INVESTIGATION OF CONVECTIVE HEAT TRANSFER IN A HEAT EXCHANGER

MUHAMMAD AZRUL AMRI BIN SHAHRIL

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

INVESTIGATION OF CONVECTIVE HEAT TRANSFER IN A HEAT EXCHANGER

MUHAMMAD AZRUL AMRI BIN SHAHRIL

This report is submitted
in fulfillment of the requirement for the degree of
Bachelor of Mechanical Engineering

FACULTY OF MECHANICAL ENGINEERING

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2019

DECLARATION

I declare that this project report entitled "Investigation of Convective Heat Transfer in a Heat Exchanger" is the result of my own work except as cited in the references

Signature : ....

Name : Muhammad Azrul Amri Bin Shahril

Date : 3 July 2019

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.

Signature	: ....
Supervisor‘s Name	: Shamsul Bahari Bin Azraai
Date	: 3 July 2019

DEDICATION

Special to

Father and mother, family, lectures and friends

ABSTRACT

The main purposes of this study is to simulate the heat transfer rate of convection for the two types of tube bank arrangements, which is the inline and staggered arrangement by using the ANSYS Fluent software. The results from the simulation that are used the cross-flow heat exchanger for both arrangements has been discussed in this report, which is by presenting the temperature, velocity and pressure profiles. From the simulation results, the comparison of performance heat transfer rate for both arrangements also determined in this study by using the air temperature difference to indicate the heat transfer for each the arrangements. The performances of both arrangements show that the staggered tube bank arrangement has a higher heat transfer rate compared to the inline tube bank arrangement. The result from simulation also has been validated by comparing with the experimental result and from the comparison, the results from both approaches are in good arrangements.

ABSTRAK

Tujuan utama kajian ini adalah untuk mesimulasikan kadar pemindahan haba konveksi bagi dua jenis susuan tiub bank, iaitu susunan sebaris dan berperingkat dangan mengunakan perisian ANSYS Fluent. Hasil daripada simulasi yang mengunakan penukar haba aliran siling untuk kedua-dua susunan tiub bank telah dibincangkan dalam laporan ini dengan menyampaikan profil suhu, hadlaju dan juga tekanan. Dari hasil simulasi, perbandingan prestasi kadar pemindahan haba untuk kedua-dua susunan juga ditentukan dalam kajian ini dengan mengunakan perbezaan pada suhu udara untuk menunjukkan kadar pemindahan haba bagi setiap susunan. Prestasi kedua-dua susunan menunjukkan bahawa susunan tiub bank berperingkat mempunyai kadar pemindahan haba yang lebih tinggi berbanding susunan tiub bank sebaris. Hasil daripada simulasi juga telah disahkan dengan membandingkan hasil daripada eksperimen dan dari perbandingan tersebut, hasil dari kedua-dua pendekatan adalah dalam susunan yang baik.

ACKNOWLEDGEMENT

Praise to Allah Almighty because His Blessing I had successfully completed my Final Year Project, which is to the fulfilment of the requirement for the degree of Bachelor of Mechanical Engineering.

I would like to express my gratitude and special appreciation to my project supervisor, Mr. Shamsul Bahari Bin Azraai because giving the guidance, support, and encouragement during completing this project. Furthermore, special thanks to my second examiner Mr. Mohd Hafidzal Bin Mohd Hanafi that also giving the comment and suggestion to improve my project in order to achieve the project objective.

Not to forget, I would like to express my thanks also to Mr. Faizal Bin Jaafar, the assistant engineer for the heat transfer laboratory for giving cooperation and guiding during the experiment was conducted for this project.

Last but not least, I also would like to express gratitude toward my beloved parents Mr. Shahril Bin Md. Dalis and Mrs. Norhayati Binti Md. Nor, which is always taking care about the project progress. Lastly, thanks to my friend and all people that willing helps and support me in completing this project report.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF APPENDICES

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamic
DM	Design Modeller
SIMPLE	Semi-Implicit Method for Pressure Linked Equations

LIST OF SYMBOL

A	Area
d	Diameter
D	Outer diameter
S_T	Transverse pitch
S_L	Longitudinal pitch
S_D	Diagonal pitch
W	Watt
β	Angle

CHAPTER 1

INTRODUCTION

1.1 Background

Heat exchanger is a device or component that are used to change temperature between two fluid that have different temperature, which is it involve the heat transfer between fluid flow with the solid wall in order to make the fluid that flow to change the temperature as that are required. This application are widely used in engineering industry such as application in energy production, chemical processing, space heating and air-conditioning, food industry, waste heat recovery and many others in order to decreases the cost of fluid used. There are many types of heat exchanger that are used in industry like counter-flow, parallel-flow and cross-flow with mixed or unmixed as show in **Figure 1.1,** where the most common type of heat exchanger in industrial application due to large number of tube that packed in a shell with their axes parallel to that of the shell.

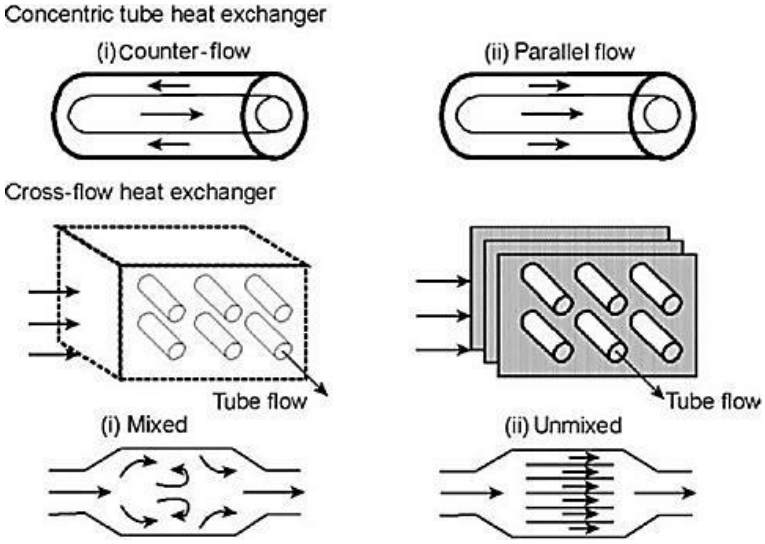

Figure 1.1 : Type of fluid flow (Incropera et al. 2007).

Mirzakhanlari et al. (2015) reported that there are classes of natural convention heat transfer over varied body separated into the internal and external flows, the internal flow is restricted by surface or crater meanwhile external flows around the body are evaluated in unrestricted area. In the heat exchanger, there have only the internal convection between the tube and two type of fluid which are in different temperature. Furthermore, Khan et al. (2006) reported that tube bank for heat exchanger in industry are commonly in an inline or staggered arrangement and there are categorised by the dimensionless transverse, longitudinal and diagonal pitches. To get the maximum of heat transfer rate on tube arrangement, the categorised must meet the perfect design and there need to study and proof by research.

The type of heat exchanger are not main concern for rate heat transfer in this study because the different of that type are only on direction of fluid flow, the main factor is the process of convection heat transfer between the tube and fluid flow. That why the design of tube arrangement in the heat exchanger also are one of the factor to achieve the maximum rate of heat transfer in that process.

1.2 Problem Statement

Mavridou et al. (2015) stated that the one of the main issues of heat exchange is that encounter throughout their operation is the deposit formation on the heat transfer area, that will increase the thermal resistance of the tube walls and decrease device potency. To archive the maximum rate of heat transfer on the process in heat exchanger, there many factors can help the process of heat transfer on fluid successful and one of the factors is the arrangement of tube bank in heat exchanger with categorised by the dimensionless transverse, longitudinal and diagonal pitches. Meanwhile, Saraireh et al. (2017) founded the type of tube bank which is a compact or widely tube bank also factor where given an effect for result rate of heat transfer in the exchanger during their operation. Furthermore, Qiu et al. (2004) stated the tube spacing also give the very significant effect on the boiling heat transfer in compact tube bank whether of inline or straggled tube bank arrangement. To increase rate heat transfer, two type design of tube bank arrangement are study which is inline tube and staggered tube bank.

From the previous study of Saraireh et al. (2017) and Qiu et al. (2004), the arrangement of tube at tube bank are given significant effect on rate of heat transfer where it the most important criteria of heat exchanger during their operation. When the heat exchangers are producing the poor rate of heat transfer, it will give effect to the other system that required the fluid became the lowest temperature such as at power plant where the heat exchanger are used to cooling down the oil lubricant for bearing inside the turbine. If the oil lubricant are not in lowest temperature, it can make the bearing became hotter and expanded due to temperature lubricant and operation which can be damage the bearing and also the rotor shaft.

1.3 Objective

In this study, there are focuses on the convective rate of heat transfer in heat exchanger between two types of tube bank arrangement that have in an industry today. The objectives of this study are:-

- To study and simulate the heat transfer in an array of inline tube and staggered tube.
- To study and compare the performance of inline tube and staggered tube.
- To study and validate the result with experiment.

1.4 Scope

For this study, the objectives is focus to simulate and comparing the heat transfer rate between two type arrangement, which is inline and staggered arrangements by referring the temperature difference of air that flow through the heat exchanger. Furthermore, the type of heat exchanger that is considered in this study is only the cross-flow heat exchanger. Moreover, to achieve the objective of this study, there have the scope of work to be done so that at in the end of this study get the proper result of rate of heat transfer by referring the temperature different of air flow of each tube bank arrangement. Firstly, there need to construct the tube bank arrangement, which is the inline and staggered tube arrangement type by set the value of longitudinal and transverse pitch using SOLIDWORK. Secondly, to simulate the heat transfer rate in an array of inline tube and staggered tube arrangements is using Computational fluid dynamic (CFD), which the software is the ANSYS Fluent. Lastly, to analyses and validate the result from simulation, which is CFD by comparing with experimental result.

CHAPTER 2

LITERATURE REVIEW

2.1 Tube Bank

Tube bank is the array of tube that are commonly design in the heat exchanger and it usually found in circular tube but it also can be other geometry like the rectangular tube. The tube banks are also attached on the plain plate surface or finned plate.

2.1.1 Tube Bank Arrangement

This study is to investigate the type of the arrangement for tube bank, which is commonly used in cross-flow of heat exchanger, so the understanding of tube arrangement are needed before do the comparison of the heat transfer rate by referring the temperature different between the tube bank arrangement. Tube is a hollow circular component of metal where are used in heat exchanger to transmit or holding a substance such as fluid or gas from inlet section to outlet section. Furthermore, in the industrial application there have a tube material that can transmit the fluid with a highest temperature like in the boiler, which is at the inside of boiler there have many tubes to produce the superheated steam to supply at turbine in order to generate the electricity. Other than that, the tube also can be found as circular component where acting as fin where to increase the heat transfer rate for cooling. By attaching the surface extended surface called as a fin, it will increase the surface area, which is used to transmit the heat to surrounding by convection process (Incropera et al. 2007).

When the array of tube are arranged in a set of tube, it is called as tube bank or tube bundle and usually it was found in the heat exchanger such as condenser, boiler, and radiator, where the applications have widely in industry such as power plant, chemical industry, food industry, air conditioning and many other. Furthermore, Qiu et al. (2004) also stated there have the tube bank have been widely used for desalination and solar power absorption chillier which flooded type or full liquid type evaporator where the fluid boils on outside of tube bank. Furthermore, there have two types of arrangement that have widely used in industry application which inline arrangement and staggered arrangement with plain surface plate of tube bank (Saraireh et al. 2017, Wang et al. 2016). In particular, the arrays of tube bank arrangement are typically used to enhance the maximum of heat transfer rate compare to the single passes tube (Bender et al. 2018, Uguru-Okorie et al. 2018).

Moreover, the inline and staggered tube bank arrangement are usually in the type of cross flow where the two type of fluid at a perpendicular angle to each other flow direction. According to the configurations of flow in model heat exchanger, the flow can be categorised into their type and there have two types of flow in cross flow heat exchanger, which is mixed and unmixed flow. If the fluid flow on surface tube and it is can move freely in a transverse direction, then this arrangement called as mixed meanwhile if both of the fluid flow passes through their own tunnel and only strictly move in a transverse direction, the arrangement called as unmixed (Rathore et al. 2011).

Furthermore, in the both of tube bank arrangement there has two type fluids with different state, which is one fluid moves over the tubes, while a second fluid different temperature passes through the tube. The fluid that are move over the tube surface will exchange the heat between with the fluid the passes through the tube in order to increase or decrease the temperature fluid that passes through the tube (Mangrulkar et al. 2017). On the other hand, there was had the studies to purpose the new design of tube bank arrangement which trapezoidal tube bank, which is the new design are for enhance the heat transfer rate in heat exchangers (Bender et al. 2018).

2.1.2 Tube Bank Characteristics

For the both tube bank arrangement, which is the inline tube bank and staggered tube bank, there have a configuration that can affect the value of heat transfer rate on the heat exchanger. According Incropera et al. (2007), the configuration parameter that consider is diameter (d), transverse pitch (S_T), longitudinal pitch (S_L), diagonal pitch (S_D), and Reynolds number. The outer Diameter (D) of the tube is a characteristic length, where are used to identified the Reynolds numbers.

Figure 2.1 shows that the parameter in the inline and staggered tube bank arrangement. Firstly, the S_T is the transverse distance between two tube on tube bank from the air flow direction, meanwhile S_L is the longitudinal distance between two tube on tube bank from air flow direction which both configuration important in tube bank design. On the other name, Gowda et al. (1998) are called as pitch to diameter for both pitch and for this two configuration, there have in both arrangement, inline tube bank arrangement and staggered tube bank arrangement. However, in staggered arrangement of tube bank there has additional configuration that need to consider, which is the S_D and it is the diagonal distance between two tube canter in the tube bank.

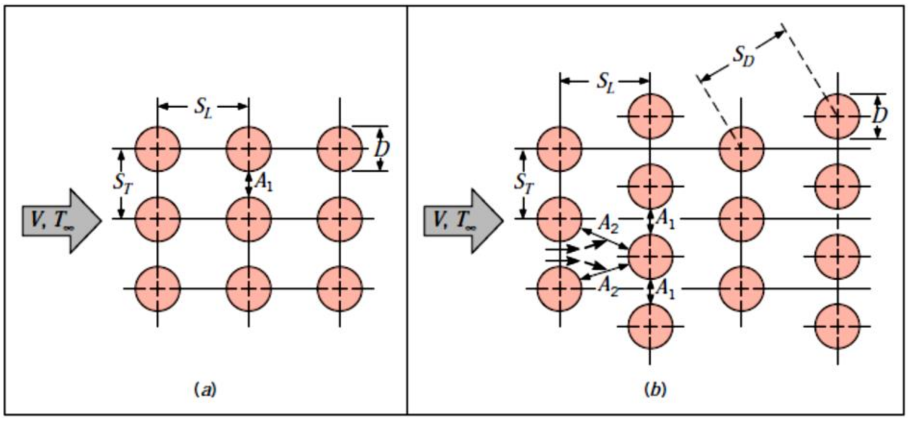

Figure 2.1 : Tube bank arrangements. (a) Inline. (b) Staggered (Incropera et al. 2007).

2.2 Convection Heat Transfer

Convection is one of modes in heat transfer process. This process are involve energy transfer between a solid surface and the fluid that flow outside of solid surface.

2.2.1 Tube Bank Arrangement

The capability of heat exchanger during operation to change the different temperature of fluid or gas was measured by the values of heat transfer rate that it will capable to archive during the process. To get the better heat transfer rate for the heat exchanger, the arrangements of tube are also one of factor to enhance the maximum heat transfer rate, so that in this study the comparison between inline and staggered tube bank arrangement were investigate. The designs of the tube bank are important to make the heat exchanger more favourable with its dimension and value heat transfer. Saraireh et al. (2017) found that the inline tube arrangement provided higher heat transfer rate compared to the staggered tube arrangement where 10.14 kW and 9.39 kW, respectively through the comparison of Computational Fluid Dynamic simulation by using FLUENT software. The configuration for the tube that are used in the study is diameter tube equal to 10 mm, transverse pitch S_T = 30 mm and longitudinal pitch S_L = 35 mm. In the terms for spacing between tube on the tube bank, **Table 2.1** shows the result that the compact of tube bank whether inline or staggered arrangement which is S_T and S_L are equal to 20 mm are indicate the greater heat transfer compare to the loosen area of tube bank.

Table 2.1 : Heat transfer rate for compact and wide tube bank (Saraireh et al. 2017).

Heat Transfer	Compact Tube		Wide Tube	
	Inline	Staggered	Inline	Staggered
q (kW)	18.48	14.56	10.14	9.39

Khan et al. (2006) stated that staggered arrangement tube bank give higher heat transfer rate than the in line arrangement for same transverse and longitudinal ratio which 1.25 mm x 1.25 mm. For the comparison between compact and widely tube bank, the result indicate the compact tube bank had higher heat transfer rate compare than widely space where 25.5 kW and 20.1 kW, respectively. Furthermore, in this study there had highlight that that heat transfer rate of tube bank was decreasing due to the distance pitch ratio of tube bank was increasing. From both literature studies, the outcome for the results have contradicted in term of tube arrangement whether inline or staggered tube bank arrangement. However, for the result of spacing of tube, both result indicated that compact tube bank give more heat transfer rate than wide tube bank heat transfer rate.

As mentioned by Kong et al. (2016), the staggered tube arrangement with continuous slotted fin surface mounting at the tube bank will give outstanding performance in heat transfer, which is investigate by numerical simulation and justify by experiment. Furthermore, the studies also stated because the inline tube arrangement had spacious dead flow zone between two adjacent of a tubes, that means the inline tube arrangement produce a poor heat transfer efficient compare to staggered tube arrangement. In other studies on staggered tube arrangement, which using the serrated-fin at the tube bank as show in **Figure 2.2** by Næss (2010), the author found that the fin specification, which is the height and pitch of fin, the result outcome indicated a tolerable effect on heat transfer rate. The first case is the increasing the fin height, it will increase the heat transfer coefficient, however in second case where the increasing the fin pitch, it will reduced the heat transfer coefficient. The author also stated this result may be because the penetrating and mixing between fluid and the serrated fin tube, which it give more efficient in heat transfer rate.

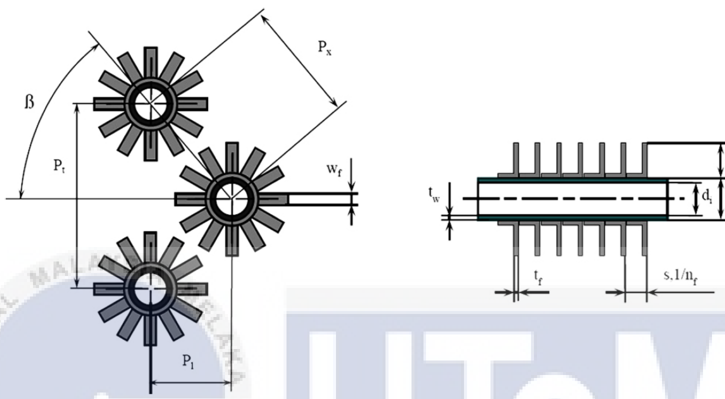

Figure 2.2 : Tube and tube arrangement design (Næss 2010).

According to Mangrulkar et al. (2017) studies, by apply the splitter plate can only relevant to the staggered tube bank arrangement of tube for efficiency of heat transfer rate compare inline arrangement tube bank arrangement, where the splitter plate had added the pitch ratio which may lead to lower heat transfer rate. The both tube arrangement had the same transverse pitch ratio and longitudinal pitch ratio, which is 1.75 and 2.0 respectively. Furthermore, the authors stated that the splitter plate gives the better enhancement may be because the formation of a symmetric vortex that produced on either side of splitter, which the splitter plate has the length that equal to the diameter cylinder tube, 25 mm with the thickness equal to 1.75 mm and result was get by experimental and numerical simulation. In these studies, the authors are focus only to evaluate the heat transfer in staggered tube arrangement with and without splitter plate as show in **Figure 2.3**.

From literature studies, which is from Kong et al. (2016), Næss (2010) and Mangrulkar et al. (2017) studies that about contact surface area, the result indicated that contact surface area between the fluid give the effects on heat transfer rate for the tube arrangement. In other words, to get the better heat transfer rate for tube arrangement in heat exchanger, it can done by expending the contact surface area between two different flow temperatures in heat transfer process.

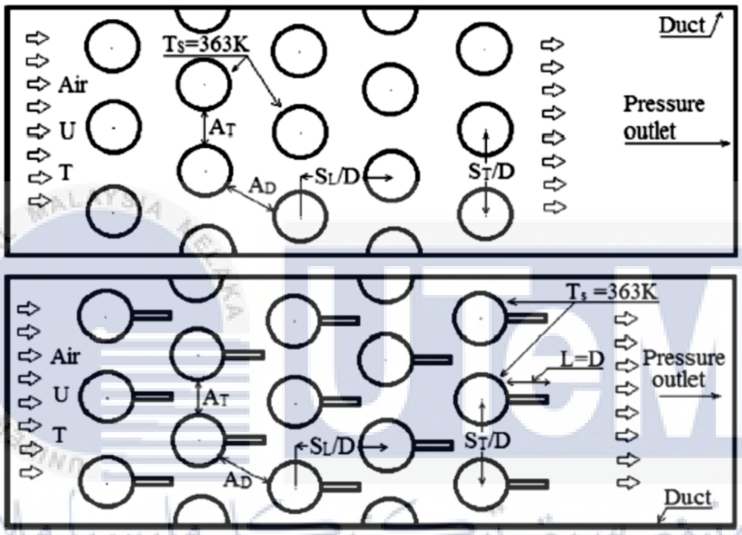

Figure 2.3 : Staggered tube bank arrangement with and without splitter plate (Mangrulkar et al. 2017).

In the studies for inline tube bank arrangement by Mirzakhanlari et al. (2015), which the studies to investigate the trade of the average Nusselt number that passing over through the inline tube bank arrangement by using the CFD, which is that average Nusselt number is dimensionless parameter that are used to describe the heat transfer rate. In these studies, the authors found that when the Rayleigh number is increased, than the average Nusselt numbers are also increase. Furthermore, for $d_T/D = 2$ and $d_L/D = 2$ with the isothermal tubes, the Rayleigh number at 10^8 generated the highest value for average Nusselt number by using 8 tubes in the tube bank. Moreover, the studies about inline tube bank arrangement of tube bank over mixed convection had been conduct and the result as complicated to describe. Other than that, for the $d_T/D = 2$ and $d_L/D = 2$, Haider et al. (2010) also found that the heat transfer over circular cylinder, for the staggered arrangement has higher compared to the inline arrangement. As stated by Gowda et al. (1998), the outcome from a study between average Nusselt Number and pitch to diameter ratio, it show that the average Nusselt number for pitch to diameter ratio equal 1.5 are higher than 2.0. This study was done by increasing in Reynolds number in both situations, which is it was increased from 150 until 150. So that, this studies indicate that the heat transfer rate for inline arrangement with compact tube bank was higher than wide tube bank.

2.2.2 Substance Flow

The substances that are flow in heat exchanger also a one of factor for archive maximum heat transfer rate in both of tube arrangement. The study was done by Murray (1993) and the author reported that achievement of the heat transfer rate in inline tube arrangement more appropriated than the staggered tube arrangement for gas-particle cross flows. This are due to the particles can enhance and reduce local and overall heat transfer coefficients in both arrangements. Meanwhile, Murakawa et al. (2018) stated that to build the heat exchangers are suitable to build with two tube arrangement, which is for upstream regions it in array of staggered tube bank meanwhile for downstream regions it in array of inline tube bank and this for vertical flow direction of heat exchanger. This experiment was study around tube in two phase flows which is in bubbly and intermittent flow regimens by using the pitch to diameter ratio is equal to 1.5.

In addition, there are previous studies that using boiling water to identified heat transfer in compact staggered tube arrangement of tube bank. Qiu et al. (2004) said that the distance between each tube on staggered tube arrangement give resulting in effect on the boiling heat transfer where to get the increment of heat transfer rate, the distance between tube need to be reduce at the staggered tube arrangement. The result was studies by experimentally, and its show a compact staggered tube arrangement with tube spacing 0.3 mm had maximum value heat transfer at low or intermediate heat flux. Furthermore, the authors also stated that it impractical to using the value of tube spacing in the engineering machinery because it too small for actual bundle evaporators.

2.2.3 Design of Tube

The tube can be design to other geometric and it also gives the effect on rate of heat transfer on the tube arrangement in the heat exchanger. According the studies from Bayat et al. (2014) found by experimental method where the staggered arrangement that using the cam shaped for tube in heat exchanger as show in **Figure 2.4**, it will lower the size of heat exchanger and it can raise the thermal hydraulic performance during heat transfer process. Unfortunately, for the heat transfer rate for this cam shaped of tube bank, the result obtain was less than to the circular shaped tube about 48% to 54% with equivalent diameter 22.44 mm, even though the heat transfer escalate up to 36% to 41% as increasing Reynolds number from 27000 to 42500. For inline arrangement of tube bank, it also had been study and the result for rate of heat transfer are also not satisfying to design, Lavasani et al. (2014) found in the experimental studies that heat transfer rate from cam shaped tube in inline tube arrangement as show in **Figure 2.5**, it is lower than circular tube about 5% to 11% with equivalent diameter 22.44 mm, but by replacement of circular tubes with cam shaped tubes in heat exchangers that will help energy conservation and minimizes heat exchanger’s size for the inline tube arrangement.

From the result of both previous studies in cam shaped tube design, it can be highlight that the cam shaped tube are not suitable to apply in order to enhance the maximum heat transfer rate for tube bank in heat exchanger.

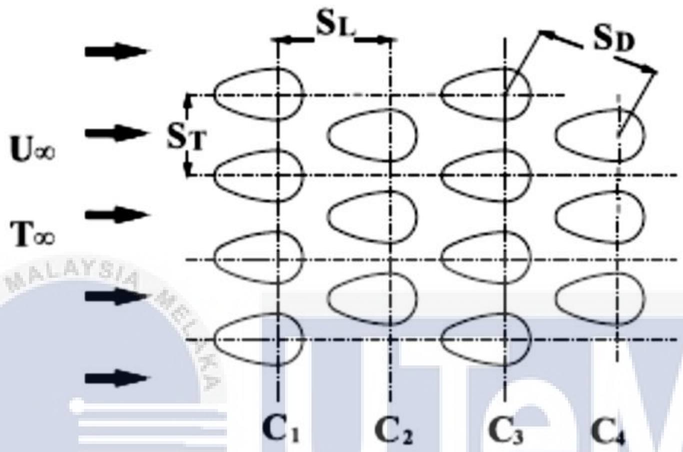

Figure 2.4 : Staggered tube bank arrangement for cam-shaped tube (Bayat et al. 2014).

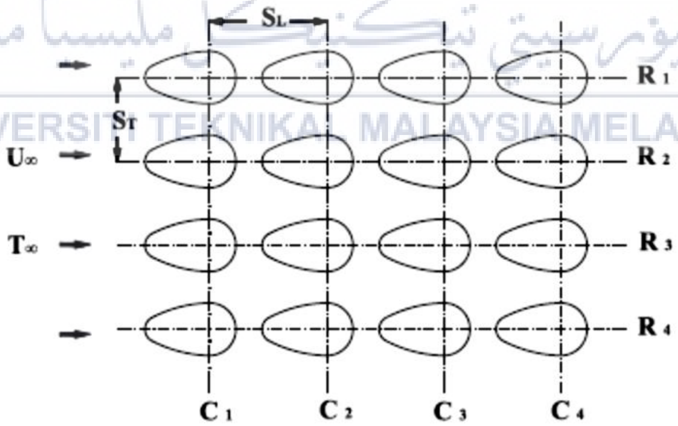

Figure 2.5 : Inline tube bank arrangement for cam-shaped tube (Lavasani et al. 2014).

Furthermore, according to Wang et al. (2016), the authors found that by change the design of tube bank which is the degree inclination of tube in tube bank as show in **Figure 2.6**, whether inline tube bank or staggered tube bank arrangement will not resulting to the any lead to heat transfer rate for both arrangements. On the other hand, according the

Mavridou et al. (2015) has studies about the design of tube bank and the authors found that the design for inline tube arrangement tube bank by using 2 unequally sized cylinders with d/D = 0.5 S_L/D = 1.5 and S_T/D = 3.6 as show in **Figure 2.7**, it was led to a 40% specific (per unit volume) heat transfer enhancement compare with standard arrangement S/D = 2.6 by evaluated experimentally.

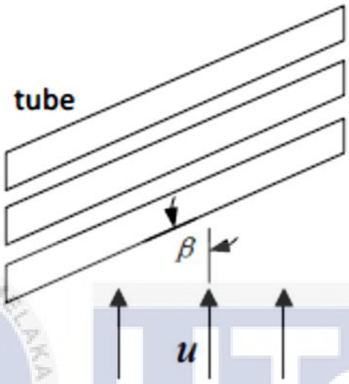

Figure 2.6 : Impacting tube angle from fluid flow (Wang et al. 2016).

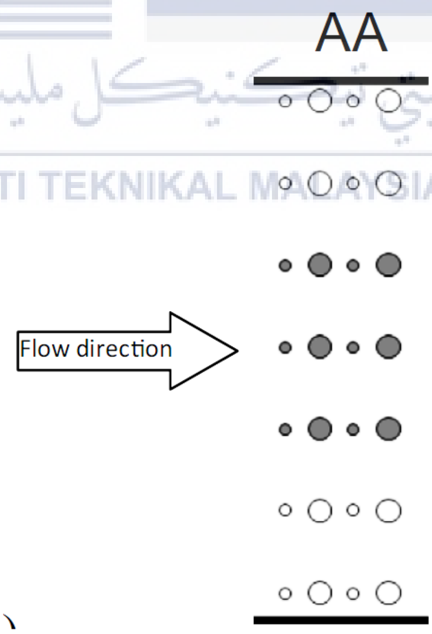

Figure 2.7 : Arrangement of tube bank with different diameter (Mavridou et al. 2015).

Besides, there have studies about new tube bank arrangement which a new trapezoidal tube bank by using numerical simulation. According to Bender et al. (2018), the studies was conduct under a force convection to investigate effect the trapezoidal factor t/d as parameter and their found that increasing the trapezoidal factor t/d as show in **Figure 2.8**, where maximum t/d = 12/16 will increasing the average Nusselt number, which is indicate high heat transfer rate show in below. In addition, the **Figure 2.9** show that the comparison the result with inline and staggered arrangement of tube bank, which is the result show that the staggered arrangement has better average Nusselt number, follow by trapezoidal arrangement and lastly inline arrangement of tube bank in the high Reynolds number.

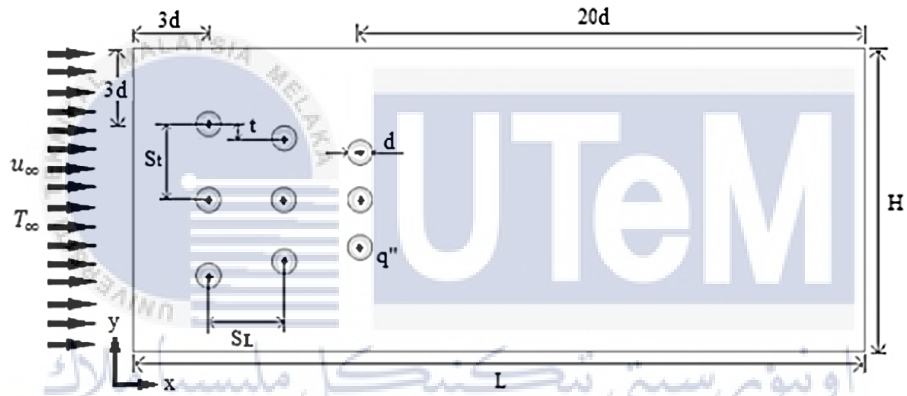

Figure 2.8 : The arrangement of trapezoidal tube bank (Bender et al. 2018).

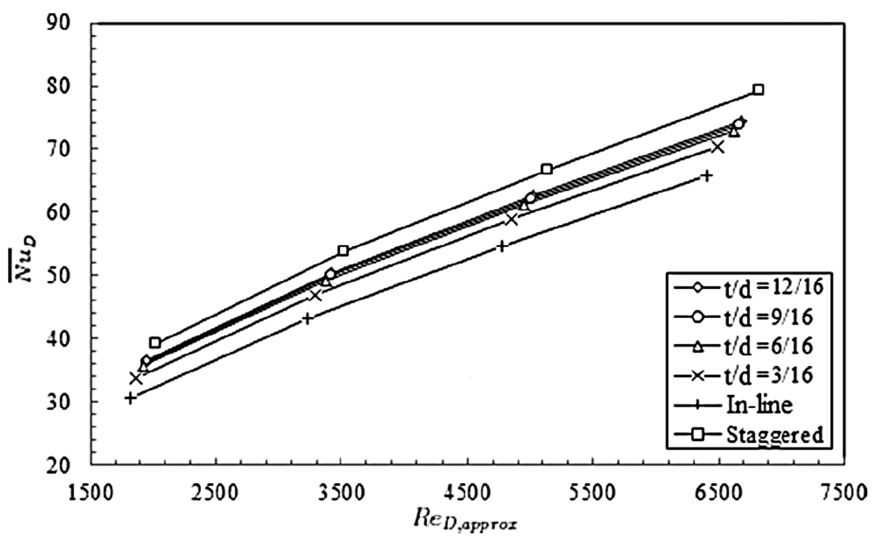

Figure 2.9 : Average Nusselt number for the parameter of t/d (Bender et al. 2018).

2.3 Computational Fluid Dynamic

Computational fluid dynamic (CFD) has been generally used in industrial and non-industrial applications, which it is a branch of fluid mechanics and solutions for fluid flow with interaction by a numerical method and mathematical model in order to identify the fluid flow. Furthermore, for engineer the CFD had become the essential in order to calculate and get the data of product for study before the prototype products were produce. This program also cheaper in terms of cost, give more accurate data and more quickly compare conventional process (Andersson et al. 2012). To identify the behaviour of fluid in the study, the simulation is best way to solve the problem and it can be done by using CFD software, which there has many type of software that can be used such as ANSYS Fluent, ANSYS CFX, Solidworks, Gambit and many more.

In previous study, Saraireh et al. (2017) was used CFD which is the ANSYS Fluent software to simulate the average Nusselt number, which indicate the heat transfer rate for staggered arrangement of tube bank. Furthermore, the authors consider this study as in three dimensional with 16 tubes, made from aluminium with the geometrical parameter as show in **Table 2.2**. The authors also state that in this software, the properties of air can be used and the values can be set as show in **Table 2.3** and the Fluent software also give solution for heat transfer rate by considering the implicit formulation, steady state calculation, laminar model and energy equation. In order to get the solution, the mesh and boundary condition for the simulation must be defined properly and the meshing in the study that was used by authors is fine mesh, which it the mesh generate elements values that consist of 1300000 elements. Furthermore, the velocity, heat flux and bulk temperature was set before the running the simulation, where the velocity of air is variable parameter it have been set between 0.6 m/s to 1 m/s at the inlet tube wall. For the heat flux, it was set on range between 1000 W/m^2 to 3500 W/m^2 meanwhile the bulk temperature of air cross flow of tube was set as constant, As a result, the simulation software generated the value of average Nusselt number for variance of value heat flux, the simulation also generated result the average Nusselt number for variance of value Reynolds number at sentence heat flux value. From the result in simulation, the authors can make the conclusion for the chose, which one had better heat transfer rate for the tube arrangement.

Table 2.2 : The parameter for staggered tubes bank (Saraireh et al. 2017).

Quantity	**Dimension**
Large diameter D (mm)	10
Small diameter d (mm)	18.5
Tube thickness t (mm)	1
Longitudinal pitch S_L (mm)	40
Transverse pitch S_T (mm)	42
Number of tubes N	16
Inlet velocity V (m/s)	0.6, 0.8, and 1

Table 2.3 : Air properties (Saraireh et al. 2017).

Thermal conductivity of air k (W/m.K)	0.0263
Density of air ρ (Kg/m^3)	1.17
Specific heat of air C_p (J/kg K)	1007
Kinematic viscosity ν (m^2/s)	15.8×10^{-6}
Prandtl number Pr	0.0707
Ambient air temperature T_∞ (^{0}C)	22

Another studies that are used CFD is by Kong et al. (2016) studies which is the authors generate the computational meshes by multi-block hybrid approach using the software GAMBIT 2.4.6. In the studies, the authors created geometrical model of the three type design of fin, which is plain fin, continuous slotted fin and alternant slotted fin on inline and staggered arrangement. Furthermore, the authors use aluminium as materials for the tube and fin in all geometrical models and to generate the mesh, there are used as hexahedral structured mesh. Next, the authors setup the boundary condition by assumed the water in tube with a steady temperature and at the inlet boundary of domain, the air velocity is in the range from 0.5 m/s to 5 m/s and the temperature is set. In these studies, the effects of buoyancy and radiation have been neglect. Next, for generate the governing equation with the boundary equation in these studies, it was solved using ANSYS Fluent, which the SIMPLE algorithm that are used to deal the coupling of pressure and velocity fields and also the Second-Order Upwind Scheme, which is employed to obtain more accurate results. As the result from the simulation, the authors get the visualisation of the temperature contours, pressure contours.

Besides that, in studies that conduct by Mangrulkar et al. (2017) by used CFD method, which is ANSYS Fluent as a solution to identify the fluid flow. Initially, the geometry models are created by insert the parameter of the tube bank with 5 row of tube in staggered arrangement with the splitter plate which the length are same with diameter of cylinder tube are attach on the tube bank. The non-uniform mesh was generating by using the quality is 0.65 and meshed using quad and tetrahedral mesh for fluid and air section. Furthermore, the authors also stated the accuracy and the reliability of the numerical simulation can be detected by the grid size of the mesh. Moreover, the authors consider the model as the turbulence flow, energy equation of momentum, turbulence kinetic energy. Next, the heat flux and air velocity were set as constant, which is the wall temperature is 363K and the velocity air was controlled from the experiment. Moreover, this simulation in studies used SIMPLE algorithm and Second-Order Upwind Scheme. As the result, the simulation generate the temperature contour, turbulent kinetic energy contour, streamline for flow cross contour for each models geometry that have been studies. Other than that, the average Nusselt number, pressure drop and friction factor that with the variance of Reynolds number was got by the authors from this simulation.

CHAPTER 3

METHODOLOGY

3.1 Introduction

This chapter describes the methodology that is used in order to archive the objective for this study. The work flow chart for this study is shows in **Figure 3.1**, which is the methods that are used to get the result heat transfer rate in study is starting with the simulation and follow up by experiment. The result from the simulation, which is by using the CFD software for both arrangements must be success before proceed to the experiment work. Next, the experiment work will be conduct by using the experimental equipment, which is WL 352 GUNT Hamburg with the test specimen of the staggered tube arrangement. In order to validate the result from the simulation by comparing the result from experiment, the data input of variable for simulation must be tally with the experiment, so that the simulation will use the input data from the experiment and this can prevent the error of the result.

3.2 Simulation

In this study, to do the simulation for the heat transfer in inline and staggered tube arrangements of tube bank, the CFD are used, which is the ANSYS Fluent software. Furthermore, this software is the most application that has been used by the authors in the previous studies and it is provide the satisfied results. For this simulation, the configurations of model are obtain from the experimental approach, which is the dimension of the air duct model, S_T, S_L, D, L for tube arrangement of test specimen.

Furthermore, the data input for the variable at the boundary condition are also obtained from the experiment in order to obtain the result that tally to the experiment result.

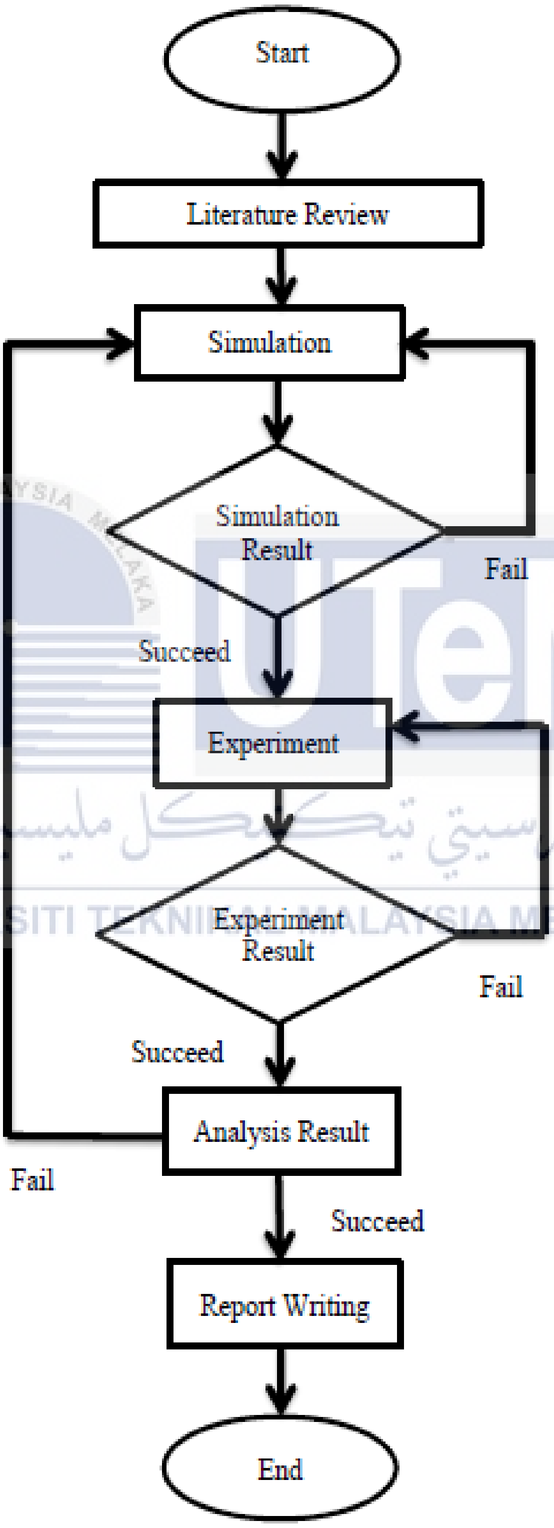

Figure 3.1 : Flow chart of the methodology.

3.2.1 Geometrical Construction

In order to do simulation using Computational Fluid Dynamic (CFD), the geometry model which know as domain must be construct. The three dimensional computational model for 2 case type of geometry model are create using Design Modeller (DM) software. The first geometry model is the plate with inline tube arrangement inside the air duct, as show in **Figure 3.2** and **Figure 3.3**, next the second geometry model is the plate with staggered tube arrangement inside the air duct that are show in **Figure 3.4** and **Figure 3.5**.

The geometry model is initially created with air duct, which is the dimension for air duct is 125 mm x 630 mm x 128 mm for both of arrangement model. The length and wide for air duct in this simulation are same with the actual test equipment for the experiment, but high of the air duct not same, which the actual is 1000 mm meanwhile the simulation is 630 mm because the distance between thermocouple inlet and outlet air is 630 mm for actual. So that, to get the accurate date, the actual distance of these two thermocouples is use as the high of the air duct for the simulation. Moreover, the distance between bottom air duct model and bottom tube arrangement model is 190 mm meanwhile the distance between top air duct model and top tube arrangement model is 320 mm.

Inside the air duct, there have the model of tube arrangement, which is by created the plain surface plate with the dimension is 119 mm x 118 mm for length and height, respectively Next, the geometry model consists of an array of circular tube with dimension for the diameter and length is 15 mm and 105 mm, respectively. The cylinder are attached on geometry of the plain surface plate which in inline arrangement and staggered arrangement. The transverse pitch and longitudinal pitch for the inline tube arrangement is 30 mm and 40 mm, respectively meanwhile the staggered tube arrangement is 30 mm and 20 mm, respectively with the both arrangement have a same cross sectional of circular tube. For the staggered tube arrangement, there have extra parameter, which is diagonal pitch and the distance between each cylinder is 25 mm. The Boolean are created in the process to make the connections between air duct and the tube bank arrangement model by using Subtract as the operation.

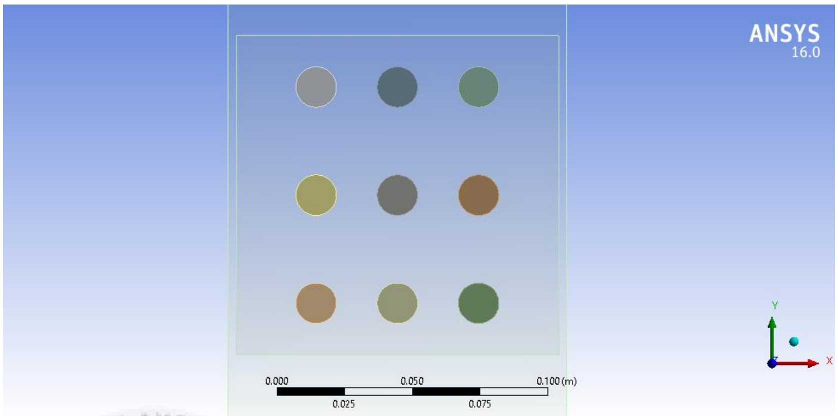

Figure 3.2 : Inline tube bank arrangement.

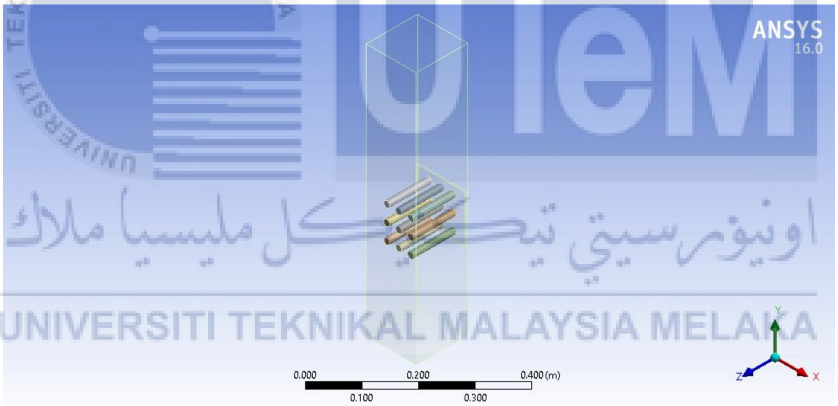

Figure 3.3 : Air duct with inline tube arrangement.

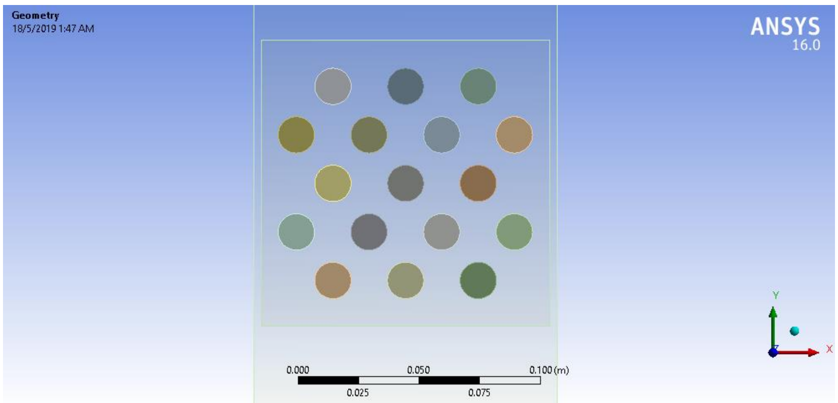

Figure 3.4 : Staggered tube bank arrangement.

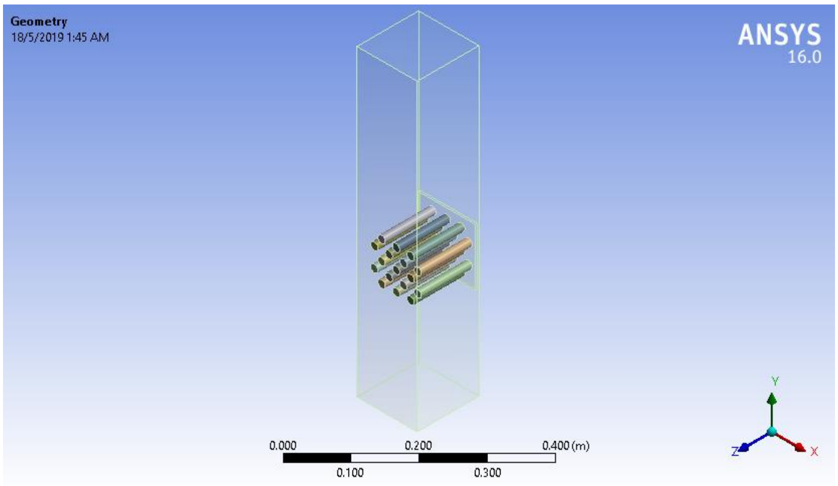

Figure 3.5 : Air duct with staggered tube arrangement.

3.2.2 Meshing

The three dimensional geometry computational models are imported to Meshing software for generating non-uniform volume mesh. For this meshing process, the advanced sizing function are set as ON with Curvature state, “Fine” for the grid of meshing and “High” for smoothing mesh in this study because it have large variation in the areas of model surface. Next, the meshing are generate for each geometry model. The inline tube bank meshing is shows in **Figure 3.6** and **Figure 3.7**, meanwhile for staggered tube bank are shows in **Figure 3.8** and **Figure 3.9**.

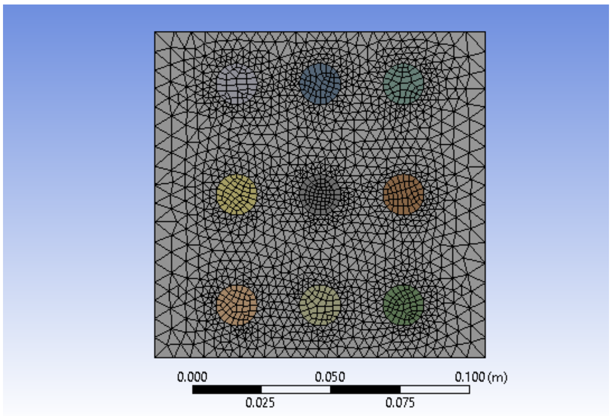

Figure 3.6 : Mesh inline tube bank arrangement.

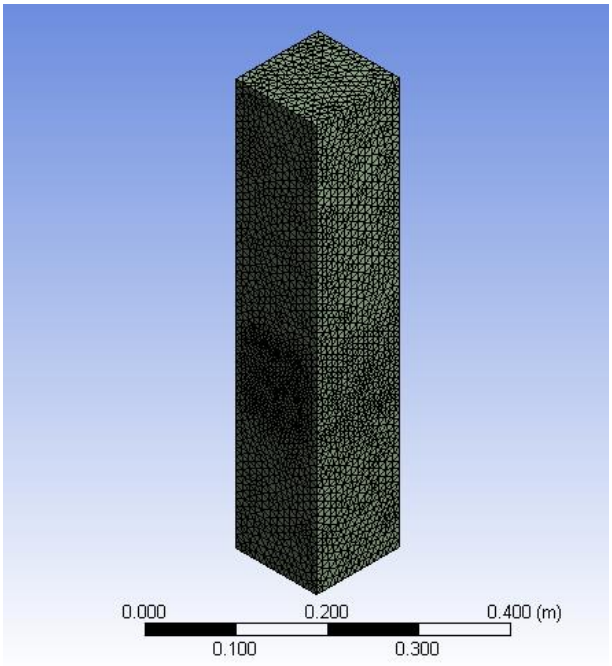

Figure 3.7 : Mesh air duct with inline tube bank.

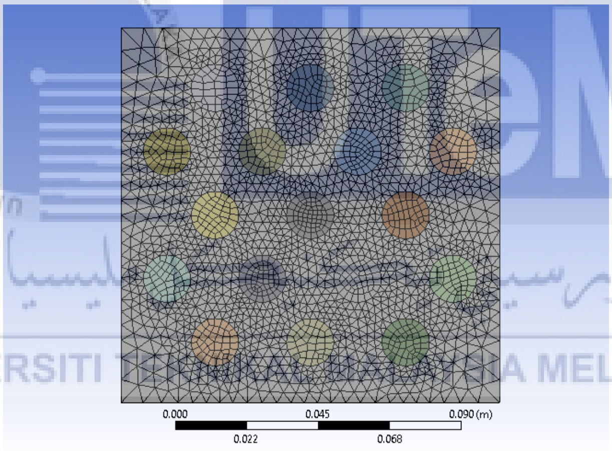

Figure 3.8 : Mesh staggered tube bank arrangement.

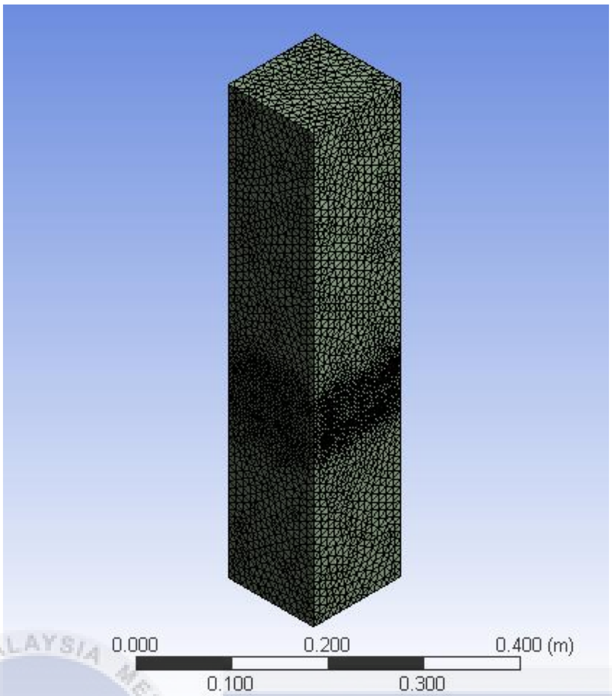

Figure 3.9 : Mesh air duct with staggered tube bank.

After generate the meshing, the each boundary that involve in fluid flow are naming by select the face of surface on model that need to name and it was name by the condition flow of fluid, which is each model has “Inlet air” and “Outlet air” on the bottom and top of air duct respectively, and the “Heated plate” on the surface plate. From the meshing process finish, the statistics detail of Nodes, Element, Skewness and Aspect ratio for the model was automatic generated and the **Figure 3.10** and **Figure 3.11** is shows the detail of statistics model for each model that has been generate.

Statistics	
Nodes	101869
Elements	494343
Mesh Metric	Skewness
Min	7.365e-004
Max	0.83387
Average	0.23892
Standard Devi...	0.12635

Statistics	
Nodes	101869
Elements	494343
Mesh Metric	Aspect Ratio
Min	1.1594
Max	12.595
Average	1.8445
Standard Deviation	0.47348

Figure 3.10 : Statistics detail for inline model.

Statistics	
Nodes	150690
Elements	691178
Mesh Metric	Skewness
Min	4.4874e-004
Max	0.82881
Average	0.23918
Standard Devi...	0.12578

Statistics	
Nodes	150690
Elements	691178
Mesh Metric	Aspect Ratio
Min	1.1277
Max	12.589
Average	1.8384
Standard Deviation	0.47009

Figure 3.11 : Statistics detail for staggered model.

3.2.3 Boundary Condition

The boundary condition is importance step in order to get the proper result from the CFD simulation. The material of the both model has been set same, which is for both tubes arrangement model is the aluminium, meanwhile fluid that cross the tube inside the air duct is the air properties. In this section, there have several boundary condition that are used on both model, which the velocity inlet, pressure outlet, interior, and wall at the "Inlet air", meanwhile pressure outlet at the "Outlet air". The "inlet air" is define as velocity inlet, meanwhile for the "outlet air is define as pressure outlet. Furthermore the "Heated plate" surface and the cylinder surface is define as no-slip wall, for the air duct surface is define as free-slip wall, meanwhile the interface between air flow and solid surface it are define as interior.

Besides that, the input data for the variable in this simulation that obtain from the experiment was set in this section and this data was applied for both arrangement models. Firstly, for the temperature of plate surface is set as a constant temperature, which is 112.09 ºC, meanwhile the temperature of inlet air that cross the tube also set as constant temperature, which 23.46 ºC and both of value are get during experiment and the value are in the average value. Next, for the speed of air velocity at the air inlet section for each model was set same both case, which is 2.32 m/s and also in constant value. The input value for the simulation must be same with experiment because in order to make sure the result for simulation are same with result from experiment, which is also used to validate the result of the simulation with the experiment.

3.2.4 Running Simulation

To solve in this simulation for both model arrangement, there have the criteria are consider in this study, which is the SIMPLE algorithm scheme with Green-Gauss Cell Based for gradient discretization and Second Order Up Wind Scheme for the momentum discretization. These methods are used to deal the coupling of pressure and also to obtain the more accurate for the result in this simulation. Next, the number of iteration are set from 1 until 1000, which are used to make the repeating calculation process until it the iteration are converge and automatically stops to complete the calculation.

3.3 Experiment

In this study, the experimental approach is conduct in order to get the result of temperature difference from the test specimen. The results from the experiment are used to validate the result from the CFD simulation.

3.3.1 Experimental Setup

The simulation results were compared with experimental data to validate the reliability of computational modelling and the method that used in this study. The heat transfer experiment ware carried out in open cycle air duct with external fan where acting as a state of force convection. The equipment of experimental facility product from WL 352 GUNT Hamburg is shows in **Figure 3.12**, which is it consist the vertical air duct, display and control unit, and the staggered tube arrangement test specimen.

The vertical air duct was connected to the display and control unit, which are used to set and measured values can be read on digital displays as show in **Figure 3.13**. On the display and control unit there have indicator of temperature for inlet and outlet of air duct, the heating temperature, heating power, fan speed. Furthermore, the fan speed for air flow in the vertical air duct can set until 100 rpm, for the heating power, the maximum power can set to heating the specimen is 200W to produce high temperature to test specimen. To measure the temperature on test specimen, there has external wire sensor that place on the test specimen during heating process.

The vertical air duct as show in **Figure 3.14** have the cross sectional of 125 mm x 1000 mm x 128 mm, for length, high, and wide respectively. Furthermore, at the top of vertical air duct it placed the external fan to control the velocity air inside the test section that is used for force convection process. On the middle of vertical air duct, there have blank space, which is the test section area with the dimension is 121 mm x 120 mm and it is use to put the test specimen that already heated by heating machine. Furthermore, There have the fixed thermocouple sensor is placed at bottom and top of vertical air duct in order to measure the temperature on that section during the process. The distance between bottom air duct and bottom thermocouple for inlet temperature is 260 mm, meanwhile for the distance between bottom thermocouple and bottom of test section is 190 mm. Next, for distance between top of test section and top thermocouple for outlet temperature is 320 mm, meanwhile the distance between top thermocouple and top air duct is 110 mm.

The experiment was only conducted on test specimen of staggered tube arrangement due to the limitation on sample for inline tube arrangement. On the test specimen, there have external port that are connected to control unit using wire and it is function to heating the test specimen using electric power. Next, for the cross-section profile of the circular tube is represented in **Figure 3.15**, which the circular diameter is 15 mm with length 105 mm and it was attached on the plain surface plate with dimension 119 mm x 118 mm in the staggered arrangement. The test tube bank with staggered arrangement bank has transverse, longitudinal and diagonal pitch, which is 30mm, 20 mm, and 25 mm, respectively by with attached with 17 circular tubes for the test in this experiment as show in **Figure 3.16**.

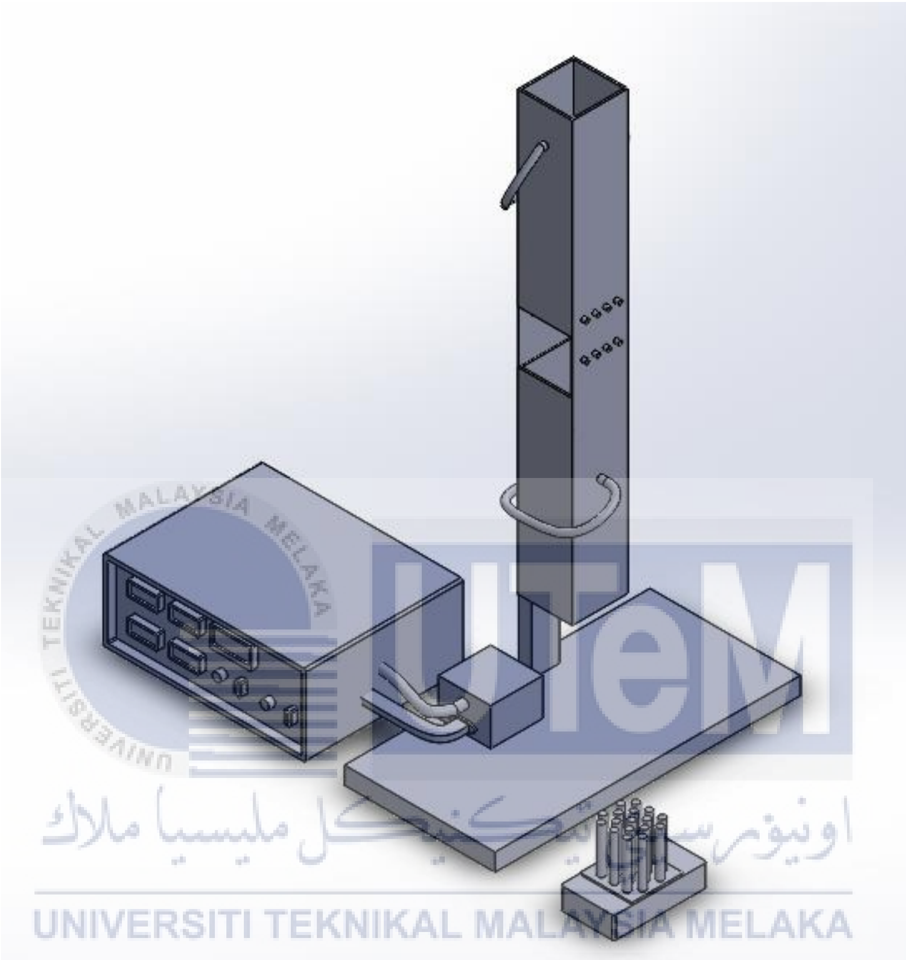

Figure 3.12 : Experiments equipment.

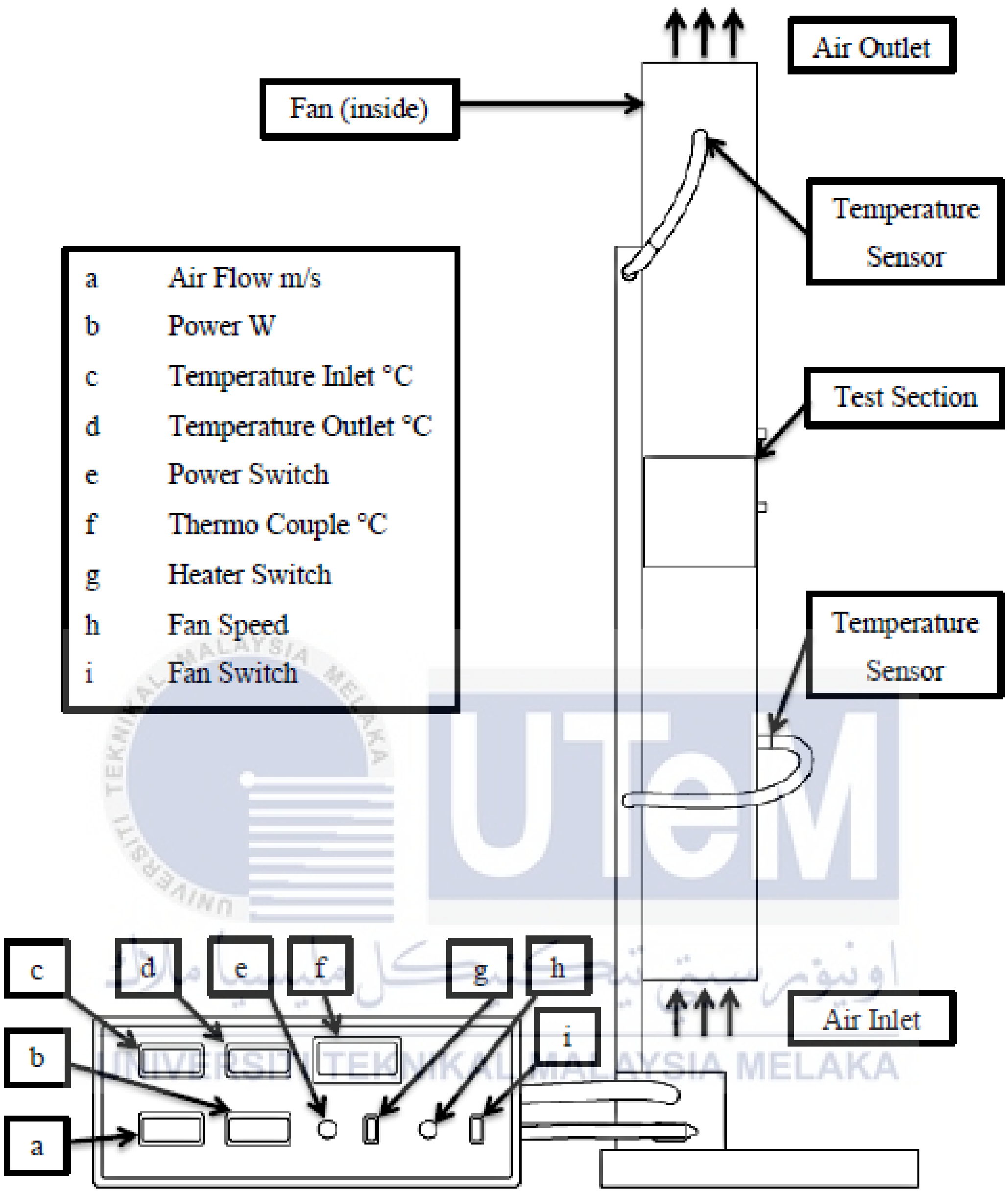

Figure 3.13 : Schematic view for experiment equipment.

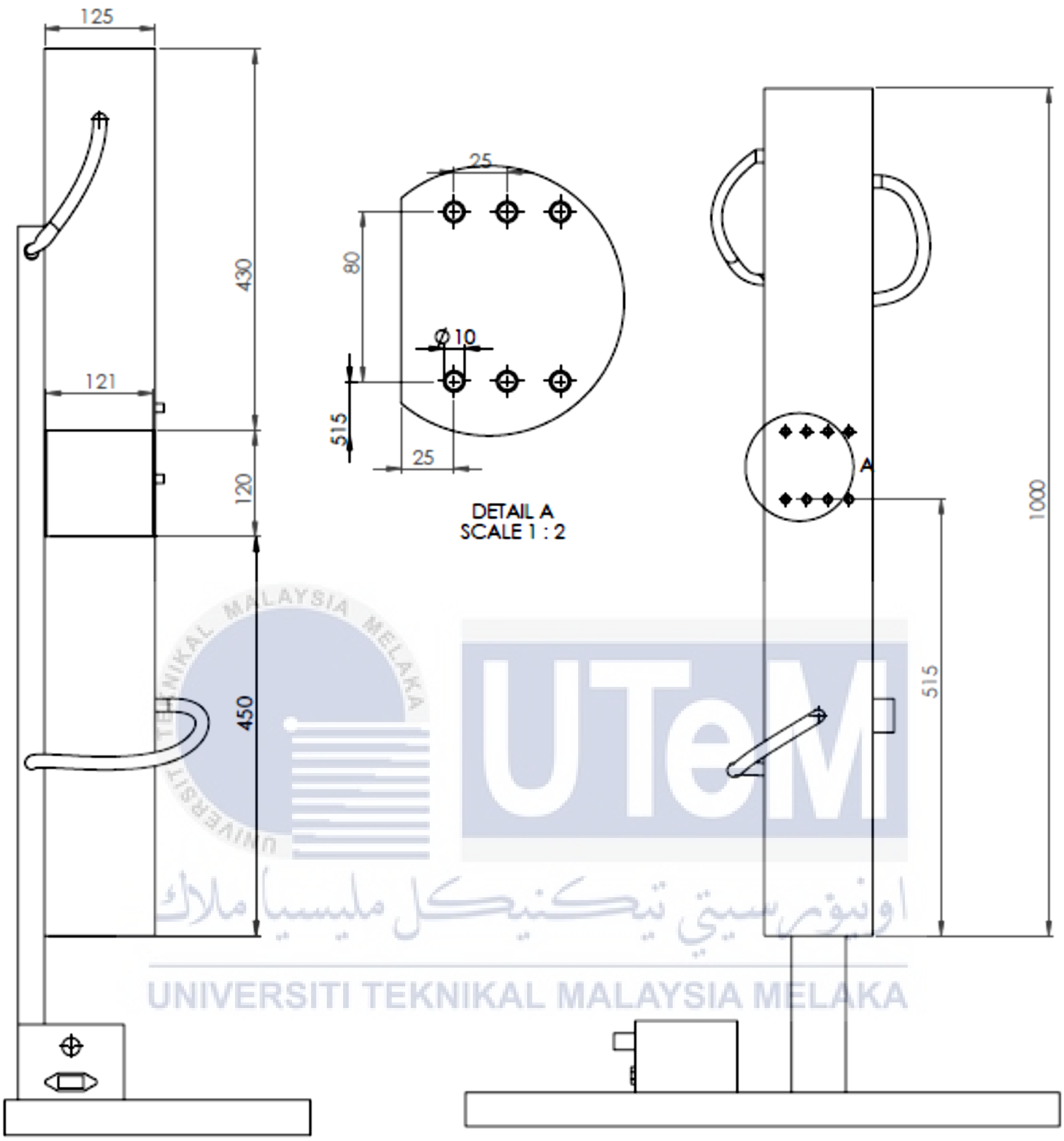

Figure 3.14 : Vertical air duct.

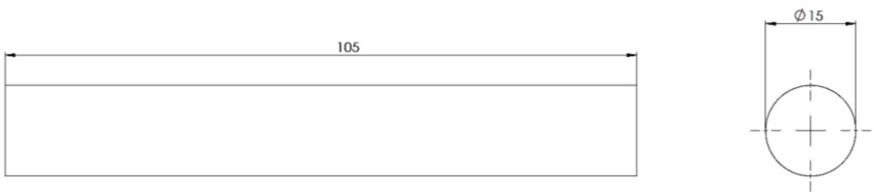

Figure 3.15 : Cross section of tube.

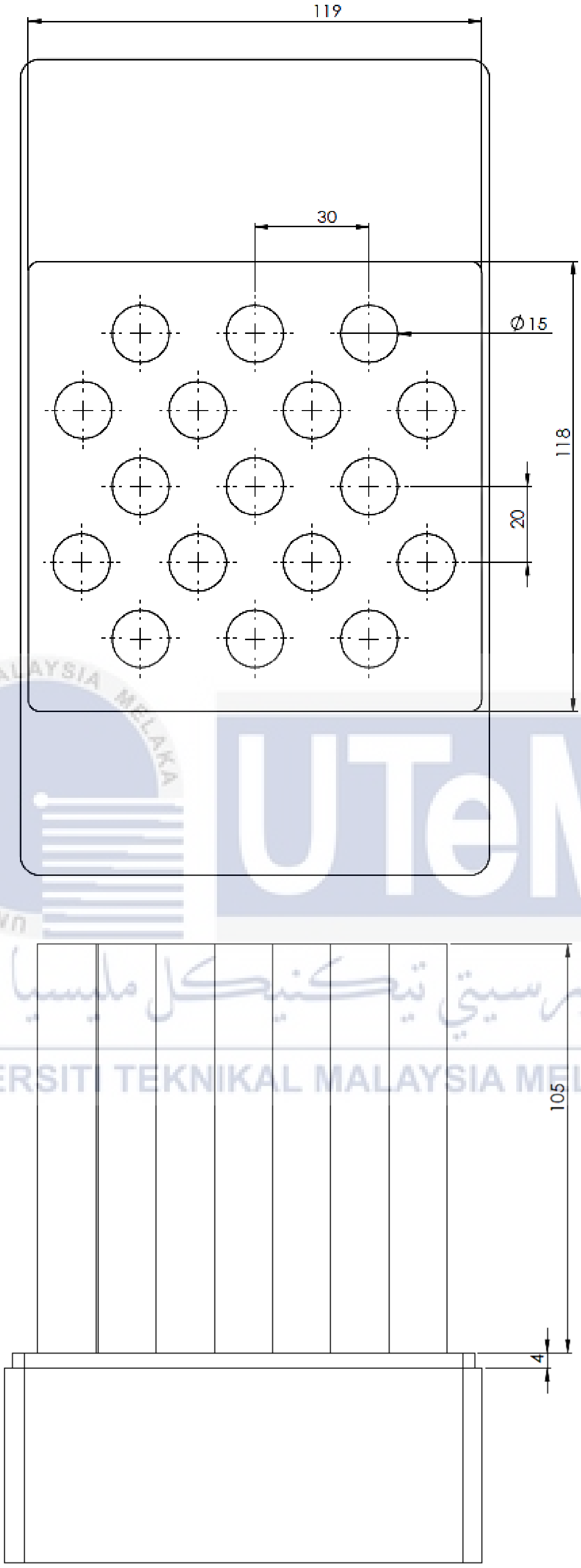

Figure 3.16 : Plate with staggered tube bank arrangement.

3.3.2 Experiment Procedure

To get accurate result for each variable that consider on test specimen in this experiment, the procedure must be follow with precautions. Furthermore, the value must be accurate and in a steady state to get the proper result. This experiment is conduct for one test specimen only, which is the plate with staggered tube bank arrangement.

The test specimen is heated using the electric heating element and the 200 W are set to heating the test specimen by set the value on the control unit. The process heating continue until the temperature of test specimen was reached the steady state as show in **Figure 3.17**. When the temperature of test specimen reach on steady state, the initial temperature of surface plate on test specimen are record and the power supply that using for heating was take out. After that, the test specimen is installed on vertical air duct and clip properly to avoid any leaking during the experiment process as show in **Figure 3.18**. After that, the switch for external fan is ON, which is used to induce the air into the vertical air duct to starting the process of force convection. By using the anemometer that show in **Figure 3.19**, the air velocity inside the vertical air duct is recorded at the inlet of air duct as shown in **Figure 3.20**. After that, the test specimen is left for a moment until it reaches the steady state temperature. Next, on the display of the control unit in **Figure 3.21**, the readings of temperature for inlet and outlet air duct are show on the indicator and the values are recorded in order to calculate the temperature different. From the values of temperature different between air inlet and air outlet, the heat transfer rate is calculated using the formula.

On the staggered tube arrangement, there have the extra data of temperature that are need to obtain during the experiment process, which is the temperature on the point hole that parallel to the tube length and the point location is show in **Figure 3.22**. The temperatures are taken during the process of cooling down the test specimen on several points along the length of tube.

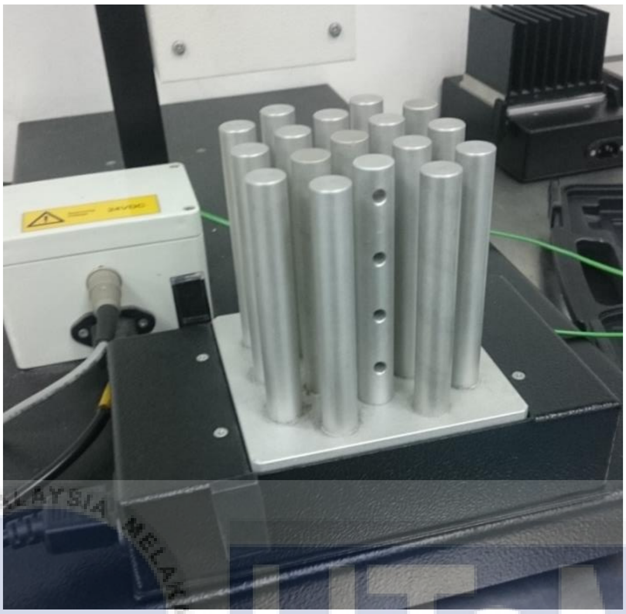

Figure 3.17 : Heating process.

Figure 3.18 : Installing test specimen.

Figure 3.19 : Anemometer device.


Figure 3.20 : Taking air velocity process.

Figure 3.21 : Reading on control unit.

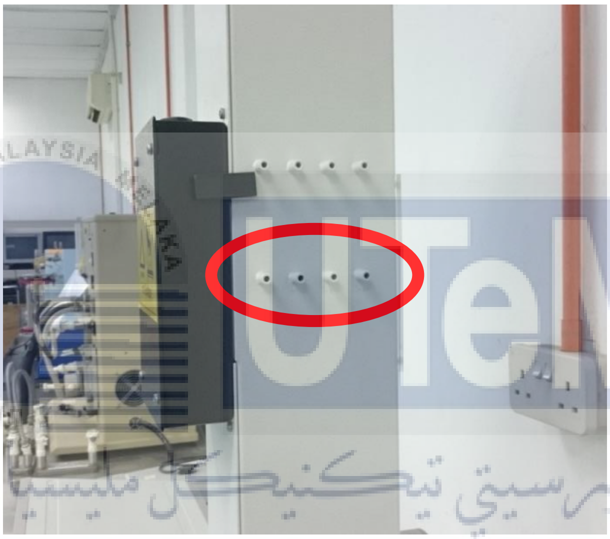

Figure 3.22 : Point location.

CHAPTER 4

RESULT AND DISCUSSION

4.1 Introduction

This chapter described the result from the simulation and experimental approach that has been conducted in order to compare the performance of heat transfer rate for each tube arrangement, which is the inline or staggered tube bank arrangement by referring the temperature different value of air flow at the inlet and outlet section of the vertical air duct. Furthermore, the comparison result between the simulation and experimental approach are also discussed in this chapter in order to validate the results obtained from the simulation approach.

4.2 Simulation

The result for simulation of CFD that are using ANSYS Fluent software has represented as an illustration of the contour, streamline and also the vector of the fluid that flows inside the geometry model. From the simulation result, the average value of a variable at a certain location also can be generated and was recorded to compare the performance for both arrangement models. Thus, the input value for boundary condition for both arrangement models must be the same, which is the inlet air velocity and temperature, and the temperature heating on the surface plate.

4.2.1 Inline tube bank arrangement

The results from post-possessing in ANSYS Fluent for the inline tube arrangement model are discussed in this section by presenting the contour of the temperature, velocity and the pressure that affected the model geometry. Furthermore, the streamline of the velocity of the model also will be presented in this section, which is this illustration will describe the fluid flow in the vertical air duct during the simulation.

4.2.1.1 Temperature profile

In this study, the temperature fluid flow in the heat exchanger model, which is the air is the main concern to prove the better heat transfers rate of the model. **Figure 4.1** shows that temperature contour of air flow from the inlet air at the bottom to the outlet air at the top of the model, where it shows the changes of temperature. On the outlet section of air flow, the temperature is increased from 23.46 °C to 27.90 °C and it shows the show the air temperature difference is 4.44 °C.

From the result of temperature differences, it was indicated that the process heat transfer between solid circular and fluid air have appeared. Moreover, from the intensity of contour airflow was also can determined the process heat transfer, which is the highest intensity indicated the higher heat transfer occurs (Mangrulkar et al. 2017). Thus, the intensity has big different around the cylinder surface and the air, which is the process of convection heat transfer occur.

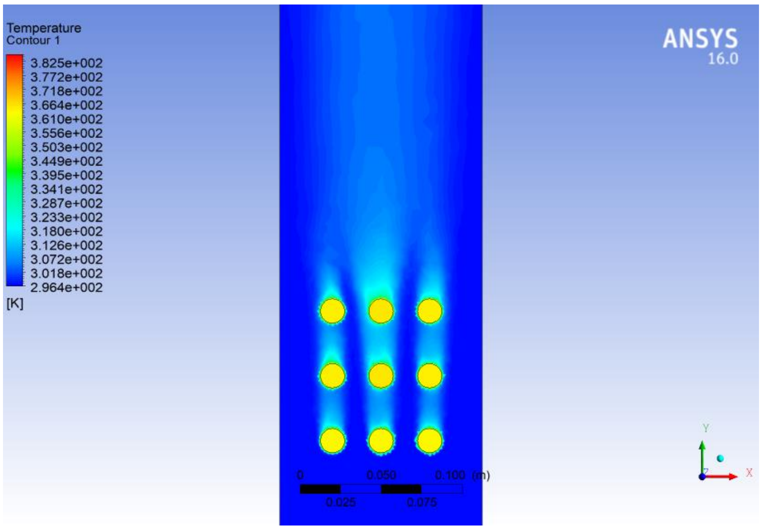

Figure 4.1 : Temperature contour inside air duct.

On the test specimen, the heat transfers also happen and it is presented by the changes in terms of temperature. The test specimen that used aluminium as material for the plate and cylinder, the temperature different between surface plate to tube is 18.14 °C, which is at the initial condition, the base plate temperature a 112.09 °C, after the cylinder is cooling down using the air the average temperature of the surface cylinder it is becomes 93.68 °C as shown in **Figure 4.2**, From the result contour, it show that heat was transfer from the plate to the cylinder and despite to the air that flow around the cylinder and plate, that means process heat transfer, which is the convection heat transfer was occur during that time.

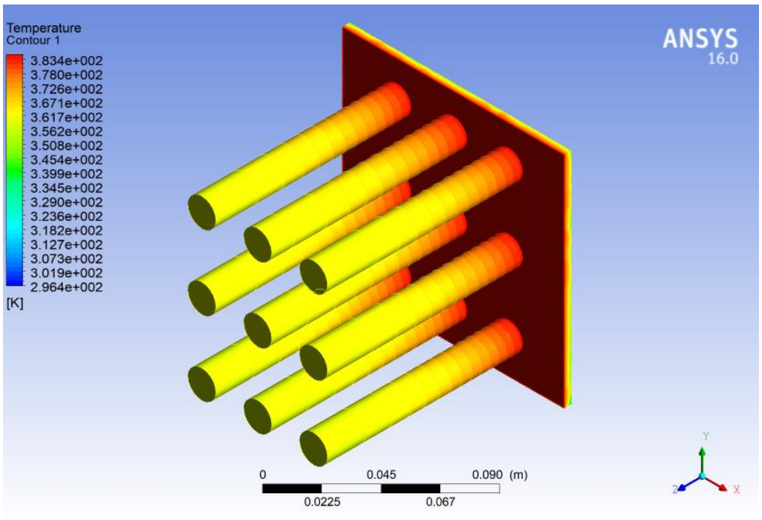

Figure 4.2 : Temperature contour on cylinder.

4.2.1.2 Velocity profile

The velocity of air was also affected during the process heat transfer between the circular tube and the air, which is the value of velocity air became decrease at the outlet of air duct as shown in **Figure 4.3**. At the inlet of the air duct, the value of average velocity inlet air is 2.32 m/s and after the air flow was pass-through with tube bundle, the value becomes slightly higher, which is 2.33 m/s at the outlet section.

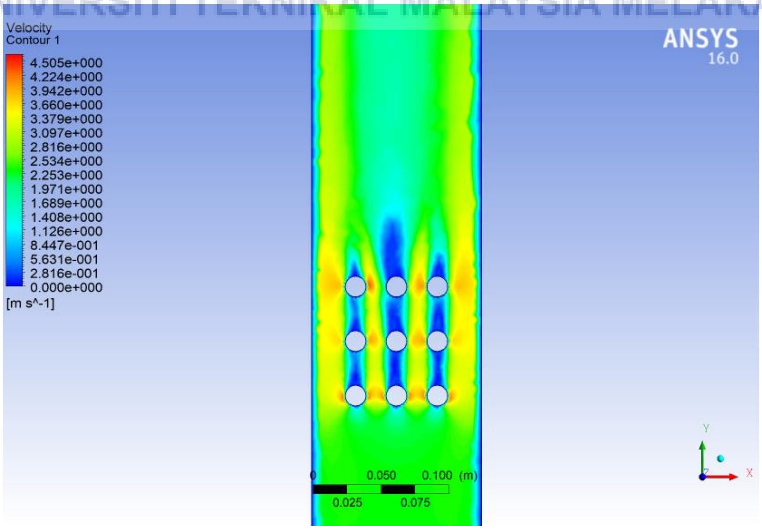

Figure 4.3 : Velocity contour inside air duct.

Figure 4.4 shows the streamline of air velocity inside the air duct that is starting from inlet to outlet air duct, which is from the bottom side to the top side. The streamline shows that the airflow became higher after striking the tubes on the first row meanwhile, on the second and third row, the airflow that hitting the tubes is small compare to the first row and the air velocity that pass through last row also similar to the first row.

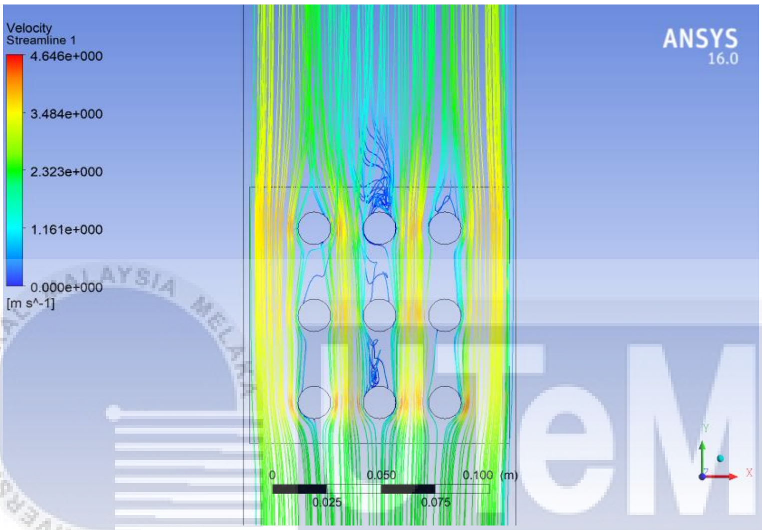

Figure 4.4 : Streamline.

Figure 4.5 shows the vector of velocity air that striking the tube bundle inside the air duct. It is show that the air velocity at the stagnation point on the tube is slower and after it separate, the speed become increase and pass through the tube bundle meanwhile, at the top section of tube show that the velocity very low.

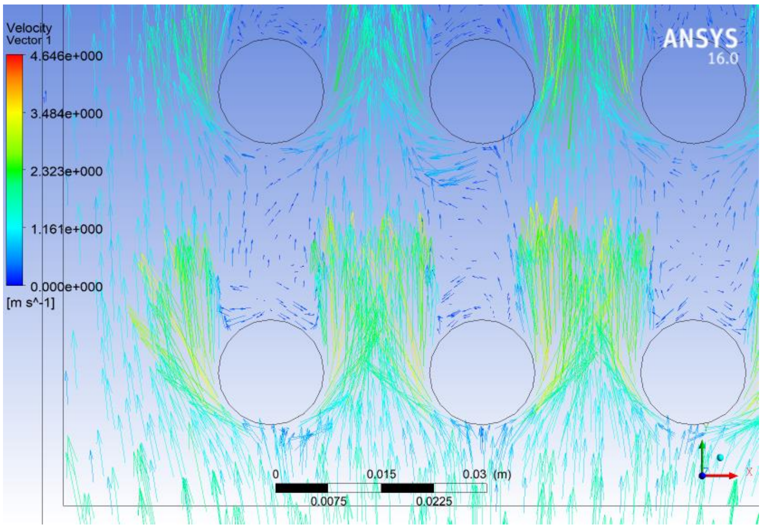

Figure 4.5 : Velocity vector.

4.2.1.3 Pressure profile

Figure 4.6 shows the pressure contour for the airflow inside the air duct, which is the pressure at the outlet section became lesser compare at the inlet section. At the inlet section, the average pressure is 3.34 Pa, after it is passes through the tube bundle the pressure decreased became 0.00 Pa at the outlet section. In the other word, the pressure drop inside the air duct was happen.

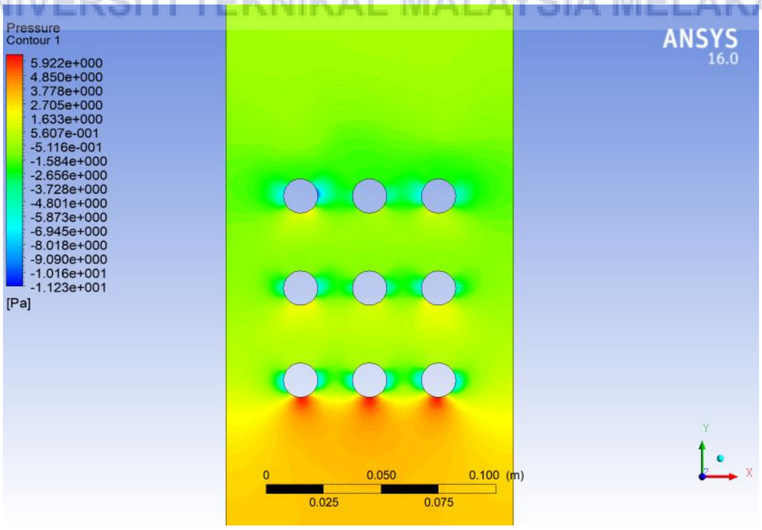

Figure 4.6 : Pressure contour inside air duct.

4.2.2 Staggered tube bank arrangement

The results from post-possessing in ANSYS Fluent for the staggered tube arrangement model are discussed in this section and same with the result from inline tube arrangement model, which is by presenting the contour of the temperature, velocity and the pressure that affected on the model geometry. Furthermore, the streamline of the velocity of the model also will be presented in this section, which is this illustration will describe the fluid flow in the air duct during the simulation

4.2.2.1 Temperature profile

The temperature of the fluid that flows inside the air duct, which air are the main concern in this study in order to represent the better heat transfer for the tube bank arrangement model. For the staggered tube bank arrangement model, **Figure 4.7** shows that temperature contour of air flow inside the air duct from the inlet air at the bottom to the outlet air at the top of the model, where it has the changes of temperature. On the outlet section of air flow, the temperature are increases became 31.80 °C from the 23.46 °C at the inlet air section and it shows the difference of temperature air is 8.34 °C.

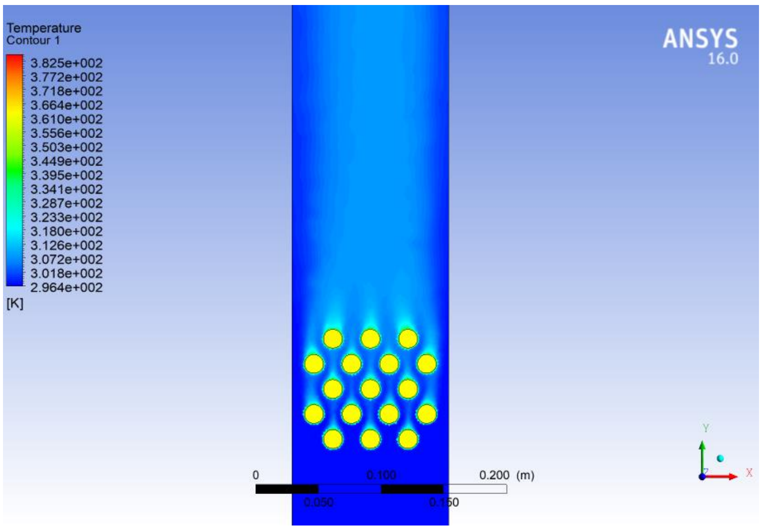

Figure 4.7 : Temperature contour inside air duct.

For the temperature of the cylinder, which is by using aluminium as the material for the cylinder, it is also presented the changes in terms of temperature. At the first initial condition, the surface of plate temperature a 112.09 °C, after the cylinder is cooling down by the moving air, the average temperature of the surface cylinder it becomes 92.03 °C as shown in **Figure 4.8** and this indicates the temperature difference for test specimen is 20.06 °C. From the result contour, it shows the same with result contour for the inline arrangement, which is the convection heat transfer was take place during that time but for the staggered arrangement, it is more quickly compare to the inline arrangement due to temperature difference of surface plate and cylinder is more higher.

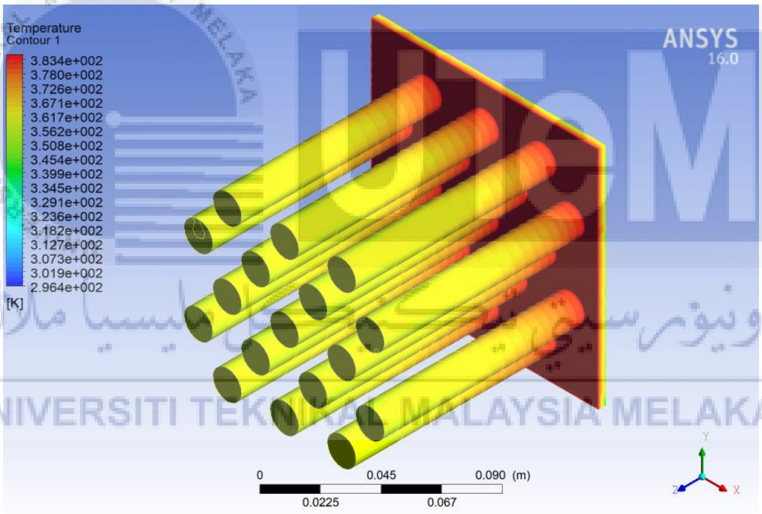

Figure 4.8 : Temperature contour test specimen.

4.2.2.2 Velocity profile

The velocity of air is also affected during process heat transfer between the tube and the air, which the value of the velocity of became decrease at the outlet of air duct as shown in **Figure 4.9**. At the inlet of the air duct, the value of inlet air is the 2.32 m/s and after the air flow was intersected with tubes, the value became slightly higher, which is 2.34 m/s at the outlet of the air duct.

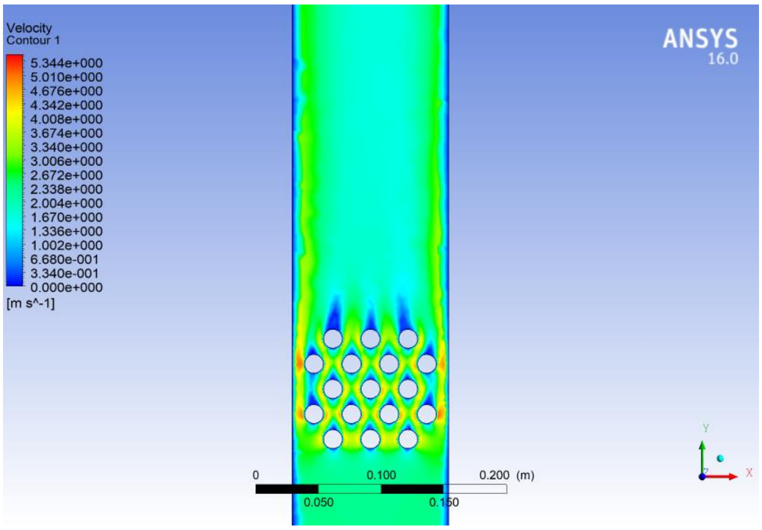

Figure 4.9 : Velocity contour inside air duct.

Figure 4.10 shows the streamline of air velocity inside the air duct that is starting from inlet to outlet air duct, which is from the bottom side to the top side. The streamline shows that the airflow became higher after striking the tubes on the first row meanwhile, on the second and third row, the airflow that hitting the tubes is more than the first row and the air velocity that pass through last row was higher compared to the first row.

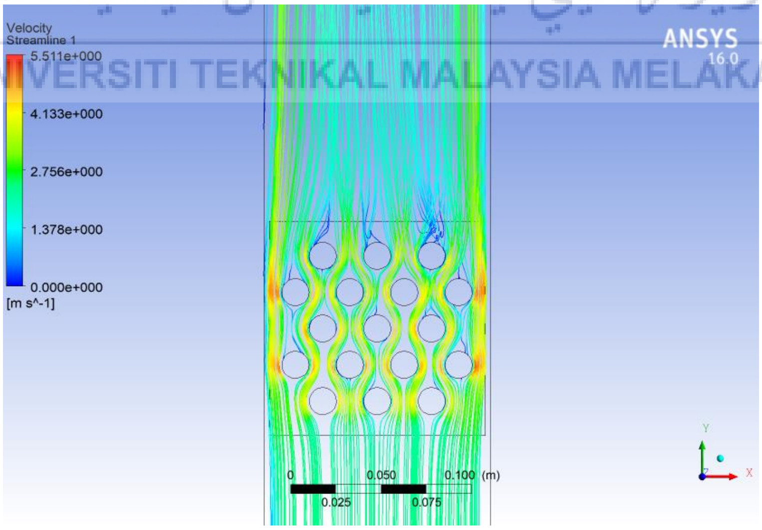

Figure 4.10 : Streamline velocity.

Figure 4.11 shows the vector of velocity air that striking the tube bundle inside the air duct. It is show that the air velocity at the stagnation point on the tube is slower and after it separate, the speed become increase and striking the next row of tube bundle meanwhile, at the top section of tube show that the velocity become more constant due to intersect the air that flow from the previous row.

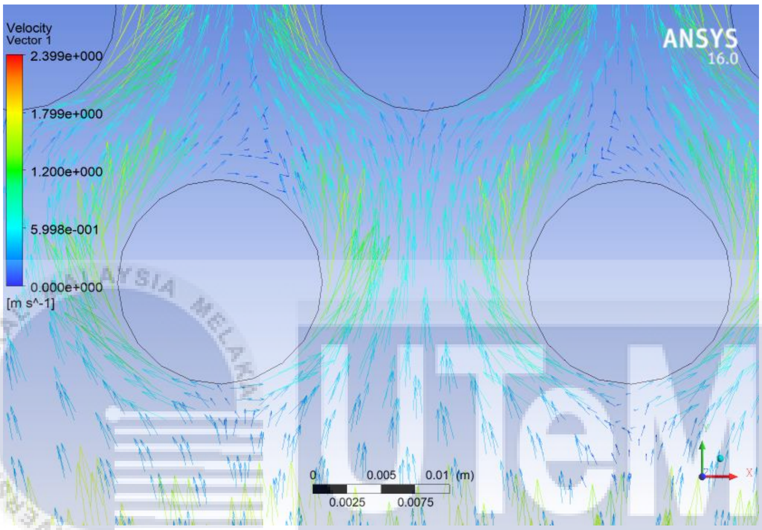

Figure 4.11 : Velocity vector.

4.2.2.3 Pressure profile

Figure 4.12 shows the pressure contour for the airflow inside the air duct, which is it the pressure at the outlet became lesser compare at the inlet. At the inlet section, the pressure is 7.55 Pa, after it is passed through the tube bundle the pressure decreased became 0.00 Pa at the outlet section. In the other word, the pressure drop inside the air duct was happened.

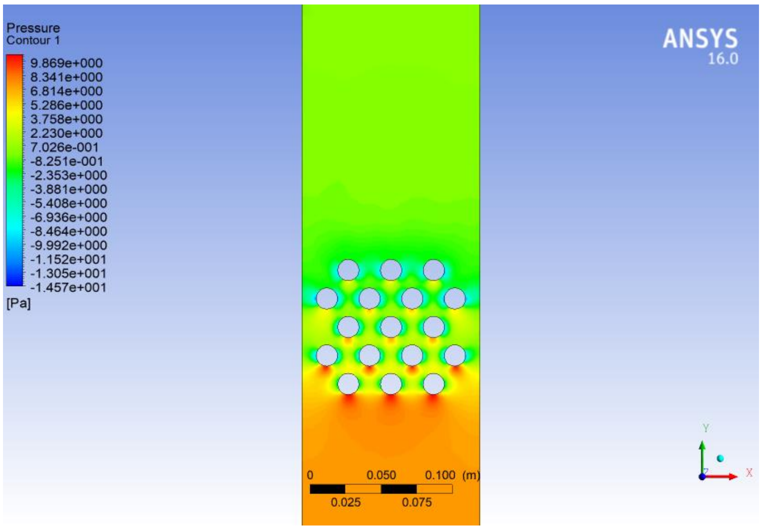

Figure 4.12 : Pressure Contour inside air duct.

4.2.3 Comparisons between inline and staggered tube bank arrangement

To compare the performance of heat transfer for both of tube bank arrangement model, which is the inline and staggered tube bank arrangement, the temperature difference of air that flow inside the air duct is used as a reference to indicate the which one the arrangement has better heat transfer properties. In fact, the main concern of changes of temperature air flow was used as a reference due to the relation of the heat energy equation, which is by assuming the constant specific heat value and mass flow rate of the air flow inside the air duct that crossflow with the tube bank. Therefore, when the value of the temperature difference of air flow increases, the heat energy produced also increases in order to transfer the heat between the air and the solid.

The **Table 4.1** below show the comparison of result for the inline and staggered arrangement for the temperature inlet, outlet, different air and heat transfer that absorb of air, which is obtained from the simulation. In addition, the temperature difference was also calculated and from the calculation, it is clearly seen that the staggered arrangement has a higher value compared to the inline arrangement by 87.84 % of the inline arrangement for the configurations parameter of the staggered arrangement for D, S_T and S_L are equal to 15 mm, 30 mm and 20 mm, respectively. Therefore, the staggered arrangement has the better heat transfer rate compared to the inline arrangement (Khan et al. 2006, Mangrulkar et al. 2017, Kong et al. 2016).

The inline tube bank arrangement produce low heat transfer rate compared to the staggered tube bank arrangement due to limited surface of solid that exposed to the air the cross the solid surface (Uguru-Okerie et al. 2018, Lee et al. 2013). Furthermore, the inline tube arrangement had formed the dead flow zone between two adjacent of a tubes, this is the one factors the inline tube arrangement produce a poor heat transfer efficient compare to staggered tube arrangement (Kong et al. 2016, Ikpotokin et al. 2014).

Table 4.1 : Table comparison of inline and staggered arrangements.

Variables	Inline	Staggered
Temperature Inlet (°C)	23.46	23.46
Temperature Outlet (°C)	27.90	31.80
Temperature Different (°C)	4.44	8.34
Heat transfer absorb by air (W)	12.54	12.72

From the ANSYS Fluent simulation, the result of temperature change on the test specimen was also determine and **Figure 4.13** below show the graph for temperature flow for both of model arrangement, which is Point 0 represented the temperature of the plate surface, meanwhile the Point 1 until 4 is temperature along the circular cylinder length. From the graph, it is illustrated that the test specimen undergoes the process heat transfer, which both models became cooling at the end of the cylinder. However, the results show that the cylinder of the staggered arrangement has become more quickly to cooling compared to the cylinder of the inline arrangement.

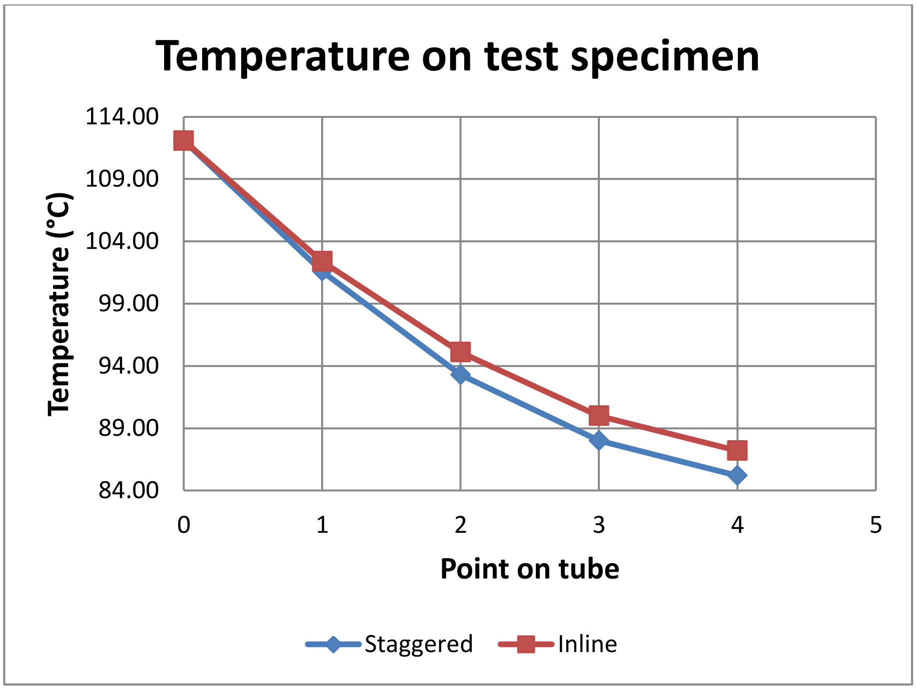

Figure 4.13 : Temperature on test specimen.

Table 4.2 shows the data of temperature for test specimen, which is point 0 represented the temperature of the plate surface, meanwhile for the temperature of cylinder surface is calculated as an average value, which is from point 1 until 4. Furthermore, it also shows where both of arrangement has undergoes temperature changes but in terms of performance, the staggered arrangement has large changes compared to the inline arrangement, which is higher about 8.96% of inline arrangement.

Table 4.2 : Table comparison of inline and staggered arrangement.

Temperature	Inline	Staggered
Plate surface (°C) (Point 0)	112.09	112.09
Average tube surface (°C) (Point 1 - Point 4)	93.68	92.03
Different (°C)	18.41	20.06

4.3 Experiment

The experiment for this study has been conducted to get the data and the actual value of the variable that is used as the input value to the simulation of the test specimen by using the ANSYS Fluent software. Furthermore, the result from this experiment will be used to validate the result from the simulation and the experiment is only conducted on staggered tube bank arrangement due to the limitation of the test specimen model of inline bank arrangement.

4.3.1 Staggered tube arrangement

In order to get more accurate data in this experimental approach, the experiment for the staggered tube bank arrangement was repeated for 5 times with the same power of heating for the each repeating experiment, which is the 200 W. From the data obtain in each repeating experiment, the average value for each variable that are considered in this study was calculated and recorded to apply into the simulation approach.

The **Figure 4.14** shows the graph of reading for air velocity inside the air duct, which is for the 5 times that reading was taken for the same of speed fan and the value of air velocity is not so differed for each repeating where was around 2.31 m/s to 2.34 m/s. From the data obtained, the average value for air velocity at the inlet of air duct is 2.32 m/s.

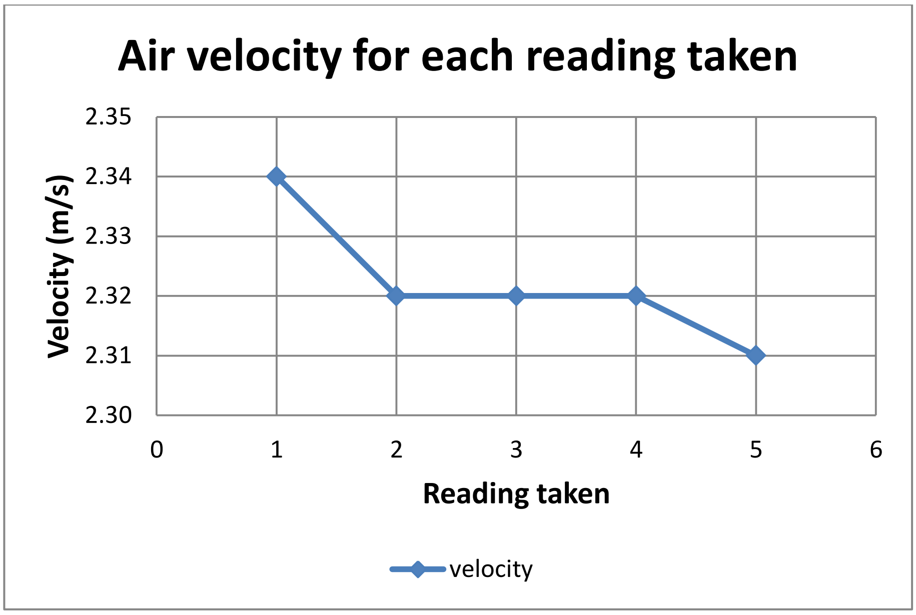

Figure 4.14 : Air velocity at inlet air duct.

For the heating temperature, which is the temperature at the surface plate of the test specimen, the data obtained are shown in **Figure 4.15** and it is around 108 °C until 116 °C by using the constant heating power, which is 200 W that was set on the control panel for each repeating experiments. From the data obtained, the average value of the temperature at the surface of the plate test specimen is 112.09 °C.

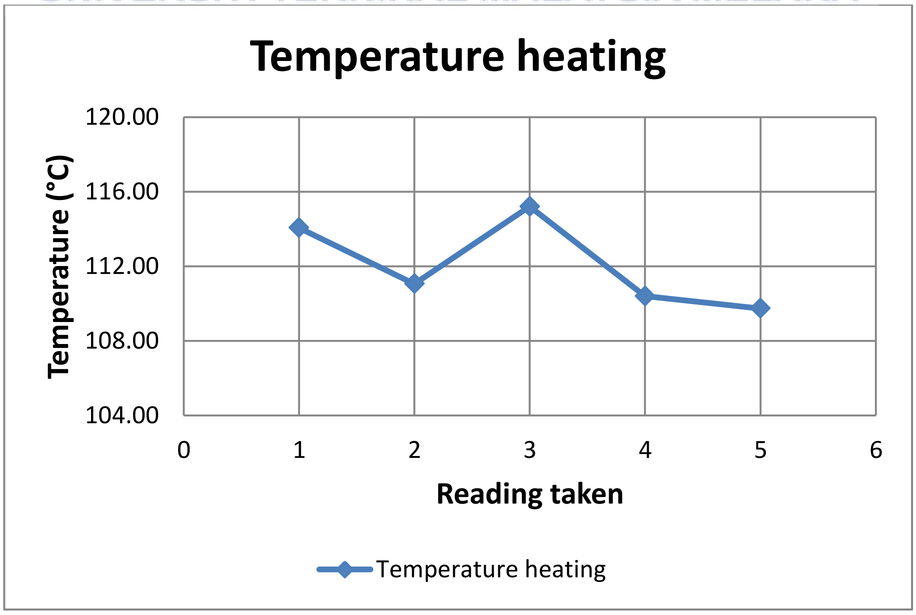

Figure 4.15 : Temperature of plate surface.

In this experiment, there has 6 variables that are recorded during the experiment was conduct, which is the temperatures for air at the inlet and outlet section of the air duct, and the temperature at 4 points along the tube length. In **Figure 4.16**, it shows the value of air temperature that flows inside the air duct, which is at the inlet and outlet section of the air duct and from the graph, the average value of air temperature at the inlet and outlet is 23.46 °C and 27.02 °C respectively.

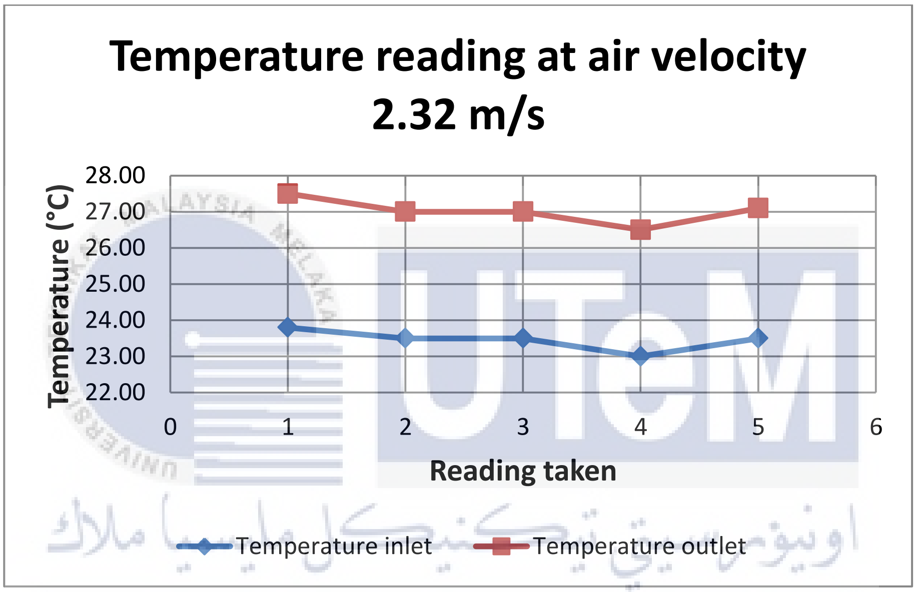

Figure 4.16 : Temperature of air.

For **Figure 4.17** show the graph for the temperature at each point that along the tube length, which is the value is taken 5 times. From the graph the average value of temperature along the tube length was determined, which is the average temperature for point 1, point 2, point 3 and point 4 is equal to 71.36 °C, 68.92 °C, 67.64 °C, and 66.36 °C, respectively.

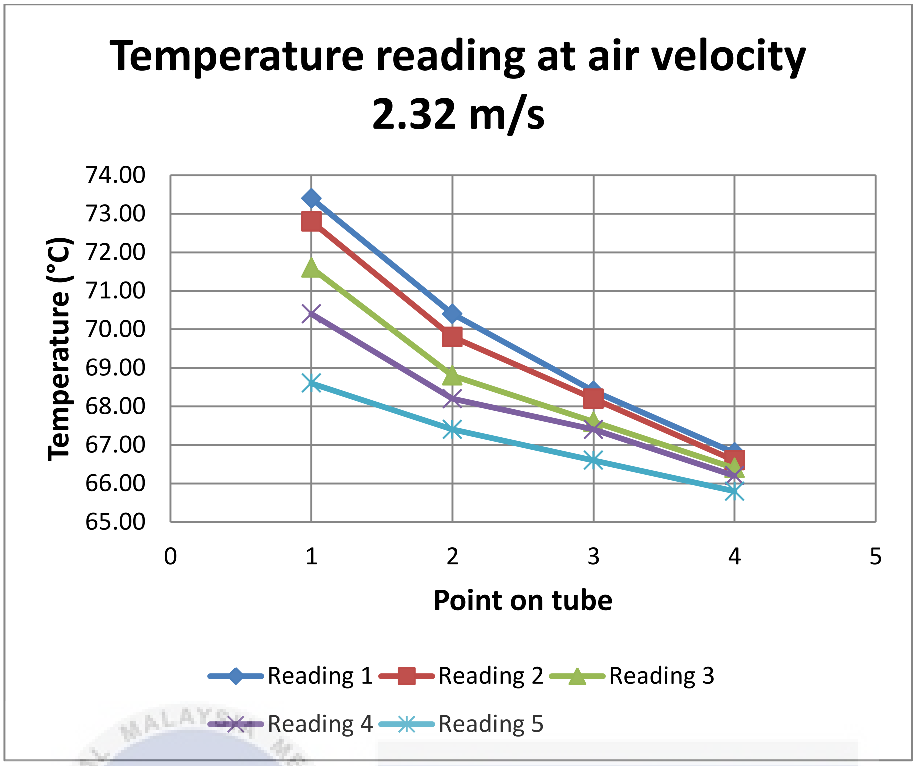

Figure 4.17 : Temperature at each point.

For the average reading of temperature for each point at along the tube length are shown in **Figure 4.18** below, which is the temperature on point 1, point 2, point 3 and point 4 and from the data obtained, the average temperature on the surface of the tube is 68.57 °C.

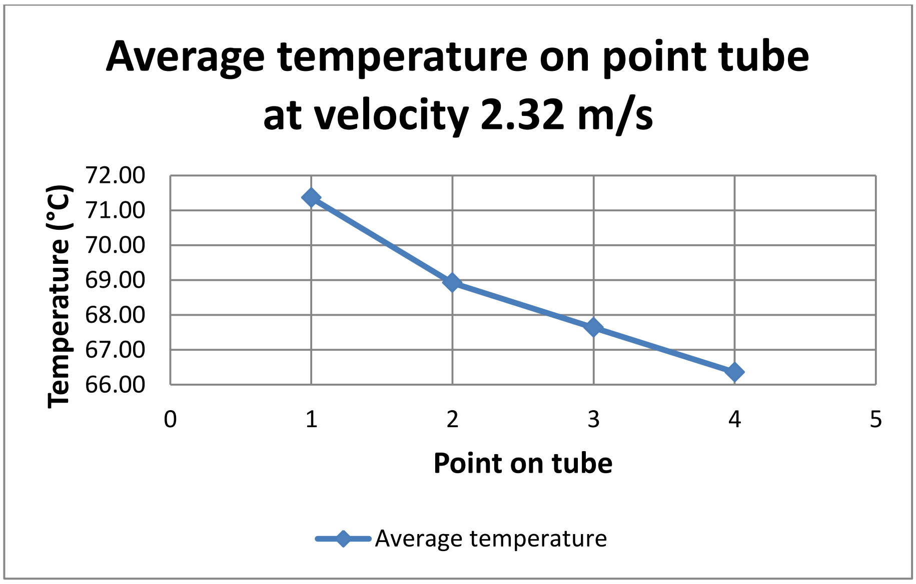

Figure 4.18 : Average temperature at each point.

From the result obtain in this experiment, which by using heating power 200 W to heated the test specimen and 2.32 m/s for the speed of air velocity at the inlet air duct, it shows the temperature of the air flow inside the air duct temperature became increase by about 15.17%, which is the temperature difference for the air flow is 3.56 °C.

Meanwhile, for the surface temperature of the staggered arrangement, it decreased about 38.83%, which is 43.52 °C from the temperature surface plate along to the surface of the tube after the test specimen undergoes the forced convection heat transfer process.

For the input value that will be used to simulate the staggered tube bank arrangement, **Table 4.3** shows the average value that obtains from the simulation in order to make sure the result are same to simulation and it is will be used as a validation step.

Table 4.3 : Staggered arrangement data.

Data obtain	Average value
Velocity air inlet (m/s)	2.32
Temperature air inlet (°C)	23.46
Temperature air outlet (°C)	27.02
Temperature surface plate (°C)	112.09
Temperature surface tube (°C)	68.57

4.3.2 Staggered tube arrangement on variance air velocity

The experiments for the variation of air velocity inside the air duct also were conducted with same heating power, which is 200 W for the staggered tube bank arrangement. There are 5 value of the air velocity at the inlet of air duct is was recorded, which is 0.77 m/s, 1.57 m/s, 2.10 m/s, 2.51 m/s, and 3.06 m/s and all the data that record using the average value.

Figure 4.19 shows the graph of data that obtain from the experiment, which is the temperature at the inlet and outlet of air that flow inside the air duct. It is shown that the values of air temperature at the outlet section were decreasing due to the increase of air velocity speed inside the air duct. From the graph, the air temperature difference for the inlet and outlet are also can be obtained for each the air velocity and it was shown in **Table** 4.4 with the value of heat transfer that absorb by air.

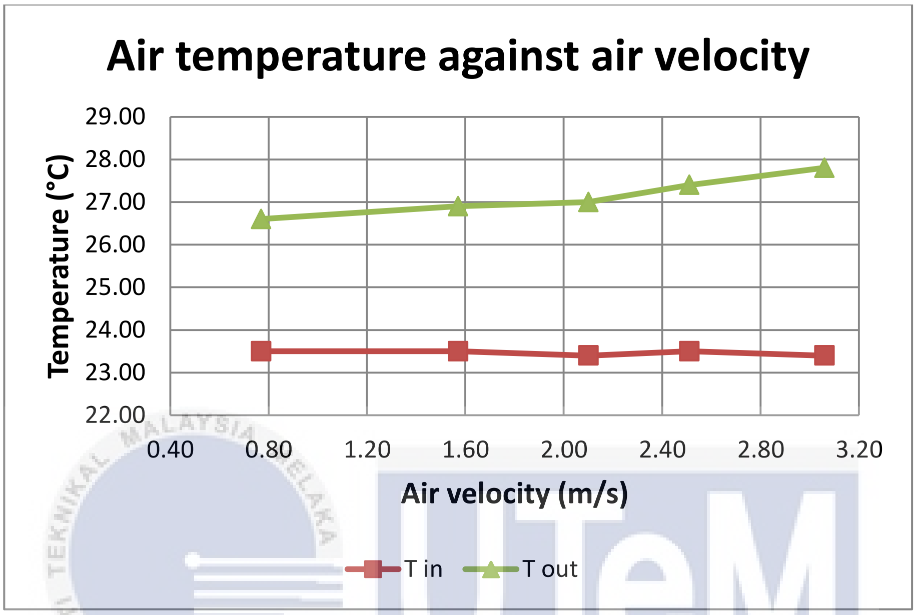

Figure 4.19 : Temperature for variation speed air velocity.

Table 4.4 : Result for variation air speed velocity.

Velocity (m/s)	Temperature different (°C)	Heat transfer absorb by air (W)
0.67	4.30	12.54
1.57	3.90	12.52
2.10	3.70	12.51
2.51	3.40	12.50
3.06	3.20	12.49

The **Figure 4.20** are shows the average value of temperature for each point that are along the tube length, which is at point 1, point 2, point 3 and point 4 and its show the value becomes decrease from point 1 to point 4 for each of air velocity that is considered in this experiment. From that graph, it is also shown that the highest value of air velocity has the lowers temperature at the end of tube.

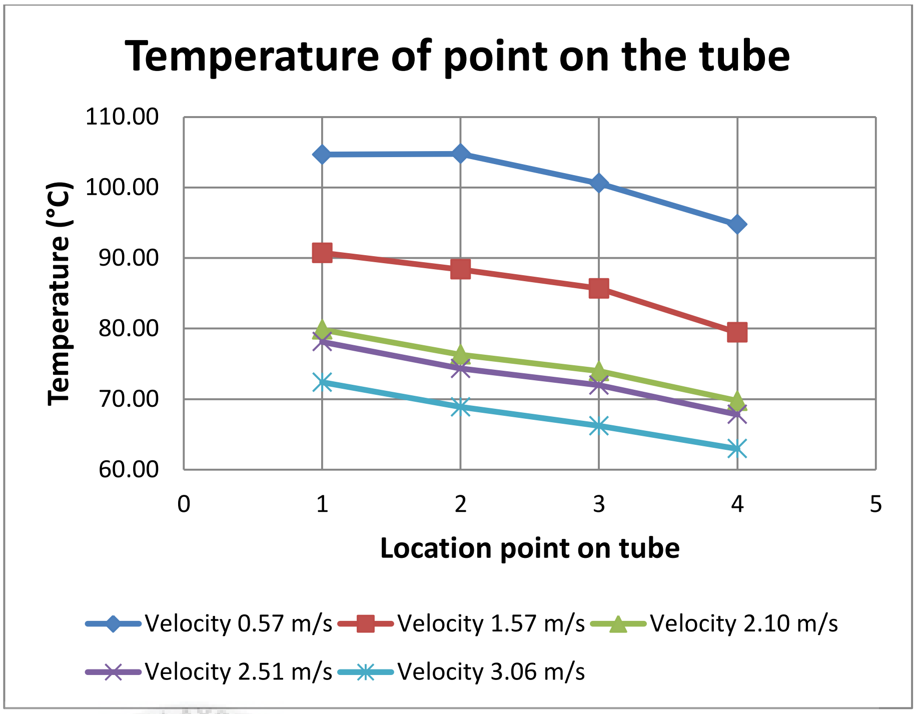

Figure 4.20 : Average temperature of point on the tube.

Figure 4.21 show the average temperature on the surface of the tube for each air velocity speed that is recorded in this experiment. From the graph, it is shown that the temperature on the surface of the tube became decrease with increasing the air velocity speed.

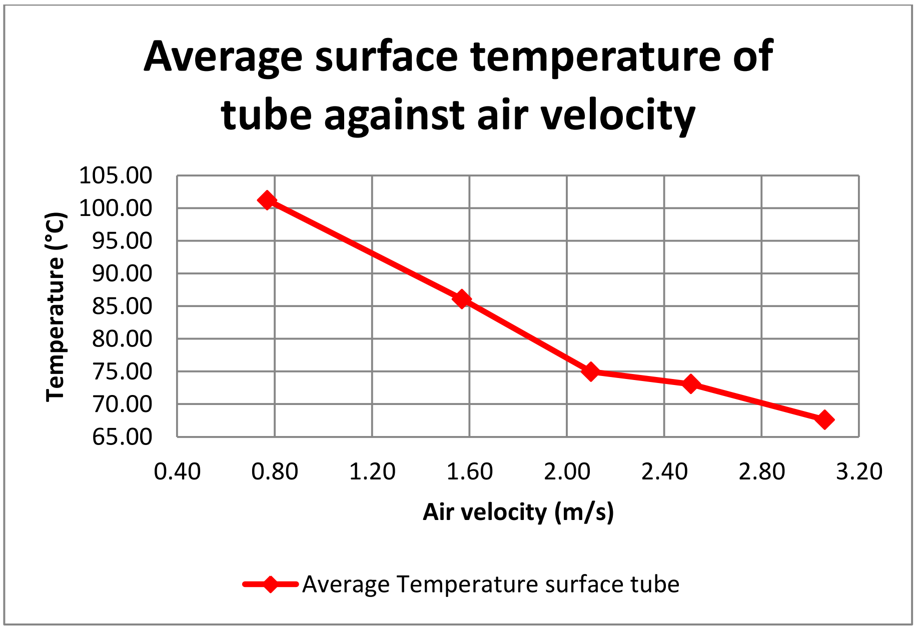

Figure 4.21 : Average surface temperature on variation air velocity.

4.4 Comparisons between simulation and experiment

In order to validate the result of simulation approach by using the experimental approach, the parameter of input value and configurations of test specimen must be must be tally between these two approaches. In detail, the input value for the simulation approach are obtain from the experiment approach, which is the air inlet velocity, air inlet temperature and plate surface temperature is equal to 2.32 m/s, 23.46 °C and 112.09 °C, respectively. Meanwhile, for the configuration are also same for D, S_T, S_L, and S_D, which is 15 mm, 30 mm, 20 mm and 25 mm, respectively.

The comparison result from simulation and experimental approach for the staggered arrangement is shows in **Table 4.5** and to illustrate the contrasts, the temperature of air and solid were taken into account, which is the temperature air at the outlet and surface of the cylinder. From the result obtained, it is indicated that result of simulation has slightly error compare the of result experiment, thus by calculating the percentage error to determine the accuracy of the result. Specifically, the error of simulation result for temperature air outlet is 17.69%, meanwhile the temperature surface cylinder is 35.62% of the experiment result and from the percentage of error, it is shows the values error was slightly higher for temperature air outlet and surface cylinder.

Table 4.5 : Comparison result for simulation and experiment.

Variables	Simulation	Experiment
Temperature air outlet (°C)	31.80	27.02
Temperature surface cylinder (°C)	92.03	68.57

In the other word, there have several factors that make the data that has been collect was error, whether in the simulation or the experiment. Generally, all the parameter, which is the dimension of the model air duct, the staggered tube arrangement, the boundary condition for both approaches are same. Furthermore, the location of temperature air inlet in simulation with thermocouple air inlet at test equipment also same, and same goes to the location of temperature air outlet in simulation with thermocouple air outlet was same. Nevertheless, the one of the reason that contributes the error is during the experiment, the temperature surrounding has effect during the reading of the temperature are recorded. This is because during the experiment, to obtain the temperature at point on the surface of cylinder, the test specimen was take out due to limited space for capture the temperature, so that the heat on the test specimen was exposed with temperature surrounding and process natural convection was happen on that time. Other that than, the experimental apparatus also the factor can contribute the error in this study, where the calibration of apparatus to capture the data for the reading of temperature of air flow and solid surface. Furthermore, the condition of surface wall inside the air duct are also contribute this error result, which the actual surface inside the air duct are slightly rough compare to the surface inside the air duct for the simulation that are assume smooth, therefore the result that produce from the simulation are more higher due to small factor of losses.

Although the result for both approaches has slightly error, the result for both the approach show the same trend and in a good arrangement, which is the temperature of air flow are becomes increase at the outlet section, meanwhile for the temperature solid, it becomes decrease at the surface of the cylinder.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

In conclusion, the objective of this study was achieved and the result has been discussed in this report. The simulation for the inline and staggered tube bank arrangement have been obtained by using the ANSYS Fluent software and has been presented in term of the contour of temperature, velocity, pressure and also the streamline. Furthermore, from the simulation result, the average value of changes for temperature, velocity and pressure was obtained and has been discussed in this report. From the result obtained, the temperature of air and solid has the changes, which the temperature air become increases meanwhile the temperature solid become decrease for both arrangements. Other than that, the inline and staggered arrangements show the same result for velocity and pressure, which is the velocity air was increase meanwhile the pressure was drop at the outlet section.

Moreover, the comparison of heat transfer for both arrangements has been discussed by referring the value of air temperature difference. The result shows that the staggered arrangement has highest temperature difference compared to the inline arrangement, which is about 87.84 % of the inline arrangement. In addition, the result from the simulation has been validated by using the experimental approach, which is the data of configuration and the boundary condition for both approaches was the same. From the result of validation, it is shown that result from simulation has slightly error, which is around 17.65% to 35.65% and the factors that contribute to this error has been discussed in this study. However, the result from both approaches shows the same characteristic for the temperature, which is the temperature of air increased at the outlet section, meanwhile the temperature of solid decrease at the surface of the cylinder. Thus, the result reliability from the simulation in this study has been validated by the result from the experiment.

5.2 Recommendation

For the recommendation of further study, the configuration for both arrangements, which is the inline and staggered arrangements can be changed for a few designs in order to get the better heat transfer rate for both arrangements. There are several parameters can be considered to improve such as reduce or increase the D or S_T and S_L distance of the arrangement in order to enhance more the heat transfer rate and produced the optimum configuration parameters without having an increase of the cycle flow, which is it can increase the size of the heat exchanger. Furthermore, the material selection is also one of the factors to enhance the heat transfer rate, which is by selecting the best material that has the highest thermal conductivity the heat transfer of material and air more effective. Therefore, for the next study of the inline or staggered tube bank arrangement, these two factors can be considered in order to enhance the heat transfer heat for the crossflow heat exchanger.

REFERENCES

Andersson, B., Andersson, R., Hakansson, L., Mortensen, M., Sudiyo, R., & Wachem, B. V. (2012). *Computational fluid dynamics for engineers*. Cambridge: Cambridge University Press.

Bayat, H., Lavasani, A. M., & Maarefdoost, T. (2014). Experimental study of thermal–hydraulic performance of cam-shaped tube bundle with staggered arrangement. *Energy Conversion and Management,* 85, pp 470-476. doi:10.1016/j.enconman.2014.06.009

Bender, A., Meier, A., Vaz, M., & Zdanski, P. (2018). A numerical study of forced convection in a new trapezoidal tube bank arrangement. *International Communications in Heat and Mass Transfer*, 91, pp 117-124. doi:10.1016/j.icheatmasstransfer.2017.12.007

Gowda, Y. K., Narayana, P. A., & Seetharamu, K. (1998). Finite element analysis of mixed convection over in-line tube bundles. *International Journal of Heat and Mass Transfer*, 41(11), pp 1613-1619. doi:10.1016/s0017-9310(97)00244-5

Haider, M. J, & Danish, S. N, Khan, W. A, Mehdi, S. U. & Abbasi, B. A.. (2010). Heat transfer and fluid flow over circular cylinders in cross flow. *Journal Of Engineering Sciences*. 3. 68.

Ikpotokin, I. and Osueke, C.O. (2014) Heat Transfer and Fluid Flow Characteristics Study for In-Line Tube Bank in Cross-Flow. *International Journal of Mechanical & Mechatronics Engineering*, 14 (3). pp. 93-105. ISSN 8282

Incropera, F. P., Dewitt, D. P., Bergman, T. L., & Lavine, A. S. (2007). *Fundamentals of heat and mass transfer* (Sixth ed). Hoboken, NJ: John Wiley.

Khan, W. A., Culham, R. J., & Yovanovich, M. M. (2006). Analytical Model for Convection Heat Transfer from Tube Banks. *Journal of Thermophysics and Heat Transfer,* 20(4), pp 720-727. doi:10.2514/1.15453

Kong, Y., Yang, L., Du, X., & Yang, Y. (2016). Effects of continuous and alternant rectangular slots on thermo-flow performances of plain finned tube bundles in in-line and staggered configurations. *International Journal of Heat and Mass Transfer*, 93, pp 97-107. doi:10.1016/j.ijheatmasstransfer.2015.10.008

Lavasani, A. M., Bayat, H., & Maarefdoost, T. (2014). Experimental study of convective heat transfer from in-line cam shaped tube bank in crossflow. *Applied Thermal Engineering*, 65(1-2), pp 85-93. doi:10.1016/j.applthermaleng.2013.12.078

Lee, D., Ahn, J., & Shin, S. (2013). Uneven longitudinal pitch effect on tube bank heat transfer in cross flow. *Applied Thermal Engineering,* 51 (1-2), pp 937-947. doi:10.1016/j.applthermaleng 2012.10.031

Mangrulkar, C. K., Dhoble, A. S., Chakrabarty, S. G., & Wankhede, U. S. (2017). Experimental and CFD prediction of heat transfer and friction factor characteristics in cross flow tube bank with integral splitter plate. *International Journal of Heat and Mass Transfer*, 104, pp 964-978. doi:10.1016/j.ijheatmasstransfer.2016.09.013

Mavridou, S. G., Konstandinidis, E., & Bouris, D. G. (2015). Experimental evaluation of pairs of inline tubes of different size as components for heat exchanger tube bundles. *International Journal of Heat and Mass Transfer,* 90, pp 280-290. doi:10.1016/j.ijheatmasstransfer.2015.06.047

Mills, A. F., & Chang, B. H. (2004). Error analysis of experiments. *UCLA, Los Angeles, CA,* 12-16.

Mirzakhanlari, S., Pourjalali, B., Sheikhrobat, A. A., & Behzadi, T. (2015). Numerical Investigation of Natural Convection Heat Transfer on Aligned Arrangement Tube Banks. *Procedia Engineering,* 127, pp 148-154. doi:10.1016/j.proeng.2015.11.315

Murakawa, H., Baba, M., Miyazaki, T., Sugimoto, K., Asano, H., & Ito, D. (2018). Local void fraction and heat transfer characteristics around tubes in two-phase flows across horizontal in-line and staggered tube bundles. *Nuclear Engineering and Design,* 334, pp 66-74. doi:10.1016/j.nucengdes.2018.05.005

Murray, D. (1993). A comparison of heat transfer in staggered and inline tube banks with a gas-particle crossflow. *Experimental Thermal and Fluid Science,* 6(2), pp 177-185. doi:10.1016/0894-1777(93)90027-g

Næss, E. (2010). Experimental investigation of heat transfer and pressure drop in serrated-fin tube bundles with staggered tube layouts. *Applied Thermal Engineering,* 30(13), pp 1531-1537. doi:10.1016/j.applthermaleng.2010.02.019

Uguru-Okorie, D. C., Ikpotokin, I., Osueke, C. O. & Fadare, A. D. (2018) Experimental Comparison of Staggered and In-line Tube-bank Thermal Performance. *International Journal of Mechanical Engineering and Technology,* 9 (13). pp. 1061-1070.

Qiu, Y., & Liu, Z. (2004). Boiling heat transfer of water on smooth tubes in a compact staggered tube bundle. *Applied Thermal Engineering*, 24(10), pp 1431-1441. doi:10.1016/j.applthermaleng.2003.11.021

Rathore, M. M., & Kapuno, R. R. (2011). *Engineering heat transfer* (Second ed.). Sudbury: Jones & Bartlett Learning.

Saraireh M. A., Alsaraireh F. M., & Alrwashdeh S. S. (2017). Investigation of Heat Transfer for Staggered and In-Line Tubes, *International Journal of Mechanical Engineering and Technology*, 8(11), pp 476–483

Wang, Y., Gu, X., Jin, Z., & Wang, K. (2016). 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, pp 584-594. doi:10.1016/j.ijheatmasstransfer.2015.10.018

APPENDICES

APPENDIX A

There is Gantt chart for Projek Sarjana Muda (PSM) 1 for 16 week:-

WEEK / TASK	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Title Selection																
Meeting & Briefing By Supervisor																
Chapter 1 (Introduction)																
Chapter 2 (Literature Review)																
Chapter 3 (Methodology)																
Final Report PSM 1 Submission																

APPENDIX B

There is Gantt chart for Projek Sarjana Muda (PSM) 2 for 16 week:-

WEEK TASK	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Meeting & Briefing By Supervisor																
Experiment																
Simulation																
Analysis result																
Recommendation & Conclusion																
Final Report PSM 2 Submission																

APPENDIX C

Sample calculation.

1) Temperature difference.

Given Temperature inlet air = 23.46 °C

Temperature outlet air = 27.90 °C

Temperature difference = Temperature outlet – Temperature inlet

Temperature difference = 27.90 – 23.46

= 4.44 °C

2) Percentage of error.

Given Simulation (temperature air outlet) = 31.80 °C

Experiment (temperature air outlet) = 27.02 °C

$$\text{Error} = \frac{\text{Simulation} - \text{Experiment}}{\text{Experiment}} \times 100\%$$

$$\text{Error} = \frac{31.80 - 27.02}{27.02} \times 100\%$$

= 17.69%

3) Heat transfer that absorb by air.

Given Temperature different (ΔT) = 277.3 K

Specific heat of constant pressure (C_P) = 1.005 kJ/kg.K

Mass flow rate = ρAV

= (1.225 kg/m^3) (0.128 m x 0.125 m) (2.32 m/s)

= 0.045 kg/s

Heat absorb by air = Mass flow rate * Specific heat * Temperature different

= 0.045 * 1.005 * 277.3

= 12.54 W