl Number, Pr**
Copper	387.6	8978	381	-	-
Aluminum	202.4	2719	871	-	-
Carbon Steel	51.9	7870	472	-	-
Graphene	3000	2250	700	-	-
Crude Oil	0.1275	810.3	2290.53	0.00218	39.16
Danol XHT	0.139	859	1957	0.0206	290.03

3.3.4 Modelling of Shell and Tube Heat Exchanger in GAMBIT

The design setup begins with the volume of shell. Real cylinder for shell was created in centred X direction. The diameter of shell is 1150mm and length of 5507mm. The radius 1 and radius 2 were set for 575mm. The height is 5507mm. A cylinder volume was created with the size of tube which is 200mm diameter. The tube then copied and arranges them symmetrically. The distance between each tube is 400mm horizontal and vertical. In order to create holes in the shell, the volume of shell was subtracted with the 4 tubes. The geometry of the SHELL is shown in the Figure 3.3 below.

Inlet and outlet nozzle for oil was created. Volume of cylinders with diameter of 22.0mm and 30.0mm height were created. These volumes were moved to position of positive Y direction (which is y = -525 & x = 2440.5 from origin) and negative Y direction (which is y = 525 & x = -2440.5 from origin). Shell and nozzles was united to become a single solid volume. The volume was define and labelled as SHELL as shown in figure below. The geometry of the SHELL with nozzles is shown in the Figure 3.4 below.

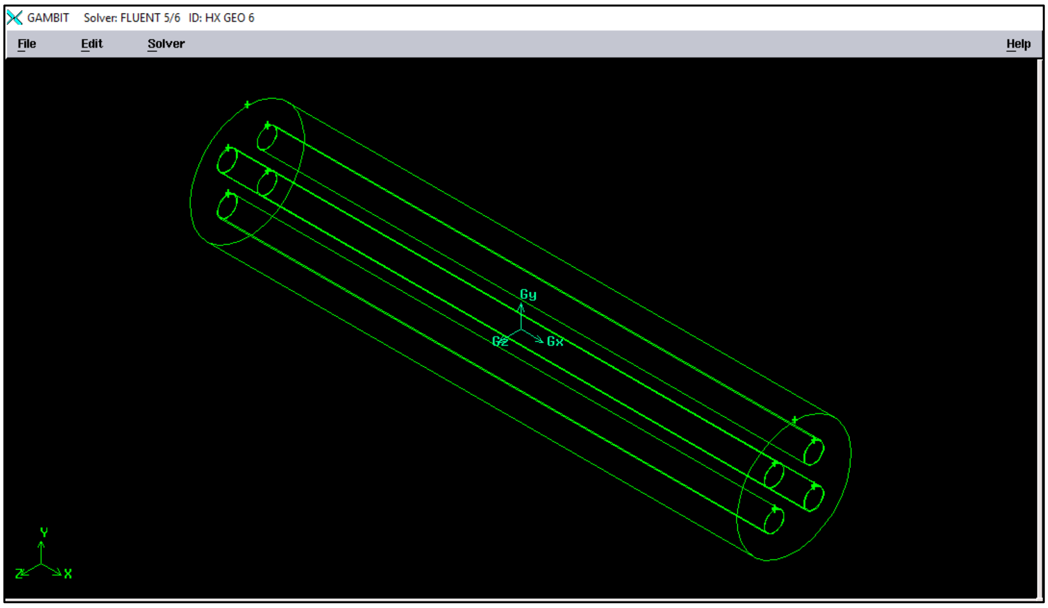


Figure 3.3 Geometry of SHELL

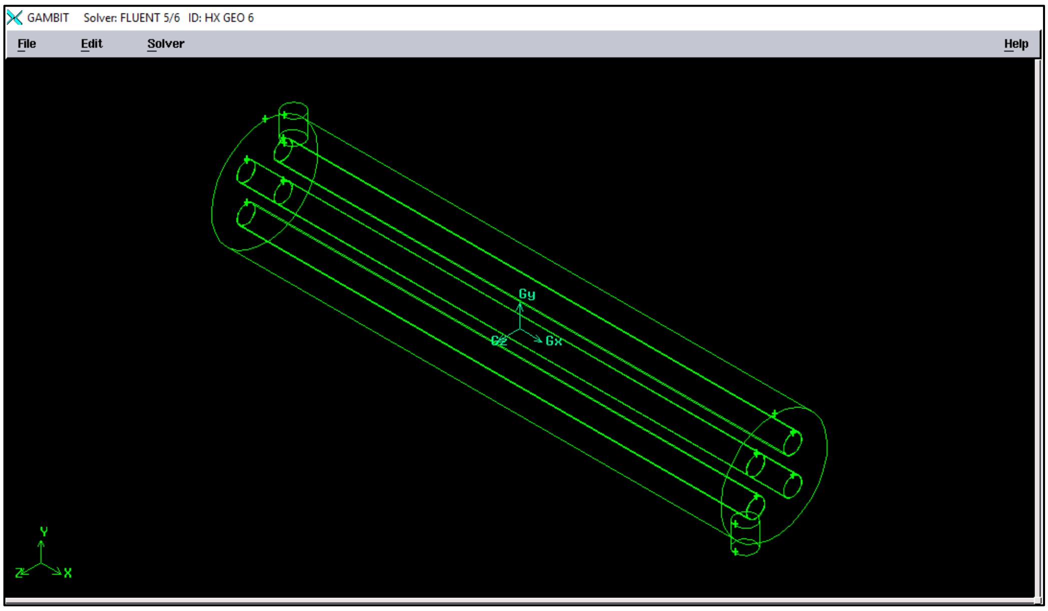


Figure 3.4 Geometry of SHELL with nozzles

After that, a cylinder tube with diameter of 20mm was created, then subtract with inner tube volume which is 190mm diameter to make a hole. The hollow volume of tube then copied to become in total of 4 tubes. Move and fixed all those 4 tubes into the holes created in the SHELL volume. The geometry of TUBES is shown in Figure 3.5 below.

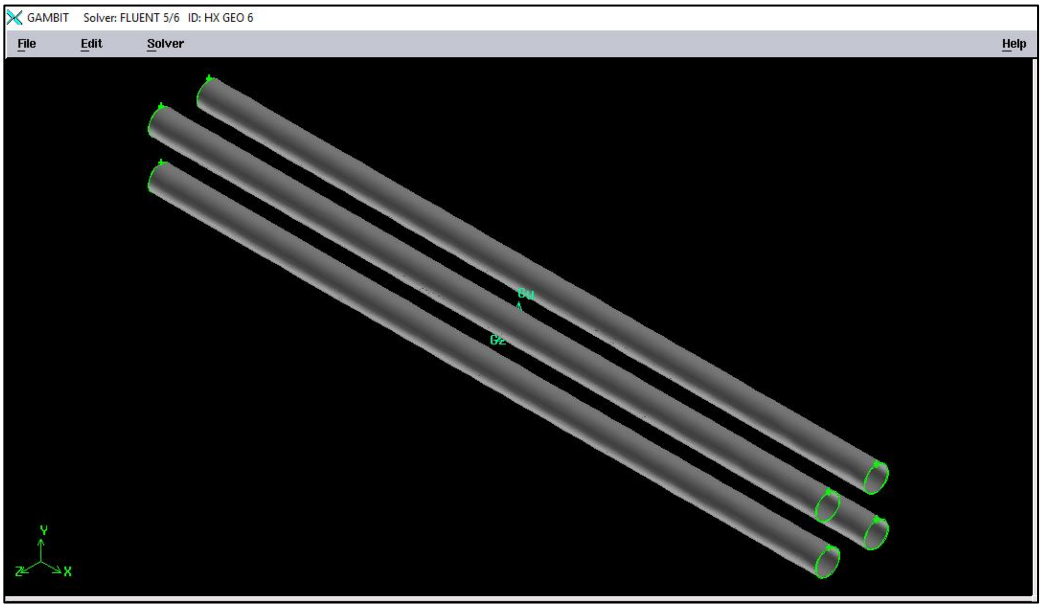


Figure 3.5 Geometry of TUBES

Next, fluid domain in the tube was created by draw solid cylinder with diameter of 190mm. The length of the INNER TUBE VOLUME was created longer than the tubes. This is because the INNER TUBE VOLUME later will be uniting with inlet channel and outlet channel to become one single body. Copy it into 4 parts then fixed in the hollow tubes. The INNER TUBE VOLUME is shown as in Figure 3.6 below.

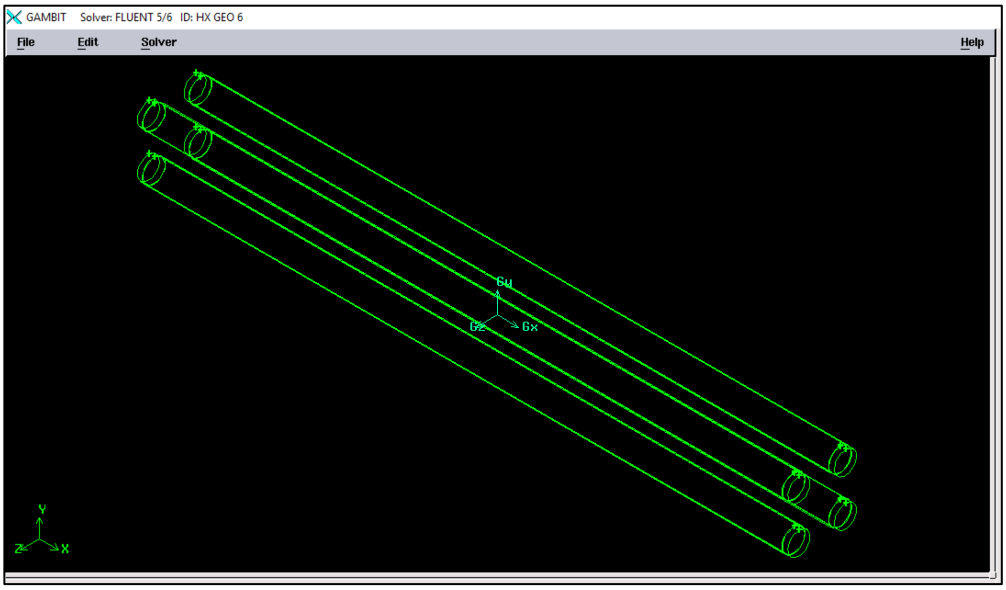


Figure 3.6 Geometry of INNER TUBE VOLUME

Then, solid cylinder with diameter of 1150mm and height of 100mm was created. This volume is the inner channel and outlet channel of the heat exchanger. The volume then moved to position x = 3253.5 and x = -3253.5 from origin. The nozzles for the inlet and outlet of crude were created. Solid cylinders with diameter of 430mm and 300mm height were created as nozzles. Move them to position of (x, y) = (-3253.5, 525) and (x, y) = (3253.5, -525). Unite the nozzles and the inlet outlet channel. The volume is shown as in Figure 3.7 below

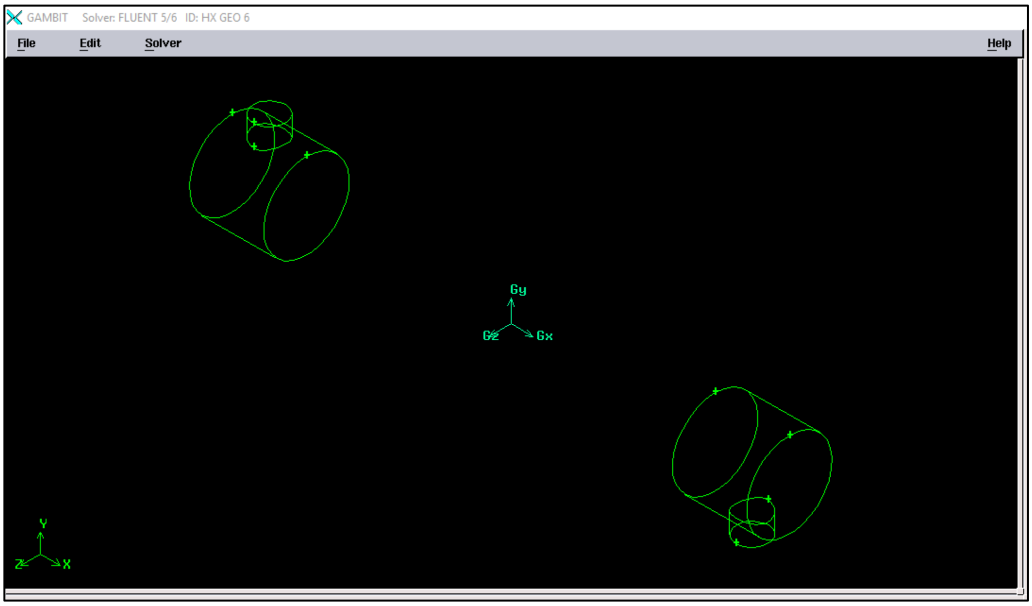


Figure 3.7 Geometry of inlet and outlet channel

INNER TUBE VOLUME, INNER CHANNEL and OUTLET CHANNEL were united to become a single body. The single volume was defined as FLUID DOMAINS. This volume is the body for the flow of crude in the heat exchanger. The geometry of the FLUID DOMAINS is shown in the Figure 3.8 below.

8 baffles were created for the heat exchanger. The radiuses of the baffles were followed as same with the SHELL's radius. The height of the baffles was modified from existing size in the drawing in figure above. The heights of the baffles is 575mm which is 50% baffle cut. Then they were arranged according to actual dimension in figure above. The baffles is shown in Figure 3.9 below.

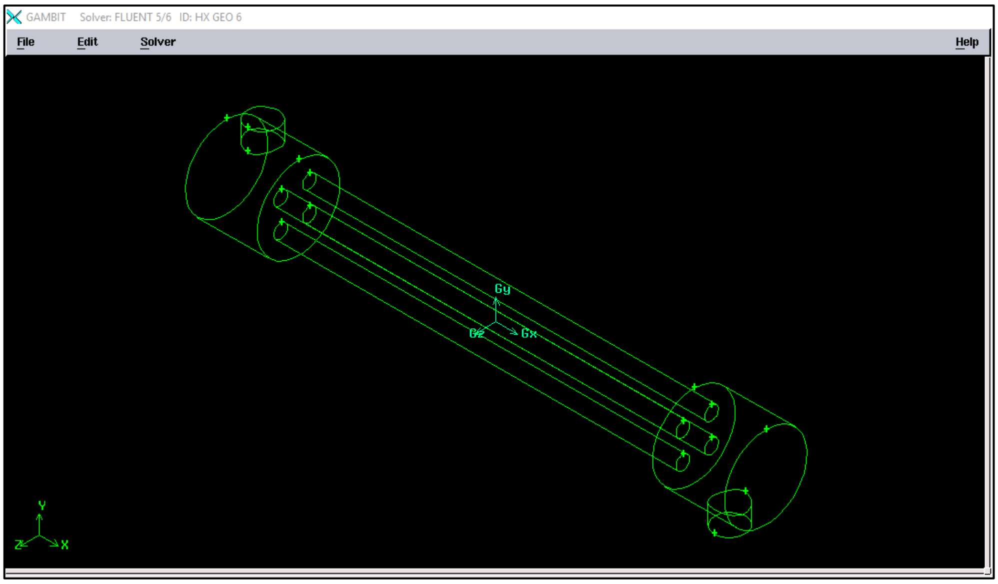


Figure 3.8 Geometry of FLUID DOMAIN

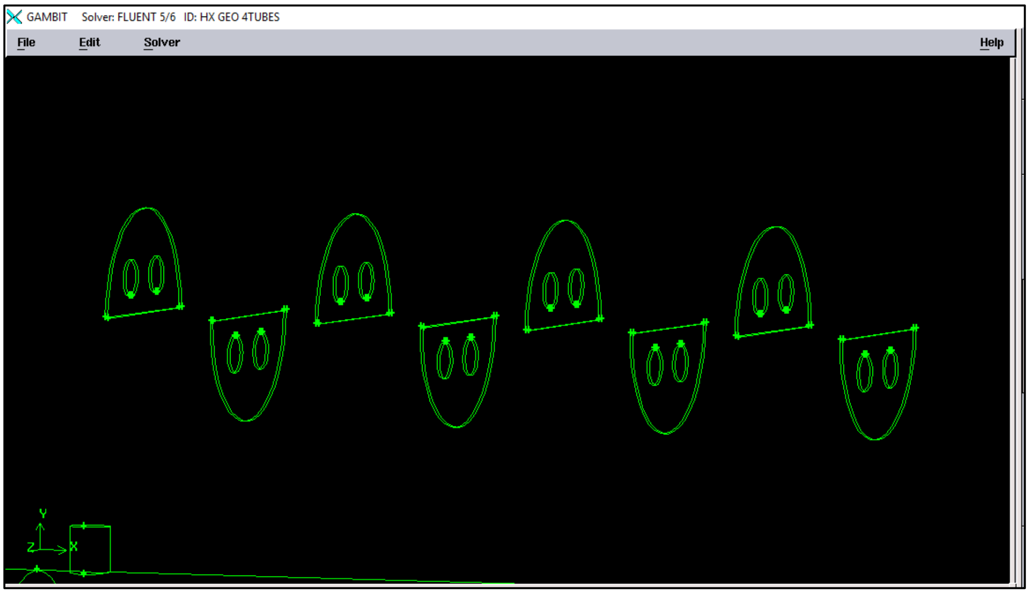
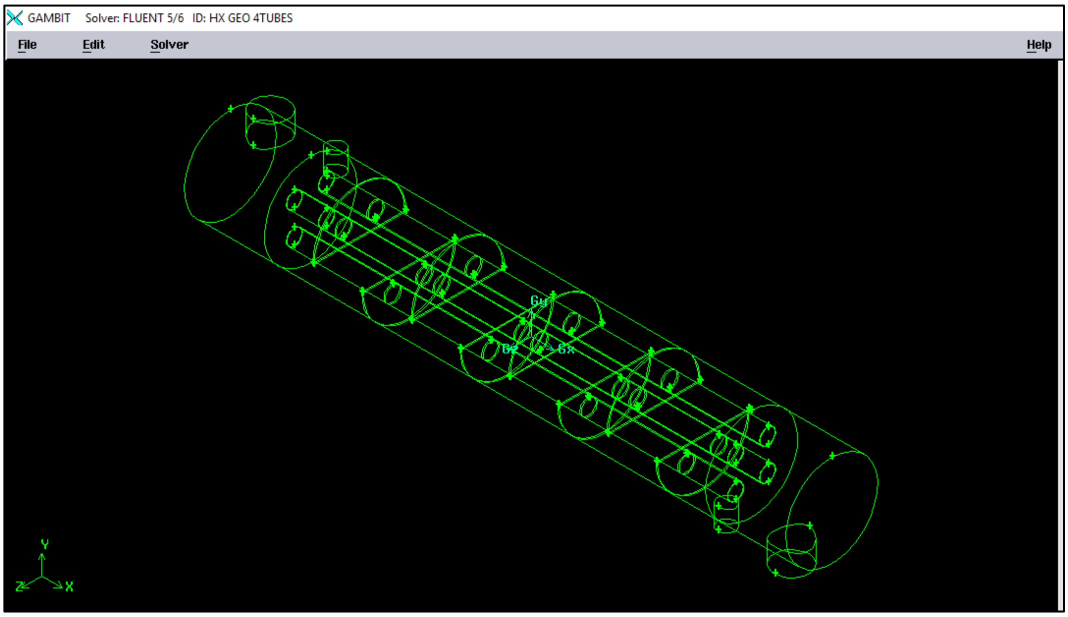
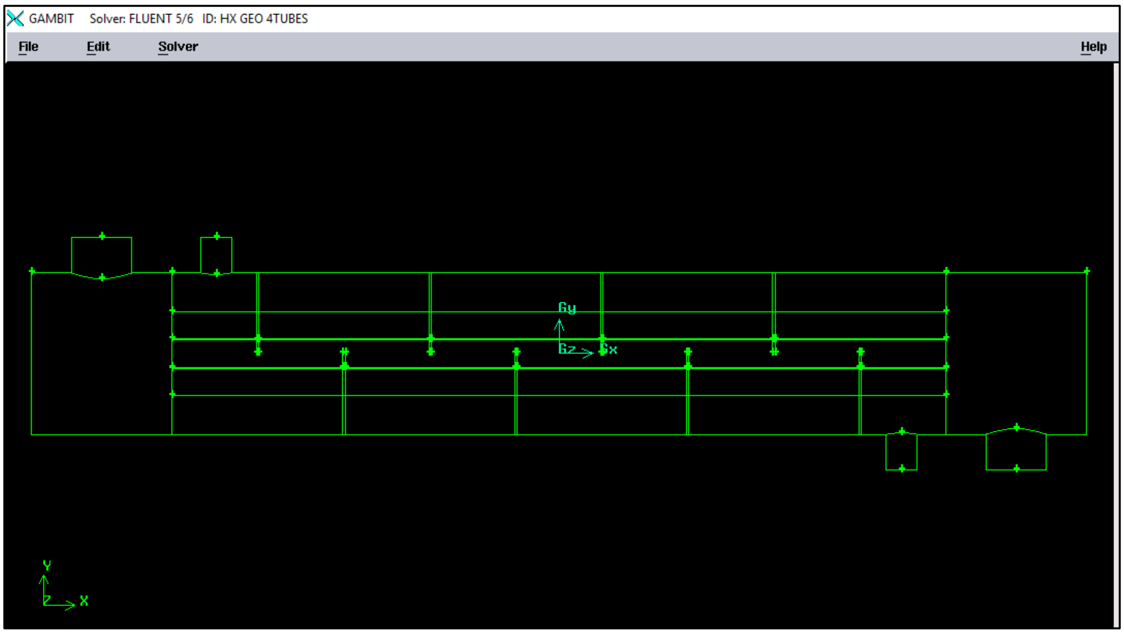


Figure 3.9 Geometry of baffles

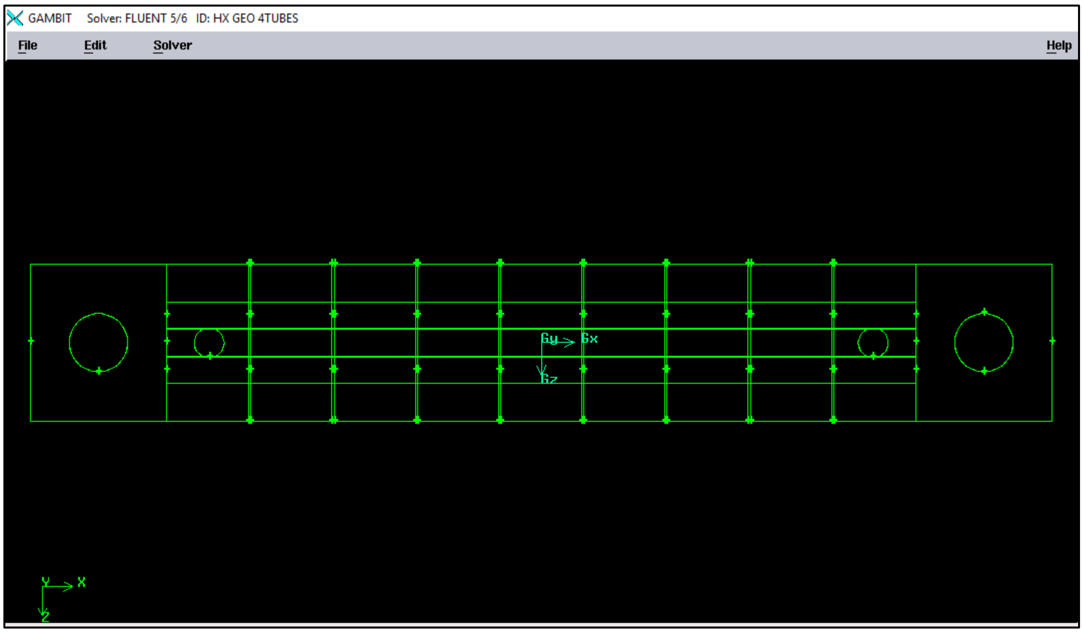
All parts in the geometry were defined in Zones Operation Setting. This command can set up to specify boundary types before import them to FLUENT. Baffles was defined as wall. The inlets for oil and crude were set as velocity inlet. The outlets for them were set as pressure outlet. SHELL and FLUID DOMAINS was set as fluid while TUBES is solid. The geometry was done and then saved as “.dbs” files type. Lastly, mesh the geometry and export as “.msh” file. Figure 3.10 below shown full design of the geometry of shell and tube heat exchanger. Figure 3.11 shows the meshing of geometry.


GAMBIT Solver: FLUENT 5/6 ID: HX GEO 4TUBES
File
Edit
Solver
Help
Gy
Gx
Y
Z
X

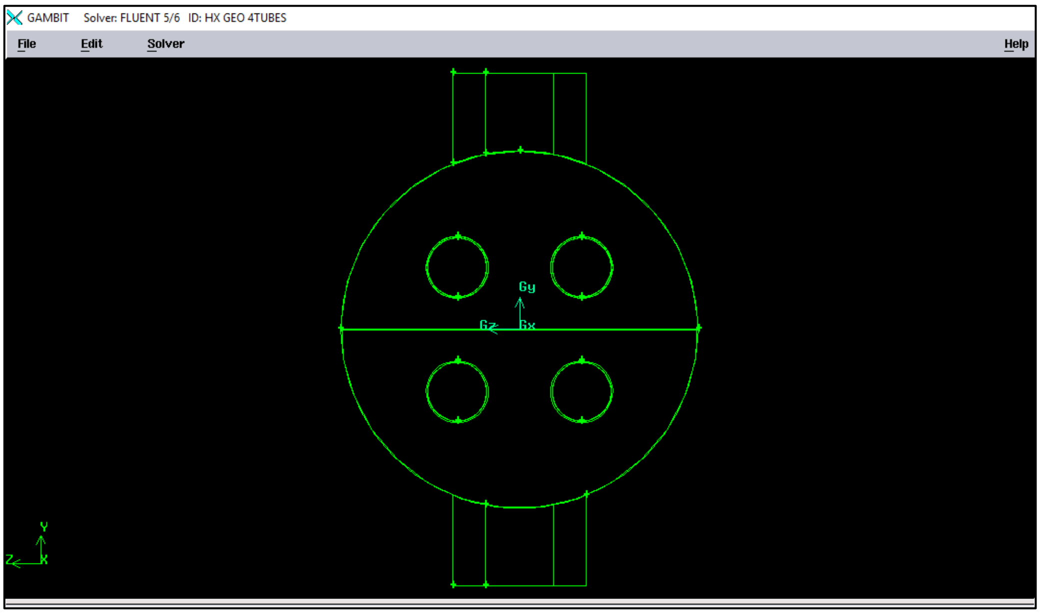
(a)


GAMBIT Solver: FLUENT 5/6 ID: HX GEO 4TUBES
File
Edit
Solver
Help
Gy
Gz
Gx
Y
X

(b)

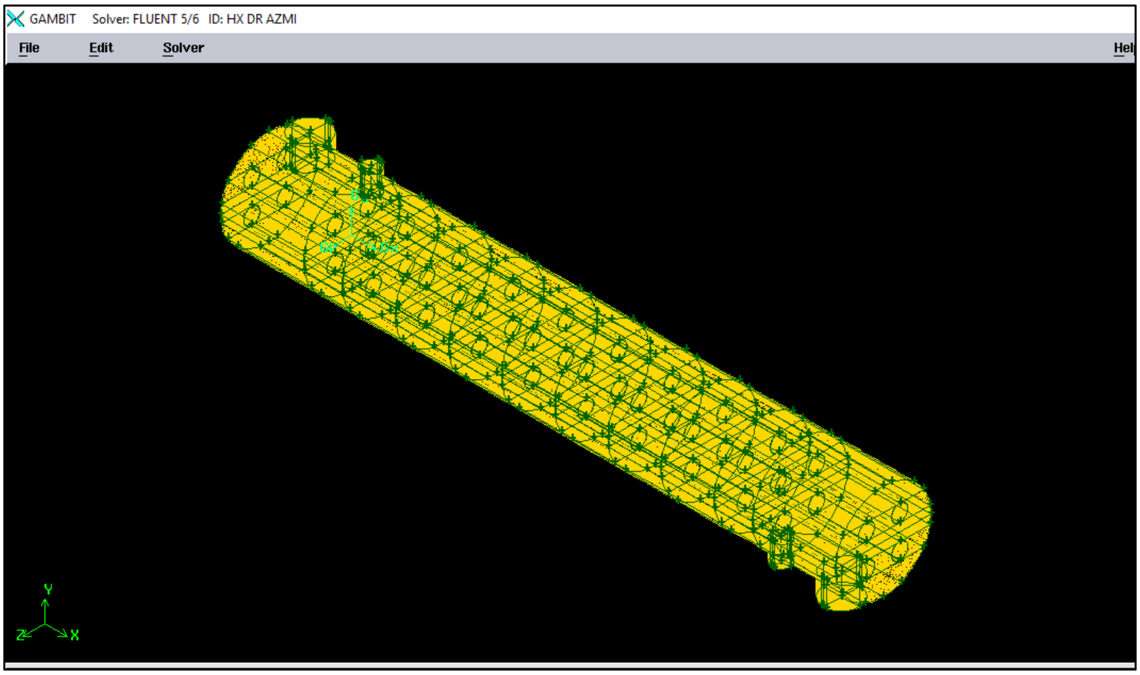


(c)

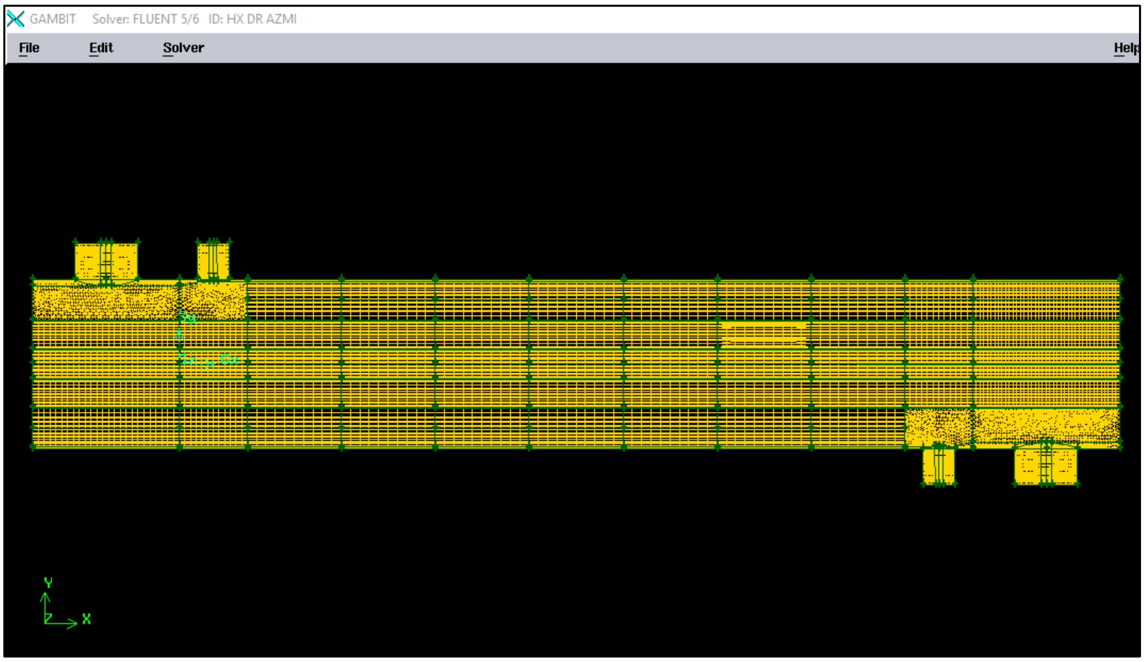


(d)

Figure 3.10 Geometry of heat exchanger (a) isometric view, (b) Side view, (c) Top view, (d) Front view


GAMBIT
Solver: FLUENT 5/6 ID: HX DR AZMI
File
Edit
Solver
Y
Z
X

(a)


GAMBIT
Solver: FLUENT 5/6 ID: HX DR AZMI
File
Edit
Solver
Help
Y
X

(b)