# NUMERICAL STUDY OF PLATE HEAT EXCHANGER FOR EREMA MACHINE


By:

**MUHAMAD HAIDAR BIN AWANG**

(Matrix no.:128950)

Supervisor:

**Dr. Mohd Azmi Bin Ismail**

Industry Supervisor:

**Mr. CK Cheah**


IN COLLABORATION WITH

PENFIBRE (FILM DIVISION) SDN. BHD.

29 May 2019

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

I hereby certified that this thesis which I now submit for assesment is entirely my own work, that I have reasonable care to ensure thet the work is original and does not to the best of my knowledge breach any law of copyright and has not been cited and acknowledged within the text of my work.

Name of student : Muhamad Haidar Bin Awang Signature :

Date : 29 May 2019

Witnessed by

Supervisor : Dr. Mohd Azmi Bin Ismail Signature :

Date : 29 May 2019

# ACKNOWLEDMENT

First of all, I would like to express my deepest appreciation to my supervisor Dr. Mohd Azmi Ismail for his supervision of my final year project. His non stop support and assistance for my final year project were from the initial to the final stage of my final year project was really helpful for me. This dissertation would be impossible to finish without his encouragement, patient guidance and constructive suggestions throughout the whole period. During this time, not only I learnt about the heat exchanger but managed to enhance my knowledge, positive attitude and other skills through him.

Moreover, I would like to thank my supervisor from Penfibre (Film Division) Sdn. Bhd, Mr CK Cheah for his useful informations, advices and assistance during my visit to the factory. He always keep me up-to-date about the project of heat exchanger for EREMA. I really appreciate his willingness to support me through the whole period of my final year project. Sincere thanks are also extended to all technician that involved for given technical support.

Besides, I would like to express my gratitude to both of my parents and all my family for their non stop moral support, encouragement for me mentally and spiritual and their time to help me with my research. A big thanks to my fellow coursemates for sharing their knowledge and ideas in the progression of this project.

Lastly, I want to express my sincere thanks to all lecturer involved in my final year project whether directly or not. Their guidances and opinions extremely valuable and help me a lot through my final yea project. This research project had become one of my biggest success and a big milestone in my career development. I will work hard and strive to use gained skills and knowledge in the best way and I will continue to improve in order to achieve my objective in my career.

TABLE OF CONTENTS

# Table of Contents

# LIST OF TABLES

## LIST OF FIGURES

# LIST OF ABBREVIATIONS

| | |
|---|---|
| FAV | Finite Volume |
| Re | Reynold’s Number |
| CAD | Computer Aided Design |
| FEA | Finite Element Analysis |
| CFD | Computational Fluid Dynamics |
| SST | Shear Stress Transport |
| FSI | Fluid Structure Interaction |

## ABSTRAK

Mesin penukar haba banyak digunakan untuk mengawal suhu sesuatu mesin yang lain. Bagi mesin penukar haba yang digunakan di EREMA, air digunakan sebagai agen penyejuk untuk air takungan. Mesin penukar haba yang digunakan (53 plat) di EREMA tidak mempunyai kapasiti yang mencukupi. Jadi, lebih banyak plat akan ditambah pada mesin penukar haba (115 plat). Kapasiti mesin penukar haba yang baru daripada HISAKAWORKS S.E.A Sdn Bhd (115 plat) perlu ditentukan kapasiti nya. Tujuan projek ini adalah untuk mendapat suhu pada saliran keluar untuk air takungan supaya dapat menetukan kapasiti mesin penukar haba itu. Mesin penukar haba inii dianalisisi secara numerikal. Simulasi ini dijalankan menggunakan ANSYS 19.0 Fluent satu hala FSI. Suhu pada saliran keluar diperoleh dan kadar tukaran haba dikirakan. Dalam kajian ini juga, keputusan yang diperoleh dari simulasi itu dibandingkan dengan data yang diperoleh daripada vendor dan didapati tidak sama.Dengan menggunakan kan kondisi sempadan yang sama, suhu pada saliran keluar didapati sebnayak 23 ˚C namun suhu yang didapati darpida vendor ialah 16 ˚C Jadi, suhu padar saliran masuk ditukar untuk mendapat suhu yang dikehendaki oleh takungan air iaitu 20 ˚C. Hasil dan kepututsan yang diperoleh akan digunakan untuk memperbaiki prestasi mesin penukar haba kelak.

# ABSTRACT

Heat exchanger is a device widely used to control the temperature of the some machines.For the EREMA's heat exchanger, the fluid used is water as a cooling fluid for the water bath.The current heat exchanger (53 plates) used in EREMA do not have enough capacity to maintain the temperature of water bath at 20˚C. The plan is to add more plates to the heat exchanger. The capacity of the new heat exchanger designed by HISAKAWORKS S.E.A Sdn Bhd (115 plates) need to be determined. The purpose of this project is to determine the outlet temperature of the new heat exchanger to be used in EREMA in order to determine the capacity of the heat exchanger. The heat exchanger is analysed numerically. The simulation is carried out using ANSYS 19.0 Fluent by one-way FSI. The outlet temperature is obtained thus the heat transfer of the heat exchanger can be calculated. Also in this study, the result is compared with the vendor's data for validation and there are some differences are noticed.With the same boundary conditions, the outlet temperature obtained for water bath is 23 ˚C where the vendor's data is 16 ˚C The inlets boundary condition is changed in order to obtain the required temperature for the water bath 20˚C. These result will serve as a basis for heat exchanger design to improve the heat transfer rate.

# CHAPTER 1 : INTRODUCTION

## 1.1 Overview

Application of heat exchangers in industry as a cooling or heating devices are very wide. 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 fluids, at different temperatures and in thermal contact [1]. There are many type of heat exchanger such as spiral tube heat exchanger, double pipe heat exchanger, plate-type heat exchanger and etc. The one that we are concern about is gasket plate heat exchanger.

The gasket plate heat exchanger have many advantages compare to other type of heat exchanger such as high heat transfer coefficient, high thermal coefficient, the higher efficiency and the smaller space occupancy compare to shell and tube heat exchanger, less pollution, ease of maintenance, no vibration and low possibility of mixing of fluid [2].

It is consist of a number of thin rectangular metal plates sealed around the edges by gaskets and held together in a frame as shown in figure 1.1 below. Each plate is made by stamping or embossing a corrugated (or wavy) surface pattern on sheet metal. On one side of each plate, special grooves are provided along the periphery of the plate and around the port for a gasket. Alternate plates are assembled such that the corrugation on successive plates contact or cross each other to provide mechanical support to plate pack through a large number of contact points.

There are few way to increase the performance of the plate heat exchanger and one of them is by adding the number of plate that will increase the heat transfer area. Increasing the heat transfer area will subsequently increase heat transfer rate as it is proportional to each other as shown in equation 1.1.

$$q = UA\Delta T_m \qquad \text{Equation 1.1}$$

Where,

q = heat transfer rate

U = overall heat transfer coefficient

A = heat transfer area

$\Delta T_m$ = mean temperature different

## 1.2 Research Background

### 1.2.1 EREMA

EREMA is a plastic recycling system which will reprocess the flakes from grinder (that grind the rejected film) into pellet chip. The process involve in EREMA system are shown in figure 2.1 below. The flakes form flakes hopper will go to scale hopper for weighing process and then it go inside the cyclone to separate the tiny particles and powder from flakes. Then flake will go inside extruder through circle feeder (to feed the flakes inside shredder) to become molten polymer and eliminate the air bubbles in the molten polymer. Then molten polymer is forced out through a die and shaped into filament form (strands). These strands are quenched in the water bath to quickly cool down their temperature. After that the blower will eliminate the water from strands and the strands are cut into chip. The screen vibrator will then filtered out the uneven cut polymer chip. Finally a blower will sent the chips to pellet silo which will used for as raw material for film making process.

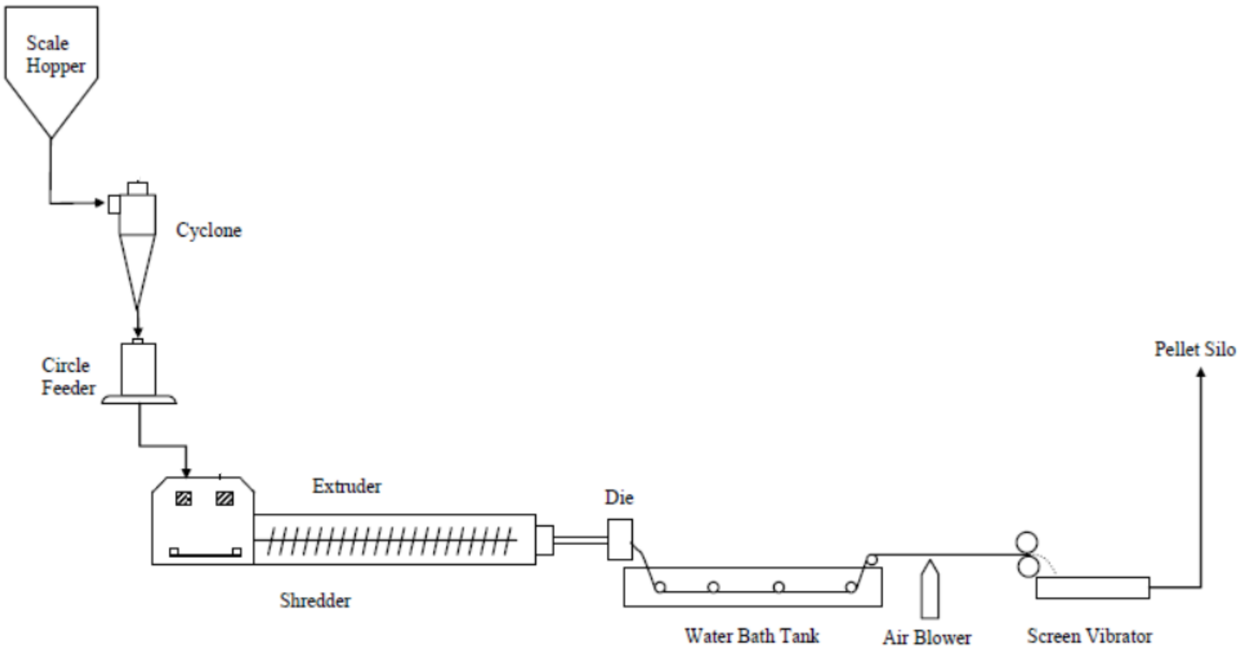


Figure 1.1 : Flow chart of EREMA

For EREMA, it is crucial to maintain the water bath at desirable temperature in order to keep the product in its best quality as for Penfibre (Film Division), are committed in producing high quality of film. If the temperature of water cannot be maintained well at setting temperature, the quenching process will disturbed and the quality of the chip produced will be effected. The strands that coming out of die need to be quenching at certain temperature that will result amorphous structure of the polymer (heat treatment) for certain reasons. [3].

The water bath temperature needed to be maintain in range of 20 ˚C ~ 25 ˚C. The method used for cooling this water bath is by using gasket plate heat exchanger (liquid-to-liquid cooling). The water bath will absorb heat from the polymer (quenching). After that the motor pump will pump the water to flow inside the heat exchanger. This heat exchanger used cooling water (14 ˚C) to absorb heat from the water (water bath) and that water will flow back to water bath. The schematic diagram is shown below.

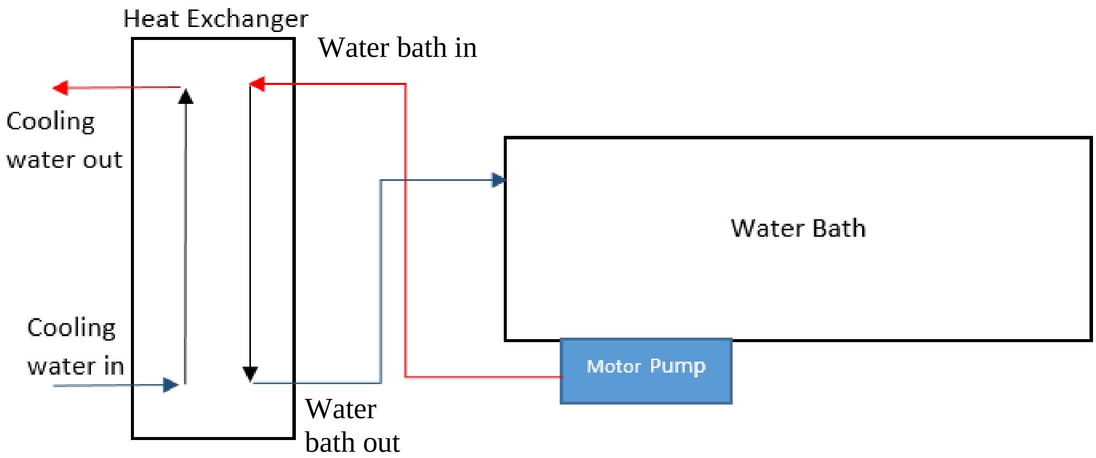


Figure 1.2: Water flow direction of water bath and heat exchanger

## 1.3 Problem Statement

Current heat exchanger used for cooling the EREMA water bath is not enough to reduce the water bath temperature. The setting temperature is 20 ˚C but the resulting temperature is around 25˚C – 28˚C. Then after a period of time, the temperature trend will keep rising which can reach 31˚C. It show that the current EREMA water bath heat exchanger capacity is not enough to maintain the water temperature at low temperature (20˚C) for a long period of time. Theoretically explained by the vendor that adding more plate to 115 plates to the EREMA heat exchanger can increasing its cooling capacity and reduce outlet temperature to 16 ˚C. The data provided by the vendor need to be validate.

## 1.4 Aim and objectives

The aim for this project is to measure the capacity and performance the new designed of ERAMA heat exchanger. The objective are:

- To study and understand about plate heat exchanger
- To modelling the designed plate heat exchanger that used for EREMA water bath
- The simulate the modelled heat exchanger
- To measure the performance of this heat exchanger
- To validate the simulation data heat exchanger with the current on site heat exchanger and with vendor data

## 1.5 Scope of the project

This project is to find the hot fluid (water bath) outlet temperature of the heat exchanger used in EREMA that will give the capacity of the heat exchanger. Therefore, the project will cover the scope as follow:

1) Collecting data of the design of the heat exchanger and its plates that consist of type fin, heat transfer area, dimension of the plates and material of the plates.

2) Collecting data of the fluid involve in the heat exchanger such as the inlet temperature for both hot and cold fluid and volume flow rate.

3) Calculating the velocity of the fluids at the intlets, mass flow rate and Reynold's number to determine the type of flow.

4) Modelled the design of the EREMA heat exchanger using the SolidWork 2017.

5) Modelled the design of the plates in the design modeller in Ansys with the heat transfer area of 0.2957 $m^2$.

6) Simulating the performance of the heat exchanger by using ANSYS 19.0 and obtain the outlet temperature of the water bath.

7) Determine the heat transfer rate for the heat exchangera and compare the results provided by the vendor and on-site results..

## 1.6 Outline of the reports

This thesis consist of five main chapters. The first chapter discusses on the heat trasnfer rate principles, working principles of the EREMA and overview of the heat exchanger. The problem statement regarding the heat exchanger also discussed in chapter one. The factor that affecting the heat transfer rate also discussed in this chapter. Lastly, this chapter also has the project scopes, project objectives, and outline of the project.

In chapter two, the literature is reviewed based on the factor affecting the capacity of heat exchanger and the modelled to be used in CFD.

For chapter 3, the methodology on modelling the heat exchanger design using SOLIDWORK 2017 and DesignModeler will be presented. The setting up of the simulation which is using ANSYS 19.0 also will be presented.

For chapter 4, the results of the outlet temperature and the performance of the heat exchanger in term of heat transfer rate will be discussed and verified with data from the vendor.

In the last chapter, the conclusion will be made in chapter 5. Some recommendations and suggestions for future research will be proposed and discussed.

# CHAPTER 2
# LITERATURE REVIEW

## 2.1 Optimum working conditions for heat exchanger

In Turkey, there was a study where 8 three-dimensional computer-assisted models are considered, with 15˚, 30˚, 45˚, and 60˚ of grooved angles, parallel/counter flow and cross/counter flow of the heat exchangers. The thermodynamic characteristic are determined by CFD method, the material selected to be AISI stainless steel and titanium which are used in geothermal applications and for the fluid the water and geothermal fluid are used. So, 32 models are have been prepared and examined numerically. As a result, the optimum conditions for maximizing the performance of the plate heat exchanger have been studied. It is found that the best operating conditions for heat transfer are a plate heat exchanger model with the parallel counter flow, water-fluid, titanium and 60˚ groove angle [2].

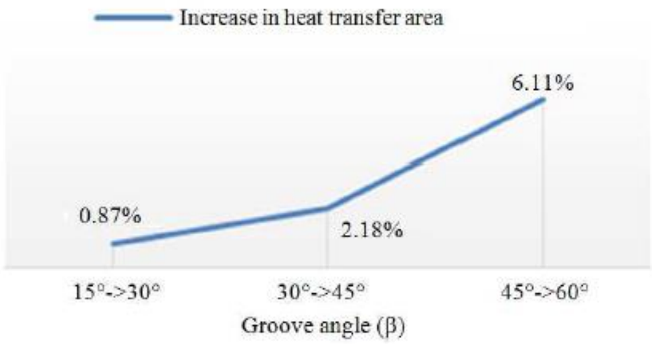


Figure 2.1: Percentage increase of heat transfer area by groove angle.

Study done by Kan (2014) to investigate optimum operating conditions of the plate heat exchanger. 9 models are simulate with the grooves angle of 30˚, 45˚ and 60˚ at the flow rates of 0.2 kg/s, 0.3 kg/s and 0.43 kg/s. By CFD analysis, the best thermal performance obtained at the grooving angle of 60˚ with flow rate of 0.2 kg/s. He conclude that the lower flow rate will have the best performance in thermal [4].

## 2.2 Turbulence modelling

The most important factor in achieving heat transfer in the plate heat exchanger is turbulent flow due to complex geometry. Therefore the turbulence model used in analyses has been chosen at an advanced level. In the study, Reynold stress model which is a model that can be physically counted as turbulence model is used. The Reynold stress model has the most appropriate modelling potential to accurately solve complex flows and takes into account flow curvature, rotation, and high-stress ratios when solving [2]

Another study by Sahan 2001, the thermal performance is increased by means of turbulent or mixed flow. The parallel or cross flow of the water flow directions are closely related to the thermal efficiency of the heat exchanger. Due to the horizontal, vertical or angled production of water circulation channels formed on the channels formed on the plate surfaces, the water flows can be supported by the parallel flow and the cross flow, as well as the mixed flow, so that the thermal efficiency can be maximized [5]

## 2.3 Effect of pressure drop on thermal performance

Chen and Chen (2006) investigated the effects on heat transfer and pressure drop of heat exchanger of five experimental factors consisting of flow rate, number of blades, blade thickness, number of pipes, number of plates and plate angle using Taguchi method. The amount of fluid, number of blades, blades thickness and number of plates were found to affect the performance of the heat exchanger as a basis and they determined that the parameter outside them played an important role in the design of the optimum heat exchanger [6].

## 2.4 Effect of Reynold’s number on thermal performance

There are many studies examining the optimum working condition of the heat exchanger. From the Arturk study, they experimentally and theoretically investigated the thermal performance of a plate heat exchanger under variable Reynolds number in their work. They used the SST k-w turbulence model for the simulation. The CFD codes are very effective for optimum design of plate heat exchanger in different geometric configurations. Their results are showed that the higher the Reynolds number of the stream, the higher the thermal performance [7].

Another study from Freund and Kabelac tried to determine the regional heat transfer coefficient in plate heat exchanger with IR thermal camera and CFD analysis. They found that different region have different heat transfer coefficient formed and they are proportional to Reynolds number. In addition, they come to the conclusion that the most suitable heat exchanger geometries can be found by CFD analysis in order to make CFD analyses using different Reynolds turbulence models [8].

# CHAPTER 3
# METHODOLOGY

## 3.1 Overview

In order to study the capacity of new heat exchanger in EREMA, this project is prepared for the simulation of the fluid flow and heat transfer on the heat exchanger. This new heat exchanger have a total of 115 plates compared to old model with only 53 plates due to not enough cooling capacity. In this study, numerical analysis are solved by finite element method, three dimensional, time independent and combined (conduction and convection) heat transfer apporach.

## 3.2 Data collection

The data about EREMA heat exchanger is collected and studied. Some of calculations are done to obtain overall information of EREMA heat exchanger. The model of the heat exchanger is SX – 194A – TNHP – 115. The details of the model are as explain below. It is based on the HISAKAWORKS S.E.A Sdn Bhd Plate Heat Exchanger Operation and Installation Manual

Table 3.1: Design of the plate

| No. | Model Name | Description |
|---|---|---|
| 1 | SX | • Herringbone pattern<br>• Pitch is fine and shallow<br>• Heat transfer plate mode |
| 2 | 1 | • The size of heat transfer plate (from 0 to 16) |
| 3 | 9 | • Chevron angle of heat transfer pattern (0 to 7 is mixed arrangement) |
| 4 | 4 | • Plate thickness which is 0.4mm |
| 5 | A | • Method for channeled gasket installation (A for slit – in and glue less) |
| 6 | TN | • Metal bended covering connection |
| 7 | H | • Class of frame resistance to pressure ( H for middle pressure) |
| 8 | P | • Universal frame model |
| 9 | 115 | • Show the number of plates which is 115 plates used |

Table 3.2: Important parameter of the heat exchanger

| | |
|---|---|
| Heat transfer area | 0.2986 $m^2$ |
| Hot inlet temperature | 35 ˚C |
| Cold inlet temperature | 14˚C |
| Plate’s material | SUS304 |
| Gasket’s material | Nitrile rubber |
| Flow rate | 20 $m^3$/h |

## 3.3 Computer modelling

According to the real design of the heat exchanger, it is quite complex and it involve a total of 115. Some simplification had to made on the model to aid and to ensure this simulation will run smooth due to limitation on the computer used as to run all 115 plates with such complex geometries will need a much powerful computer. So two model prepared which the 1$^{st}$ model is for better understanding on the heat exchanger and the second model is for the simulation. For the simulation, it will involve four basic steps. First, drawing the model, meshing the element, setting up the boundary conditions and computing the heat transfer and fluid flow. This project utilized the computer – aided drawing (CAD) program, SolidWorks and DesignModeler to draw and model the heat exchanger. The rest of simulations were done using commercial computer fluid dynamics (CFD) software, ANSYS Fluent.

### 3.3.1 Modelling using SOLIDWORK 2017

The geometry and design of this heat exchanger were produce by HISAKAWORKS S.E.A Sdn Bhd which have a major different between old heat exchanger in term of plates’ number. The model name for this heat exchanger is SX – 194A – TNHP – 115. Interm of the design of the plates, the drawing is not provided by the vendor Hisaka due to confidential issue. So, the plate is modelled based on the model number and the drawing of the cover plate which is given by Hisaka Ltd. Figure below show the model of the plate using SOLIDWORKS 2016.

From the deatail drawing of heat exchanger, the body, frame and support of the heat exchanger are shown

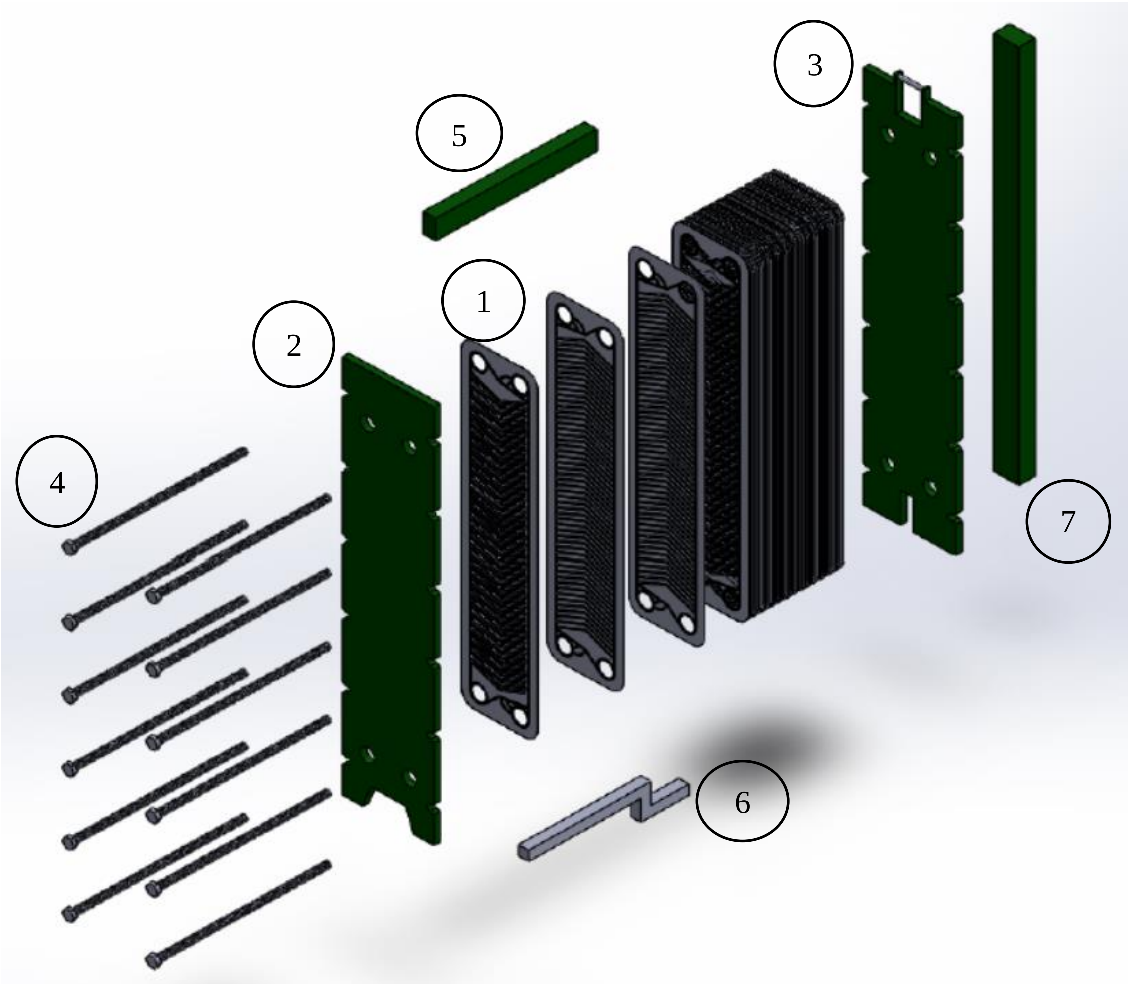


Figure 3.1: Exploded view of heat exchanger

Table 3.3: Part name with numbering

| No | Part Name |
|---|---|
| 1 | Plate |
| 2 | Front casing |
| 3 | Back casing |
| 4 | Bolt and Nut |
| 5 | Top support |
| 6 | Bottom support |
| 7 | Back support |

Calculation below using the information provided by the Penfibre (Film Division) Sdn. Bhd. The flow of the fluid is turbulence flow with Reynold's number of 62956 which is higher than 1400 (laminar flow for flow over plates).

Characteristic length, L = $5.99 \times 10^{-4}$m
Volume flow rate, Q = 20 $m^3/h$
= 0.005556 $m^3/s$
Kinematic viscosity of water, v = $0.801 \times 10^{-6}$ $m^2/s$
Cross-sectional area, A = (0.0003 x 0.220) $m^2$
= 0.000066 $m^2$

$$Re = \frac{QL}{vA}$$

$$Re = \frac{(0.005556)(5.99 \text{ x } 10^{-4}\text{m})}{(0.801 \times 10^{-6})(0.000066)}$$

$$Re = 62952$$

### 3.3.2 Design using DESIGN MODELLER

DesignModeler in ANSYS 19.0 is used to model the plates. The actual design deisgn of the plate is unknown. So some assumption is made and a simplified design is modelled for better mehsing accuracy.

For this model, the heat transfer area obtained for single plate is 0.28125 $m^2$. From the data gained the heat transfer area for single plate is 0.2987 $m^2$. The difference is small and negligible for this case. The angle of the fin also been simplified to be a horizontal fin. This is to ensure the meshing process will be faster and more accurate.

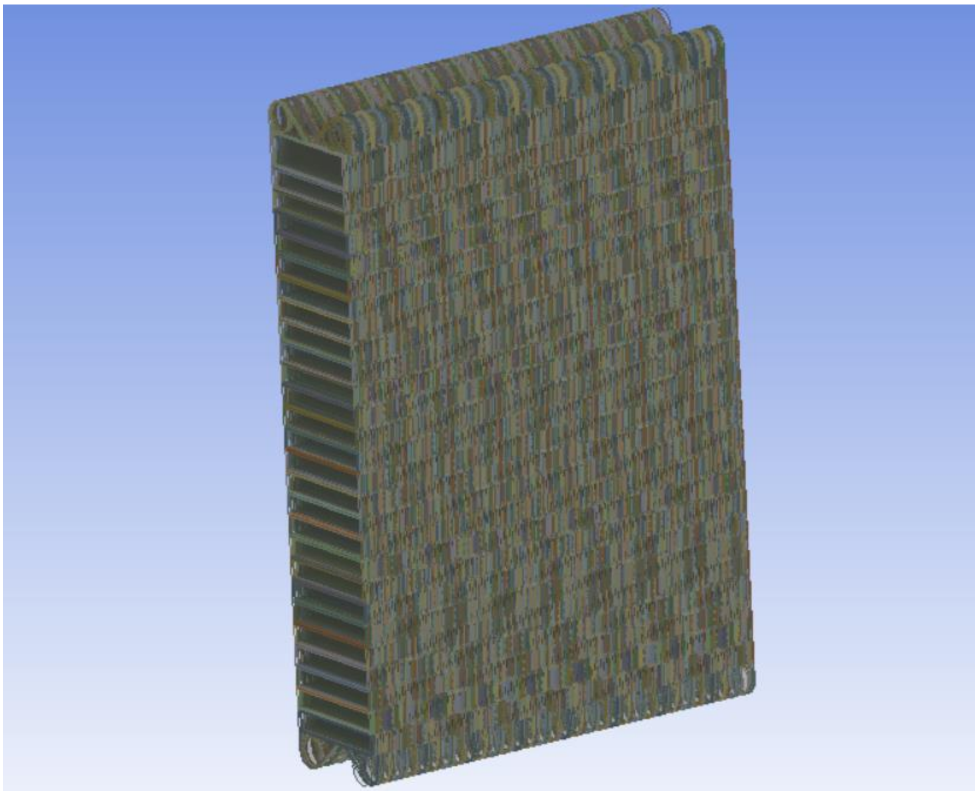

Figure 3.2: Isometric view of the plates involved for heat exchanger

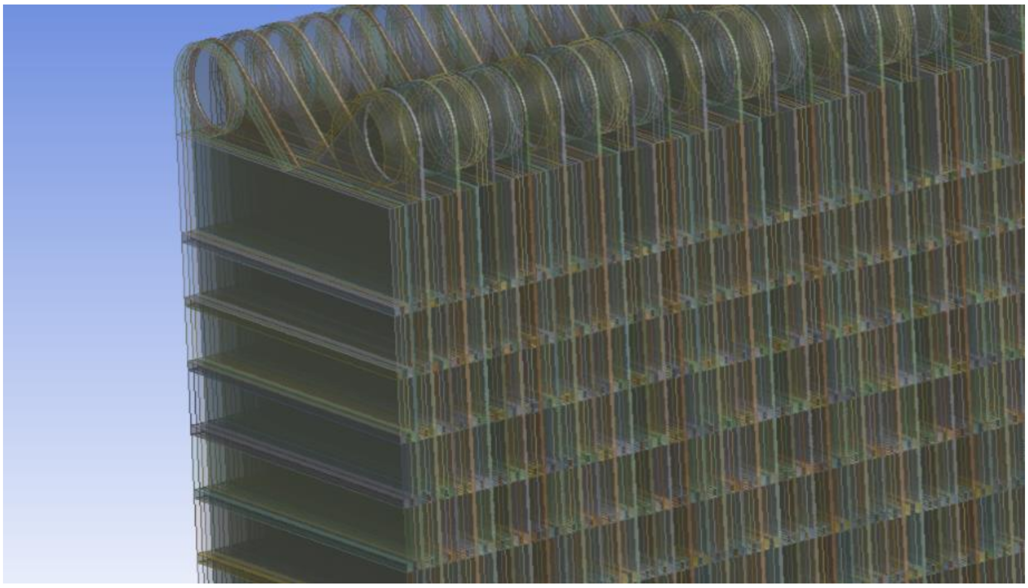

Figure 3.3: Close view of the plates

## 3.4 Computational simulation procedure

### 3.4.1 Meshing

All parts of the geometry are meshed and the type of meshed used is automatic mesh with edge sizing and multizone of hexahedron meshing and the setting used are as below.

| Details of "Mesh" | |
|---|---|
| **Defaults** | |
| Physics Preference | CFD |
| Solver Preference | Fluent |
| Element Order | Linear |
| Element Size | 2.5e-003 m |
| Export Format | Standard |
| Export Preview Surface Mesh | No |
| **Sizing** | |
| Use Adaptive Sizing | No |
| Growth Rate | Default (1.2) |
| Max Size | Default (5.e-003 m) |
| Mesh Defeaturing | Yes |
| Defeature Size | Default (1.25e-005 m) |
| Capture Curvature | Yes |
| Curvature Min Size | Default (2.5e-005 m) |
| Curvature Normal Angle | Default (18.0°) |
| Capture Proximity | No |
| Bounding Box Diagonal | 1.5034 m |
| Average Surface Area | 2.6337e-003 $m^2$ |
| Minimum Edge Length | 2.e-003 m |

Figure 3.4: Details of the mesh

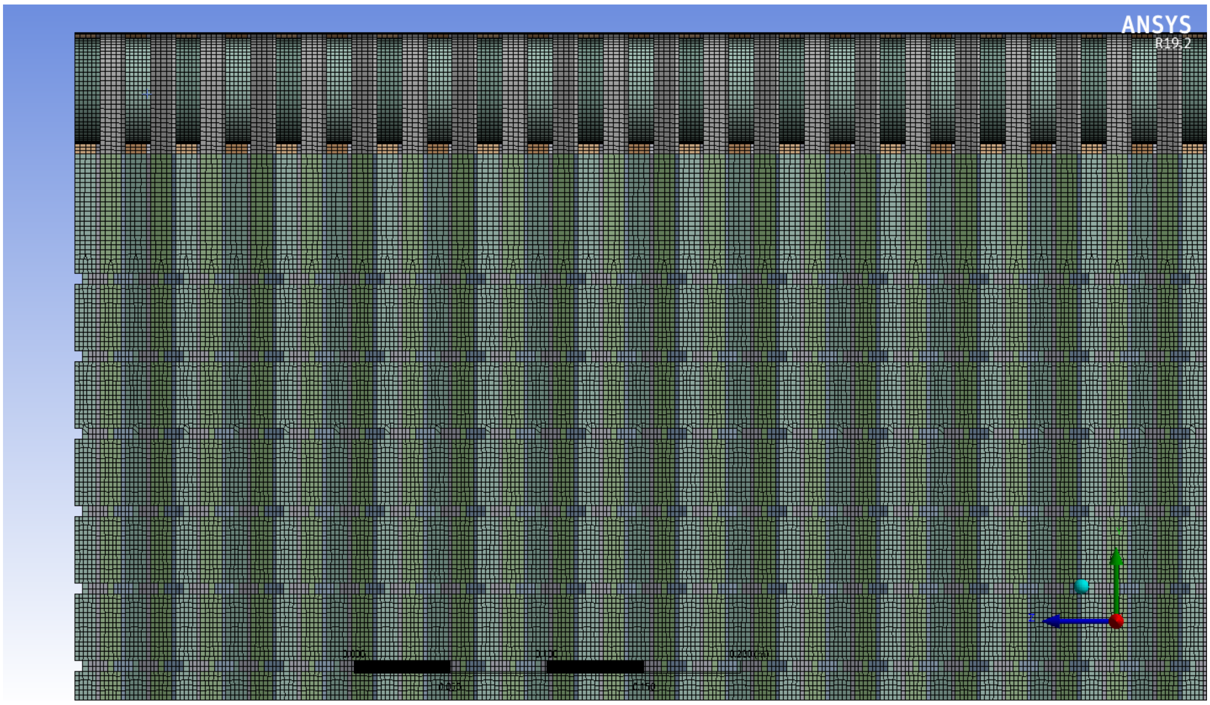


Figure 3.5: Side view of the meshing

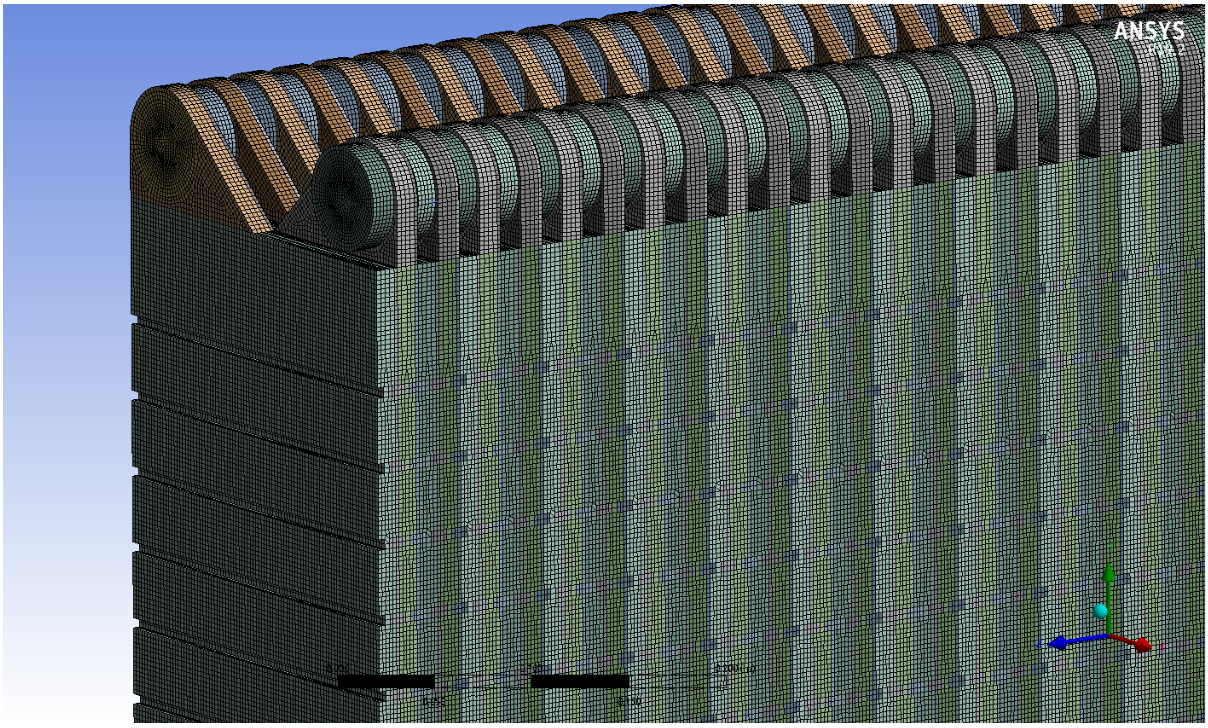


Figure 3.6: Isometric view of the meshing

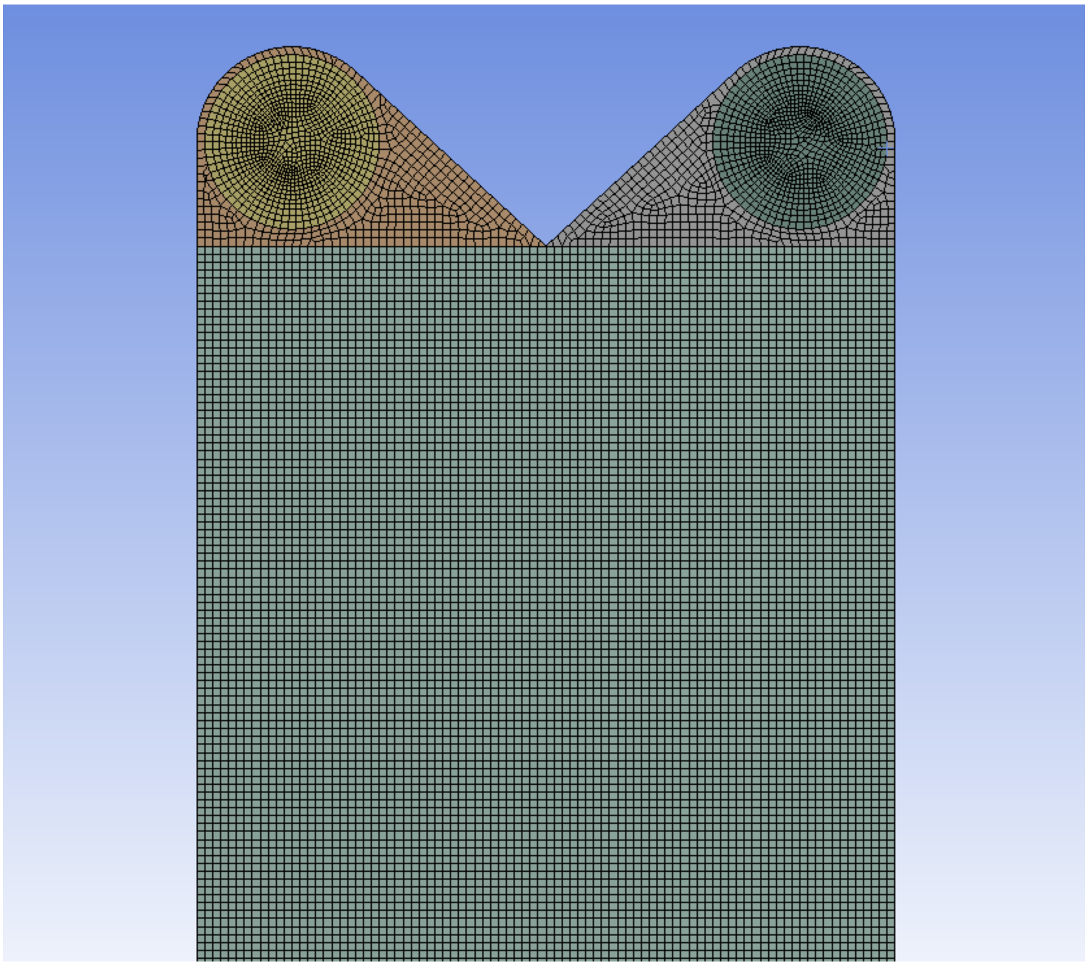

Figure 3.7: Front view of the meshing

The result of the mesh is good as the skewness obtained is 0.63949. To have a good quality meshing, the maximum skewness value must remain below 0.8. The number of nodes are 28541712 and the elements are 23418646.

| | |
|---|---|
| Export Format | Standard |
| Export Preview Surface Mesh | No |
| Element Order | Linear |
| **Sizing** | |
| **Quality** | |
| **Inflation** | |
| **Assembly Meshing** | |
| **Advanced** | |
| **Statistics** | |
| Nodes | 28541712 |
| Elements | 23418646 |

Figure 3.8: Number of nodes and elements

| | |
|---|---|
| **Sizing** | |
| **Quality** | |
| Check Mesh Quality | Yes, Errors |
| Target Skewness | Default (0.900000) |
| Smoothing | Medium |
| Mesh Metric | Skewness |
| Min | 1.3057e-010 |
| Max | 0.63949 |
| Average | 3.9775e-002 |
| Standard Deviation | 6.2763e-002 |
| **Inflation** | |
| **Assembly Meshing** | |

Figure 3.9: Skewness of the meshing

The boundary condition is chose in the meshing as show in the figure below. There are two inlets and two outlets. The inlet 1 and outlet 1 are set for hot fluid which is from water bath. The inlet 2 and outlet 2 are set for cold fluid which is from cooling water

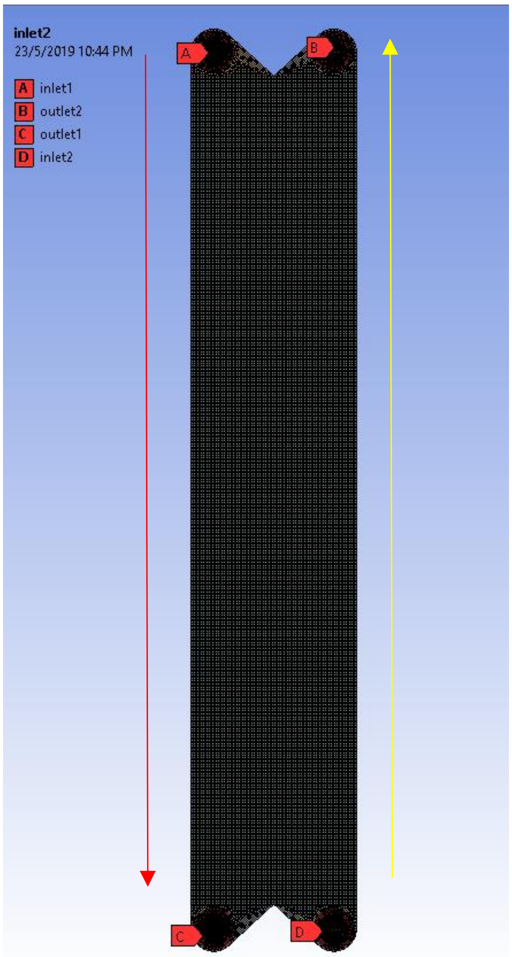


Figure 3.10: The inlet and outlet boundary condition

### 3.4.2 Setting up simulation

For the model, energy is on and the fluid model is chose to be standard k epsilon model as the fluid flow calculated is to be turbulence flow. The material for the plates are set to SUS304 and the fluid is set to water. The details of the viscous model and methods used are as shown below.

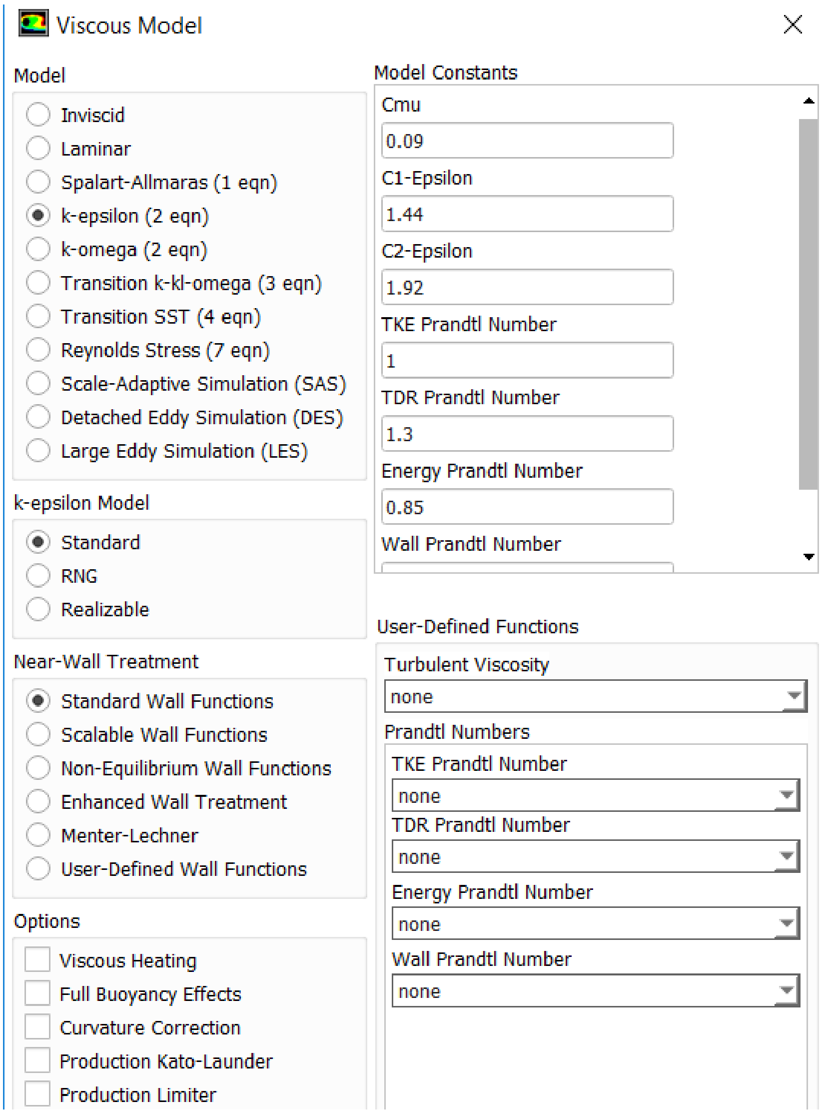


Figure 3.11: The viscous model

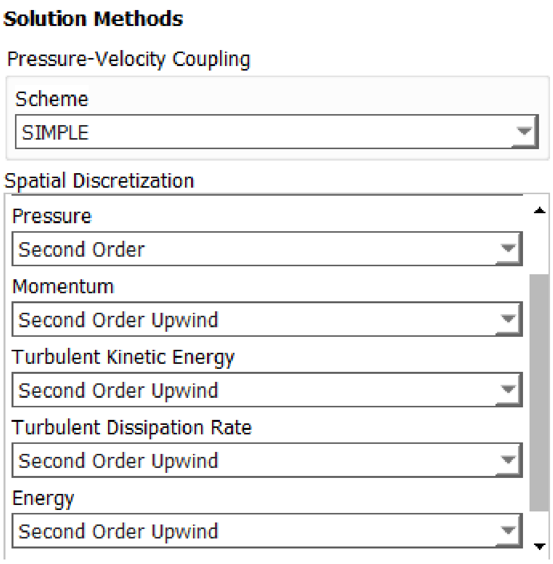


Figure 3.12: The solution methods used

For boundary condition at inlet and outlet, the value choose based on the data provided by vendor as shown in table 3.3.

Table 3.4: Inlet and outlet boundary condition

| Type | Zone | Values |
|---|---|---|
| Water bath inlet | Inlet1 | Temperature = 308 K<br>Velocity = 2.34 m/s |
| Water bath outlet | Outlet1 | Mass flow rate = 5.445 kg/s |
| Cold water inlet | Inlet2 | Temperature = 287 K<br>Velocity = 2.92 m/s |
| Cold water outlet | Outlet2 | Mass flow rate = 6.931 kg/s |
| Type of flow | All zones | Turbulence flow<br>Reynold's number = 62952 |

### 3.4.3 Run the simulation

The flow is set to steady flow because we just need to know the outlet temperature of water bath after exit the heat exchanger. The time is not a consent in this project. The number of iteration is set to 200 and from the graph of residual it is observed for the result to converged.

### 3.5 Governing equation

The FVM is based on the principle of finding the solution of the problem by dividing the geometry to be solved into solutions for each of these parts and then combining these solutions. The finite volume method uses a control volume-based technique to transform conservation equation into numerically solvable systems of algebraic equations [2].

Continuity momentum

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad \text{Equation 3.1}$$

x- coordinate

$$\rho\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right) = -\frac{dp}{dx} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad \text{Equation 3.2}$$

y- coordinate

$$\rho\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right) = -\frac{dp}{dx} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad \text{Equation 3.3}$$

z- coordinate

$$\rho\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right) = -\frac{dp}{dx} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \qquad \text{Equation 3.4}$$

Energy

$$u\frac{\partial u}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial w}{\partial z} = \frac{k}{\rho c_p}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) \qquad \text{Equation 3.5}$$

### 3.6 Calculating the capacity of heat exchanger

For the capacity of heat exchanger, the heat transfer rate is used to determine the capacity. The heat transfer rate is calculated for both simulation and for the result from vendor (HISAKAWORKS S.E.A Sdn Bhd). The results of heat transfer rate is compared for both three results. The formula used to calculate the heat transfer rate and effectiveness are as shown below.

Heat Transfer Rate Calculation,

$$q = \dot{m} c_p (\Delta T) \qquad \text{Equation 3.6}$$

Where, q = heat transfer rate

$\dot{m}$ = mass flow rate of fluid

$c_p$ = specific heat at constant pressure

$\Delta T$ = Temperature difference between outlet and inlet

Maximum Heat Transfer Rate,

$$q_{max} = C_{min} \, (T_{hot\ in} - T_{cold\ in}) \qquad \text{Equation 3.7}$$

Where, $q_{max}$ = maximum heat transfer rate

$C_{min}$ = Lowest heat capacity rate

$T_{hot\ in}$ = Temperature of hot water inlet

$T_{cold\ in}$ = Temperature of cold water inlet

Heat transfer effectiveness,

$$\varepsilon = \frac{q}{q_{max}} \qquad \text{Equation 3.8}$$

# CHAPTER 4
# RESULT AND DISCUSSION

## 4.1 Results of simulation

### 4.1.1 Result of simulation for condition 1

For the simulation 1, the inlet 1 and inlet 2 is set using the data provided by vendor. Figure 1 and 2 shows the temperature of the heat exchanger at the outlet 1 and outlet 2. The temperature from the contour for outlet 1 is 296 K and for outlet 2 is 291 K. The temperature is chose based on the colour of the contour.

Heat transfer rate, Q

$$q = \dot{m}c_p(\Delta T)$$
$$q = (5.445)(4182)(308 - 297.38)$$
$$q = 241.83\ kW$$

Maximum heat transfer rate, $q_{max}$

$$q_{max} = C_{min}\ (\,T_{hot\ in} - T_{cold\ in})$$
$$q_{max} = (6.93)(4182)(308 - 287)$$
$$q_{max} = 606.61\ kW$$

Heat transfer effectiveness, $\varepsilon$

$$\varepsilon = \frac{241.83}{606.61}$$
$$\varepsilon = 0.3974$$

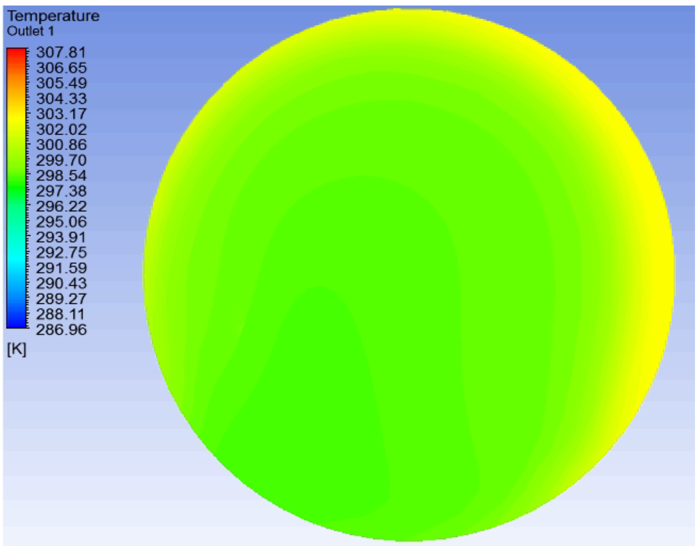


Figure 4.1: Contour of outlet 1 temperature (water bath)

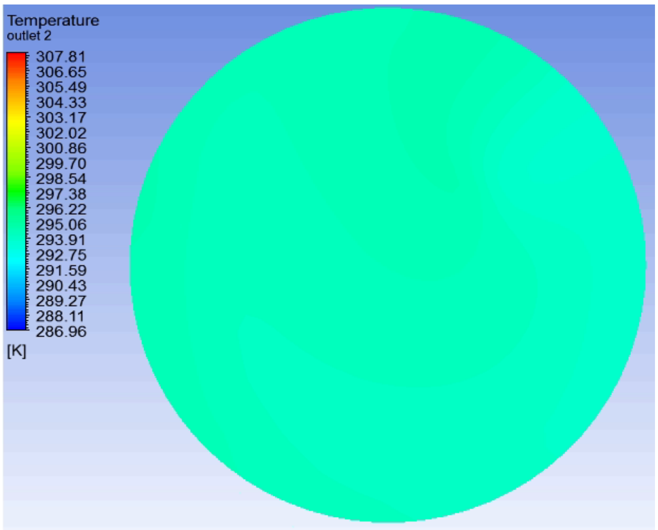


Figure 4.2: Contour of outlet 2 temperature (cooling water)

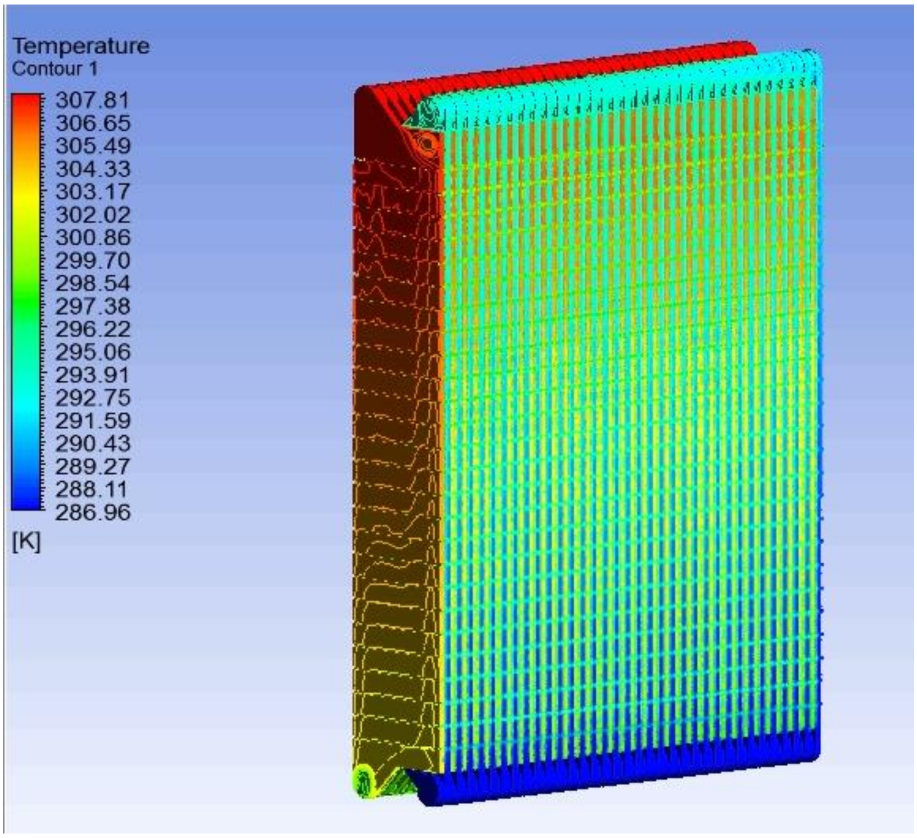


Figure 4.3: The overall temperature contour of the heat exchanger.

Figure 4.3 shows the overall temperature for the heat exchanger. The red colour contours show the fluid flow at inlet temperature and as the fluid flow down of the temperature, the temperature drops due to heat transfer occur between the cold fluid and the hot fluid via the plates. The heat from hot fluid will pass through convection to the plates, then the heat will pass to other side of plates through conduction and will pass to the cold fluid resulting in temperature drop of hot fluid and temperature rise of cold fluid.