RELATIVE ATTACK ANGLE OF FIN AFFECTS THE CONVECTIVE HEAT TRANSFER

By

Muhammad Reezwan Shah bin Razalee

22705

Interim report submitted in partial fulfilment of
the requirements for the
Bachelor of Mechanical Engineering
With Honours

FYP

JANUARY 2020

Universiti Teknologi PETRONAS

32610 Seri Iskandar

Perak Darul Ridzuan

CERTIFICATE OF APPROVAL

Relative Attack Angle Of Fin Affects The Convective Heat Transfer

by

Muhammad Reezwan Shah bin Razalee
22705

A project dissertation submitted to the
Mechanical Engineering Programme
Universiti Teknologi PETRONAS
in partial fulfilment of the requirement for the
BACHELOR OF ENGINEERING (Hons)
(MECHANICAL)

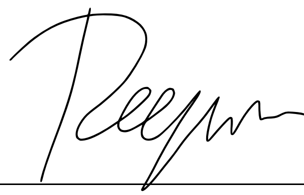
Approved by,

____
(Dr Khurram Altaf)

UNIVERSITI TEKNOLOGI PETRONAS
TRONOH, PERAK
JANUARY 2020

CERTIFICATE OF ORIGINALITY

This is to certify that I am responsible for the work submitted in this project, that the original work is my own except as specified in the references and acknowledgements, and that the original work contained herein have not been undertaken or done by unspecified sources or persons.



MUHAMMAD REEZWAN SHAH BIN RAZALEE

ABSTRACT

Analytical investigation is made for three-dimensional fluid flow and convective heat transfer for designed fin. This project aims to achieve the highest rate of convective heat transfer by changing the relative attack angle of fin. For analytical investigation, simulation software was used to save time and cost of the project by running multiple simulations on different attack angle of the fin. The results are compared with other simulation running with different velocity magnitude. The designed fin with perfect relative attack angle can be used in future industry that needs heat transfer application as it is more reliable and cost economical.

ACKNOWLEDGEMENT

First and foremost, Alhamdulillah, praise to Allah the mighty, for his showers of blessings throughout my project work to complete my final year project.

I would like to express my deep and sincere gratitude to my final year project supervisor, Dr Khurram Altaf, Senior Lecturer of Mechanical Engineering Department, for giving me the opportunity to do research and providing guidance throughout this project. His vision, intelligence, sincerity, and motivation have deeply inspired me. He has taught me the methodology to carry out the final year project and complete the project successfully. It was a great privilege and honor to work and study under his guidance. I am extremely grateful for what he has offered me. I would like to thank him for his leadership, and guidance. I am extending my heartfelt thanks to his master research student, Mr Adeel, for his patience and guidance during the discussion and throughout the project. Without them, I could not complete my final year project successfully.

I would like also to thank my family for their prayers, love and sacrifices for educating and preparing me for future. Also, I express my thanks to all my friends for guiding me throughout my study.

Table of Contents

LIST OF FIGURES

TABLE OF CONTENTS

CHAPTER 1

INTRODUCTION

1.1 Background Study

Heat transfer can be described as flow of heat due to temperature difference and the subsequent temperature distribution and changes. In other words, it is a process of heat flow from high temperature towards the low temperature. In terms of thermodynamic, heat transfer is the movement of heat along the system's boundary due to temperature difference between the surroundings and the system. The heat will continue to flow across the boundary until reaching the same temperature, which is called as thermal equilibrium.

There are three modes of heat transfer; conduction, convection and radiation. Conduction occurs when two solids contact with each other at different temperature. The heat is transferred when adjacent atoms vibrate against one another. The heat transfer from high temperature region to lower temperature region. The heat will continue to flow until both solids achieve thermal equilibrium, at point which both solids have the same temperature.

Convection is transfer of heat by mass movement of a fluid when the heated fluid is caused to move towards the opposite of the heat's source, carrying energy with it. Heat convection is often the primary mode of energy transfer in liquids and gases. Convection occurs because the particles of hot air expands and become less dense, which will rise and replaced by the cold air. It created circulation patterns in a system, causing convection currents to transport energy within the air.

The rate of convection heat transfer can be increased by increasing the temperature difference between the surface and fluid, increasing the coefficient of convection heat transfer or increasing the exposure or contact area between the surface and fluid. In this study, we will increase the area of contact by introducing the fins to the system.

In the study of heat transfer, fins are extended surface protruding from a surface to increase the rate of convection heat transfer. Adding fins to an object can be an solution that has the most economical to heat transfer problem, compared to increase the temperature difference or increase the coefficient of convection heat transfer. Enhanced fin geometries can increase the rate of convection heat transfer compared to plain fin. In addition, adjusting the attack angle of the fin can also increase the rate of convection heat transfer.

Fin efficiency can be interpreted as the ratio of actual heat transfer through the base of fin divided by highest possible heat transfer rate through the base of fin, which can be obtained if the entire fin is at base temperature. It is very useful in designing a system or in estimation of the system's performance if we know the fin efficiency. The fin efficiency can be derived from uniform heat transfer coefficient, constant fluid temperature, and heat conduction in one dimensional of the fin.

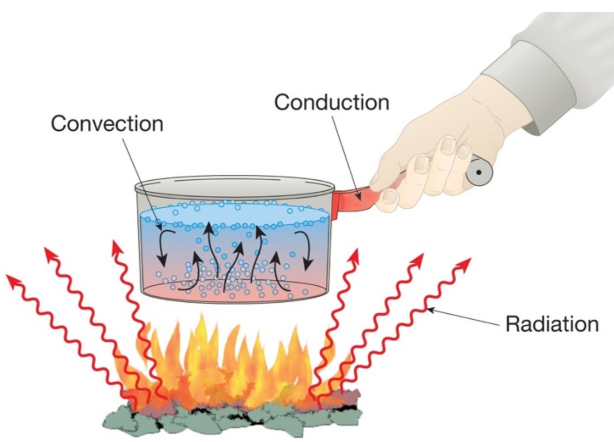


Figure 1.1: Types of Heat Transfer

1.2 Problem Statement

Different designs of fins create different rate of heat efficiency. Changing the design, with relatively different attack angle of fins can create higher rate of heat efficiency. In this project, at which angle of attack for the fins can create the highest rate of heat efficiency. This project also will determine parameters that affect the heat efficiency.

The energy transmitted as mechanical power is less than the energy generated during combustion in an IC engine. Only 25 % of the power generated, is converted into useful work. Energy lost due to friction (during transmission of power as mechanical energy) and via exhaust gases are 10 % and 35 % of the power produced, respectively. Remaining 30 % is excess heat, which needs to be dissipated through cylinder walls. A new four-bar mechanism for an engine called crank-rocker engine (CRE). Various kinds of fins are used for heat dissipation through engine cylinder block. Basic purpose of using fins is to provide more surface area for heat to be convected away from engine walls.

1.3 Objectives

The objective of this project is to determine at which angle and parameter affect the convective heat transfer, as different angle leads to different rate of heat efficiency. This project also will determine which attack angle has the highest convective heat transfer on designated fin design. The determined relative attack angle of fins can be used for the crank-rocker engine through the engine cylinder block.

1.4 Scope of Study

The main focus of this project is to determine the relative attack angle of fin to achieve the highest heat transfer efficiency. The study covers analytical and experimental analysis. The software used for analytical analysis is ANSYS, which create solid modelling and run simulation with thermal condition through it. By using this software, we can save time and cost of the project. The attack angle of fins with the highest heat efficiency will be selected for experimental purpose.

We need to run the experiment of this project for validation purpose. We will compare both analytical and experimental results to validate the simulation and real life situation. During the experiment, the attack angle of fins with highest efficiency will be used based on the simulation on ANSYS.

CHAPTER 2

LITERATURE REVIEW

Heat can be transferred in three modes which are conduction, convection and radiation. For conduction, the equation as below:

$$Q = \frac{kA(T_2 - T_1)}{d}$$

Where, (1)

Q = Amount of heat transferred

K = Thermal conductivity of the material

A = Heat transfer area

T_2 - T_1 = Temperature gradient, the difference between two temperatures

d = material thickness

For convection, the equation as below:

$$Q = hA(T_2 - T_1)$$

Where, (2)

Q = Amount of heat transferred

h = Convective heat transfer coefficient

A = Heat transfer area

T_2 - T_1 = Temperature gradient, the difference between two temperatures

For radiation, the equation as below:

$$Q = \sigma A(T_1^4 - T_2^4)$$

Where, (3)

Q = Amount of heat transferred

σ = Steftan - Boltzmann constant (5.6703 x 10^{-8})

A = Heat transfer area

T_1 = Hot body absolute temperature

T_2 = Cold surroundings absolute temperate

For this project, the important equation to be considered is convective heat transfer equation. In convective heat transfer, a major role in overall energy transfer process is the bulk fluid motion of the fluid. There are two types of convective heat transfer, which are forced convection and free convection. In forced convection, the fluid is forced to flow over a surface by external force, while for free convection the fluid motion is caused by buoyancy effect.

According to Jasim and Soylemez (2018), introduction of perforated fin can improve the thermal performance of heat sink. Perforated fin is designed fin with holes in the solid to increase the area of heat transfer, so that the amount of heat transferred can be increase. Hence, it will increase the heat transfer efficiency of the fin. Slightly changed the inclination orientation of the perforated fins can increase the amount of heat transferred compared to parallel orientation. Jasim and Soylemez (2018) have found that few advantages with inclined perofation can be achieved by increasing the inner convection area, increasing the external perforated area and change the inclination angle. The result of this modifications lead to change of the effectiveness of heat transfer.

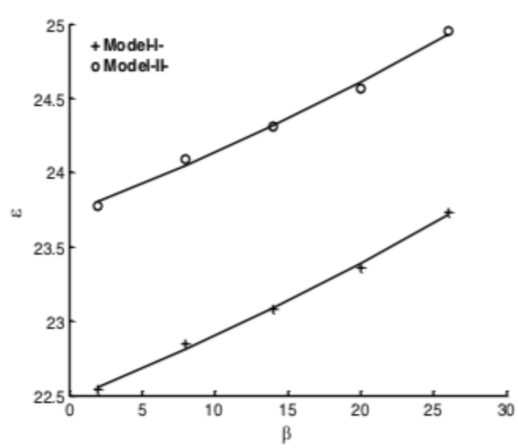


Figure 2.1: Change the effectiveness with the inclination angles

Improving the thermal performance of pin fin by 65% and decreasing of thermal resistance are the results of applying inclined perforation fins.

According to Kadbhane and Palande (2016), the convective heat transfer of fin depends on various factors such as geometrical and operating parameters, number of fins, attack angle of the fins and material of the fins. Due to limitation to increase size of the fins, optimal fins spacing were found as the most important factor to enhance convective heat transfer. Plus, forced convection has better enhanvement in convective heat transfer compared to natural convection.

According to Rasel, Islam, and Hasanat (2016), the forced convection applied to rectangular body with staggered arranged circular fins affect the flow characteristics. When the nusselt number and heat transfer coefficient increase, the velocity increase. As the velocity increases, the heat transfer rate from fin decreases and from the base increases.

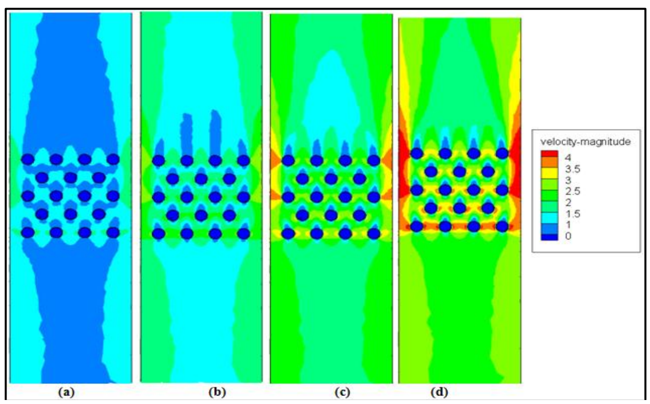


Figure 2.2: Velocity profile at midsection of the duct
(a) v = 1 m/s (b) v = 1.5 m/s (c) v = 2 m/s (d) v = 2.5 m/s

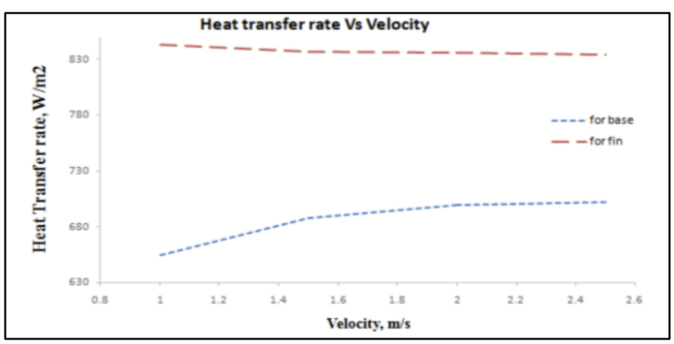


Figure 2.3: Heat transfer rate vs velocity of air for base and fin

As the velocity increases, the heat transfer for base increases and the heat transfer for fin decreases.

According to Narato, Wae-hayee, Vessakosol, and Nuntadusit (2017), the inclined angle of pin arrays could affect the heat transfer characteristics. They made a research and experiment for case of angle at 60˚, 90˚, 120˚ and 135˚ at constant Reynolds number at Re=32,860. They used simulation software ANSYS Workbench to gain the data.

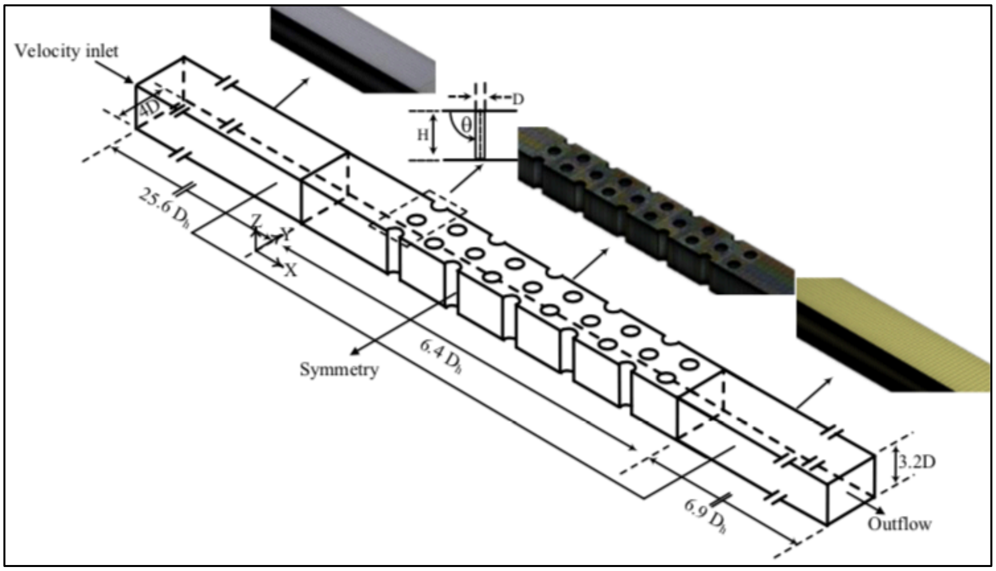


Figure 2.4: Computational domain and grids

The Nusselt number is the ratio of convective to conductive heat transfer at boundary in fluid. The higher the Nusselt number, the higher the convective heat transfer. The equation for Nusselt number as below:

$$Nu = \frac{convective\ heat\ transfer}{conductive\ heat\ transfer} = \frac{hL}{k}$$

Where, (4)

k = Thermal conductivity of the fluid

L = Characteristic length

h = Convective heat transfer coefficient

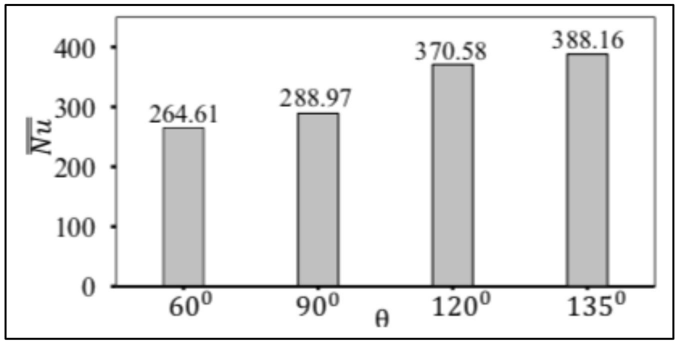


Figure 2.5: Pin inclined angles vs Nusselt number

Based on experiment by Narato, Wae-hayee, Vessakosol, & Nuntadusit (2017), the higher the inclination angle, the higher the Nusselt number. This means the inclination increases the convective heat transfer.

CHAPTER 3

METHODOLOGY

3.1 Project Details

This project consists of simulation study. The simulation needed to determine the relative attack angle of fin with highest convective heat transfer. Computational Fluid Dynamics (CFD) is the discipline to predict fluid flow, heat transfer and related phenomena by solving mathematical models. The reasons of using the simulation software are to lower the cost and optimize time of the project.

Few simulations will be run to determine the angle of attack with high convective heat transfer efficiency with different velocity magnitude. All results will be compared and analysis will be done.

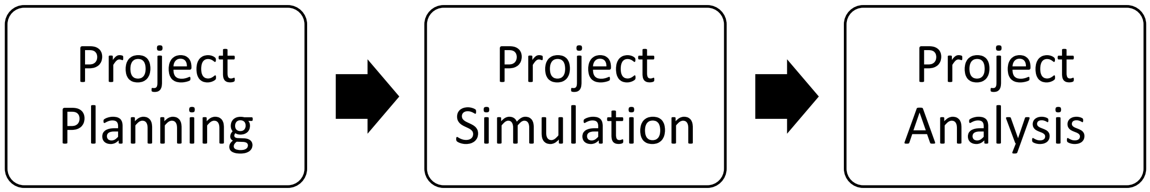
Initially, the project was planned to execute experimental study. The experimental study consists of running the fabricated fin design in the wind tunnel. The objective of running the experiment is to validate the result gained from the simulation process. The wind tunnel consists of air from inlet with few velocities magnitude, fabricated fin design, heat source, and thermocouples to record the temperature. Due to pandemic outbreak COVID19, the experiment could not be executed in this project.

3.2 Tool

The tools used for this project is simulation software. For simulation software, ANSYS Workbench has been selected as this software is the most reliable and suitable to run the simulation of convective heat transfer. ANSYS Workbench platform is suitable to deliver comprehensive and integrated simulation. It can be used to perform various types of thermal and fluid analyses. It is used in wide application range and many industry field including manufacturing, fabricating material and car development industry.

3.3 Project Approach

This project can be divided into three major phases as its timeline which are; project planning, project simulation and project analysis.



For project planning phase, problem statement of the project has been identified. Through the problem statement, the objective and scope of study has been constructed to solve the problem. During this phase, knowledge and ideas of the project has been obtained from lecturers, researcher students, and literature review from various authors in order to execute the project successfully.

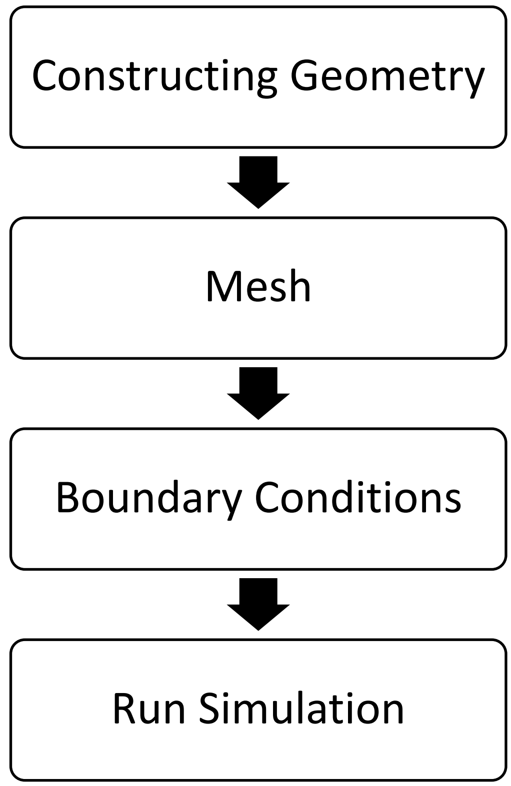
During the second phase of the project which is project simulation, ANSYS Workbench will be used to run the simulation. The first step in this phase is modelling the geometry of the fin according to its dimension. Next, mesh of the geometry will be developed, and then the boundary condition will be introduced. The relative attack angle of the fin will be adjusted by trial and error. Simulation will take place when angle has been adjusted. The repetitive process of adjusting the attack angle and running simulation will be occurred until the results achieve the highest rate of convective heat transfer.

The results will be taken account into further analysis. It will be compared as the objective is to determine the relative attack angle with highest convective heat transfer.

3.3.1 Project Understanding

Project understanding starts with finalizing and confirming the objective and scope of study of this project. Few discussions have been carried out in order to choose the desired focus for this project study. Studying and learning on simulation software also a part of understanding.

3.3.2 Running the Simulation



Simulation software that will be used for this project is ANSYS Workbench. A specific function in assisting fluid flow study that will be used is Computational Fluid Dynamic (CFD) Fluent function. Constructing a geometry model in this software is the first step to do. The geometry and dimension of fin for this project will be provided by another ongoing Final Year Project.

Then, meshing will be developed on the geometry model of the fin, as it is very important because the simulation will solve mathematical equation through the mesh introduced on the model. Meshing is subdivision of geometry into discrete geometry so that it can undergo simulation process. The higher the value of mesh, the higher the accuracy but it will take longer time to complete the simulation.

Next, boundary conditions will be setup which consist of flow inlet and outlet boundaries, the thermal of heat source, and wall. Boundary conditions are essential component of a mathematical model. This step is important to set the parameters required to the geometry model.

Lastly, the software will be ready to run the simulation by making mathematical calculations. This process depends on the number of mesh and complexity of the design. The results with highest rate of convection heat transfer will be used in the next experiment.

3.3.3 Conducting the Experiment

Wind tunnels are used primarily to test the aerodynamics of a vehicle or vehicle components. This is because the wind tunnels can assess how gaseous substance behaves around a geometry. For this project, few requirements for wind tunnel are needed as temperature requirement of the convective heat transfer experiment.

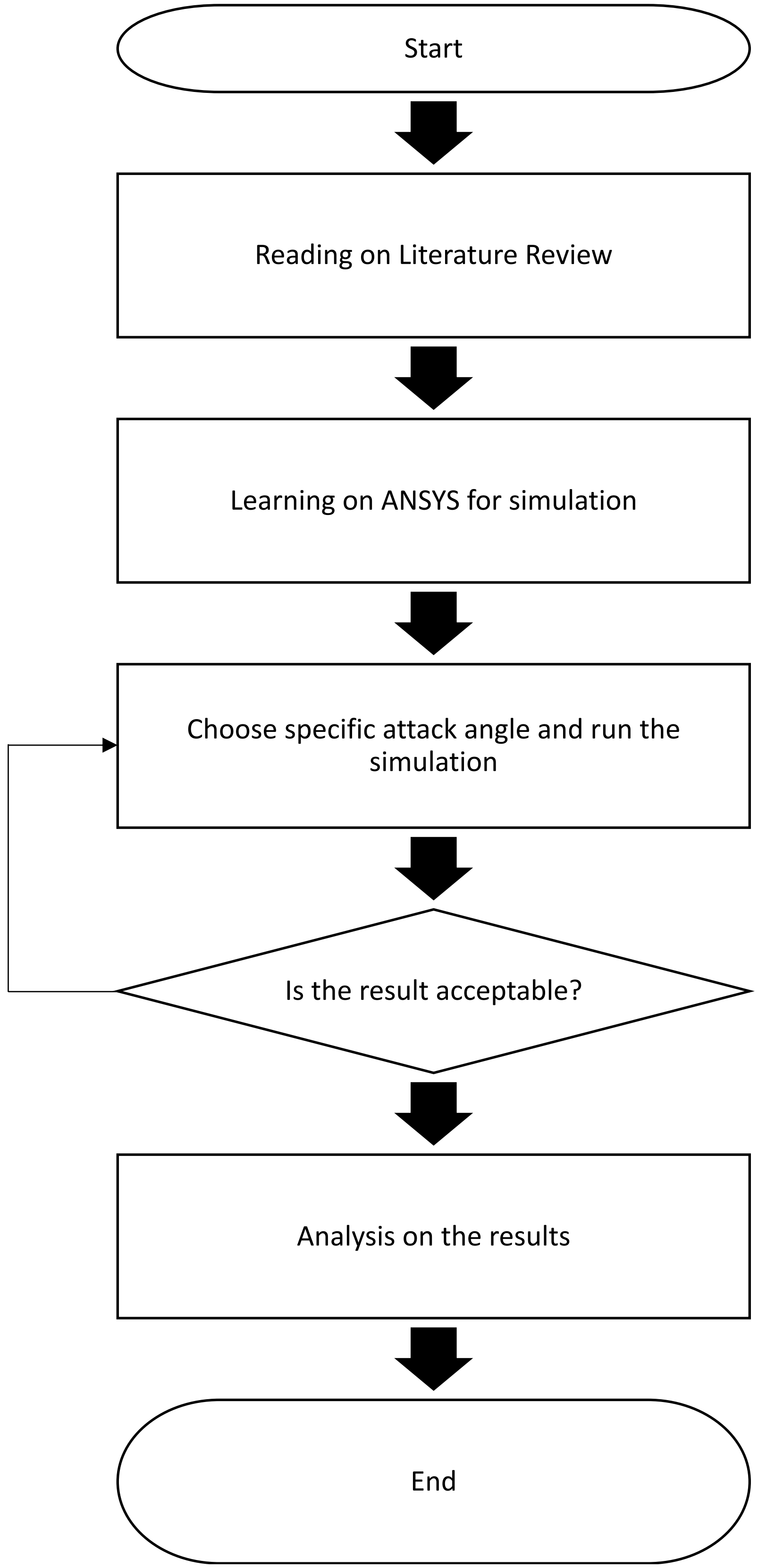
The wind tunnel must be capable to withstand temperature within the test section, to avoid heat loss along the wind tunnel. Data acquisition tools needed is to collect data on how temperature changes over time and distance between two points. For this data collection, thermocouples will be used along the wind tunnel.

The designed fin will be placed with specific angle according to the simulation with the highest convective heat transfer. Temperature will be recorded at inlet and outlet of the wind tunnel.

3.3.4 Validating the Project

Validation will be made on both results acquired from simulation and experiment. It is a basic tool for quantitatively assessing the accuracy of the computer code and the mathematical model. This process is to prove the validity of this project.

3.4 Project Flow Chart



3.5 Gantt Chart and Key Milestone

Task	Week													
	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Selection of project title														
Identification of problem														
Extensive literature review														
Proposal for the project														
Selecting methodology														
Familiarization with ANSYS														

Task	Week													
	1	2	3	4	5	6	7	8	9	10	11	12	13	14
Setting up the software based on data														
Set the boundary conditions of the simulation														
Run the simulation														
Results analysis of the simulation														
Report and documentation of the project														
Run the experiment in wind tunnel														
Validate simulation and experiment result														

Legends:

Project Progress

Key Milestone

Could not execute due to pandemic outbreak (COVID-19)

3.6 ANSYS Workbench

For this project study, ANSYS 2019 R2 was used to run the simulation processes. It has new user interface that speed up user adoption and reduces learning time to transform engineers' productivity. Few improved interface such as ease of use feature, newer capabilities around customization and instant search and find tool are provided in this version. This will improve the productivity of user when running the simulation through this software.

The analysis system used for this project is Fluid Flow (FLUENT). It contains developing of geometry, creation of mesh, setting up boundary conditions, solution and results.

3.6.1 Development of Model Geometry in ANSYS

The model geometry of fin design was produced with the assistance of ANSYS SpaceClaim. It is an intuitive 3-D modelling software solution that enables user to create, edit and repair geometry in the workflow. Several conditions were given while developing the model geometry of fin design in this project:

a) Design with the application feasibility

b) Base of the model geometry is fix, with initial purpose of running it in the wind tunnel

Properties	
Length Unit	Millimetre, mm
Solid/Fluid	Solid (Defined by geometry)
Base	34mm x 24mm x 5mm
Fin Dimension	7mm x 8mm x 34mm
Number of Fin	12 fins

Table 3.1: Geometry dimension

Figure 3.1 and 3.2 show the model geometry of the fin design. The material used for this fin design is aluminium.

Figure 3.1: Front view of fin design

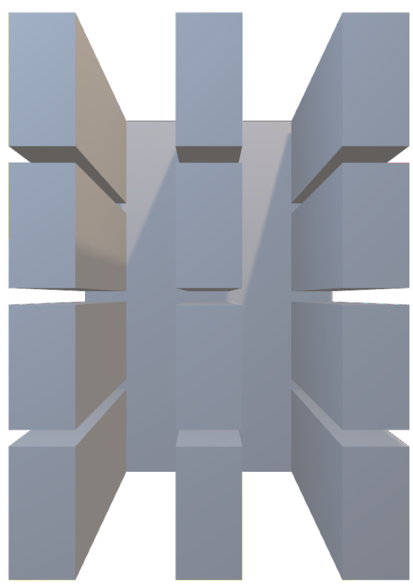


Figure 3.2: Top view of fin design

The relative attack angle of fin used in this project are 15˚, 30˚, 45˚ and 60˚ from the heat source. The designing of domain is necessary because it acts like adiabatic wall so that no heat loss will occur throughout the system starting from inlet to outlet.

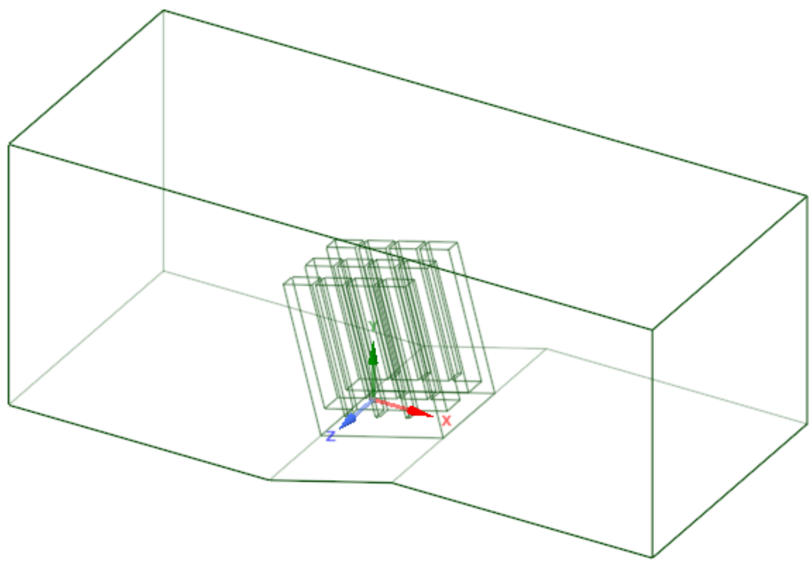


Figure 3.3: Isometric view of fin design with domain

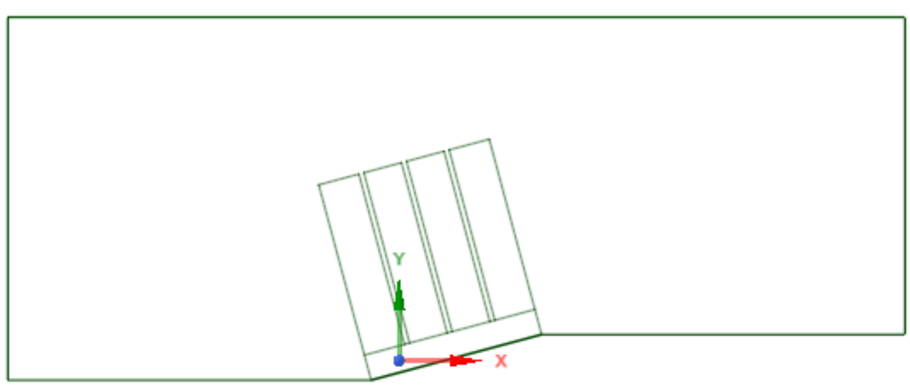


Figure 3.4: Rear view of fin design with domain

3.6.2 Design and Creation of Mesh in ANSYS

Computational Fluid Dynamics requires discretization or meshing of a model that continuous in time and space originally. The meshing of a model impacts the accuracy of the result at the end of the simulation process. It also influences the convergence and speed of the solution when running the calculation. In computer aided engineering simulation process, any computational cells such as prisms, tetrahedrons, or hexahedral are few examples of meshing integral part. Higher number of mesh's cells will consume longer time for solver to compute.

Thus, mesh independency is important step to reduce the time taken throughout the whole project. Few mesh were made to find the perfect mesh for the designed fin.

Sizing	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
Element Size	0.3mm
Growth Rate	1.2
Max Size	10mm
Size Function	Curvature
Relevance Center	Coarse
Smoothing	Medium
Transition	Slow

Table 3.2: Mesh properties

Part Name	Inlet	Outlet	Heat Source	Solid Interface	Fluid Interface
Geometry (Faces)	1	1	1	52	52

Table 3.3: Named selection list

3.6.3 Setting Up Boundary Conditions

Initial and boundary conditions are needed to solve any computational fluid dynamics problems. Boundary conditions are essential component of a mathematical model. In ANSYS Fluid Flow (FLUENT), it is equipped with standard boundary conditions such as inlet, outlet, heat source, wall and medium of the system. For this project, several boundary conditions were set up in order to calculate the results.

<table>
<tr><th>Domain</th><th colspan="2">Boundaries</th></tr>
<tr><td rowspan="6">Main Body (Solid)</td><td colspan="2">Boundary: Air Inlet</td></tr>
<tr><td>Type</td><td>Velocity-Inlet</td></tr>
<tr><td colspan="2">Boundary: Air Outlet</td></tr>
<tr><td>Type</td><td>Outlet-Vent</td></tr>
<tr><td colspan="2">Boundary: Heat Source</td></tr>
<tr><td>Type</td><td>Wall</td></tr>
</table>

Table 3.4: Type of boundaries

<table>
<tr><th colspan="2">Part: Air Inlet</th></tr>
<tr><td>Parameter</td><td>Value</td></tr>
<tr><td>Velocity Specification Method</td><td>Magnitude, Normal to Boundary</td></tr>
<tr><td>Reference Frame</td><td>Absolute</td></tr>
<tr><td>Velocity Magnitude (m/s)</td><td>2/4/6 (Each attack angle run with different velocities)</td></tr>
<tr><td>Thermal Temperature</td><td>24˚ C</td></tr>
<tr><td colspan="2">Part: Air Outlet</td></tr>
<tr><td>Parameter</td><td>Value</td></tr>
<tr><td>Velocity Specification Method</td><td>Magnitude, Normal to Boundary</td></tr>
<tr><td>Reference Frame</td><td>Absolute</td></tr>
<tr><td>Initial Gauge Pressure (Pa)</td><td>0</td></tr>
<tr><td>Thermal Temperature</td><td>24˚ C</td></tr>
</table>

Part: Heat Source	
Parameter	**Value**
Thermal Conditions	Heat Flux
Heat Flux (W/m^2)	8578
Heat Generation Rate	0
Material Name	Aluminium

Table 3.5: Boundary condition for all parts

3.6.4 Computation of Solution

In this stage, the computation of the solution is accomplished in ANSYS FLUENT. The objective is to generate a technique of simulating temperature and pressure distribution in the system precisely. The accuracy and speed of the solver depends on geometry configuration and meshing quality. Then, the results can be obtained from the computation to analyse further.

For Pressure-Velocity Coupling solution method, SIMPLE algorithm was used as it is a steady-state calculations. This scheme is referred to as pressure-based segregated algorithm. SIMPLE algorithm is the default for Pressure-Velocity Coupling solution. It is suitable for one or more skewness correction schemes up to value of 0.7 and for simple flow involving turbulence or laminar physical model only.

Spatial Discretization	
Gradient	Least Squares Cell Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Turbulent Dissipation Rate	First Order Upwind

Table 3.6: Pressure-Velocity Coupling Spatial Discretization

Standard initialization was used and the computation was from inlet. The number of iterations for calculation is 50, with reporting interval and profile update interval of 1.

CHAPTER 4

RESULTS AND DISCUSSION

In this chapter, results obtained from the simulation software will be analyse further. Few important parts in this project were investigated including temperature at the base of the fin at different attack angle and pressure drop.

The results and outcomes obtained based on simulation calculation are exported in visualization and data in order to acquire the anticipated information. The ANSYS results were exploited in form of contour plots and probe tool for temperature and pressure value at certain points.

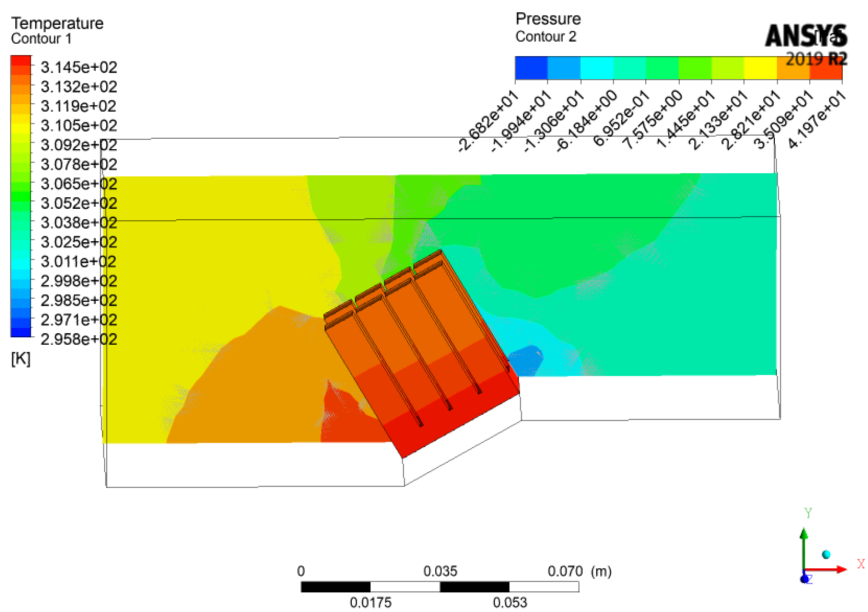


Figure 4.1: Example of result with contour

In figure 4.1, this is one example of contour distribution of temperature and pressure from ANSYS. The temperature distribution is portrayed at the solid fin, while the pressure distribution is portrayed at the plane along the domain starting from inlet to outlet.

4.1 Results on Temperature Distribution

Angle (degree)	Velocity (m/s)	Temperature (Celcius)		
		Inlet	Base	Outlet
15	2	24.00	50.13	26.93
	4	24.00	40.04	25.34
	6	24.00	38.66	24.79
30	2	24.00	59.55	26.38
	4	24.00	47.51	25.33
	6	24.00	42.20	25.01
45	2	24.00	58.21	25.09
	4	24.00	45.94	25.45
	6	24.00	42.07	24.96
60	2	24.00	59.65	26.07
	4	24.00	47.50	25.40
	6	24.00	42.31	24.73

Table 4.1: Temperature distribution

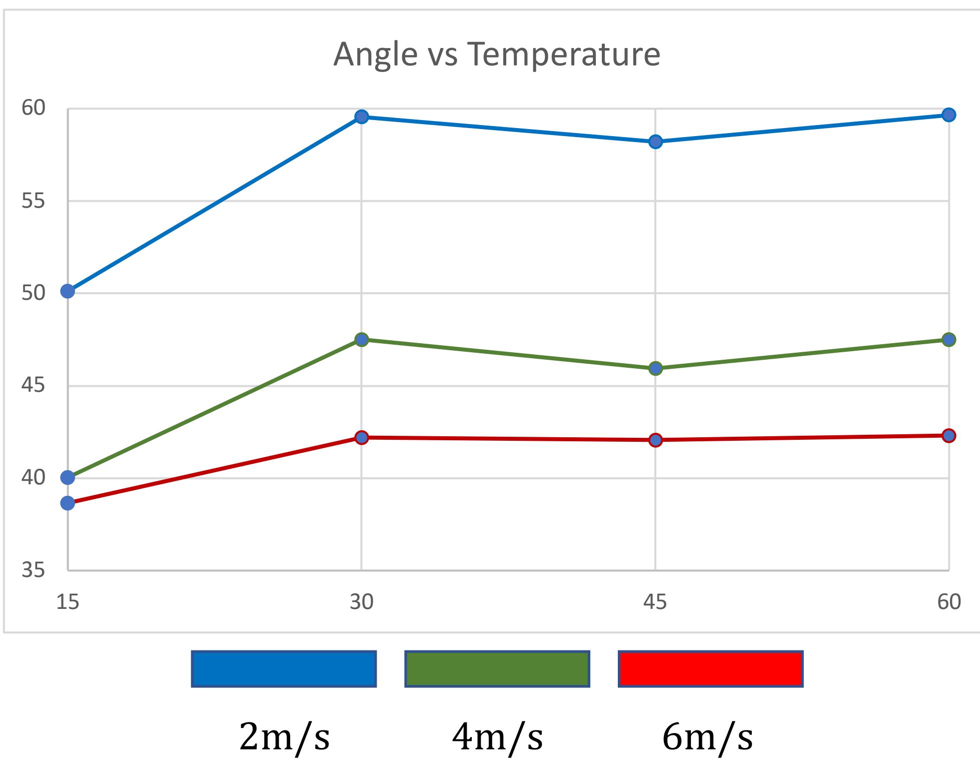


Figure 4.2: Angle versus temperature graph

4.2 Results on Pressure Distribution

Angle (degree)	Velocity (m/s)	Pressure (Pascal)		
		Inlet	Outlet	ΔP
15	2	1.75	0.00	1.75
	4	7.04	-0.01	7.05
	6	15.44	-0.01	15.45
30	2	3.08	0.00	3.08
	4	11.92	-0.02	11.94
	6	26.69	-0.02	26.71
45	2	5.16	-0.01	5.17
	4	20.21	-0.03	20.24
	6	43.74	-0.06	43.80
60	2	9.32	-0.01	9.33
	4	36.83	-0.06	36.89
	6	82.56	-0.20	82.76

Table 4.2: Pressure distribution

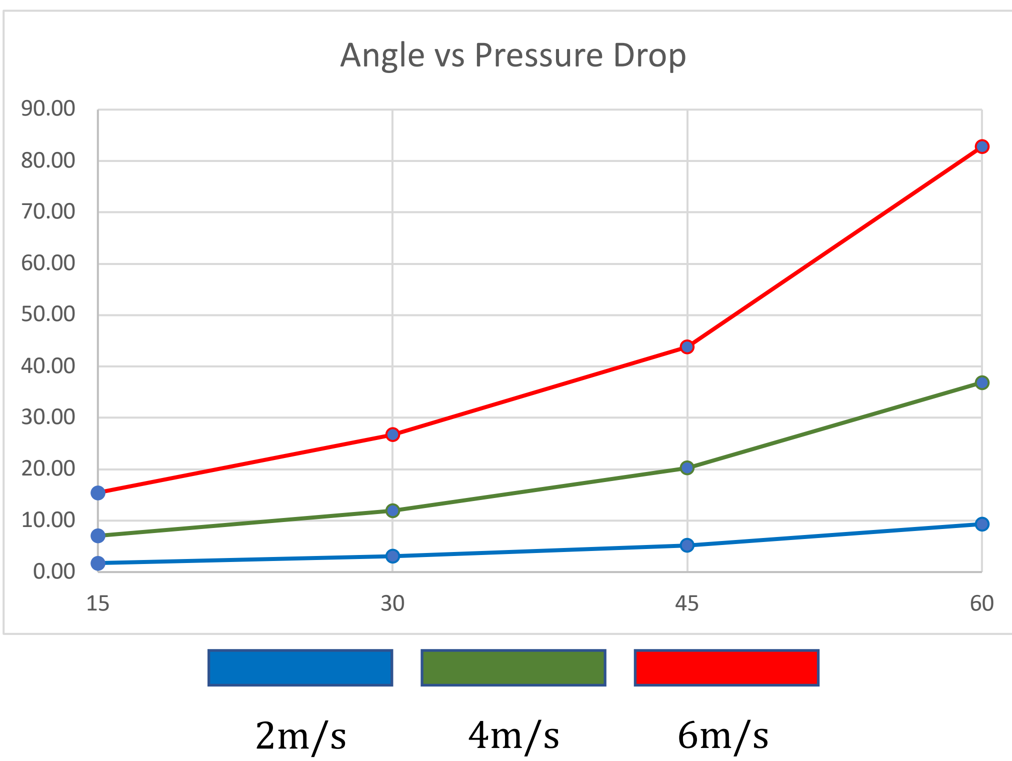


Figure 4.3: Angle versus pressure drop graph

4.3 Discussion

From the results obtained, we can conclude that changing the relative attack angle of fin can affect the convective heat transfer. In Table 4.1, the highest convective heat transfer efficiency has been achieved by fin with angle of 30˚ and 60˚. The lowest to achieve high convective heat transfer efficiency is fin with angle of 15˚.

The velocity magnitude affects on base and outlet temperature, as the higher the velocity magnitude, the lower the final temperature. This is due to forced convective heat transfer from inlet to outlet, passing the fin.

The fin with highest pressure drop in this project is fin with angle of 60˚ at 6m/s. The pressure drop is influenced greatly by the velocity magnitude. As we can see in Figure 4.3, all results with velocity magnitude of 6m/s have high pressure drop.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

From the results obtained, we can conclude that changing the relative attack angle of fin can affect the convective heat transfer. In Table 4.1, the highest convective heat transfer efficiency has been achieved by fin angle with angle of 30˚ and 60˚. The lowest to achieve high convective heat transfer efficiency is fin angle with 15˚.

Changing the relative attack angle of fin can affect the convective heat transfer. A deep study needs to carry on in order to understand how suitable angle with specific design affect the convective heat transfer. This study would be helpful for other engineers to design fins for a system for world industry for the future.

The usage of simulation software is very helpful for engineers and simulator around the world. We can save time and cost along the project, plus it is a bonus to solve a problem in the most effective way. Simulation software for engineers not only good for solving problems, but can improve finding the best solutions.

REFERENCES

Ibrahim, T. K., Mohammed, M. N., Mohammed, M. K., Najafi, G., Che Sidik, N., Basrawi, F., . . . Hoseini, S. S. (2018). Experimental study on the effect of perforations shapes on vertical heated fins performance under forced convection heat transfer. *International Journal of Heat and Mass Transfer, 118*, 832-846.

Jasim, H. H., & Soylemez, M. S. (2018). The Effects of The Perforation Shapes, Sizes, Numbers and Inclination Angles on The Thermal Performance of A Perforated Pin Fin. *Turkish Journal of Science & Technology, 13*(2), 1-13.

Kadbhane, S. V., & Palande, D. D. (2016). Review of Convective Heat Transfer from Plate Fins Under Natural and Mixed Convection at Different Inclination Angle. *International Research Journal of Engineering and Technology (IRJET), 3*(2), 467-473.

Narato, P., Wae-hayee, M., Vessakosol, P., & Nuntadusit, C. (2017). Effect of inclined angle of pin arrays on flow and heat transfer characteristics in flow channel. *Materials Science and Engineering* (pp. 243-254). Songkhla, Thailand: IOP Publishing.

Rasel, M. A., Islam, M. Z., & Hasanat, A. (2016). Analysis of heat transfer characteristics under forced convection in a rectangular body with circular fins. *American Journal of Engineering Research (AJER), 5*(10), 311-316.

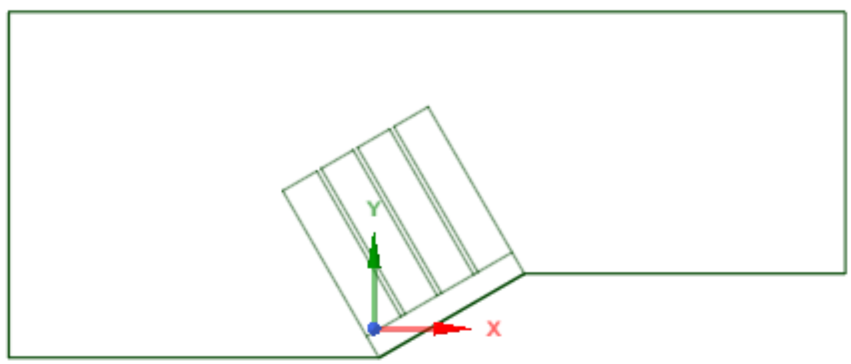
Shaeri, M. R., & Yaghoubi, M. (2009). Numerical analysis of turbulent convection heat transfer from an array of perforated fins. *International Journal of Heat and Fluid Flow, 30*, 218-228.

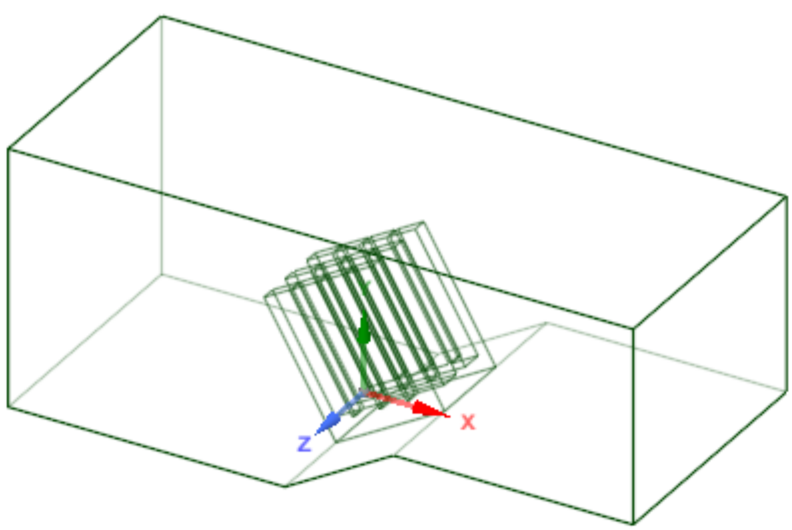
Sonawane, R., & Palande, D. D. (2016). Heat Transfer Enhancement by Using Perforation. *International Research Journal of Engineering and Technology (IRJET), 3*(4), 2624-2629.

Yuan, Z. X., Zhao, L. H., & Zhang, B. D. (2007). Fin angle effect on turbulent heat transfer in parallel-plate channel with flow-inclining fins. *International Journal of Numerical Methods for Heat & Fluid Flow, 17*(1), 5-19.

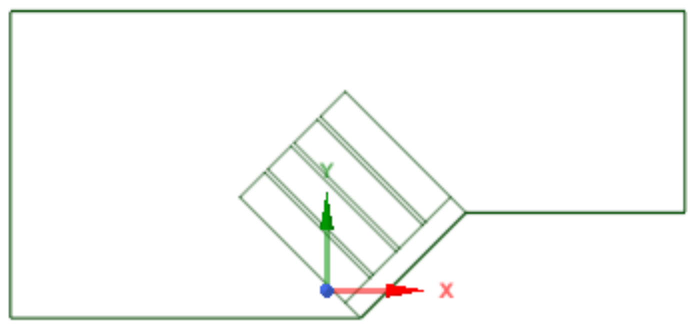
APPENDICES

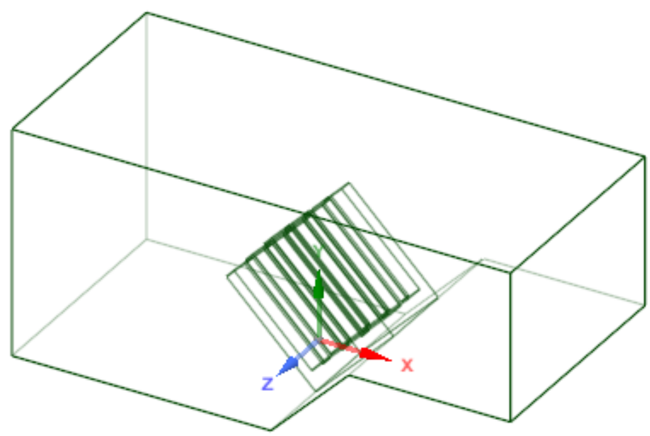
APPENDIX I: GEOMETRY OF FIN DESIGN AT 30° FROM HEAT SOURCE



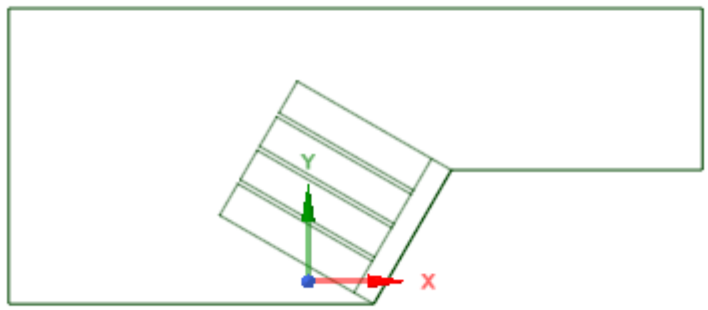


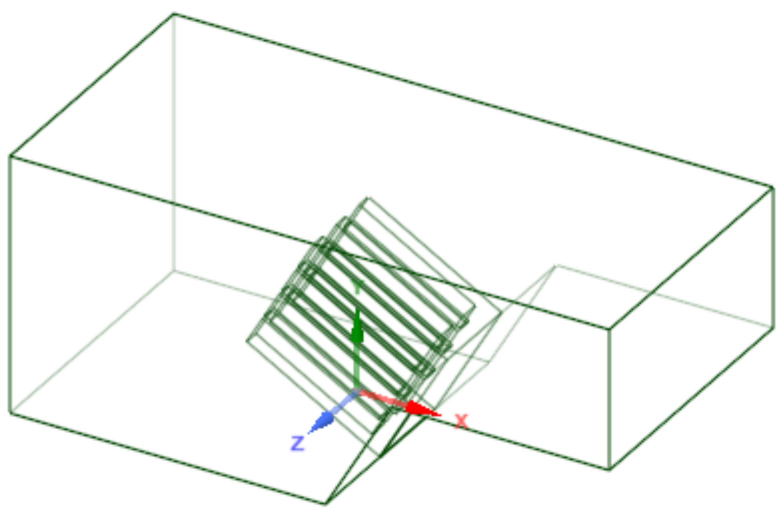
APPENDIX II: GEOMETRY OF FIN DESIGN AT 45˚ FROM HEAT SOURCE



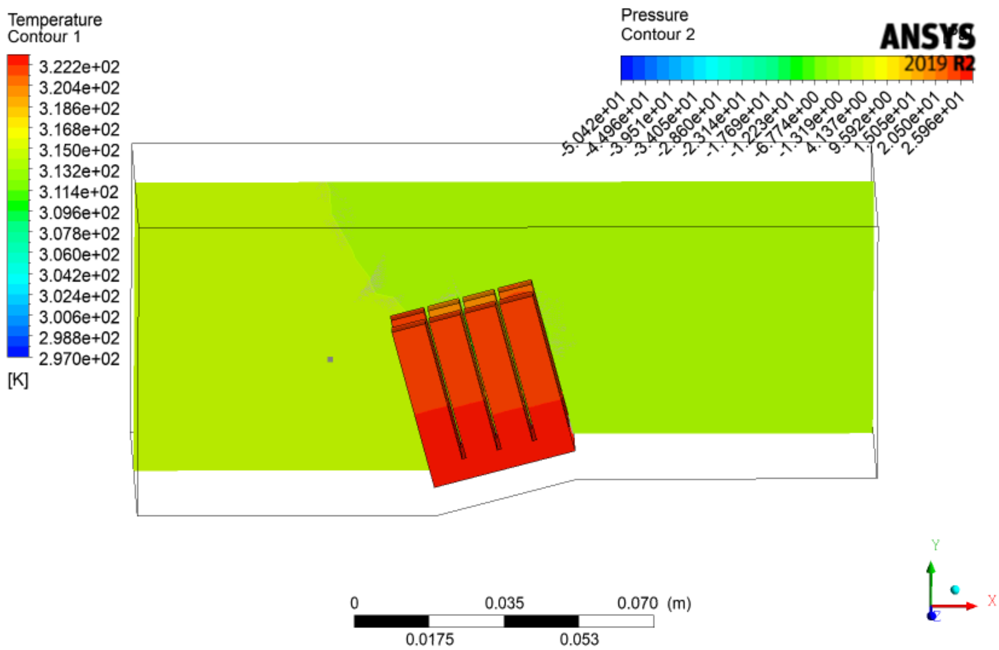


APPENDIX III: GEOMETRY OF FIN DESIGN AT 60˚ FROM HEAT SOURCE

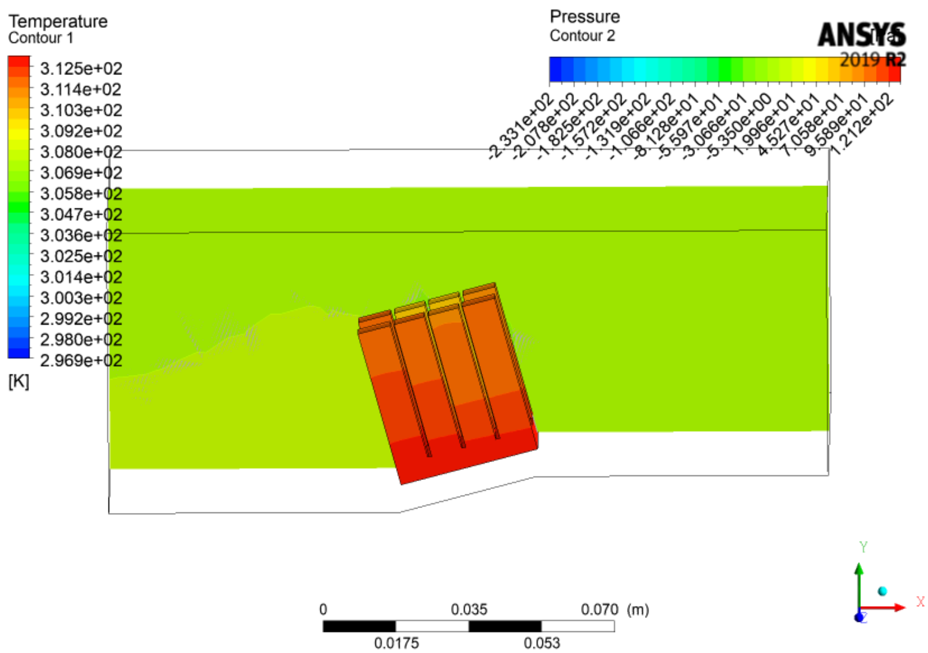




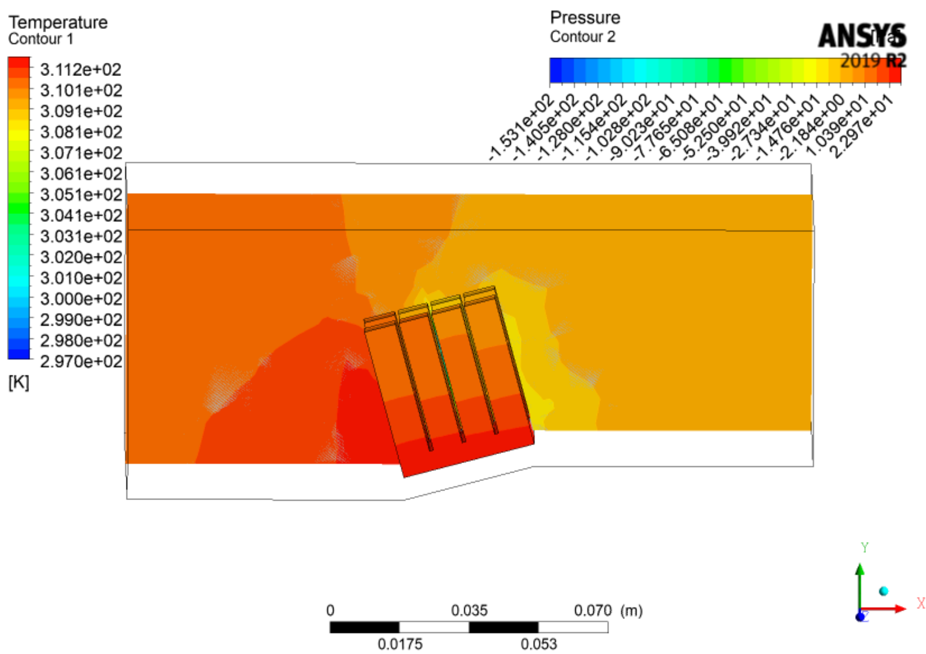
APPENDIX IV: CONTOUR RESULTS OF FIN DESIGN AT 15°



a) At 2m/s

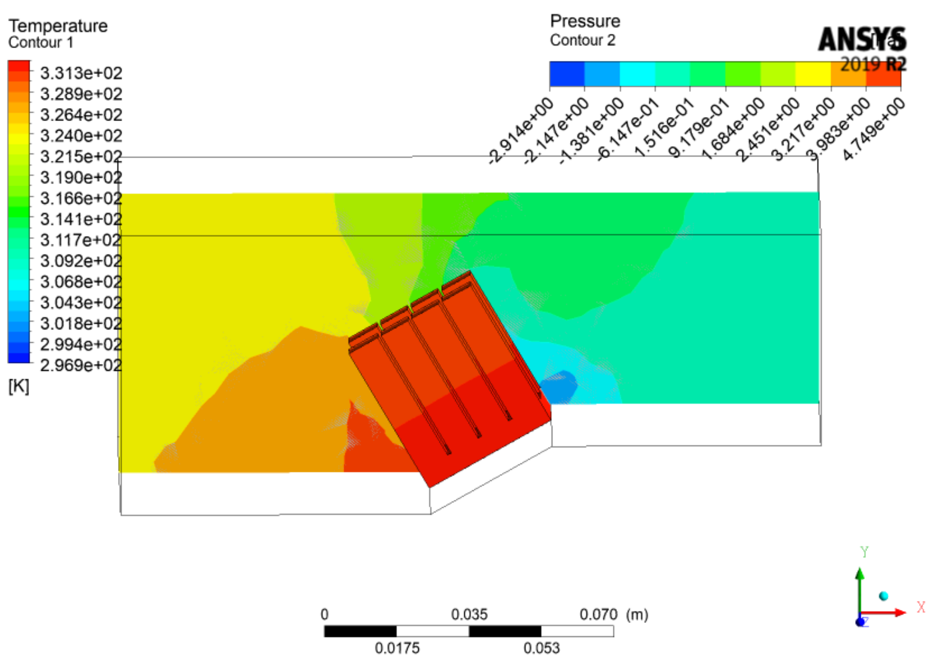


b) At 4m/s

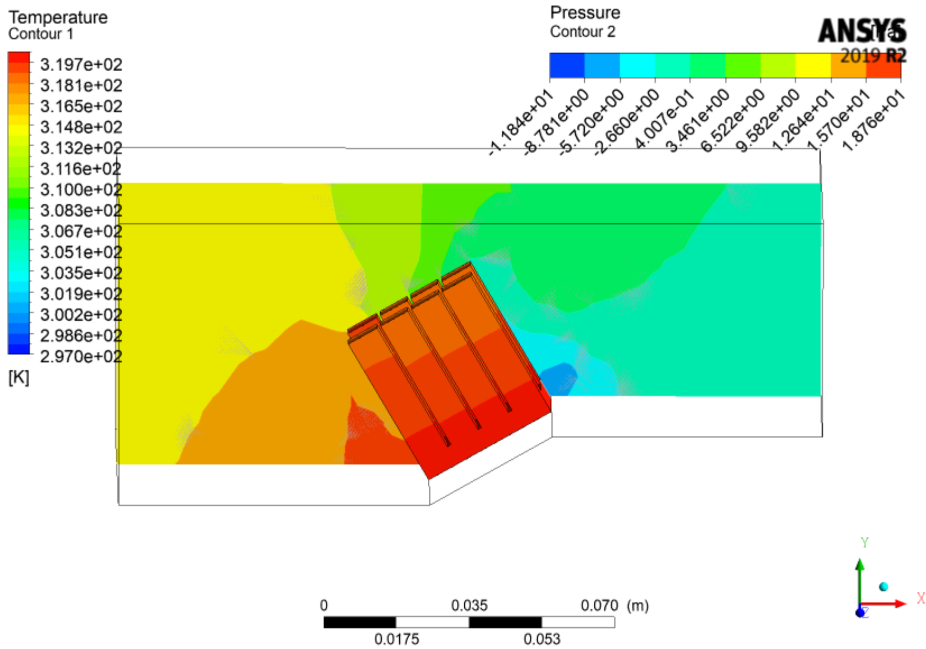


c) At 6m/s

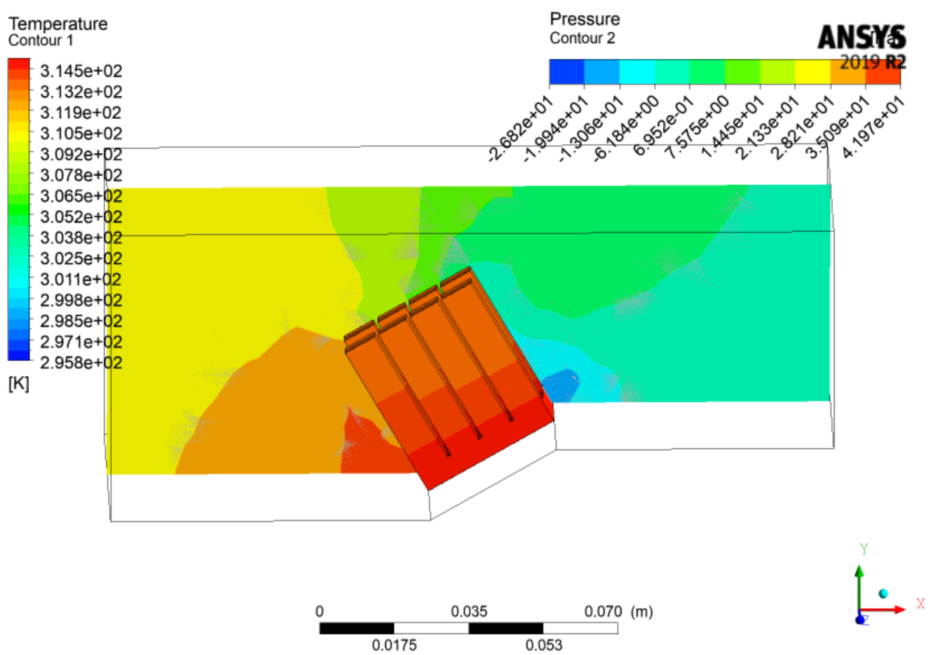
APPENDIX V: CONTOUR RESULTS OF FIN DESIGN AT 30°



a) At 2m/s

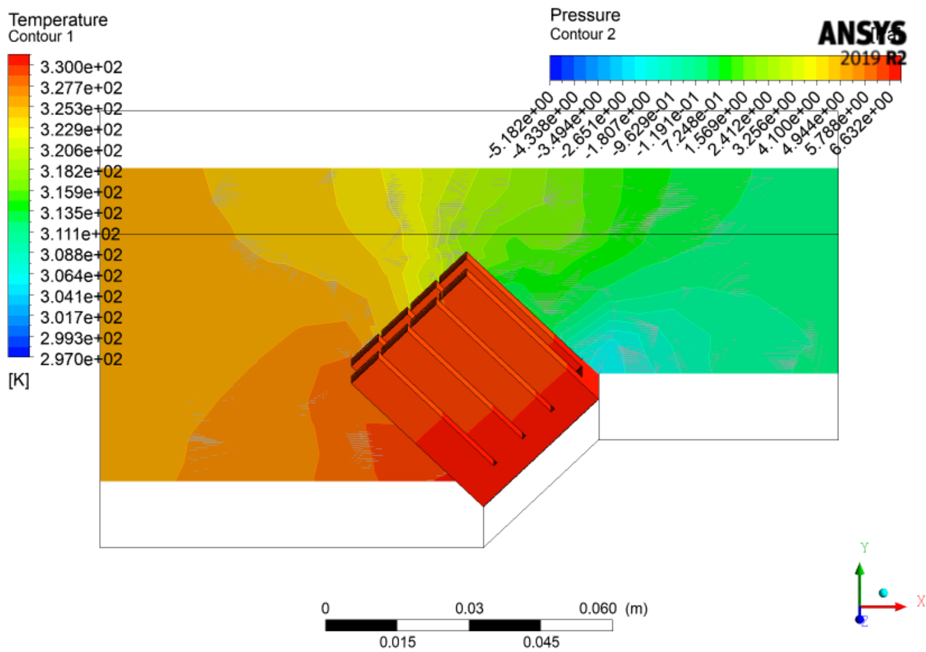


b) At 4m/s

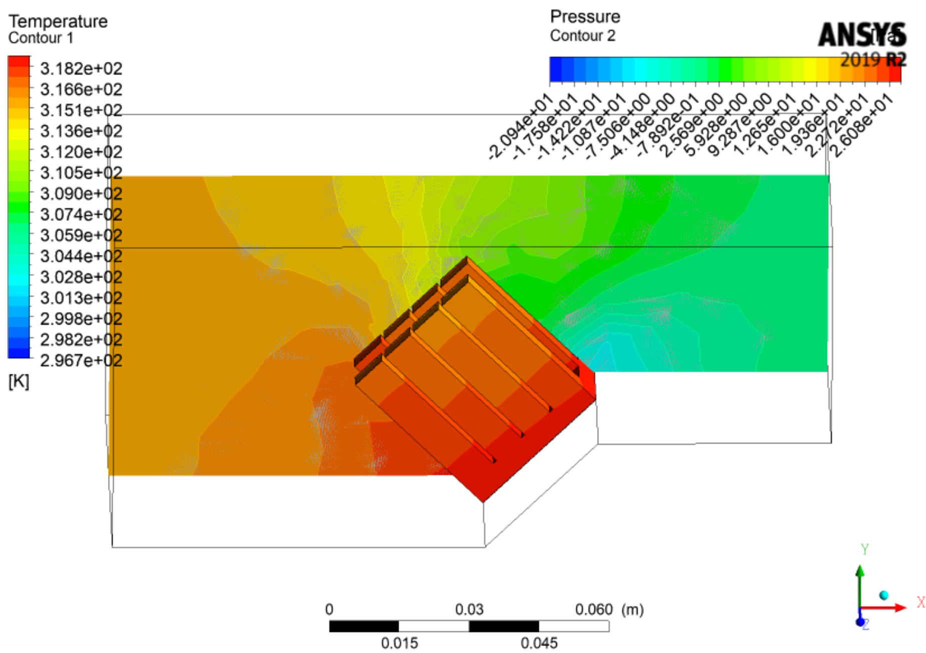


c) At 6m/s

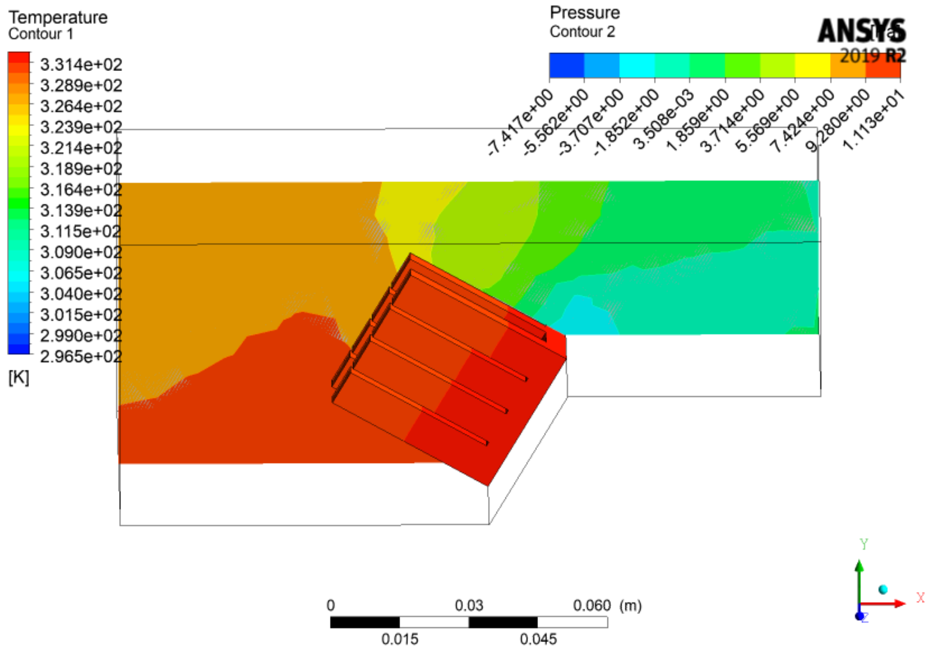
APPENDIX VI: CONTOUR RESULTS OF FIN DESIGN AT 45°



a) At 2m/s

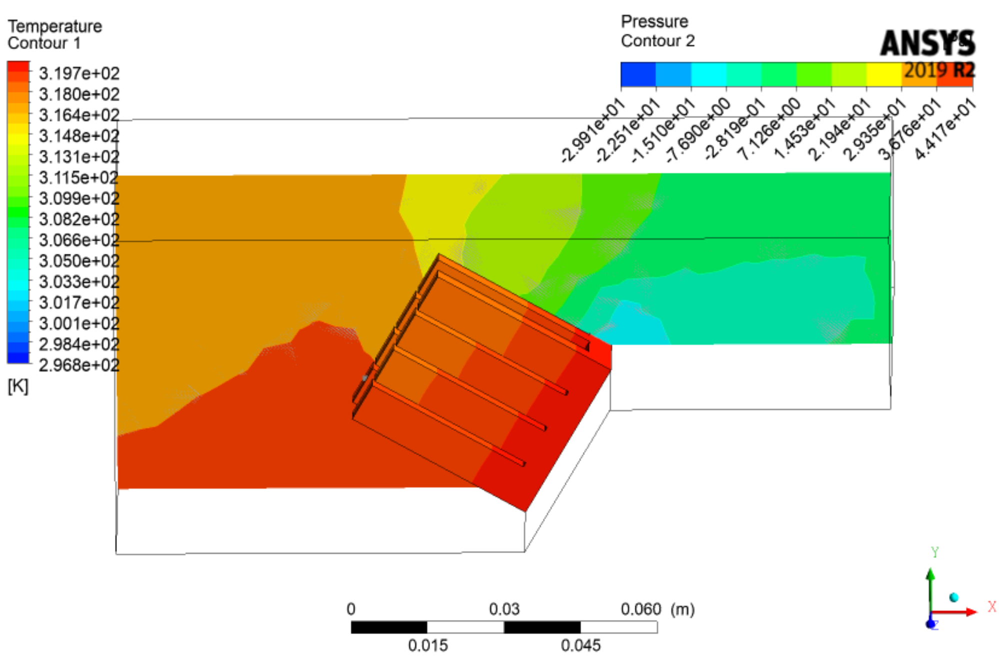


b) At 4m/s

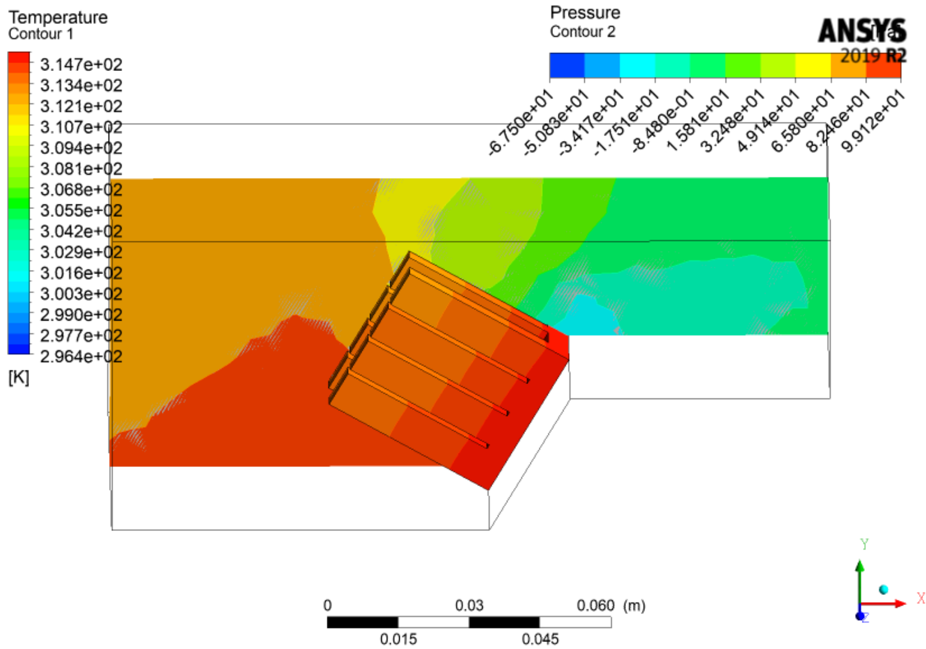


c) At 6m/s

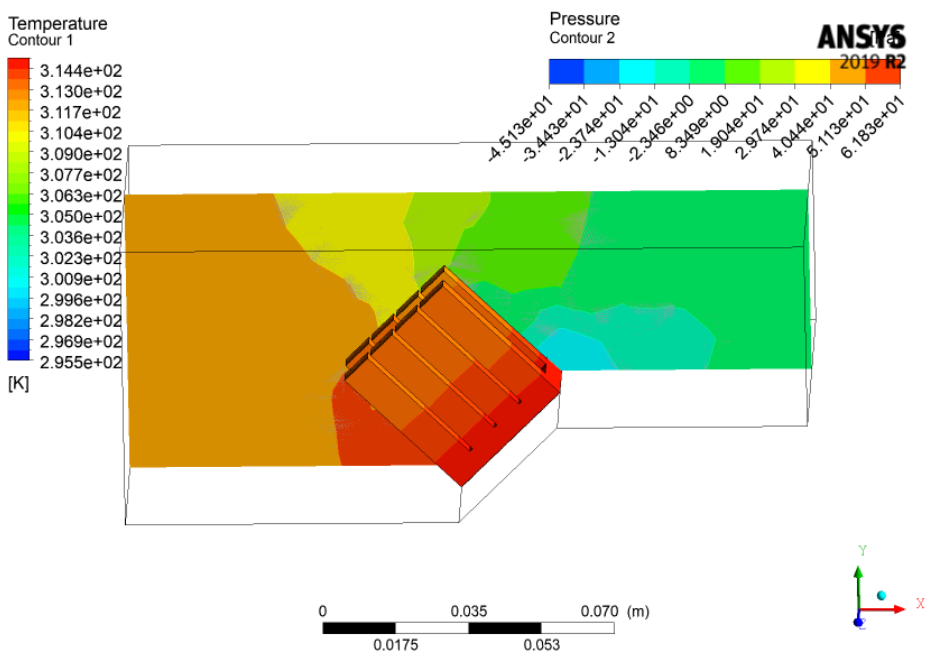
APPENDIX VII: CONTOUR RESULTS OF FIN DESIGN AT 60˚



c) At 2m/s



b) At 4m/s



c) At 6m/s

APPENDIX VIII: OVERALL RESULTS ON TEMPERATURE AND PRESSURE

Angle (degree)	Velocity (m/s)	Temperature (Celcius)			Pressure (Pascal)		
		Inlet	Base	Outlet	Inlet	Outlet	ΔP
15	2	24.00	50.13	26.93	1.75	0.00	1.75
	4	24.00	40.04	25.34	7.04	-0.01	7.05
	6	24.00	38.66	24.79	15.44	-0.01	15.45
30	2	24.00	59.55	26.38	3.08	0.00	3.08
	4	24.00	47.51	25.33	11.92	-0.02	11.94
	6	24.00	42.20	25.01	26.69	-0.02	26.71
45	2	24.00	58.21	25.09	5.16	-0.01	5.17
	4	24.00	45.94	25.45	20.21	-0.03	20.24
	6	24.00	42.07	24.96	43.74	-0.06	43.80
60	2	24.00	59.65	26.07	9.32	-0.01	9.33
	4	24.00	47.50	25.40	36.83	-0.06	36.89
	6	24.00	42.31	24.73	82.56	-0.20	82.76