

Active Liquid Based Thermal Management of PV Modules

Khalifa Alshaiba¹, Ahmad Alkaabi²

Abstract: The world today is heading towards renewable energy and replacing fossil fuel energy and nonrenewable energy, and part of renewable energy is the photovoltaic cells because it's using the sunlight as a primary energy source, and then it will convert the sunlight into electricity using a technology based on the photoelectric effect, and it's a type of inexhaustible and non-polluting energy that can be produced in installations ranging from small generators for self-consumption to large photovoltaic plants. Furthermore, the problem of photovoltaic cells is it suffers from overheating and that causes in decline in the efficiency. In our project we are trying to solve this problem by designing a heat sink that will cool down the photovoltaic cells and make it work more efficiently, so to start with our project, first did a lot of research for articles about this topic, to gather more information and know what have people done in the past to solve this problem, then we were able to come up with our requirements, after that we did a survey to gather more information from the customers and decided our final specifications, then we used MATLAB software to perform the calculation of the heat sink, after we are done with our calculation we were able to design the CAD model of our heat sink, then we tried to estimate our cost for this project to know the budget needed for this project. Furthermore, after finishing designing the CAD model of our heat sink, we send our CAD design for fabrication, and after that we started working on ANSYS Fluent software to get more detailed and accurate calculations, and we chose the ANSYS approach because our governing equations are too complex for an analytical solution, and the purpose of using ANSYS is to see the change in the thermal resistance with the Reynolds number, and to see the change in the pumping power with the Reynolds number, and we can achieve this by simulating our channel in ANSYS. Moreover, after we got our results from ANSYS, we have received our heat sink after fabrication, so we went to the lab to start testing our heat sink and then perform the heatsink experiment, and through the testing we have faced some problems such as water leakage, but we were able to solve this problem, and after that we started working on the experiment, and the goal of this experiment was to see the change in the temperature difference with the Reynolds number.

Keywords: Thermal Management, PV Module

List of symbols, abbreviations, nomenclature

Symbol	Definition
K_w	Thermal conductivity of water
C_p	Heat capacity of water
K_{al}	Thermal conductivity of aluminum
W_{sp}	Channel spacing
W_{ch}	Channel width
L_{ch}	Channel length
T_{fi}	Coolant inlet temperature
D_{ch}	Channel depth
TOW	Thickness of water
W_{hs}	Width of the heat sink
L_{hs}	Length of the heat sink
Q	Total heat output
V_t	Flow rate
N	Total Number of Channels
A_{cr}	Cross-section area of a single channel
P_{ch}	Perimeter of a single channel
D_{hg}	Hydraulic Diameter of a single channel
A_{SR}	Surface Area of a single channel
α	Aspect Ratio
h	Heat Transfer coefficient
R_{conv}	Convective Thermal Reactance
V_{ch}	Flow Rate Per Channel
m_{ch}	Mean Flow Rate Per Channel
Q_{ch}	Heat output per channel
$T_{f,out}$	Outlet temperature of fluid
T_{surr}	Channel Temperature at the outlet section
R_{cond}	Conductive Thermal Resistance
T_{cm}	Maximum temperature of the heat sink
U	Velocity of Flow
Re	Reynold Number
f	Friction constant
ΔP_{ch}	Pressure Drop

T_{cmc}	Maximum temperature of the heat sink
u	Velocity of the fluid along the x - direction
v	Velocity of the fluid along the y - direction
w	Velocity of the fluid along the z - direction
P	Pressure of the fluid
μ	Viscosity of the fluid
ρ	Density of the fluid
V_{avg}	Average velocity
PP	Pumping power
R_{th}	Thermal resistant

1. Introduction

1.1 Problem statement and purpose

Heat exchangers are devices used for transferring heat between two or more fluids or between one fluid and a surface and is widely used in both cooling and heating processes in several industries. Furthermore, photovoltaic cells are charged for conversion of light into electricity using semiconducting materials and used in powering homes, commercial buildings and even large power stations and much more. Sunlight is the primary input to generating electricity using PV, but they suffer a decline in performance as the operating temperature increases. Therefore, improved efficiency must be achieved through appropriate cooling systems. The efficiency of solar PV systems should also be enhanced while ensuring a longer life. Our project is a heat transfer project to construct a heat sink for cooling PV cells for improving its efficiency and leading to an increase in the life cycle of these cells. It involves modeling (MATLAB and ANSYS Fluent) and experimental work. The project will involve creating a heat sink for the cooling of PV cells; we

will deal with just the heat sink and not the PV solar cells [1].

1.2 Project and Design Objectives

Our project is about designing and building a heat sink for cooling the PV cells. Follow the systematic design process by gathering requirements, product concept, solution concept, embodiment design and detailed design, and provide data and information of the heat sinks from articles and survey, and the objectives that will be achieved by the end of our project are listed below.

- 1) Provide data and information of the heat sinks from articles and survey
- 2) Simulate the working of the heat sink using MATLAB and ANSYS Fluent
- 3) Perform performance calculations of the heat sink
- 4) Design the final model using CAD software
- 5) Capital cost estimation on the project
- 6) Build and test the heat sink



Figure 1.1: Design process chart

Then we generated design alternatives and options with different materials used, cooling fluid, channel geometry and type of pump and followed every option with a conceptual design. After that we selected the best fit design by stating each option advantages and disadvantages related to our aim of this heat sink and constructed the decision matrix which will also aid our selection. The next step was the preliminary design in which the MATLAB code was prepared, and givens were stated. After we analyzed the MATLAB code results and got the specifications values which allowed us to design our CAD model of our heat sink. However, in GPII we will simulate the working of the heat sink using ANSYS Fluent and then we will be able to construct the actual heat sink and run experiments on it in the lab.

2. Summary of Achievements in GPI and Updated Background Theory

In order for us to achieve our goal which is cooling the photovoltaic cells, we need first to do a research about what has been done in the past to solve the problem of photovoltaic cells overheating, so in this section of the report we will talk about three different types of cooling that have been used in the past to cooldown photovoltaic cells which are hydraulic cooling, liquid immersion cooling, and air active cooling.

• Hydraulic Cooling

Zakharchenko in 2004 presented a PV/T design, placing PV cell on the surface of solar collector in which cold water flows. Aluminum (Al) substrate was chosen with an adhesive coating to provide best possible thermal contact between the PV and solar collectors. The PV panel was cooled by approximately 10 K, and its conversion efficiency increased by 10% [2].

1.3 Intended outcomes and deliverables

The expected outcomes and deliverables of our project are as follows:

- 1) To get more experience in using simulation software for designing the heat sink
- 2) Ability to develop and conduct an experiment, analyze and interpret data to draw conclusions
- 3) Gain experience in the heat sink market (prices, types, performance)
- 4) Apply knowledge gained from previous courses into our project
- 5) To design and fabricate the heat sink

1.4 Summary of design process used

To design the heat sink first we collected data and information from scholar articles, surveys and past projects related to our heat sink to be able to produce the best design. Relating them to the requirements and specification adjusted from GPI.

• Liquid immersion cooling

Russell in 1981 configured an elongated tube filled with the non - conductive liquids as optical concentrator as well as a cooling system for PV arrays. PV cells are mounted inside the tube and immersed in a clear non - conducting liquid and the elongated tube with the liquid has a refractive index suitable for concentrating the sunlight onto the cells [3].

• Air active cooling

Air cooling offers less expensive cooling of PV systems where heat can be removed by either natural convection or forced convection of air. Air is the preferred media for low - cost cooling despite its poor thermo - physical characteristics.

One of the air cooling methods is the ground coupled central panel cooling system. The projected scheme serves to cool the solar panels by forced convection of air driven by a blower, the blower being run by another dedicated PV panel. Air flows through a ground - coupled heat exchanger and decreases its temperature. The cooled air soothes the solar panels while passing beneath them. Nozzles are attached to the pipes in order to ensure that streamline flow in desired directions [4].

2.1 Design criteria and specifications

2.1.1 Survey

The best way to collect information about our heat sink is to conduct a survey to decided our design specifications, and the figures shown below are the question we asked in our survey and the result of the survey is shown as well. Figure 2.1 is showing question one in our survey, and the question were talking about the cost of the heat sink, and 52.4% of the people prefer it to be less than 5000AED.

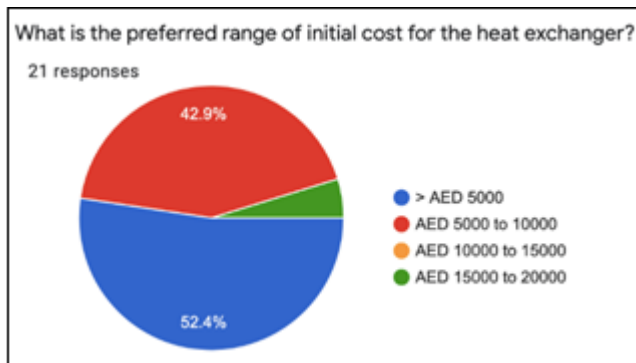


Figure 2.1: Question 1 in the survey

Figure 2.2 is showing question two in our survey, and the question were talking about the weight of the heat sink, and 52.4% of the people prefer it to be between 2.5 to 5 kg.

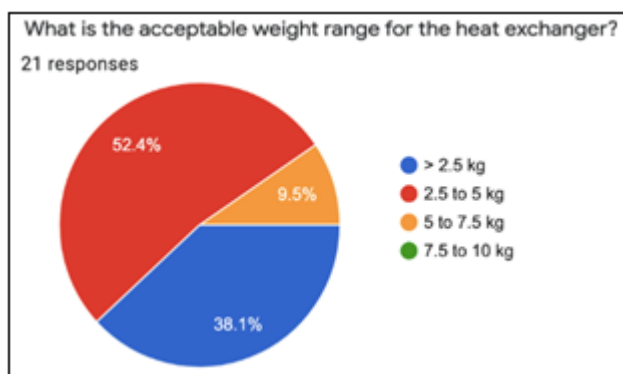


Figure 2.2: Question 2 in the survey

Figure 2.3 is showing question three in our survey, and the question were talking about the lifetime of the heat sink, and 38.1% of the people prefer the heat sink life to be between 10 to 15 years.

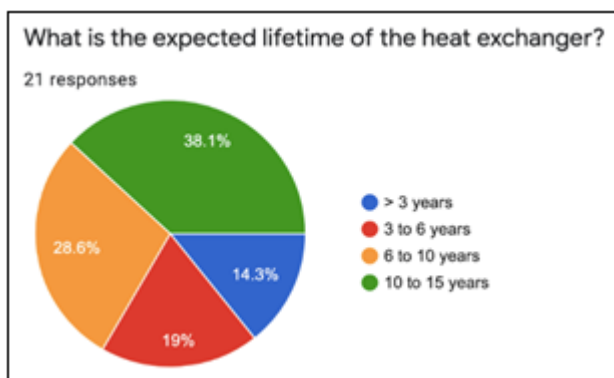


Figure 2.3: Question 3 in the survey

Figure 2.4 is showing question four in our survey, and the question were talking about the cooling fluid of the heat sink, and 38.1% of the people prefer the cooling fluid to be air and the 33.3% prefer it to be water.

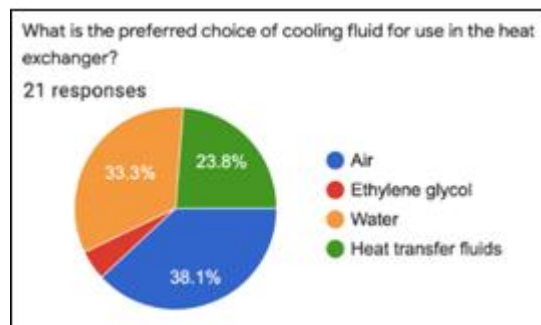


Figure 2.4: Question 4 in the survey

Figure 2.5 is showing question five in our survey, and the question were talking about the flow rate of the heat sink, and 65% of the people prefer the flow rate to be between 0.5 to 1 l/s.

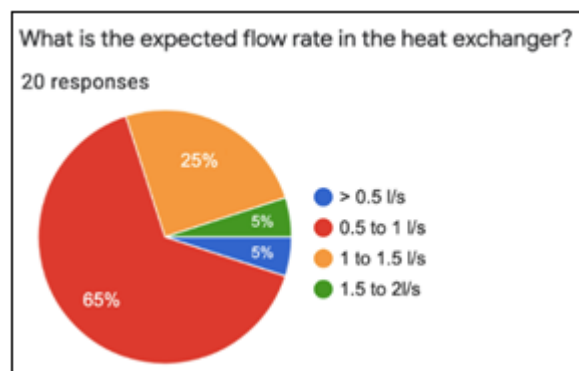


Figure 2.5: Question 5 in the survey

Figure 2.6 is showing question six in our survey, and the question were talking about the length of the heat sink, and 47.6% of the people prefer the length of the heat sink to be between 50 to 75 cm.

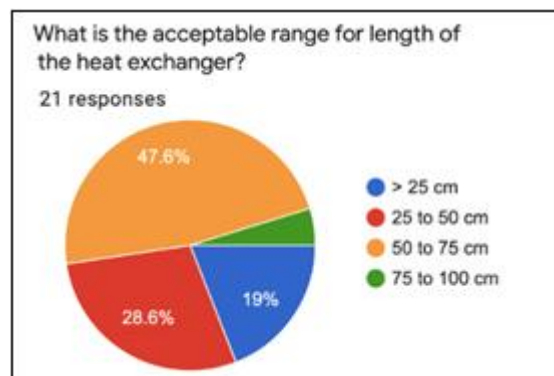


Figure 2.6: Question 6 in the survey

Figure 2.7 is showing question seven in our survey, and the question were talking about the width of the heat sink, and 47.6% of the people prefer the width of the heat sink to be between 25 to 50 cm.

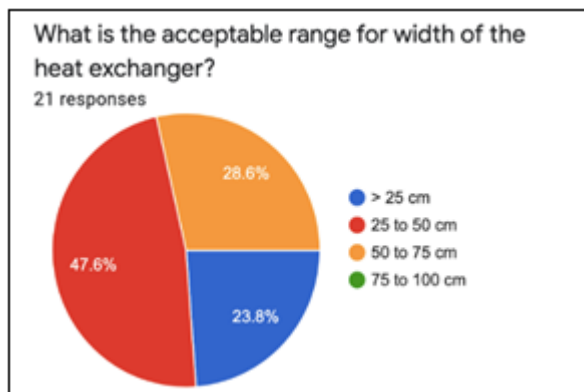


Figure 2.7: Question 7 in the survey

Figure 2.8 is showing question eight in our survey, and the question were talking about the material of the heat sink, and 42.9% of the people prefer the material of the heat sink to be between stainless steel, and 33.3% prefer it to be aluminum.

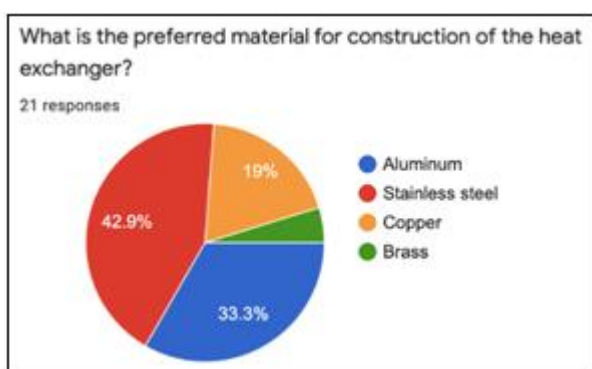


Figure 2.8: Question 8 in the survey

Figure 2.9 is showing question nine in our survey, and the question were talking about the power consumption of the heat sink, and 38.1% of the people prefer the power consumption of the heat sink to be between 1 to 2.5 kW, and another 38.1% prefer it to be between 2.6 to 5 kW.

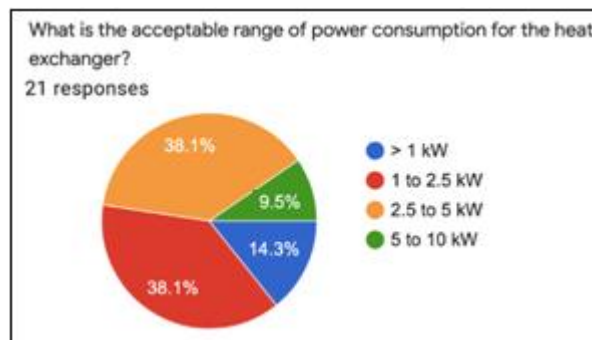


Figure 2.9: Question 9 in the survey

Figure 2.10 is showing question ten in our survey, and the question were talking about the pump type of the heat sink, and 61.9% of the people prefer the pump type of the heat sink to be centrifugal pump, and 28.6% prefer it to be gear pump.

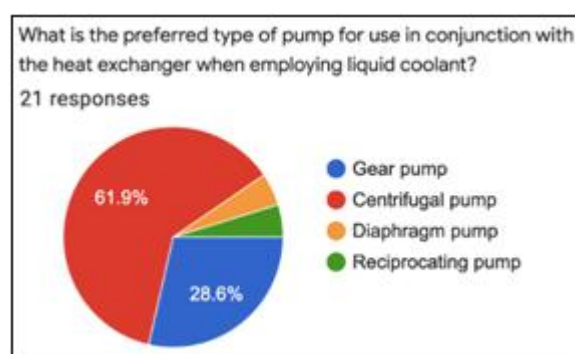


Figure 2.10: Question 10 in the survey

2.1.2 QFD house of quality

In this section of the report we build our house of quality as shown below in table 2.1, and the house of quality is one of the important steps in design, because it translate the customer requirements into measurable design targets, and the QFD above is showing each number and what it indicates, for example nine indicates the strongest, and six indicates the medium, and three indicates small, and one indicates barely related.

Table 2.1: QFD

Customer Requirement	Customer Weights	Metallic Case	Cooling Fluid	Pump	Channel Profile	Dimensions	Efficiency	Manufacturability Method
Durability	3	9		3				1
Performance	9	3	9	6	3		9	
Small Size	3	9				9		
Low Cost	9	6	9	6			3	6
Environment Friendly	6		6					
Reliable	6						6	
Maintainability	6		6	3	6			
Absolute Rating	-	126	234	165	64	27	144	57
Relative Rating	-	16.01%	29.73%	17.10%	8.13%	3.43%	18.29%	7.24%

Correlation	Value
Strong	9
Medium	6
Small	3
Barely Related	1

2.1.3 Design specifications

Our aim from this project is to produce an efficient, durable and low cost heat sink that will be able to cool the PV cells. Specifications were obtained by quantifying requirements through survey, experts and communicating with companies and they are listed below:

- Cost: > AED5000
- Lifetime: 10 to 15 years
- Weight: 2.5 to 5 kg
- Cooling fluids: water - air
- Flow rates: 0.5 to 1 l/s
- Length: 50 to 75 cm
- Width: 25 to 50 cm

- Construction material: Aluminum and Stainless steel
- Pump Type: Centrifugal pump and Gear pump Operating
- Power consumption: 1 to 10 kW

2.2 Overview of alternative assessment

In this section of the report, we will show the four design alternatives that we generated in GP1, we will show for each alternative design its material, cooling fluid, channel cross section, channel modifications, manifold shape, and the pump. Then, we will briefly talk about our reasoning for each design alternative. Table 2.2 below is showing each design's alternative characteristics.

Table 2.2: Design alternatives

Options	Alternative 1	Alternative 2	Alternative 3	Alternative 4
H. S Material	Stainless steel	Copper	Aluminum	Brass
Cooling fluid	Air	Water	Heat transfer fluid	Ethylene glycol
Channel cross - section	Semi circular	Rectangular	Square	Rectangular
Channel modifications	Zig - zag channel	Pin - fin in straight channels	Wavy channels	Straight channel
Manifold shape	Right triangular	Rectangular	Semicircular	Rectangular
Manifold arrangement	Z - type	U - type	I - type	I - type with distributor
Pump	Diaphragm pumps	Gear pump	Centrifugal pump	Reciprocating pump

Option 1: The first choice was stainless steel, it has some advantages which are low thermal conductivity, high thermal resistance and it can resist high temperature. And the profile for this choice was a circular cross - sectional profile which has a good Nusselt number and a high heat transfer coefficient. But this choice has also some disadvantages such as high pressure drop, expensive product and the diaphragm pump has no continuous flow, low speed and efficiency.

Option 2: The second choice was copper, cooling fluid is water, and the profile for this choice was rectangular profile. This choice has some advantages which are high thermal conductivity. Water is a good transfer fluid and has high heat capacity, high density and its viscosity is low. Also, rectangular channels can be manufactured easily. Moreover, the gear pump has great properties.

Option 3: The third choice was aluminum, its advantages is: cheap, lightweight and it has layer of coating for protection. Square channels are easy to manufacture. Cooling fluid for this choice is heat transfer fluid and it is the best heat transfer characteristics. Also, wavy channels are used to increase heat transfer coefficient.

Option 4: The fourth choice was brass, it has advantages such as durability, high temperature resistance and lifetime. A rectangular profile was chosen for this choice. Also, a centrifugal pump was chosen for this option.

2.3 Proposed conceptual designs

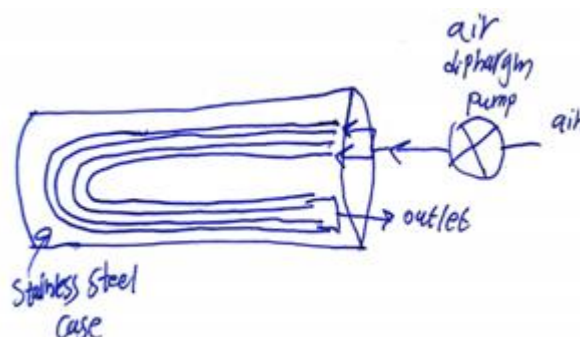


Figure 2.12: Option 1

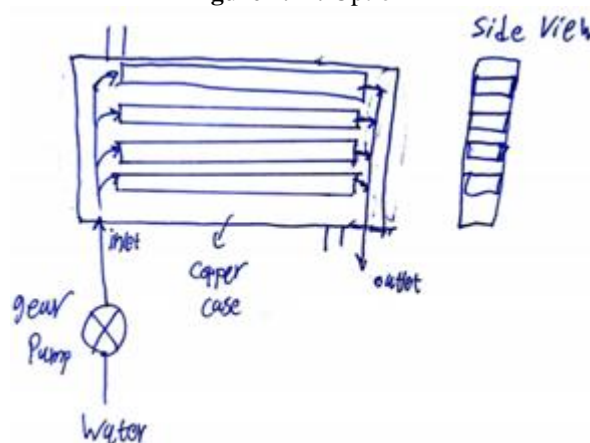


Figure 2.13: Option 2

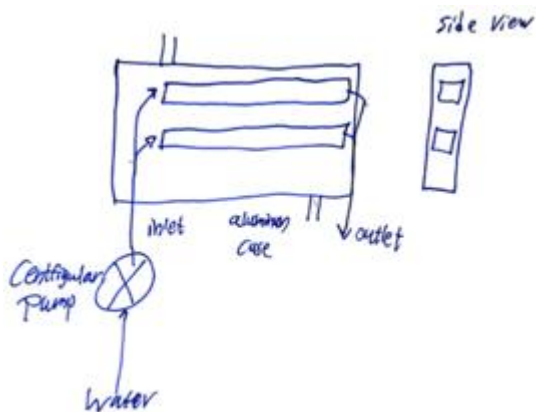


Figure 2.14: Option 3

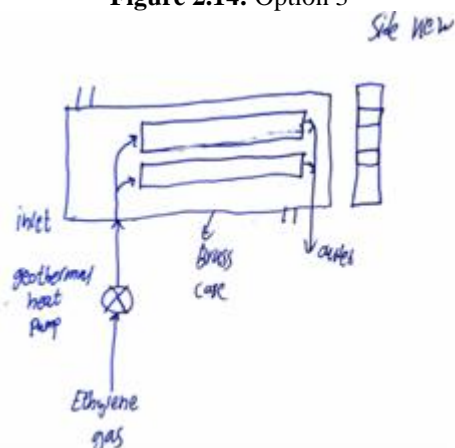


Figure 2.15: Option 4

2.4 Selected conceptual design and preliminary justification of choice

In this section of the report, we are going to talk about first what is the selected conceptual, and how we were able to select it, and then we are going to talk about a brief summary of the preliminary design that we did in GP1.

2.4.1 Selected conceptual design and reasoning

The main idea of the decision matrix is to list the customer requirements and relate it to the conceptual design and the design that gets the most points will be chosen to be our design for this project, this was one of the ways that helped us in choosing the most appropriate design, and as we can see from Table 2.3 below its showing our decision matrix, and its showing that conceptual design 3 was the best fit. Moreover, we have come up with this conclusion by looking at the advantages, disadvantages, working fluid, channel profile, type of pump of each choice, and we found that we can choose conceptual design 3 as the best choice for the heat sink for many reasons; aluminum has the best thermal conductivity and weight ratio and it is cheap, and also water is a good choice because it has high density and found everywhere, and the centrifugal pump was chosen because it is available in the university which will decrease the cost, and lastly the square channel profile is easy to design so we have chosen it.

Table 2.3: Decision matrix

Customer Requirements	Costumer Weights	Conceptual Design 1	Conceptual Design 2	Conceptual Design 3	Conceptual Design 4
Durability	3	3	1	6	9
Performance	9	6	6	9	3
Small size	3	3	3	9	6
Low Cost	9	6	1	6	6
Environment Friendly	6	3	6	6	1
Reliable	6	6	1	3	3
Maintainability	6	1	6	9	1
Absolute Rating	-	186	153	288	156

Correlation	Value
Strong	9
Medium	6
Small	3
Barely Related	1

2.4.2 Preliminary design summary

In the preliminary design section of the GP1 report, we were focused to achieve one goal, which is to write a MATLAB code that can generate different types of graphs, and from these graphs we can determine the heat sink dimensions that are best for the cooling process. Moreover, as we can see in Figure 2.16 it's showing our programming flow chart, where this flow chart is describing our MATLAB code in details, so from the flow chart we can see that the inputs and outputs are declared, and in the process section we can see that it is divided into four section because we have four variables, and the variables parameters are W_{hs} , L_{hs} , Q , and V_t , and we considered these parameters as variables because they are in ranges, where the program job is to generate graphs for us where each variable will be represented with the maximum

temperature of the heat sink, and from these graphs we can determine the best value that will give us the most cooling for each variable, and Table 2.4 below shows the ranges of the variables, also in order to generate our graphs we need to have only one variable per section and make the other variables as constant values, and in each section the program will solve for the variable three times, each time the program will set different values for the constant values, where the values will be the smallest, the middle, and the largest values in the variables range, and each time the program solve it will plot a line representing the value of the constants so at the end we will have in each graph three lines, and this process will be repeated four times as we can see in the four sections in the flow chart, so at the end we will have four graphs with three lines in each graph, and Table 2.5 below

shows the smallest, the middle, and the largest values for the four variables. Furthermore, as we can see from Figure 2.17 to Figure 2.20 it's showing our MATLAB code output, where Figure 2.17 below it's showing the max temperature VS. flow rate graph, and after we analyzed the figure we found that 1 L/sec of flow rate will give us the most cooling, and in Figure 2.18 it's showing the max temperature VS. the total heat output, where we found in this figure that has 1 kW as total heat output is the best for the cooling process, and in Figure 2.19 it's showing max temperature VS. length of the heat sink, where we found in this figure that having the length of the heat sink as 75 cm is the best for the cooling process, and in Figure 2.20 it's showing the max temperature VS. the width of the heat sink, where in this figure we found that has the width of the heat sink as 50 cm will give us the most cooling. In addition to that, Table 2.6 below is showing the best dimensions we found for our heat sink to get the most cooling after we analyzed our MATLAB code results.

Table 2.4: Ranges of the variables

Variable	Range
Flow rate (V_t)	0.5 – 1 L/sec
Total heat output (Q)	1 – 10 kW
Length of the heat sink (L_{hs})	50 – 75 cm
Width of the heat sink (W_{hs})	25 – 50 cm

Table 2.5: Variables constant value

-	Smallest value	Middle value	Largest value
W_{hs}	25 cm	37.5 cm	50 cm
L_{hs}	50 cm	62.5 cm	75 cm
Q	1 kW	5 kW	10 kW
V_t	0.5 Litter/sec	0.75 Litter/sec	1 Litter/sec

Table 2.6: Best dimensions found

Specifications	Value
Flow rate	1 Litter/sec
Total heat output	1 kW
Length of the heat sink	75 cm
Width of the heat sink	50 cm

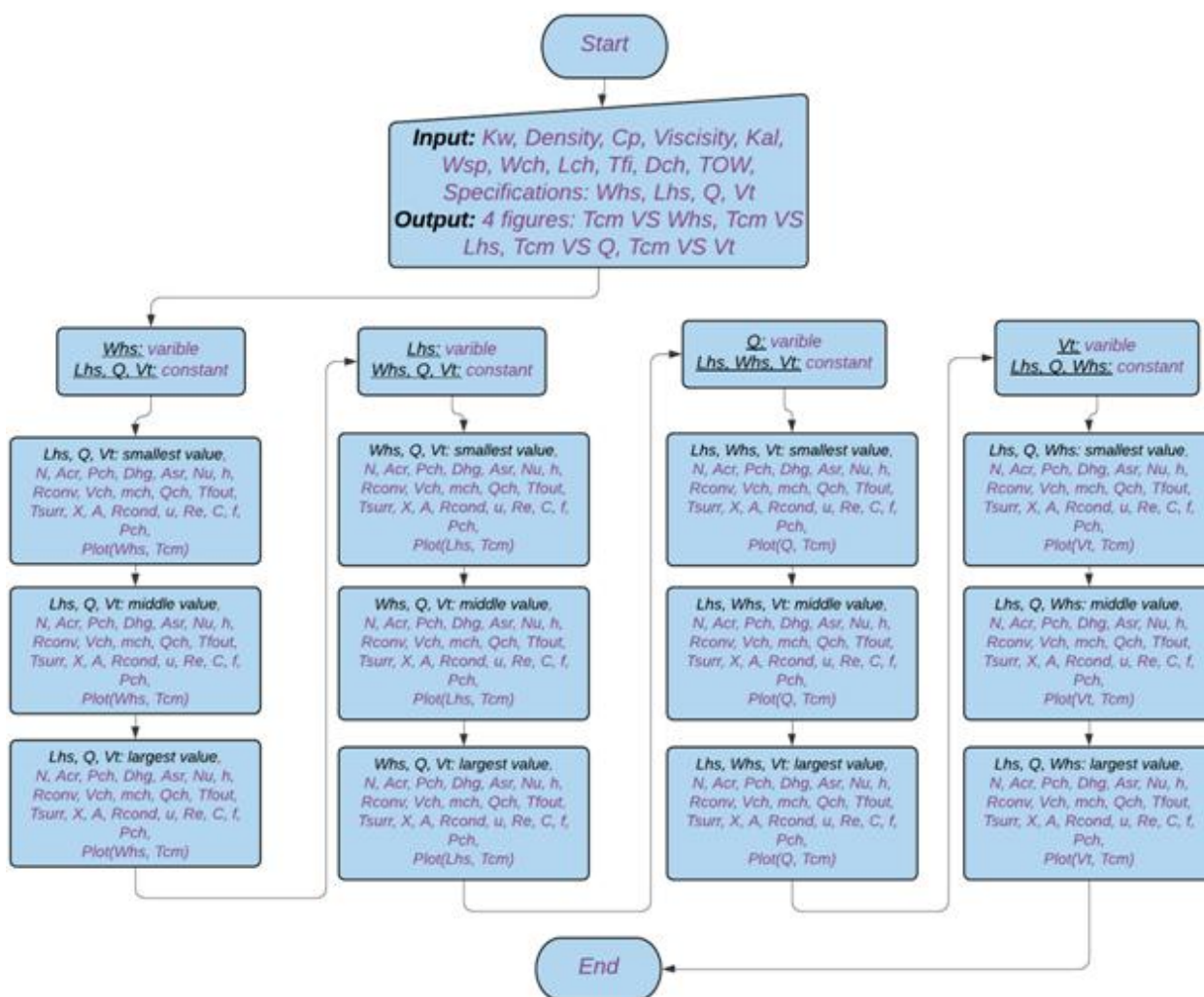


Figure 2.16: Programming flowchart

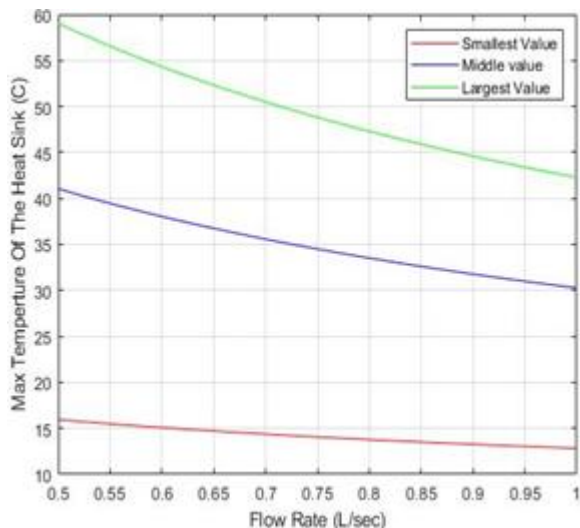


Figure 2.17: Flow rate vs. max temperature of heat sink

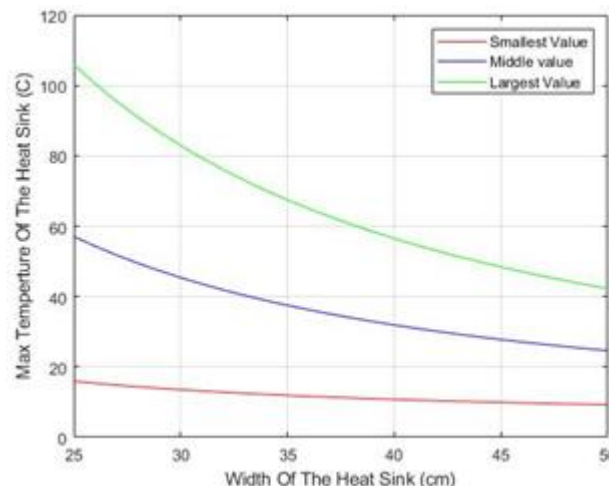


Figure 2.20: Width of heat sink vs. max temperature of heat sink

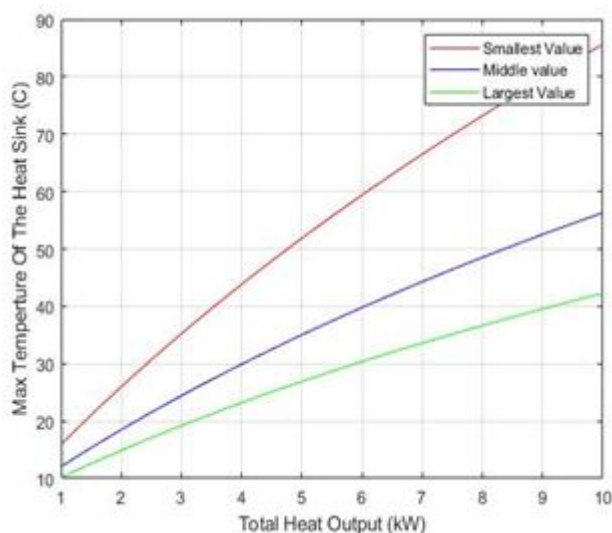


Figure 2.18: Total Heat Output vs. max temperature of heat sink

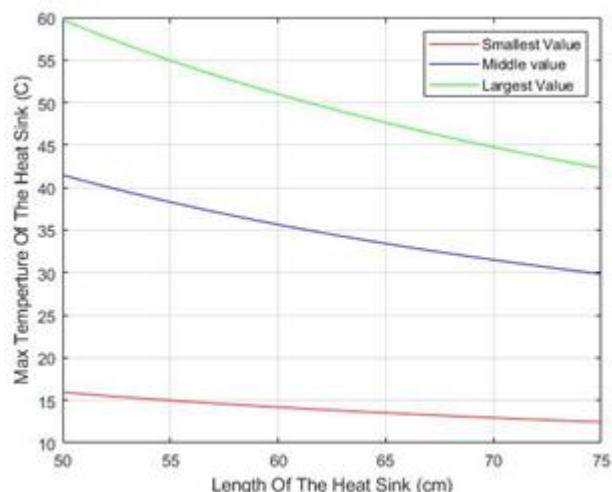


Figure 2.19: Length of heat sink vs. max temperature of heat sink

2.5 Embodiment design

After we finished the MATLAB code in the preliminary design section, and then finished analyzing the MATLAB code results we found the best dimensions for our heat sink, and now we can say that we have all the dimensions to create the CAD design of our heat sink; Table 2.7 below shows all the dimensions needed to create our heat sink model. Moreover, to design our heat sink model, we used Catia software which is used for CAD design, using the dimensions in Table 2.7 we were able to design our heat sink model. Furthermore, Figure 2.21 is showing the isometric view of our heat sink model, and Figure 2.22 is showing the side view of our heat sink model, and also Figure 2.23 is showing the top view of our heat sink model, and Figure 2.24 is showing the front view of our heat sink model, and lastly Figure 2.25 is showing the isometric view of the final CAD module of the heat sink model.

Table 2.7: Heat sink dimensions

Parameters	Dimensions
Length of the heat sink	750 (mm)
Width of the heat sink	500 (mm)
Hight of the heat sink	100 (mm)
Channel Spacing	2 (mm)
Channel width	6 (mm)
Channel Depth	3 (mm)
Total Number of Channels	62

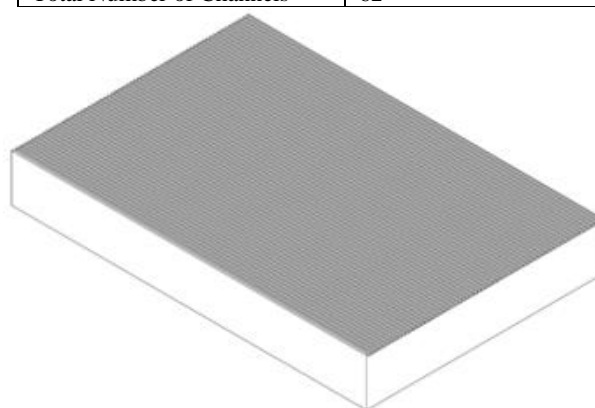
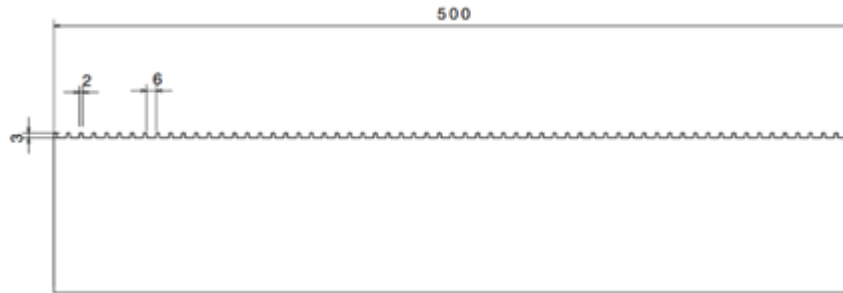


Figure 2.21: Isometric view of the heat sink model



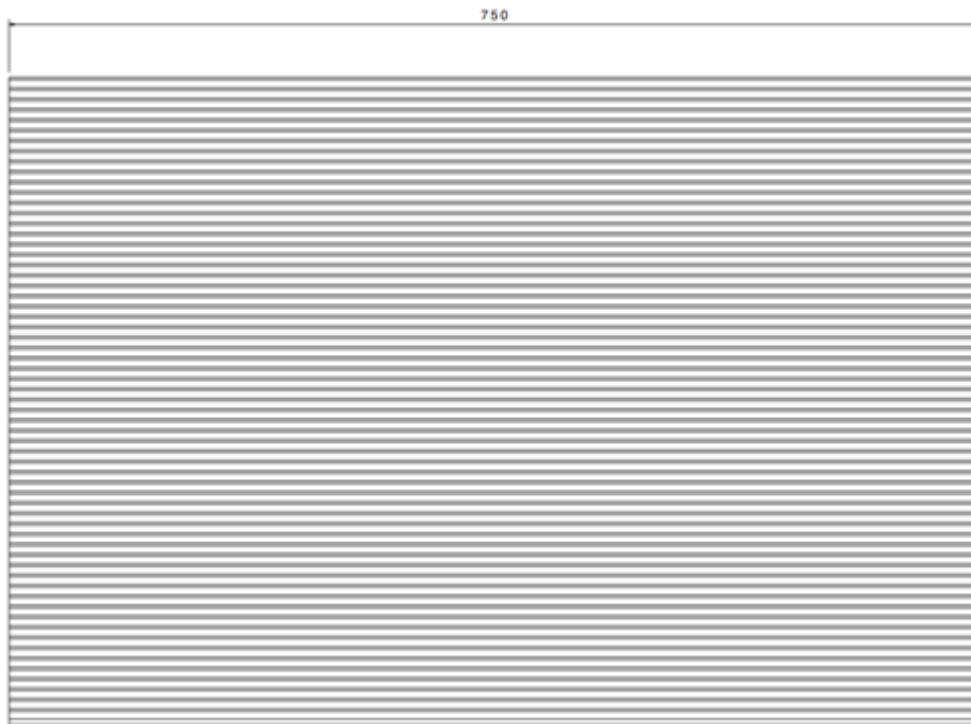
Front view
Scale: 1:2

Figure 2.23: Top view of the heat sink model



Left view
Scale: 1:2

Figure 2.22: Side view of the heat sink model



Top view
Scale: 1:2

Figure 2.24: Front view of the heat sink model

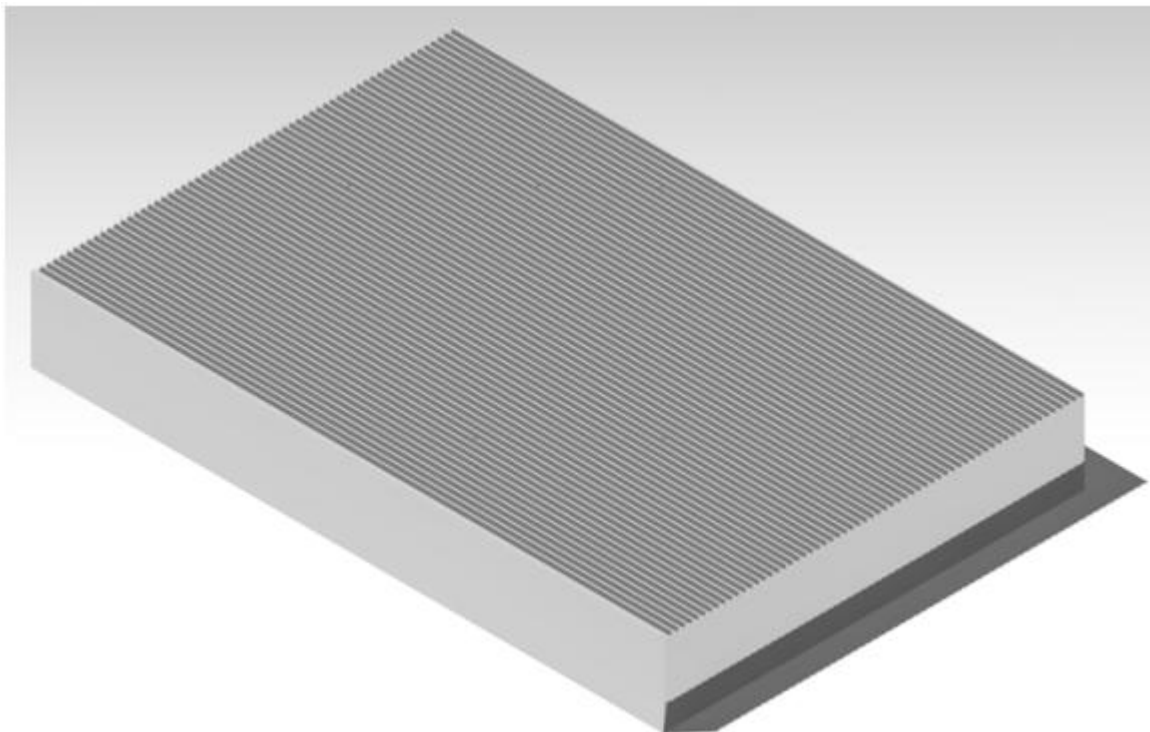


Figure 2.25: Isometric view of the final CAD module of the heat sink model

2.6 Final deliverables and preliminary cost

We looked online to find the prices of our components that we need to build the heat sink. Here are our findings:

Heat sink aluminum stock:

We found the price of the aluminum we need in kilograms. the price of aluminum is around 2 USD. We need to use around 100 kg of aluminum stock so that will be AED750.

Centrifugal pump:

We need the Centrifugal pump for our heatsink to operate. We can get the centrifugal pump online the will around AED650.

PVC tubes:

We need two PVC tubes for the inlet and the outlet. We chose this material because it has good insulation will cost around AED100.

Pressure gage:

We will use a pressure gauge to test the performance of the centrifugal pump. That will cost us AED100.

Thermocouples

We will use four thermocouples for experimental testing of the heat sink. The price of the unit is around AED100 so the total price will be AED400.

Table 2.8: Quality function deployment

Component	Cost per unit	Quantity	Total cost
Centrifugal pump	AED650	1	AED650
Aluminum	AED350	1	AED750
Tubes	AED50	2	AED100
Pressure gauge	AED100	1	AED100
Thermocouple	AED100	4	AED400
Total price	-	-	AED2000

3. Design Setup

In this section of the report we are going to design our final heat sink, and then design the needed parts to make the heat sink useable; First in this section of the report we are going to talk about our heat sink parameters, and how the parameters have changed from GP1 to GP2 and why. Then we are going to show our final heat sink model with its drawings, after that we are to talk about the parts needed to make our heat sink useable, then we are going to show the parts models with their drawings. Moreover, we designed four main parts in order to make our heat sink useable which are the bottom part, the top part, the bolt and the nut, and in this report we will discuss each part in detail.

3.1 Bottom part

The bottom part of our design is the actual heat sink, but before we show our final dimensions of the heat sink, we need first to show our initial dimensions that we concluded from GP1 and see how they have changed, and Table 3.1 is showing our initial heat sink dimensions that we got in GP1, In GP1 we got these dimensions by creating a MATLAB code that will generate graphs for us, and from these graphs we determined our heat sink dimensions that give us the most cooling.

Table 3.1: Initial heat sink dimensions

Parameters	Dimensions
Length of the heat sink	750 (mm)
Width of the heat sink	500 (mm)
Hight of the heat sink	100 (mm)
Channel Spacing	2 (mm)
Channel width	6 (mm)
Channel Depth	3 (mm)
Total Number of Channels	62

From Table 3.1 we can see that the length of the heat sink was 750 mm, and width of the heat sink was 500 mm, and the height of the heat sink was 100 mm. However, we wanted to change these dimensions due to budget fund issues, once we change these three parameters then the total number of channels will change. We tried to decrease the size of our heat sink and we have come up with the dimensions shown in Table 3.2, and this table is showing the new dimensions of our base heat sink.

Table 3.2: Base heat sink dimensions

Parameters	Dimensions
Length of the heat sink	250 (mm)
Width of the heat sink	250 (mm)
Hight of the heat sink	5 (mm)
Channel Spacing	2 (mm)
Channel width	6 (mm)
Channel Depth	3 (mm)
Total Number of Channels	31

In order to make our heat sink useable and can connect other parts to it we need to modify our base heat sink dimensions, and we used the term “base” because we are going to add to these dimensions in order to make the heat sink useable. Furthermore, in our heat sink we want the flow through the channels to be as parallel flow because it will give us the most cooling, and to do so we decided to add 50 mm in the front of the heat sink, and 50 mm in the back of the heat sink as this will give us the parallel flow that we want, and Figure 3.1 below is showing our top view of the final heat sink model after we modified the base dimensions, and the blue area in the figure is showing the 50 mm added to have parallel flow in the channels, and the orange area is the base dimensions of our heat sink from Table 3.2, To have the heat sink useable and so we can connect the bottom part with the top part later, we add 20 mm in each of the heat sink so we can make the holes for the bolt, which will allow us to connect the bottom part with top part, and in Figure 3.1 the green area is representing the 20 mm added from each side so we can connect the bottom part with top part. Lastly Table 3.3 below is showing our final heat sink dimensions after the modifications to the base dimensions in Table 3.2.

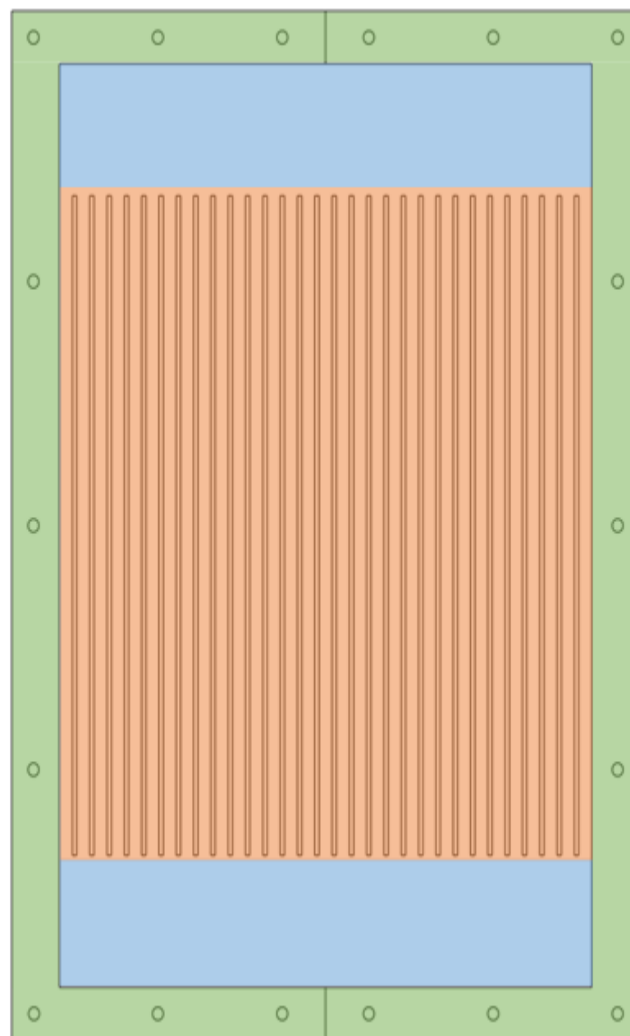


Figure 3.1: Top view of the modified base dimensions

Table 3.3: Final heat sink dimensions

Parameters	Dimensions
Length of the heat sink	390 (mm)
Width of the heat sink	290 (mm)
Hight of the heat sink	5 (mm)
Channel Spacing	2 (mm)
Channel width	6 (mm)
Channel Depth	3 (mm)
Total Number of Channels	31

- Bottom part drawings

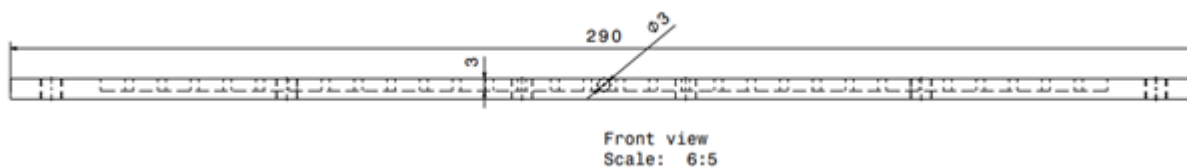


Figure 3.3: Side view of the bottom part



Figure 3.2: Front view of the bottom part

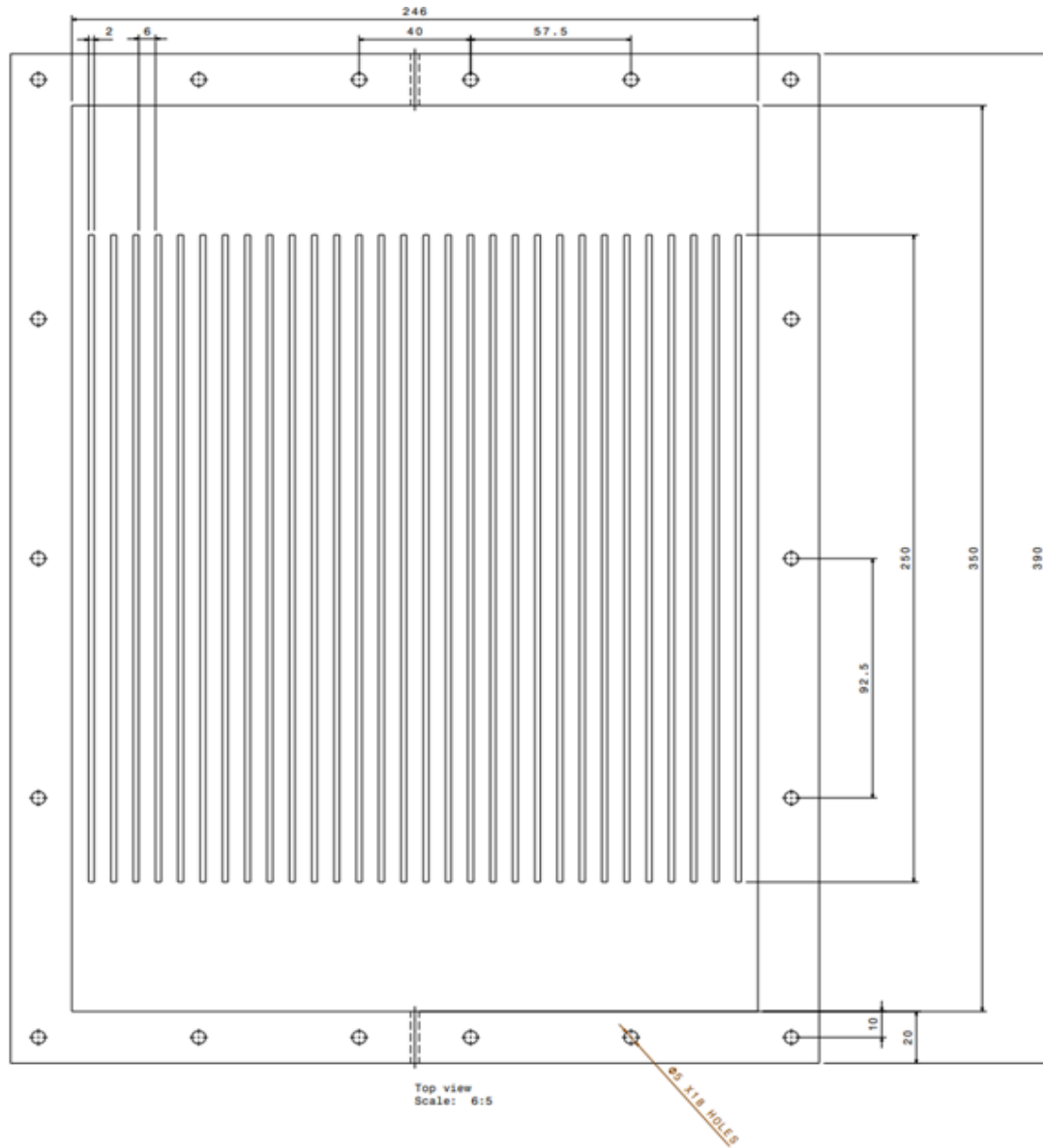


Figure 3.4: Top view of the bottom part

Figure 3.4: Top view of the bottom part

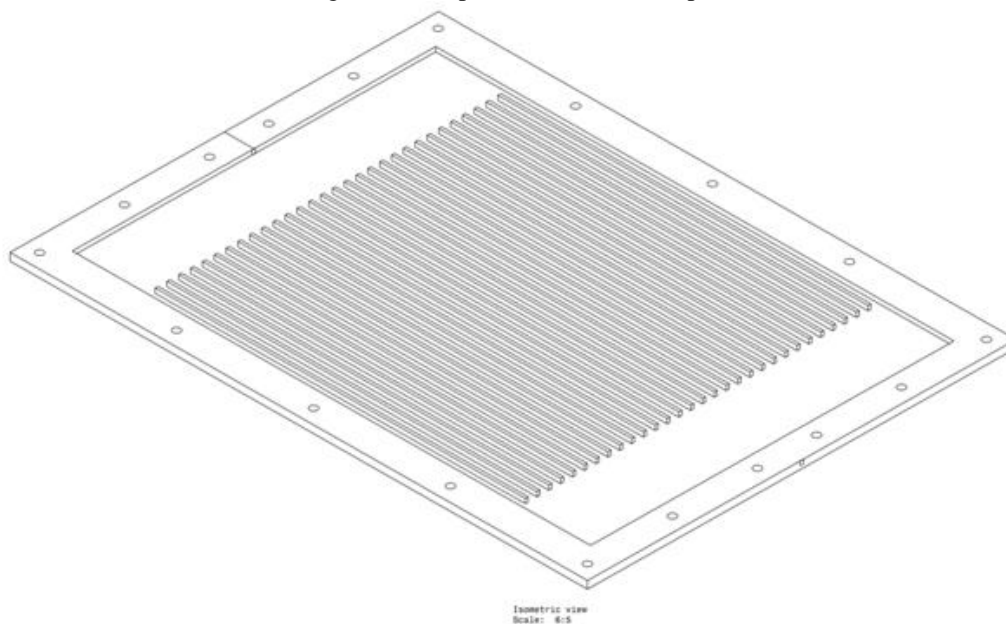


Figure 3.5: Isometric view of the bottom part

3.2 Top part

We decided to design the top part, to have our bottom part covered so the fluid can flow, and the top part is identical to the bottom part. In the top part there are 18 holes where the bolt will go in just like the bottom part, below is the

drawings of the top part where the dimensions are clearly shown in drawings.

- Top part drawings

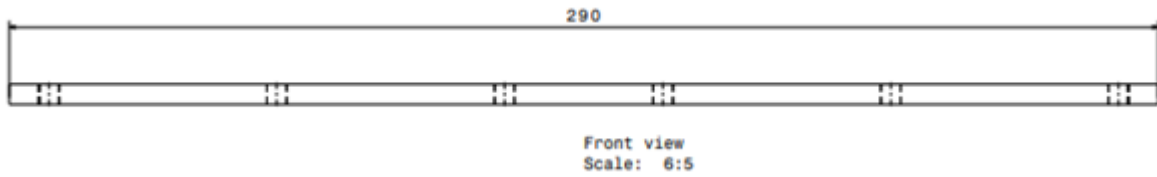


Figure 3.6: front side view of the top part

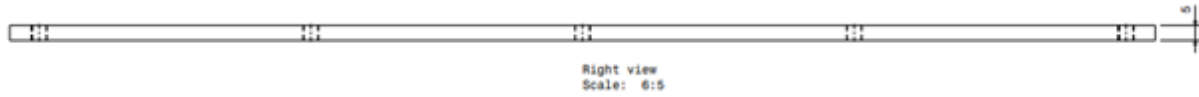


Figure 3.7: Right side view of the top part

Figure 3.7: Side view of the top part

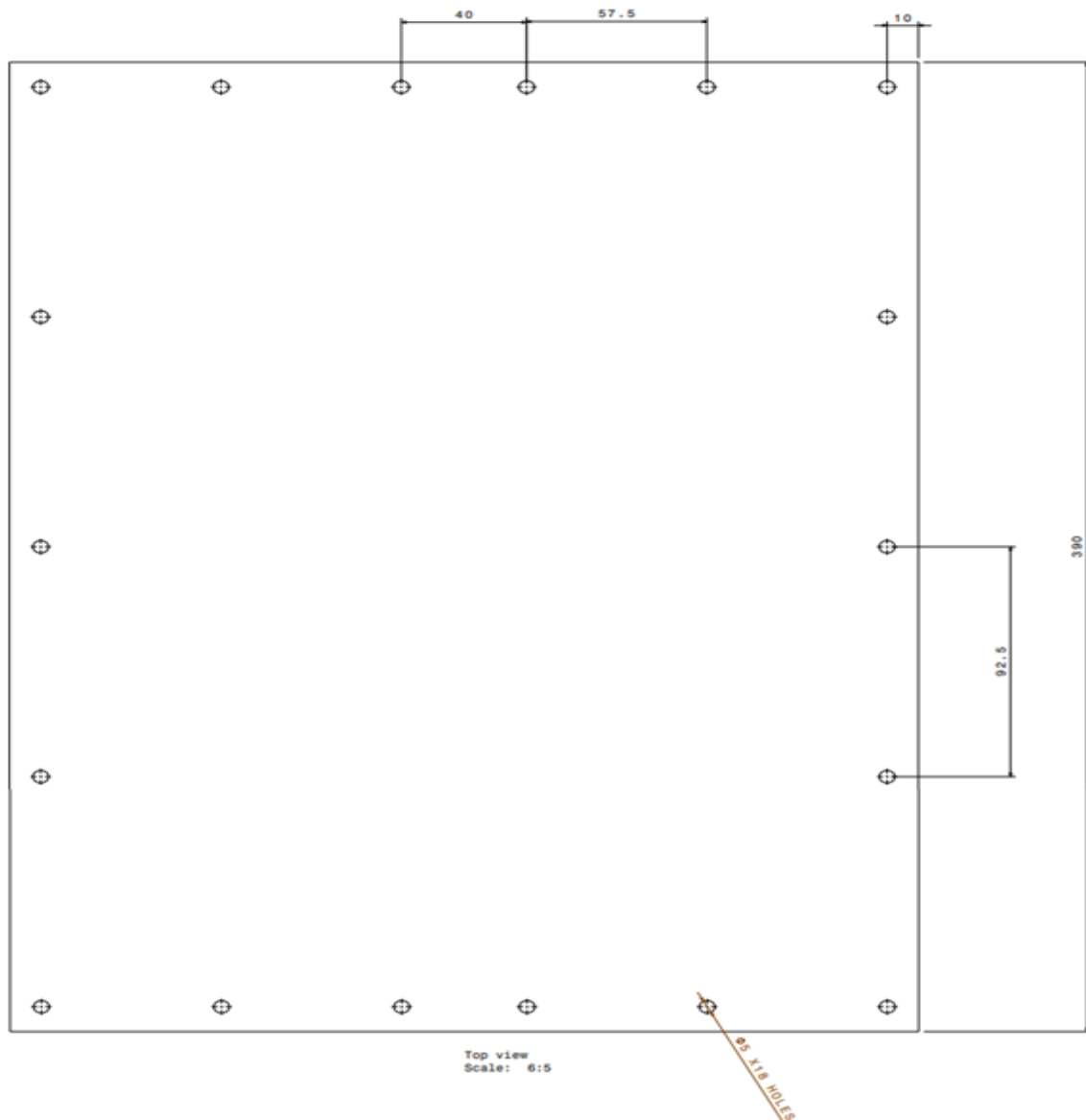


Figure 3.8: Top view of the top part

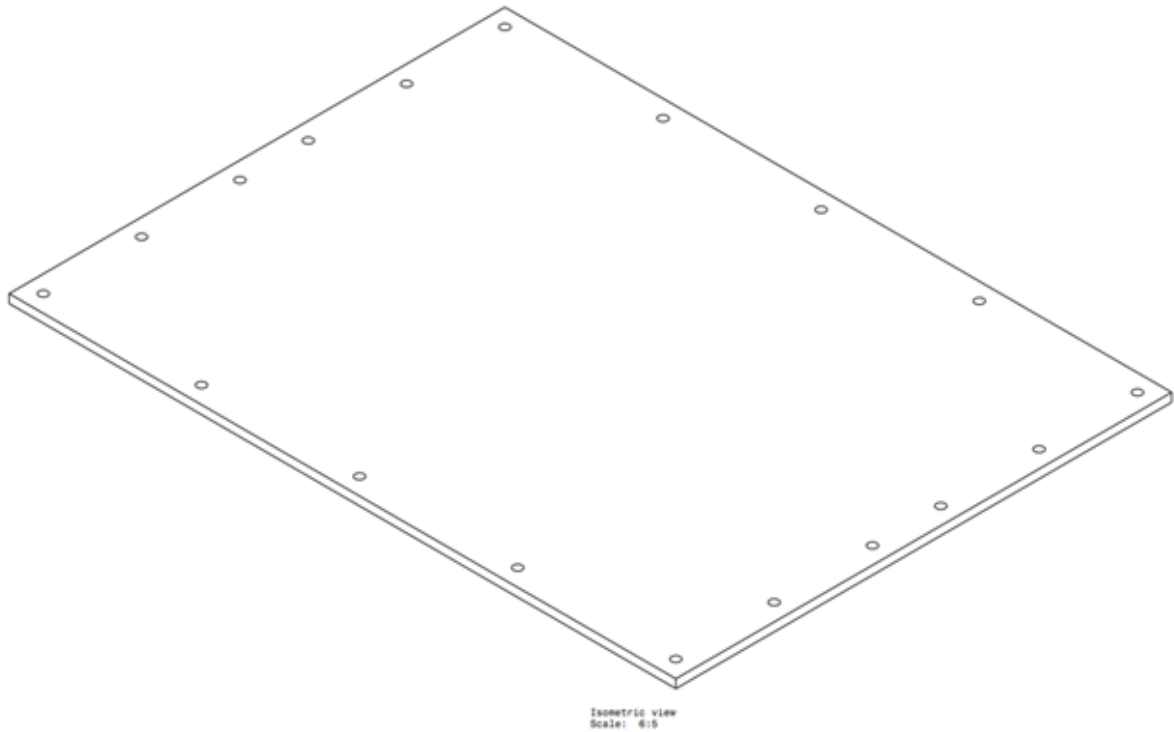


Figure 3.9: Isometric view of the top part

3.3 Bolt

The bolt was designed to keep the top and bottom parts connected together, and there are 18 holes in the top and bottom parts so there should be 18 bolts connected to keep the top and bottom parts together, below are the drawings for the bolt where the bolt dimensions are clearly shown in drawings.

- Bolts drawings

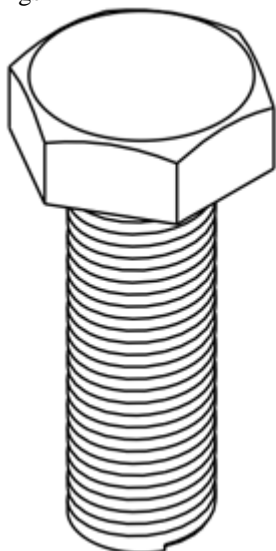


Figure 3.10: Isometric view of the bolt

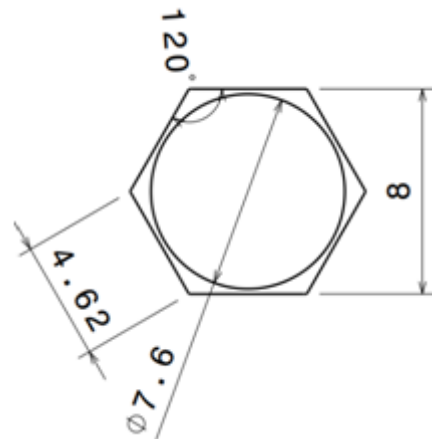


Figure 3.11: Top view of the bolt

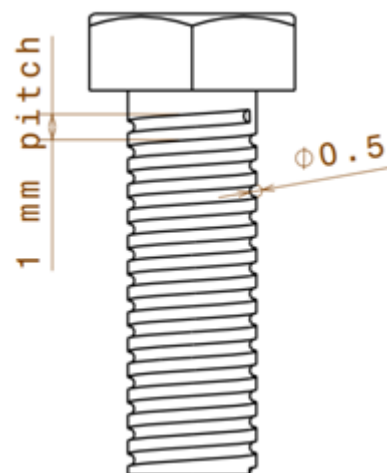


Figure 3.12: Side view of the bolt

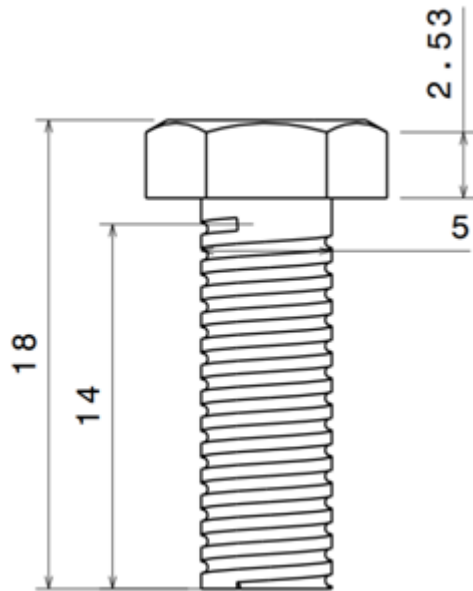


Figure 3.13: Front view of the bolt

3.4 Nut

We designed the nut to keep all the parts connected and stable, and there were 18 holes in the top and bottom parts so there should be 18 nuts that will be joint with 18 bolts. In this way the heat sink will stable, and all the parts will be connected together, and below are the drawings for the nut where the nut dimensions are clearly shown in drawings.

- Nut drawings

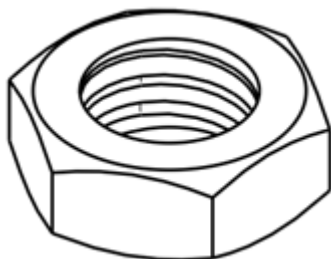


Figure 3.15: Top view of the nut

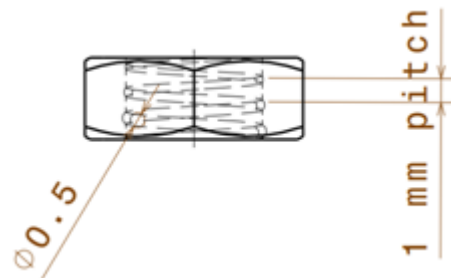


Figure 3.16: Side view of the nut

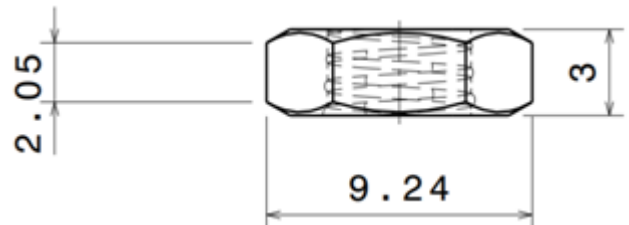


Figure 3.17: Front view of the nut

3.5 Assembly

In this section of the report, we are going to make the assembly drawings, and these drawings will tell us if all the parts of our heat sink can connect or not. In this section we will show two different types of assembly drawing, one isometric view of our heat sink with all the parts connected together, and one isometric view of our heat sink with the parts not connected together; to show the bill of the materials table. Moreover, as we can see in Figure 3.18 it's showing the isometric view of our heat sink with all the parts connected together perfectly, which means that all of our dimensions for all of the parts were correct. Figure 3.19 is showing the isometric view of our heat sink where all the is separated from each other, so we can the bill of material table which is Table 3.4.

- Assembly drawings

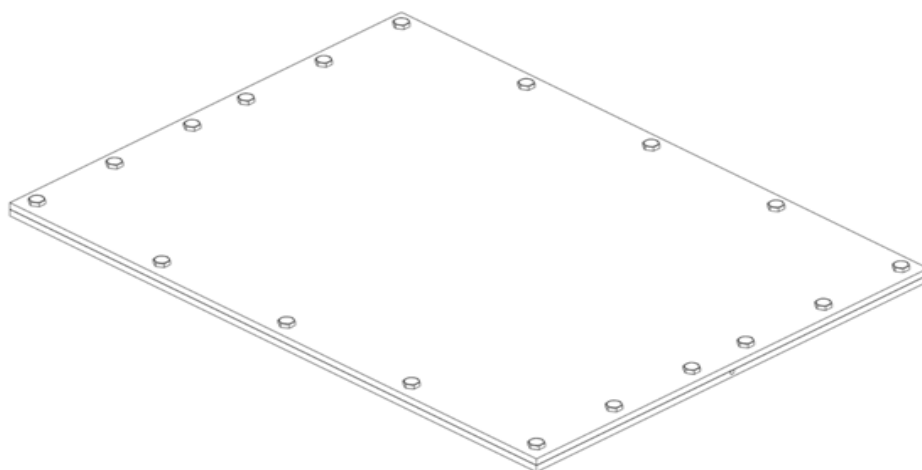


Figure 3.18: Isometric view of the heat sink (all parts connected)

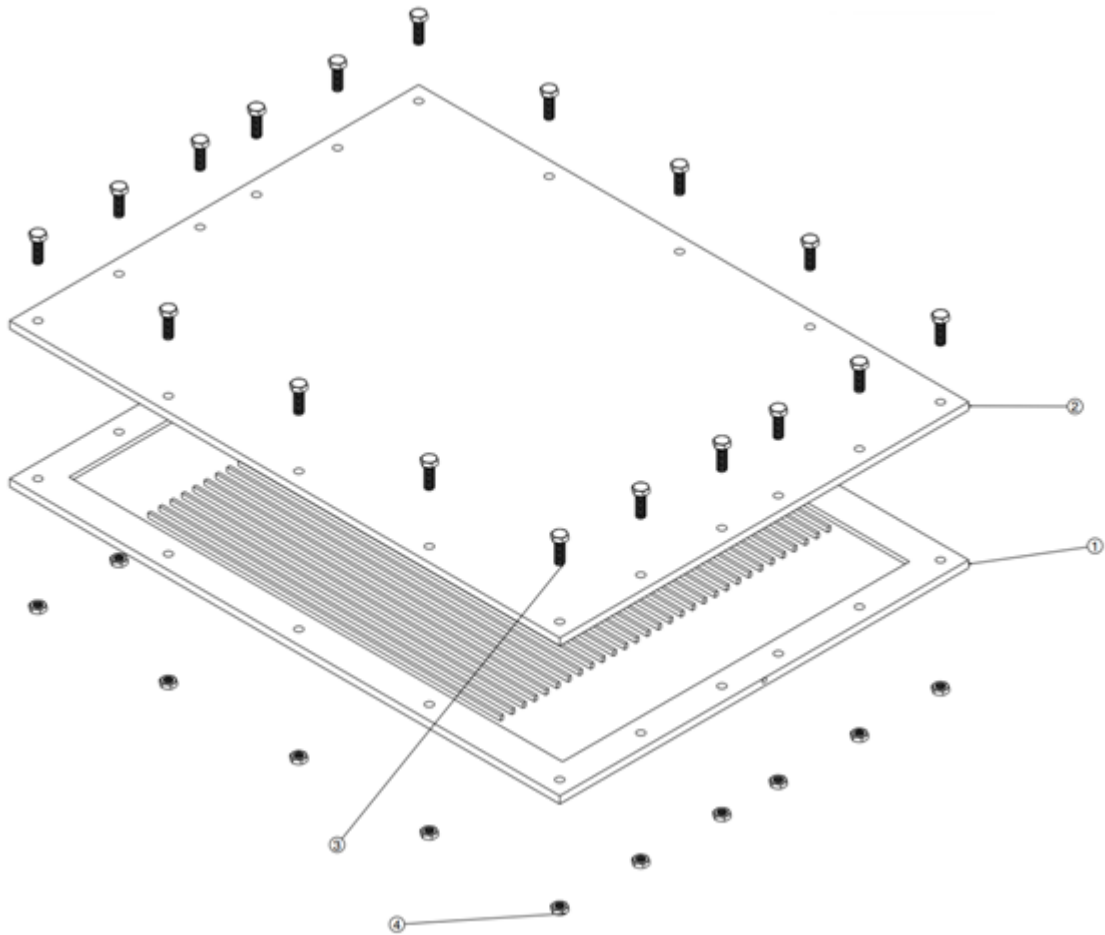


Figure 3.19: Isometric view of the heat sink (all parts separated)

Table 3.4: Bill of material

Sr No.	Component	Quantity
1	Bottom part	1
2	Top part	1
3	Bolt	18
4	Nut	18

4. Detailed Design

In this section of the report, first we will show our governing equations, where they are too complex for analytical solution, then we will try get the solutions using ANSYS Fluent software by simulating our channel, and the purpose of this is to see the change in the thermal resistance with the Reynolds number, and to see the change in the pumping power with the Reynolds number.

4.1 Mathematical modeling

The mathematical model of our heat sink consists of several governing equations including the continuity equation, Navier - Stokes equations, and the energy equation for our fluid and solid, and the continuity equation is shown below in Eq. (1) while the Navier - Stokes equations are shown below in Eq. (2), moreover the energy equation of our fluid is shown below in Eq. (3), and the energy equation of the substrate is shown in Eq. (4); where s is representing substrate, in addition there were some assumptions taken in place in our heat sink such as our heat sink is operating under steady state conditions, operation our heat sink falls

within the continuum regime, the fluid are not undergoing any phase change between their corresponding inlet and outlet, the fluid are assumed to be incompressible, and there is no viscous heat generation, external heat transfer and flow maldistribution associated with the fluid [5].

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

$$u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \frac{\mu}{\rho} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \quad (2.1)$$

$$u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \frac{\mu}{\rho} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \quad (2.2)$$

$$u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \frac{\mu}{\rho} \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \quad (2.3)$$

$$u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} = \frac{k}{\rho c_p} \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) \quad (3)$$

$$\frac{\partial^2 T_s}{\partial x^2} + \frac{\partial^2 T_s}{\partial y^2} + \frac{\partial^2 T_s}{\partial z^2} = 0 \quad (4)$$

Where u (m/s) is the velocity of the fluid along the x -direction, v (m/s) is the velocity of the fluid along the y -

direction, w (m/s) is the velocity of the fluid along the z -direction, P (N/m²) is the pressure of the fluid, μ (Pa.s) is the viscosity of the fluid, ρ (kg/m³) is the density of the fluid, c_p (J/kgK) is the specific heat capacity of the fluid, T (K) is the temperature of the fluid, moreover water is used as the fluid in our heat sink, and aluminum is used as the substrate in the study state, and the properties of the fluid are assumed at its inlet temperature.

The boundary conditions associated with the governing equations include known average velocity (V_{avg}), and the velocity at the inlet of the channel for our fluid is determined from its Reynolds number, hydraulic diameter of the channel, and fluid properties such as density and viscosity as shown below in Eq. (5) [5].

$$V_{avg} = Re \frac{\mu}{\rho D} \quad (5)$$

The hydraulic performance of the heat sink is represented in terms of total pumping power per channel (PP) as shown in Eq. (6), the thermal resistant (R_{th}) of the heat sin is represented in Eq. 7 [5].

$$PP = (P_{in} - P_{out}) \dot{Q} \quad (6)$$

$$R_{th} = \frac{T_{max} - T_{in}}{q} \quad (7)$$

Where T (K) is the average temperature over the cross-section of the fluid, \dot{Q} (m³/s) is the flow rate of the fluid,

and P (Pa) is the average pressure across the cross-section of the fluid; in and out represents the inlet and outlet of the channel, and T_{max} (K) is the maximum temperature of the heat sink, and q (W/m²) is the heat flux applied at the bottom of the heat sink. Furthermore, we can see that we got very complex differential equations for the analytical solution, so going from this point we decided to use ANSYS Fluent software as it will help us solve these equations by simulating our channel.

4.2 Sample ANSYS Fluent steps

In order for us to achieve our goal in ANSYS Fluent we must follow certain steps, for example as we can see in Figure 4.3 the software is divided into several parts which are geometry, mesh, setup, solution and results, so to start our project in ANSYS Fluent we need to start with the geometry first.

4.2.1 Geometry

ANSYS Fluent software provides the user with a tool that allows to create the geometry, however we found that this tool is not user friendly tool, so we decided to use Catiato draw our geometry and then save the file as “. igs” and then we will import the geometry file to ANSYS Fluent. Moreover, as we mentioned before our heat sink is made of multiple repeating channels, so we only need to study one channel as the results for this channel would be the same for the other channels.

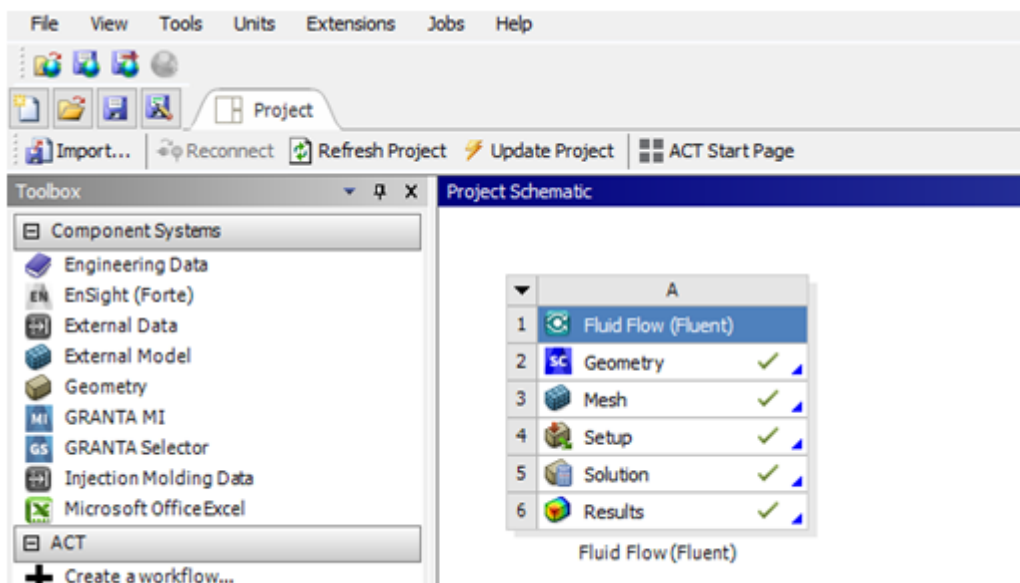


Figure 4.3: Creating our project

Furthermore, to create our geometry we need to draw two drawings, one for the solid part and one for the fluid part and then we will assemble them together as one drawing. we did this because in ANSYS Fluent we need to tell the program which part is the fluid and which part is the solid. As we can see in Figure 4.4 its showing our drawing in Catia software

for the solid part of one channel. Figure 4.5 is showing the fluid part that is flowing in the solid part of Figure 4.4, and Figure 4.6 is showing the assembly drawing of the solid part and the fluid part together, and then we save the assembly drawing as a “. igs” file to import it in the ANSYS Fluent software.

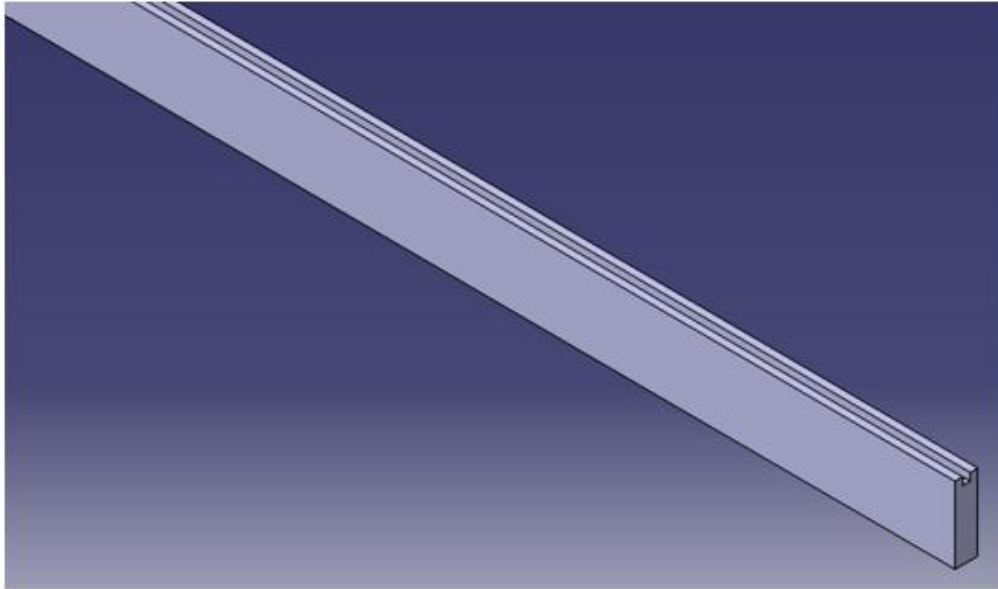


Figure 4.4: Solid part

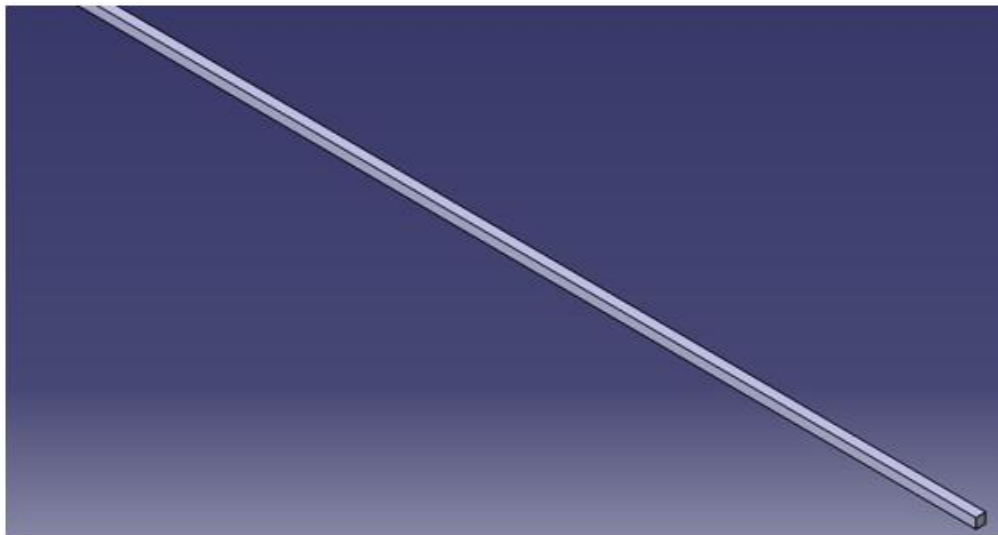


Figure 4.5: Fluid part

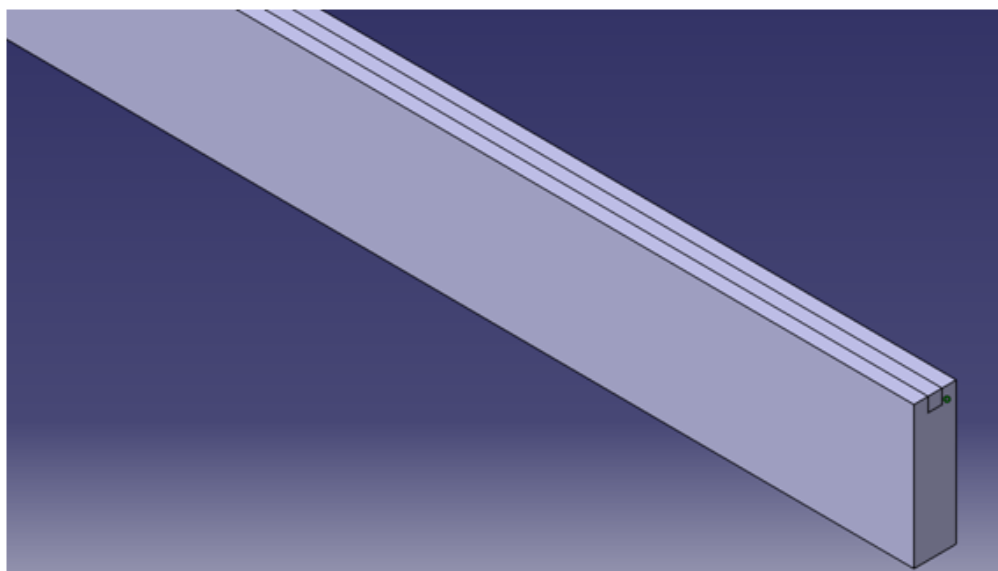


Figure 4.6: Assembly part

After that, we are almost done with the geometry part except after we import our assembly part to ANSYS Fluent software, we need to tell the program which part is the solid part and which part is the fluid part, and as we can see in Figure 4.7 showing the assembly drawing after it's imported to ANSYS.

to tell the program which part is fluid and which part is solid, we need to select the part body of the solid for example as shown, and then make a new material for the selected part body, and then go to the fluid tab and make it false in the case of the solid part, For the fluid part we repeated same with fluid tab as true in this case the program will know which part is solid and which part is fluid.

Figure 4.8 is showing the settings that we changed to declare the fluid and the solid parts. As we can see from Figure 4.8,

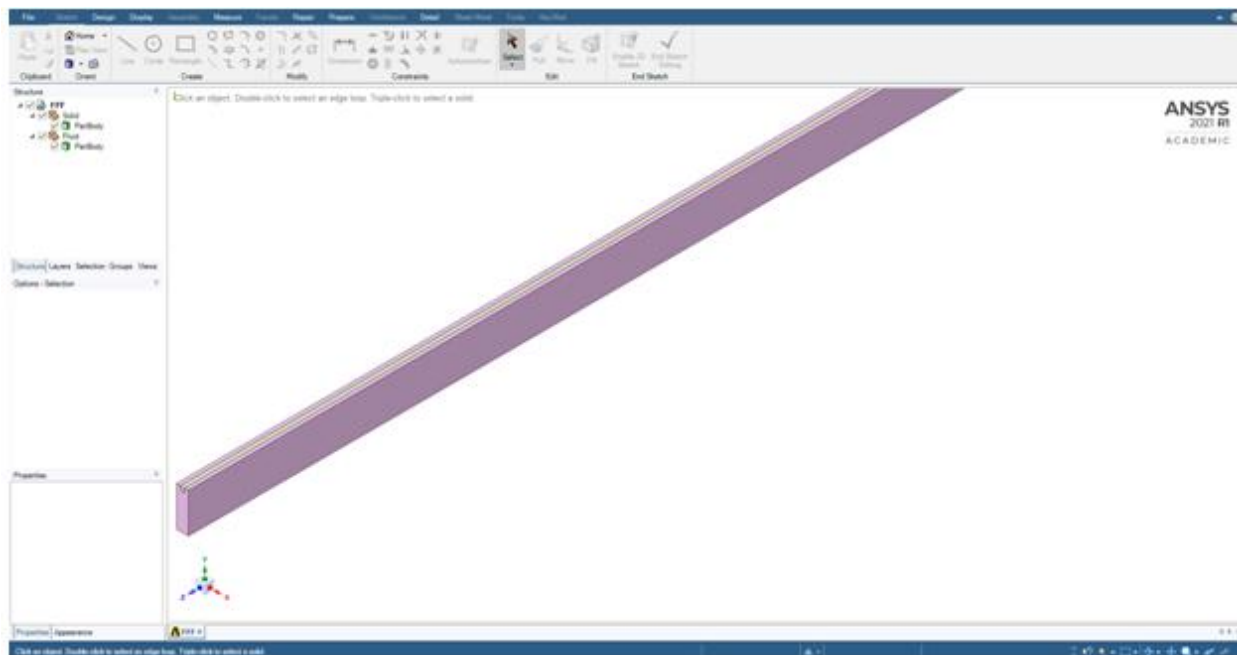


Figure 4.7: Imported geometry drawing

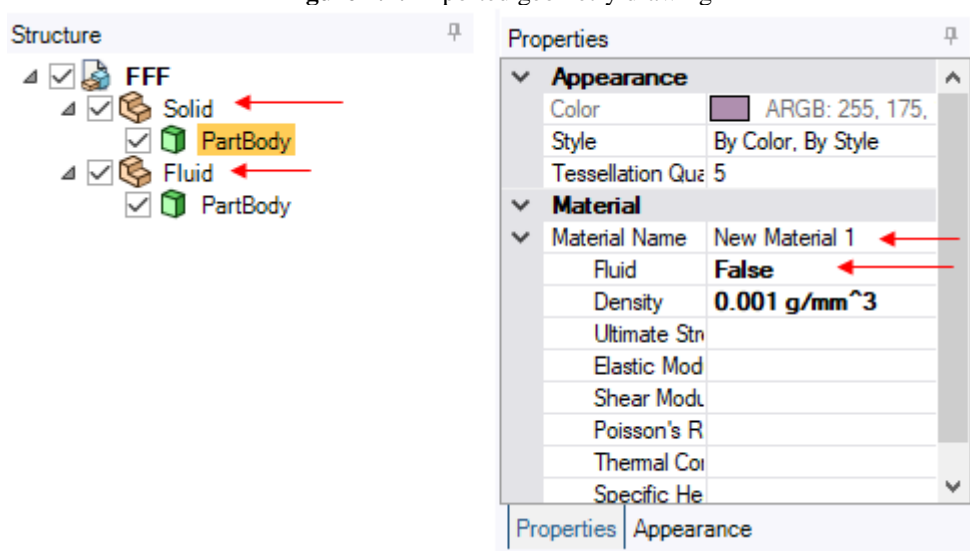


Figure 4.8: Declaring the solid and the fluid parts

4.2.2 Mesh

In the mesh part the goal is to divide our geometry into small elements to get accurate results at the end, the smaller the elements we divide the more accurate results we will get, however since we are using the student version of ANSYS there are some limitations to this version, for example the maximum number of elements allowed is 500, 000 elements. If we exceeded this number we will not be able to continue, but if we divided our geometry to a number of elements

close to 500, 000 elements we will get accurate results. In addition to that, to start creating our mesh we need first to declare the inlet, outlet and the bottom wall so that in the setup part we can change the parameters associated with them. Figure 4.9 is showing the steps that we did to declare the inlet, and it's the same steps were done to declare the outlet and the bottom wall, so first we selected the face we want to declare as an inlet as shown in Figure 4.9 and then gave it a name as "Inlet".

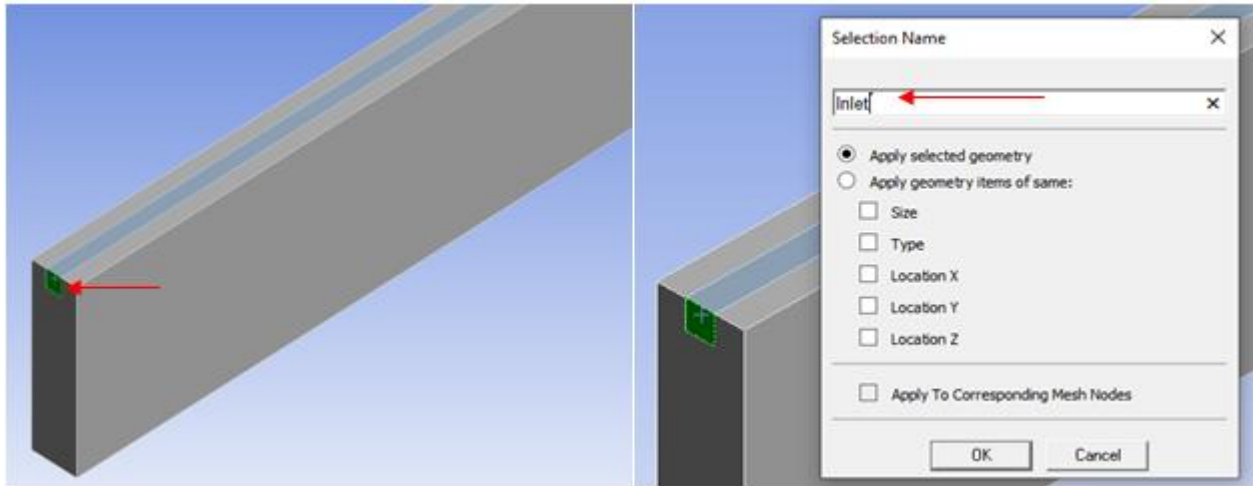


Figure 4.9: Declaring the Inlet

Then we selected the mesh tab to define the element size, and we set it to be 0.000159 m to get the number of elements close to 500,000. As we can see in Figure 4.10 we got the number of elements to be 438,588 which is close to 500,

000 and it's acceptable, and it will give us accurate results at the end, and Figure 4.11 is showing the geometry after the mesh is applied.

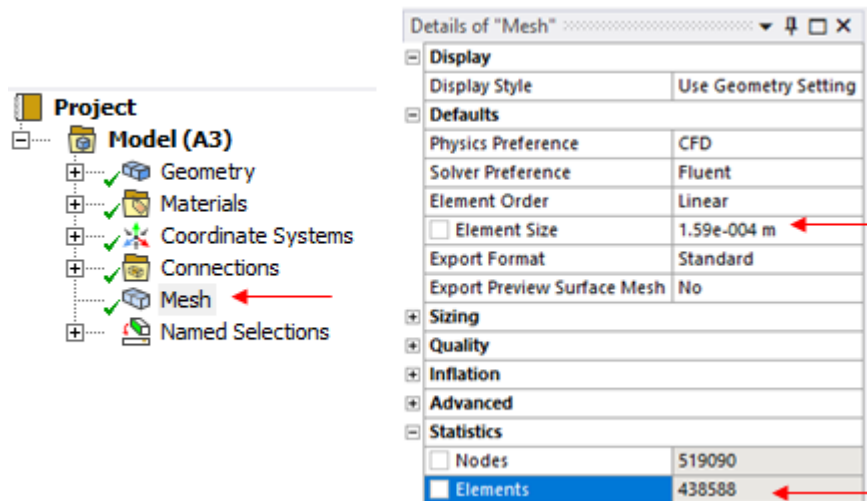


Figure 4.10: Mesh settings

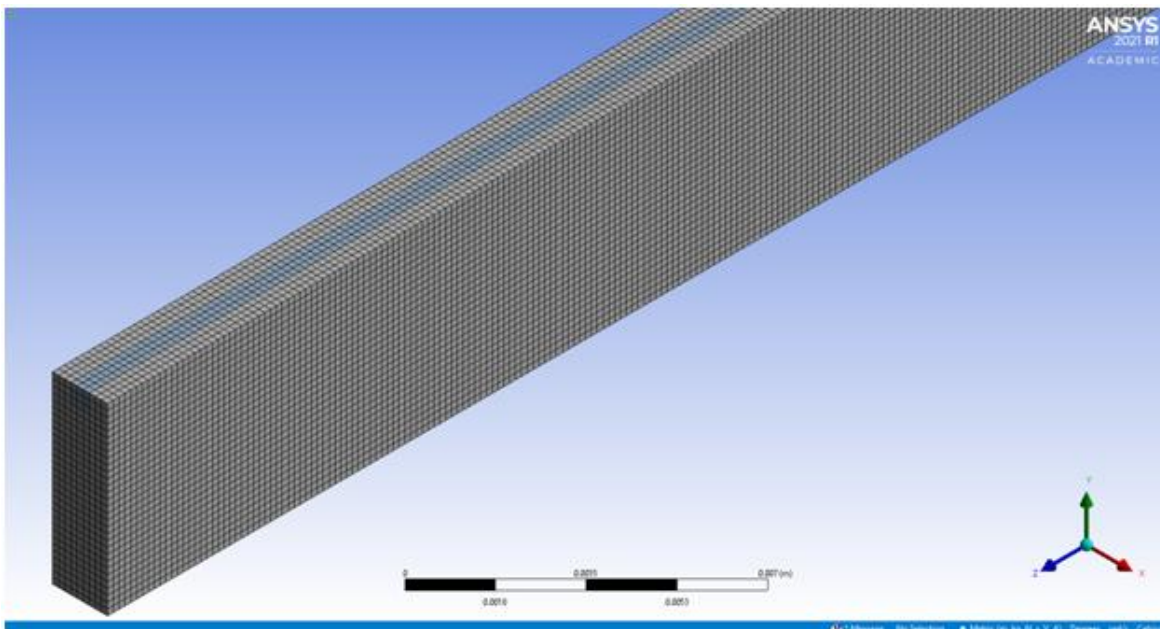


Figure 4.11: After the mesh is applied

4.2.3 Setup

In the setup part we can declare our parameters at the Inlet, outlet and the bottom wall, and to do so we must first go to the model's tab and change the energy to on, because we want the program to solve the energy equations because we have heating, then in the model's tab we changed the viscous to laminar, because we will deal laminar flow in our heat sink and the Reynolds number will be between 50 to 1500, as shown in Figure 4.12.

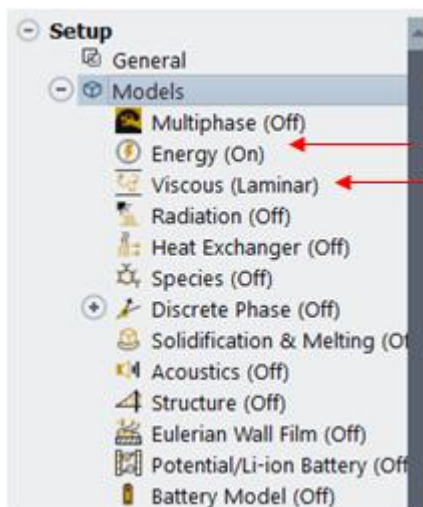


Figure 4.12: Models settings

Then we wanted to add water as fluid and aluminum as solid, and to do so we have to change it from the materials tab and then double click the fluid to add water as a fluid. Then we changed the properties of water to the inlet water temperature properties which were 5°C, and we did the same steps for the solid, and Figure 4.13 shows the materials settings.

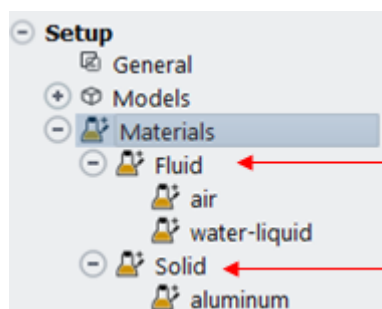


Figure 4.13: Materials settings

After that, in the cell zone conditions tab we want to change the fluid to water and the solid to aluminum, to do so we double clicked the fluid and change the material name to water as shown in Figure 4.14, and we did the same steps for the solid.

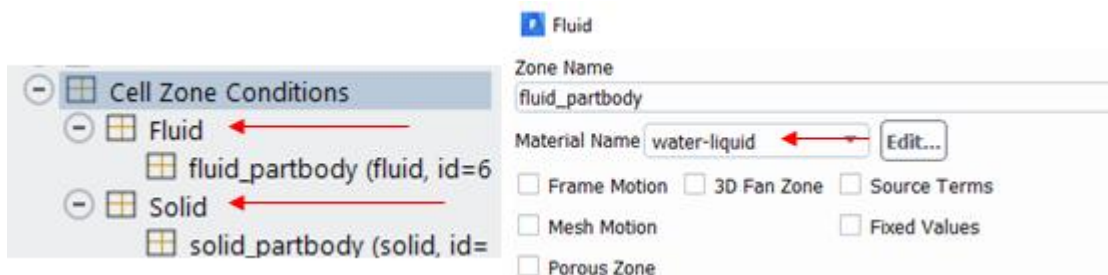
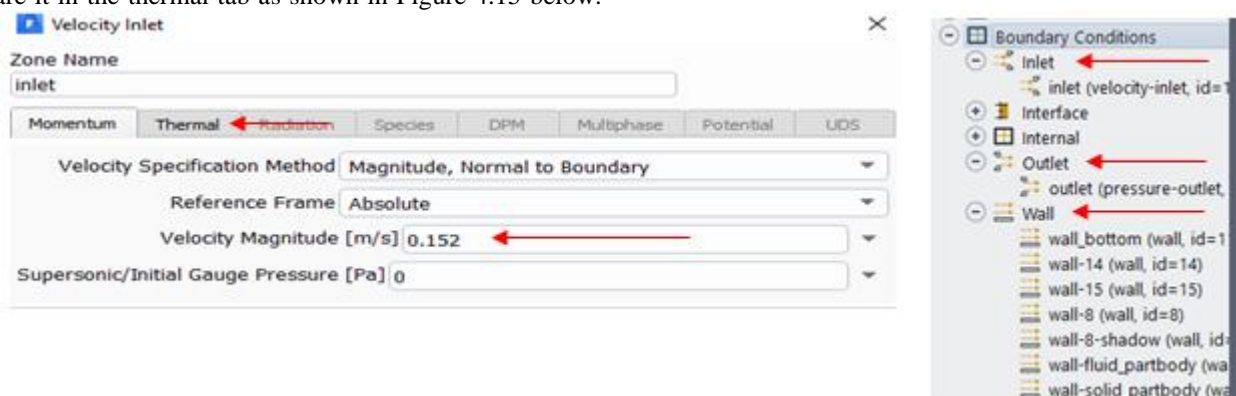


Figure 4.14: Cell zone conditions

Then we need to define the boundary conditions at the inlet, outlet and the bottom wall, so in the boundary conditions we double clicked the inlet to change the inlet velocity and the inlet temperature, and we have all the parameters to find the inlet velocity where the Reynolds number in this case was 50. The inlet temperature is defined as 5°C and we can declare it in the thermal tab as shown in Figure 4.15 below.

In the outlet we don't have to change anything because by default the outlet pressure is set zero. In the bottom wall we need to add the heat flux, to so we double clicked "wall_bottom" and set the heat flux to 1000 w/m² as shown in Figure 4.15 below.



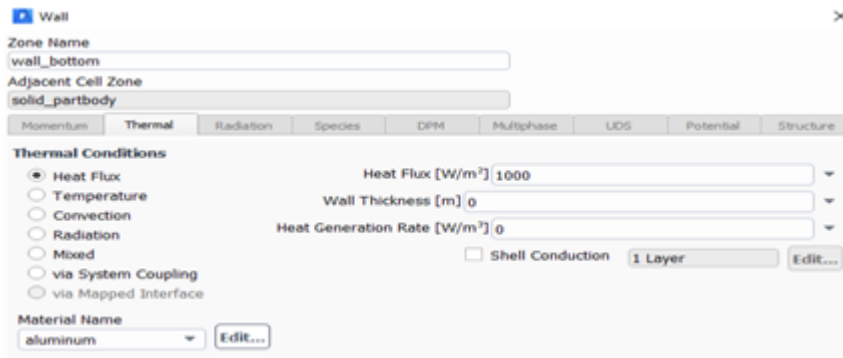


Figure 4.15: Boundary conditions settings

After that, we need to initialize the solution, and to do so we need to double click the “Initialization” tab, and then select “Standard Initialization”, and then select the inlet from the

“Compute from” section, and lastly click “Initialize”, and Figure 4.16 is showing the settings of the “Initialization” tab.

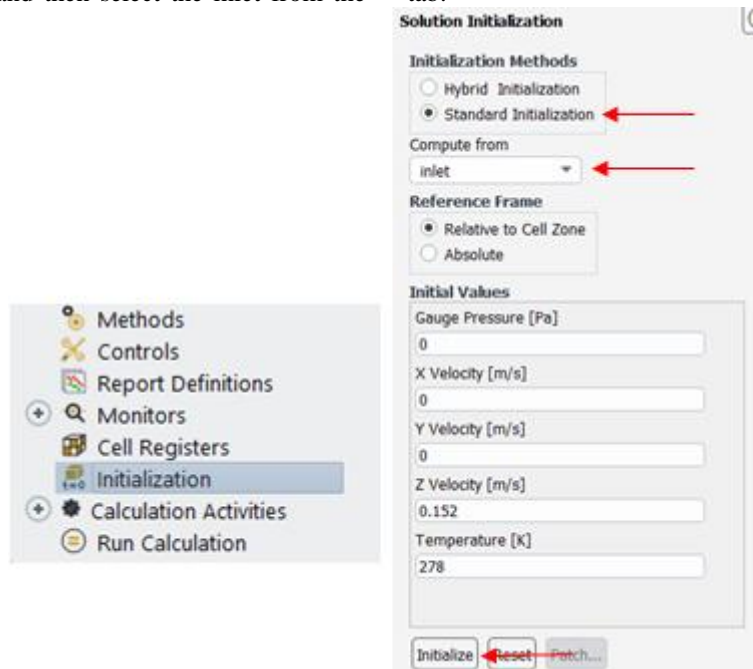


Figure 4.16: Initialization settings

And lastly in the setup part, we need to change the settings of the “Run Calculation” tab, and to do so need to double click “Run Calculation” and then change the number of

iterations to 350 so that we can get an accurate result, and Figure 4.17 is showing the settings of “Run Calculation” tab.

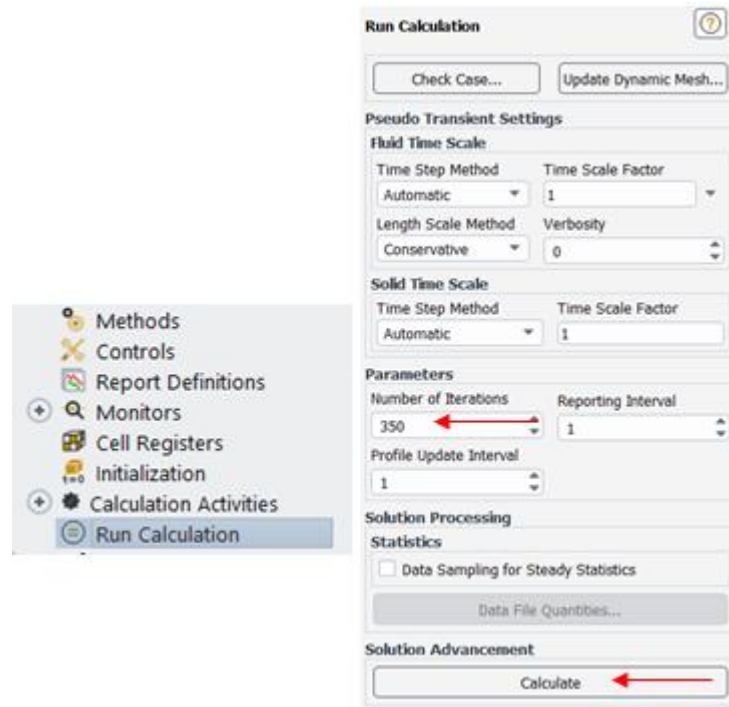


Figure 4.17: Run calculation settings

Then we can run the calculation by clicking “Calculate” as shown in Figure 4.17 to start getting results, and once the program starts calculating a graph will start generating, this graph will show the residuals vs. iteration. Figure 4.18 is showing graph after the program is done calculating the solutions, and as we can see from the figure in the x axis we

have Iterations and it goes until 350, that’s because we set it to 350 in “Run Calculation” tab, and as we can see from the graph as the number of iterations increases the graph starts to become a straight line, and this indicates that our solution is getting more accurate.

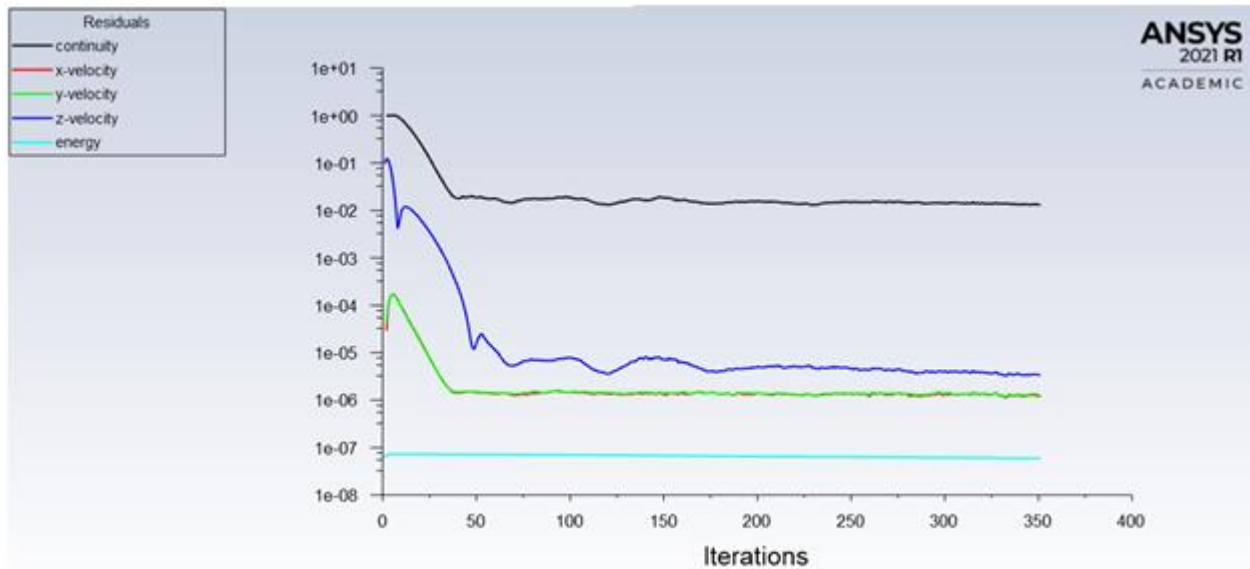


Figure 4.18: Residuals VS. Iteration graph

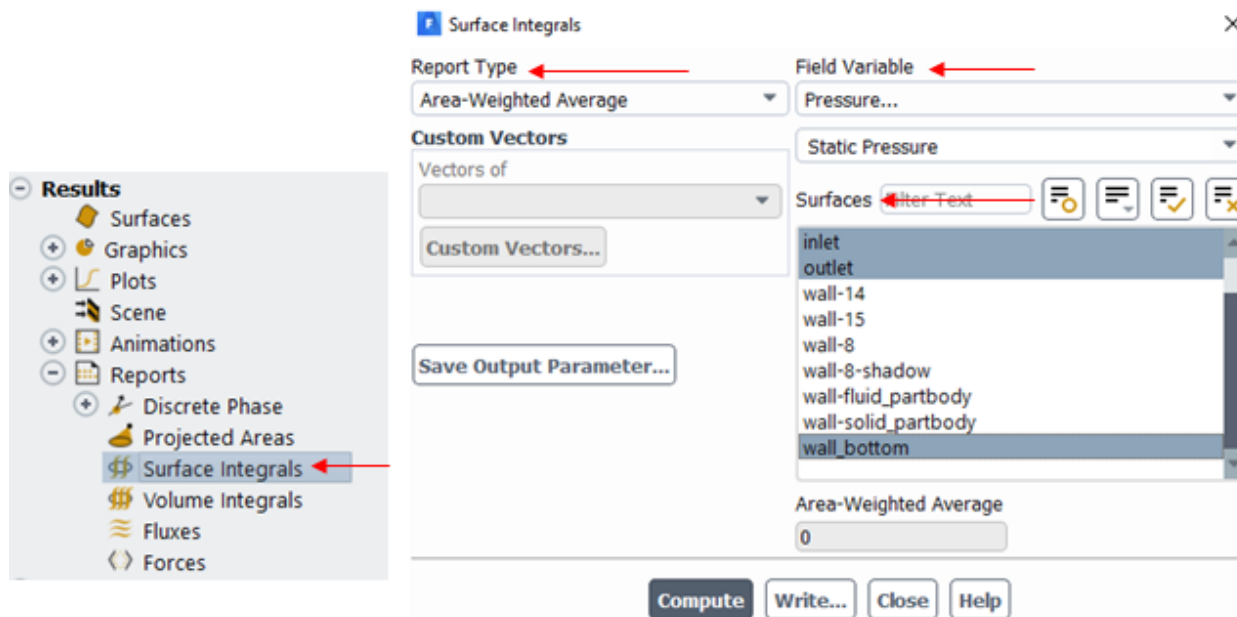
4.2.4 Solution

After we are done with the setup, and the program is finished calculating, now we are able to collect the data that we need from the solution part, the data that we are interested in and want to collect is the outlet temperature, the maximum temperature, the inlet pressure, the inlet and outlet mass flow rate. To do so we need double click “Surface integral” in the results section as shown in Figure 4.19 below, then we will get a pop window as shown in the figure

below. To find the outlet pressure we need to select “Area - Weighted - Average” from the report type section, and we need to select pressure from the field variable section, and also we need to select the outlet from surfaces section as shown in the figure below, and lastly we just need to press compute and then collect the data. Moreover, to find the other parameters that we are interested in we need to follow the same steps except we need to change small settings. For example, for the outlet temperature we need to

select “Mass - Weighted Average” from the report type section and select temperature from the field variable section, and then select the outlet from surfaces section, then to find the maximum temperature we need to select “Facet Maximum” from the report type section, and then select “wall_bottom” from the surfaces section. Lastly to find the inlet and outlet mass flow rate we need to select “Mass Flow Rate” from the report type section, and select the inlet and outlet from the surfaces section. Furthermore, after we are

done collecting the data we can now repeat all the ANSYS steps discussed in this section of the report, for Reynolds number ranging from 50 to 1500 to be in the range of the laminar flow. We are doing this so we can generate the graphs that we want after collecting the data for each Reynolds number, which are the change in the thermal resistance with the Reynolds number graph, and the change in the pumping power with the Reynolds number graph.



4.3 Channel designs

Our goal in using ANSYS Fluent software is to see the change in the thermal resistance with the Reynolds number, and to see the change in the pumping power with the Reynolds numbers. Based on the ANSYS steps. We are ready to achieve our goal, and we are going to do so by simulating different channel designs in ANSYS Fluent software, and then collecting their data, and after that we can generate the graphs. That will show us the change in the thermal resistance and the change in the pumping power with the Reynolds number.

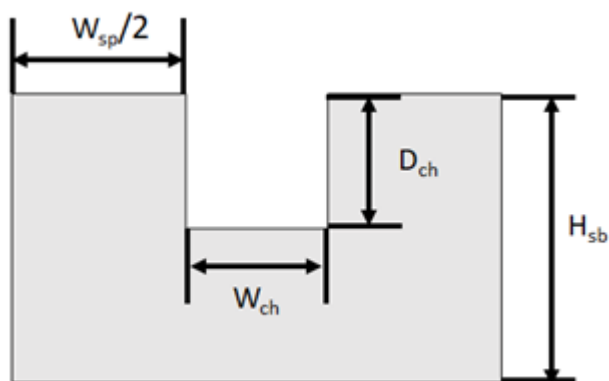


Figure 4.19: Surface integrals settings

Table 4.1: Channel designs

S. No	$W_{ch} = D_{ch}$	W_{sp}	H_{sb}	Re
1	0.5 mm	1 mm	5 mm	$50 \leq Re \leq 1500$
2	1 mm			
3	2 mm			
4	1 mm	0.5 mm	5 mm	
5		1 mm		
6		2 mm		
7	1 mm	1 mm	2.5	
8			5	
9			7.5	

Table 4.1 is showing nine different designs parameters that we chose in order to achieve our goal, where W_{ch} is the width of the channel, D_{ch} is the depth of the channel, W_{sp} is the spacing of the channel, H_{sb} is the thickness of the substrate, and Re is the Reynolds number. Moreover, as we can see from Table 4.1 design one to three only the width of the channel is changing the other parameters are fixed, this is to see how the width of the channel will change with graphs that we want to generate which are thermal resistance vs. Reynolds number graph, and the pumping power vs. the Reynolds number graph. And the Reynolds number is changing from 50 to 1500 with every data collection in every design from one to nine. Also, in design four to six the same thing is happening except this time the spacing of the channel is changing and the other parameters are fixed, and in design seven to nine the thickness of the substrate is changing and the other parameters are fixed.

4.4 Results

In Figure 4.20 above shows the relation between Reynolds number with values between 50 to 1500 with respect to the resulting thermal resistance. In respect of three different width and depth of the channel which are 0.5mm, 1mm and 2mm. We can observe that the thermal resistance will decrease exponentially as the Reynolds number increase which is directly proportional to the volume flow rate. The

reason behind this is as the mass flow rate increases the heat removed, will lead to a lower outlet temperature. Moreover, in the figure above there are three different values of the width and the depth of the channel, we can observe as the bigger the value of the width and depth the lower the thermal resistance. This is due that larger channel width and depth the larger the mass flow rate which leads to a lower thermal resistance.

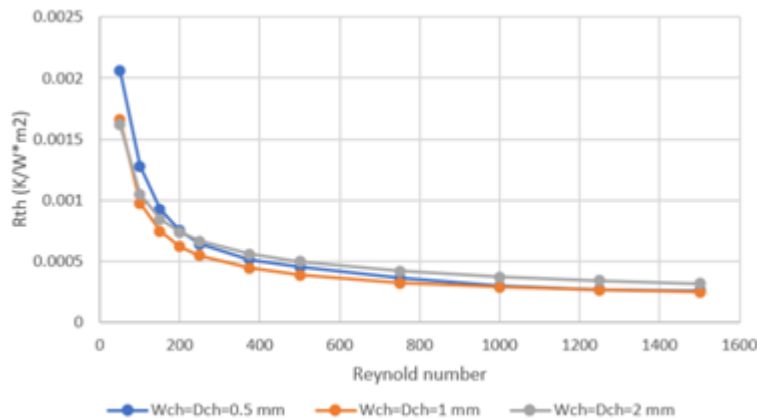


Figure 4.20: Thermal resistance and Reynolds number graph (changing width and depth of the channel)

In Figure 4.21 which shows the relation between Reynolds number with range of 50 to 1500 and the corresponding pumping power in milliwatt. In respect of three different width and depth of the channel which are 0.5mm, 1mm and 2mm. We can observe that as the Reynolds number increases which is proportional to the volume flow rate increases the pumping power will increase exponentially. This relation happens because as the flow rate increase the pressure drop increases exponentially So the required

pumping power will increase. Also, from the figure we have three different values of the width and the depth of the channel. We can observe as the width and the depth of the channel increases the pumping power will decrease. The reason behind this is because bigger values of width and depth will lead to a larger area this will decrease volume flow rate per channel resulting in a reduction of the required pumping power.

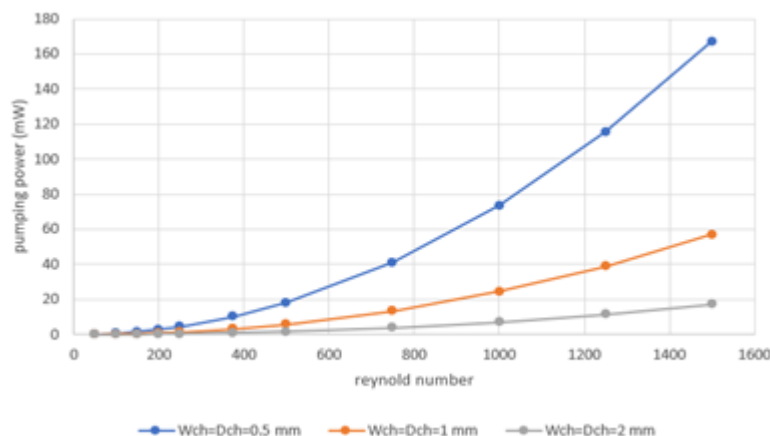


Figure 4.21: Pumping power and Reynolds number graph (changing width and depth of the channel)

In the Figure 4.22 above which shows the relation between Reynolds number with range of 50 to 1500 and the resulting thermal resistance with three different values of spacing widths which are 0.5mm, 1mm and 2mm. We can observe

that as the spacing width increase the thermal resistance will increase. The reason behind this is because as the space width increase the thermal resistance will increase.

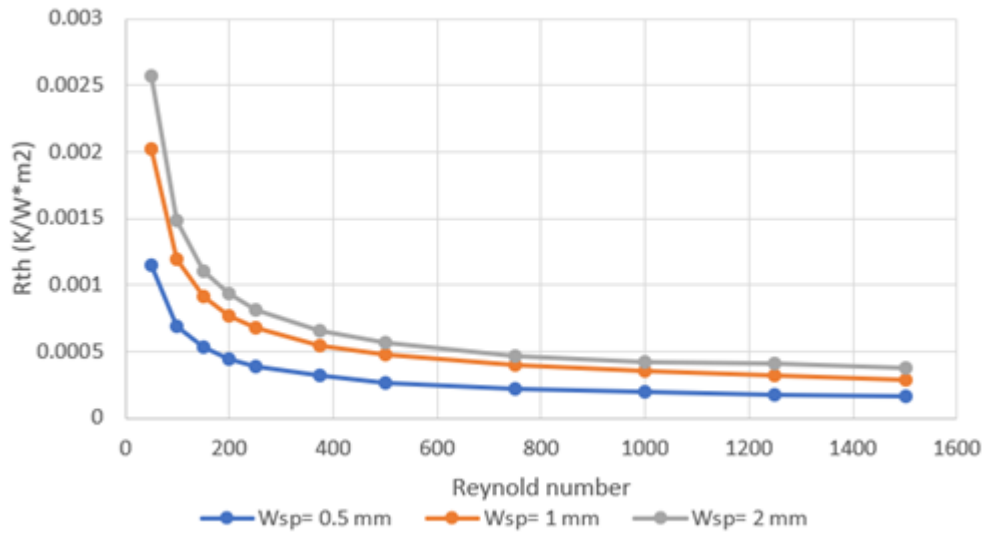


Figure 4.22: Thermal resistance and Reynold number graph (changing spacing width)

In the Figure 4.23 above which shows the relation between Reynold number with range of 50 to 1500 and the resulting Pumping power in (mW) with three different values of spacing widths which are 0.5mm, 1mm and 2mm. We can observe that as the spacing width increase the pumping

power will remain constant. The reason behind this is because as the space width increase or decrease the volume flow rate will remain constant therefore the pumping power will not be affected.

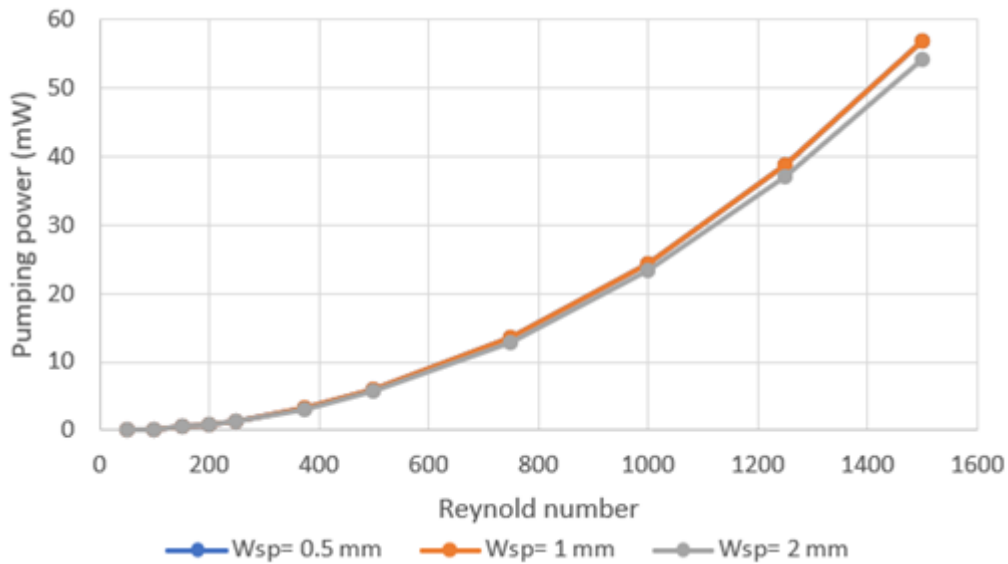


Figure 4.23: Pumping power and Reynold number graph (changing spacing width)

In Figure 4.24 above which shows the relation between Reynolds number with range of 50 to 1500 and the resulting thermal resistance with three different values of thickness of substrate which are 2.5mm, 5mm and 7.5mm. We can

observe from the figure above that all the three values have the same thermal resistance. This happens because the thermal resistance does not depend on the thickness of the substrate.

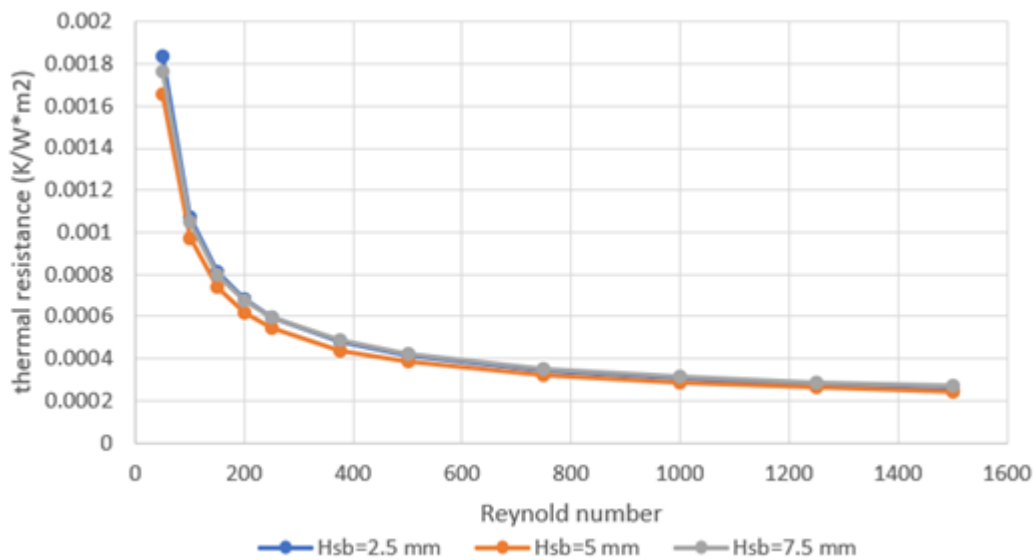


Figure 4.24: Thermal resistance and Reynold number graph (changing thickness of the substrate)

In the Figure 4.25 above which shows the relation between Reynold number with range of 50 to 1500 and the resulting Pumping Power in (mW) with three different values of thickness of substrate which are 2.5mm, 5mm and 7.5mm.

We can observe from the figure above that all the three values have the same Pumping Power. This happens because the pumping power does not depend on the thickness of the substrate.

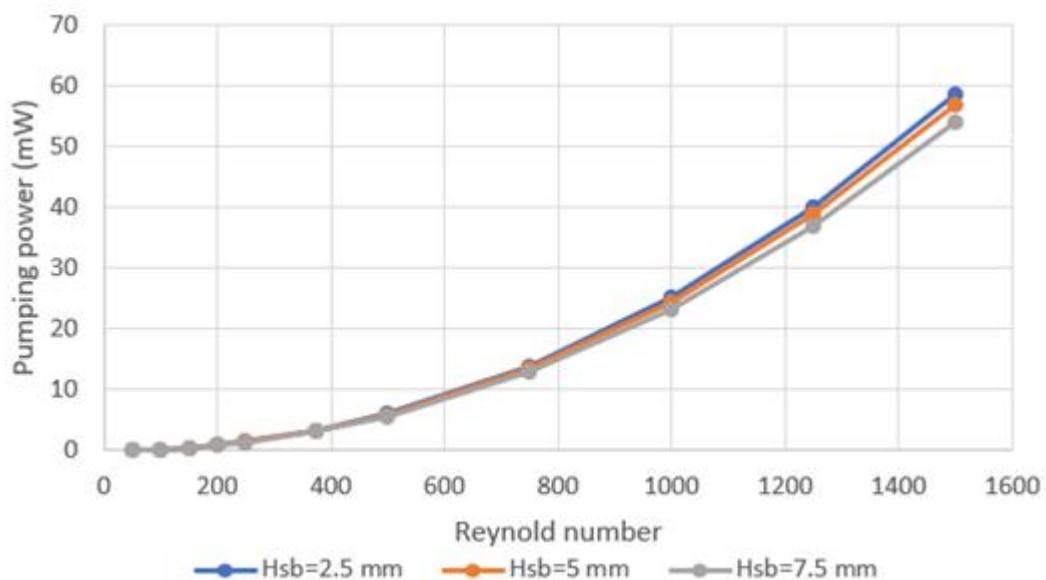


Figure 4.25: Pumping power and Reynold number graph (changing thickness of the substrate)

5. Ethical, Social, Environmental, Economical and Sustainability

In this part of the project we will be discussing how our project is subjected to these issues ethical, social, environmental, economical and sustainability.

5.1 Ethical Affairs

The ethical affairs in the engineering sector are various and are related to some fundamentals that all engineers are obliged and committed to following. We will discuss what are they and how they were dealt Within our project in case of the safety of the environment, public citizens and life also discussing some Moral prospects that engineers should perform.

5.1.1 Engineers shall at all times strive to serve the public interest.

As Engineers our goal from any research or a project is to help and aid the public, make life easy for them and invent or upgrade or even modify any idea that will help the engineering sector and the public for a bright future and developed life. In our project we tried to help the economic, environmental and public life illustrated in making sure that the heat sink will perform its job in a perfect form without harming anything and also our project will help the industries and companies who manufacture photovoltaic cells automotive industries, buildings, marine equipment's and much more will benefit from our project.

5.1.2 Engineers shall hold paramount the safety, health, and welfare of the public

At the time of designing our project safety was the most important factor that altered our final design. We have ensured and secured the safety of the environment in which we tried to minimize the land or water pollution and the emission of gases in the environment also handled the fouling of the heat sink during the process. Our project achieved this principle in making sure that our heat sink operates in an environmentally friendly form.

5.1.3 Engineers shall perform services only in the areas of their competence

As engineers, our expertise and integrity force us that we are only to perform in the area in which we are specialized. In this project as mechanical engineers we have followed this canon in which we worked on a heat exchanger which we have strong grip and background on the subjects related to this device by studying all the core topics of variables that will affect the heat sink or how it will perform in case of heat transfer and some other knowledge of how to manufacture or model the design.

5.1.4 Ethical dilemma

One of the things that we as engineers must take care of is to provide all the requirements of the customer and present them to him. It is our responsibility to inform the customer that the heat exchanger is thermally efficient, sustainable, and environmentally friendly. We have to make sure and verify that the heat exchanger has the ability to deliver high efficiency value and that it has a broad and significant impact on sustainability. In addition, we must focus and make sure that the construction, shape and design of the heat exchanger are able to work well in order to avoid and reduce pollution, high pressure drop and leakage, which also causes economic problems because it needs more pumping power and this thing is very expensive.

5.2 Safety Consideration

We must ensure that any designed system performs its duty in a safe and stable manner, even if this system fails. When we operate and handle our equipment, we handle liquids, heat and electricity, so taking caution and taking safety precautions is very important to reduce and prevent problems, accidents and hazardous situations. As a result, when the device's protection is designed, it is designed to avoid leakage and heat transfer outside of the channels. We also ensured that the fluids can safely exit the channels and into the tank, preventing the fluids and electrical equipment from colliding. Furthermore, since there are no contacts between the device and people during its manufacture, aluminum is a very safe way of manufacturing.

5.3 Social Impact

Social impact is mainly considered as the aspects that affect human life and social life. Also, it includes holding paramount the safety, health and welfare of the public. When designing our project, we have put into consideration not only the safety of the environment but the safety of the social life by making sure that the heat exchanger will not pollute the environment in a harmful way that will

eventually will lead to bad emissions of gasses. It is highly important to ensure and secure the protection of future generations air, water, and land quality, and to do so we should contribute as much as possible to sustainability. Due to the use of additive manufacturing, our heat exchanger will yield into using less resources which will lead to sustainability and reduction of emissions which will improve the environmental life and health of the people and stabilize the conditions of the climate finally it will improve the living conditions in which it is a great way in manufacturing.

5.4 Economics

Environmental sustainability refers to the desire to leave a legacy for future generations, not just the protection of our resources. We are obliged to reduce our emission of harmful gases and be reasonable of how much material we use. We believe in the efficient use of high - quality materials and the development of lifelong, high - quality goods. So, in comparison to current heat exchangers our design is quite small but highly efficient \that require less material, and require less operating energy, all of which result in lower emission of harmful gases to economic life.

5.5 Sustainability

There is a significant need for more efficient thermal management strategies for current and future energy systems in order to overcome the challenges associated with waste heat generation within energy systems. Innovative approaches to addressing thermal management issues within energy systems are introduced in this study via innovative heat storage systems for a wide range of end - use applications. This study aims to examine and compare traditional and innovative thermal management systems for various end - use applications while taking key aspects of sustainability into account. With this goal in mind, this study takes a smart approach to investigate existing and emerging thermal management systems, and smart targets for better thermal management systems are introduced. Furthermore, as case studies, some novel thermal management systems are discussed, and finally, future directions in terms of better thermal management options for a sustainable future are reviewed.

6. Selected Design Realization, Evaluation and Improvements

6.1 Fabrication

After we finished the designing our heat sink then we needed to fabricate each part, and to do so we needed to go to the lab in the University to fabricate all parts that was designed, however lab at that time we're closed due to coronavirus, so then we decided to send our design of the heat sink to a company and they will fabricate it for us, and to minimize the cost we only sent the designs for the top part and bottom part to the company for fabrication, and the other parts which are the bolt and the nut we decide to buy them separately, and Figure 6.1 showing the top part drawing and Figure 6.2 is showing the bottom part drawing. Moreover, to fabricate the top and bottom part of our heat sink the company need to use a milling machine, which is

one of the very common manufacturing processes used in machinery shops and industries to manufacture high precision products and parts in different shapes and sizes, and milling machine is machine in which the cutters rotate to remove the material from the work piece present in the direction of the angle with the tool axis, and Figure 6.3 is showing a sample image of a milling machine, and Figure 6.4 is showing the top part after fabrication and Figure 6.5 is showing the bottom part after fabrication.

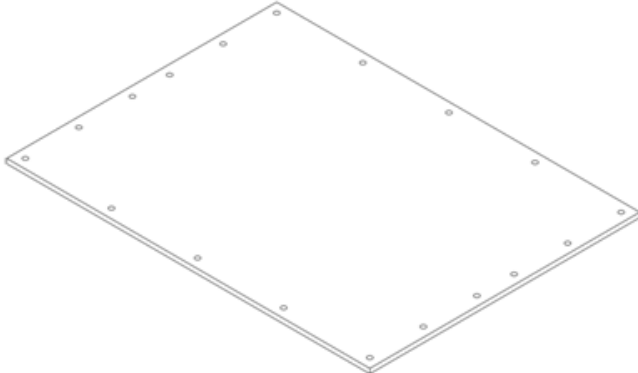


Figure 6.1: Top part drawing

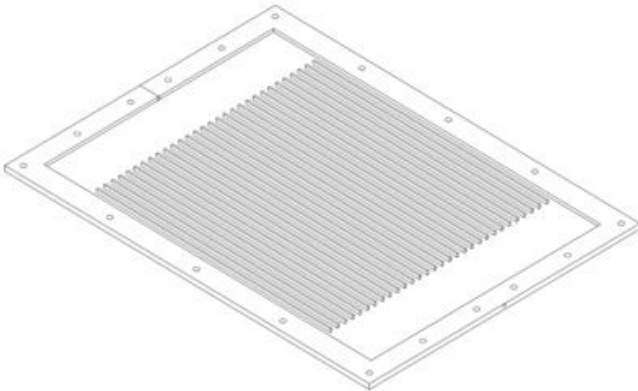


Figure 6.2: Bottom part drawing



Figure 6.3: Milling machine



Figure 6.4: Top part after fabrication

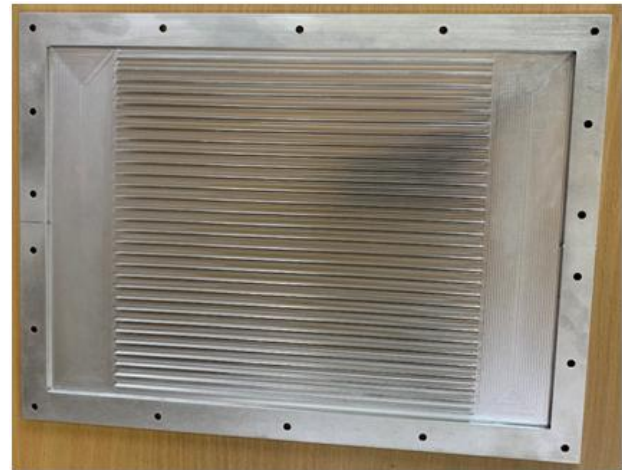


Figure 6.5: Bottom part after fabrication

6.2 Testing and Experimenting

6.2.1 Testing the heat sink

After we got our top and bottom parts from fabrication, we wanted to test our heat sink before we do the actual experiment to see if the heat sink is working or not, the first thing we noticed is that the inlet and outlet holes are 3 mm in diameter and we didn't find any pipes in this size to fit in the holes, so what we did took the top part and we threaded two holes that are diagonal to each other to install a pipe base in these two holes, and Figure 6.6 is showing the pipe base that will fit in the two threaded holes, and in this case we need to use a gun glue and glue the in the inlet and outlet holes to prevent any water leakage from these holes. After that, we wanted to connect the top part with the bottom but before we do that, we installed a plastic sheet between the top and bottom parts to prevent a water leakage from the sides, and Figure 6.7 is showing a sample image of the plastic sheet that were used. Furthermore, we connect all the heat sink parts together and we were ready to test the heat sink, and once we turned the pump on to let the water flow in the heat sink, we found that there is a water leakage from the sides and from the glued holes, so what we did after that we increased the amount of glue in the inlet and outlet holes to make it stronger, and we drilled three more holes from the sides to add more bolts and nuts, so in this way we can prevent the water leakage from the sides, and Figure 6.8 is showing our heat sink after we made the modification to it, and after that we tested the heat sink to see if there is still a water leakage or not, and we found that there is still a water

leakage from the glued holes but it's a very small amount, so we ignored it, and now we can start our experiment.



Figure 6.6: Pipe base

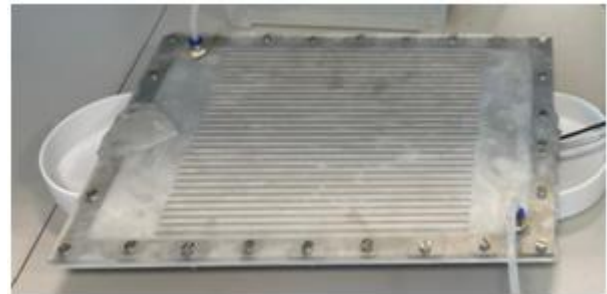


Figure 6.10: Heater

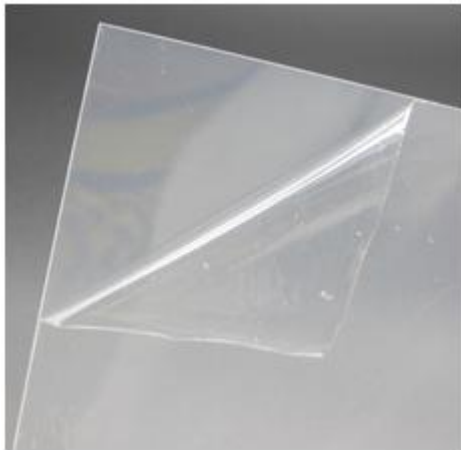


Figure 6.7: Plastic sheet

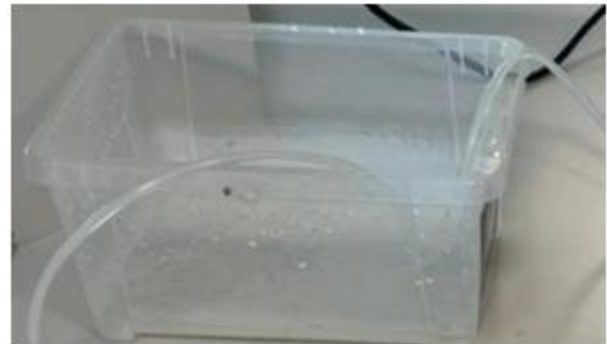


Figure 6.11: Water vessel

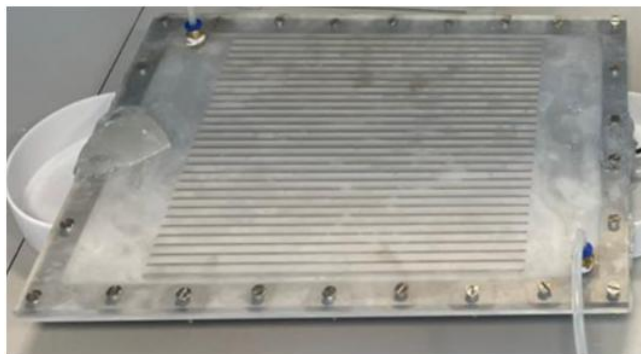


Figure 6.8: Heat sink after modifications



Figure 6.12: Pump

6.2.2 Apparatus

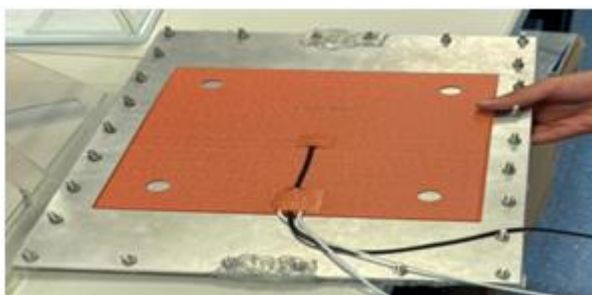


Figure 6.9: Heat sink



Figure 6.13: Beaker



Figure 6.14: Stop watch



Figure 6.15: Thermocouple

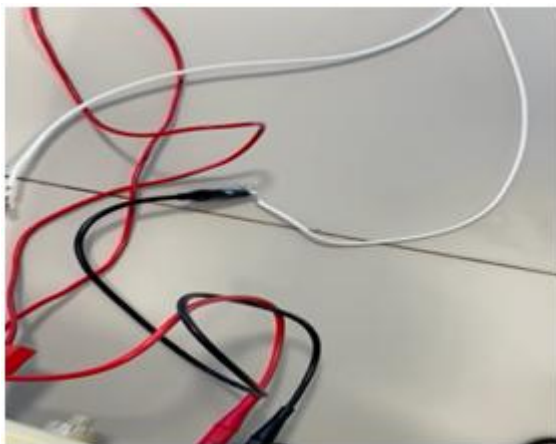


Figure 6.16: Connecting wires

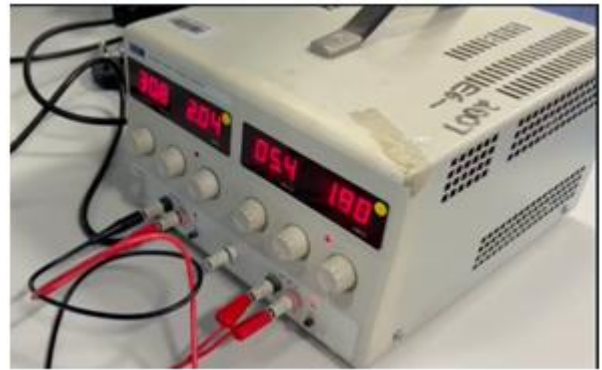


Figure 6.17: Power source



Figure 6.18: Glue gun



Figure 6.19: Tube connector

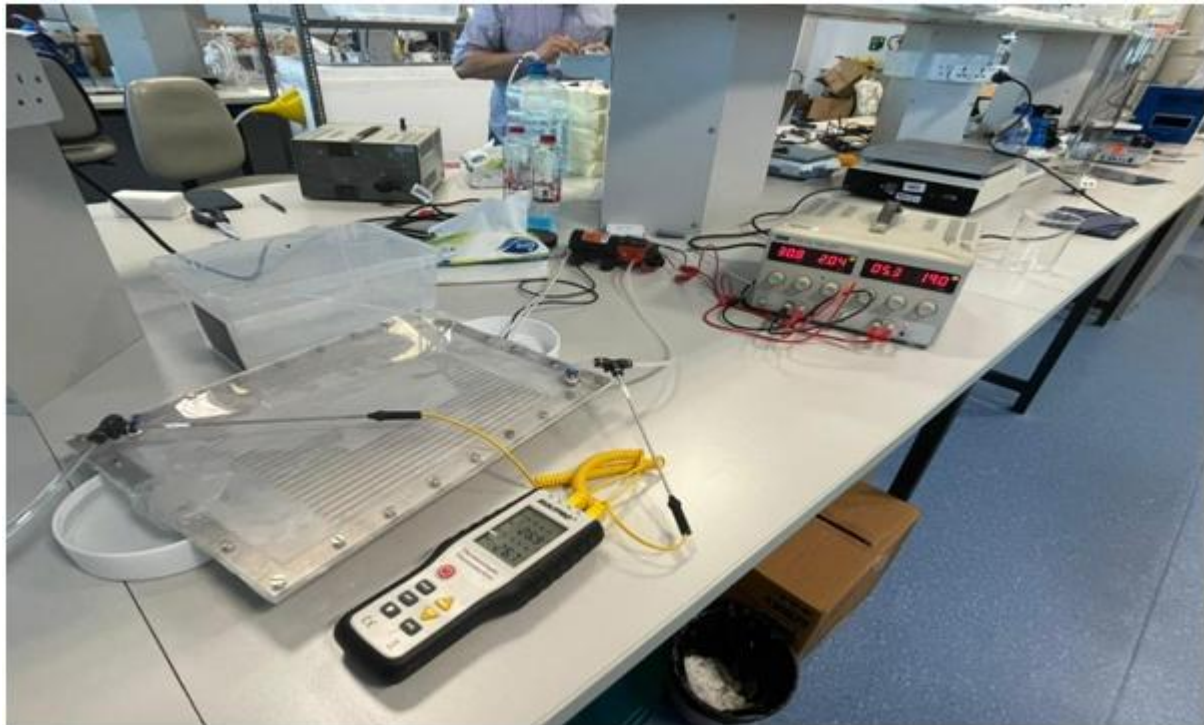


Figure 6.20: Experiment full setup

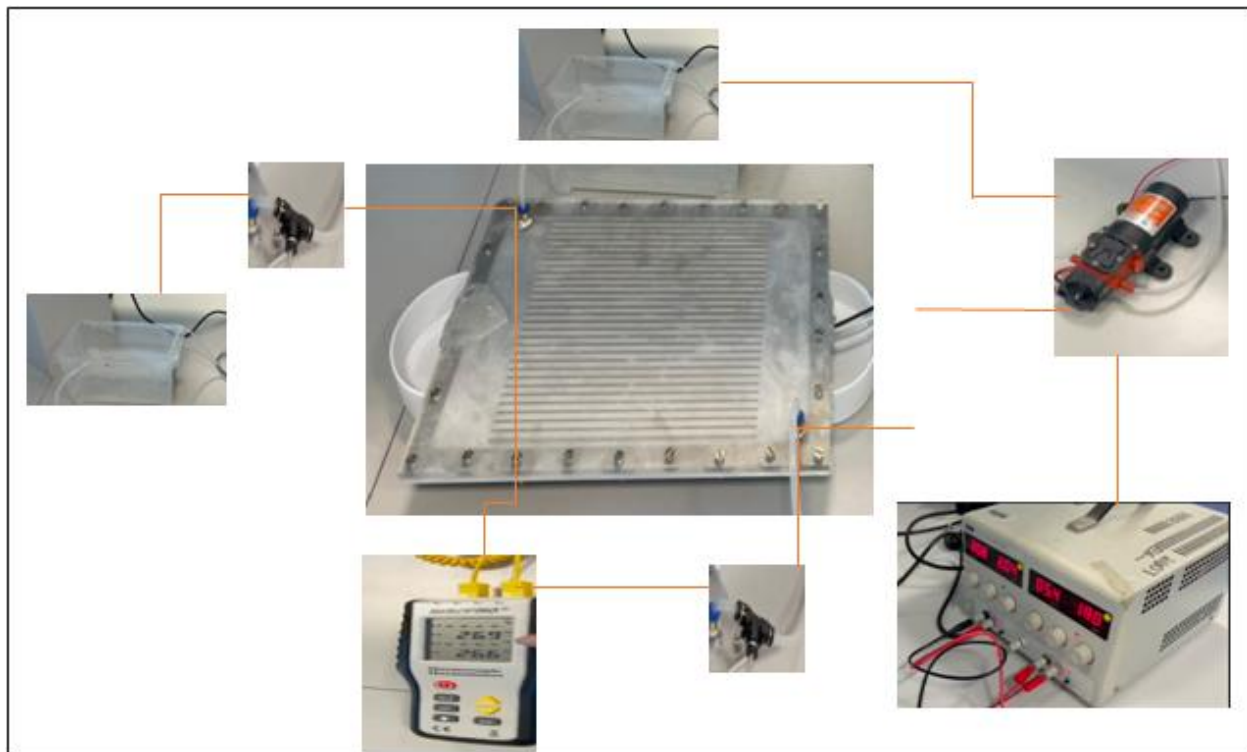


Figure 6.21: Connection diagram

6.2.3 Experiment procedure

First of all, we assigned the power source to give 4.2 V to the pump and the heater to be 30.8 V. After that the fluid was moving through the heat sink from the inlet to the output and in this time we waited for the thermocouple to give us steady value of temperature of both inlet and outlet and then recorded them in a table. These steps were repeated for four different readings in each reading we changed the power of the pump to five different values 3.2, 4.2, 5.5, and 7.8 V, which will change the flow rate of the fluid while keeping the power of the heater constant. Finally, the data

table was conducted and the mass flow rate was determined by setting up the beaker shown in Figure 6.13 to take some amount of water in a certain amount of time measure by the stopwatch as shown in Figure 6.14 which will yield the mass flow rate of the whole system.

6.3 Performance evaluation

The objective of this section of the report is to collect the data from our experiment so then we can analyze it, and as we can see in Table 6.1 below its showing the data

collection of our experiment, where V_{heater} is the heater voltage, I_{heater} is the heater current, V_{pump} is the pump voltage, I_{pump} is the pump current, T_1 is the inlet temperature, T_2 is the outlet temperature, m_{water} is the mass of the water inside the beaker, m_{beaker} is the mass of the beaker when its empty, and t is the time it took to fill the baker with water. Moreover, we were able to collect the data

for the V_{heater} , I_{heater} , V_{pump} , and I_{pump} with our power supply as their values would be the inputs to our experiment, and we were able to collect the data for the T_1 and T_2 using thermocouple as it's a device that can measure the temperature, and we were able to collect the data of m_{water} and m_{beaker} using a scale, and lastly we were able to measure the time using a stopwatch.

Table 6.1: Data collection

Reading	V_{heater} (V)	I_{heater} (A)	V_{pump} (V)	I_{pump} (A)	T_1 (°C)	T_2 (°C)	m_{water} (g)	m_{beaker} (g)	t (sec)
1	30.8	1.26	4.2	0.8	24.7	27.2	1228.31	984.61	30.24
2	30.8	1.26	3.2	0.85	24.7	26.9	1980.45	984.61	120
3	30.8	1.26	5.5	0.8	24.7	26.7	1189.96	984.61	30.35
4	30.8	1.26	7.8	0.8	24.8	26.3	1544.85	984.61	40.89

6.4 Data analysis and discussion

6.4.1 Data analysis

Equations used to find Reynold number:

$$\dot{m}_{water} = \frac{m_{water} - m_{beaker}}{time}$$

$$v = \frac{\dot{m}_{water}}{\rho A}$$

$$Re = \frac{\rho v L}{\mu}$$

Sample calculation (first reading):

$$\dot{m}_{water} = \frac{m_{water} - m_{beaker}}{time} = \frac{1.22831 - 0.98461}{30.24} = 0.00806 \text{ kg/s}$$

$$v = \frac{\dot{m}_{water}}{\rho A} = \frac{0.00806}{997 * \frac{\pi}{4} (0.006)^2} = 0.2859 \text{ m/s}$$

Table 6.2: Result of Reynolds number and temperature difference

Reading	RE	$T_2 - T_1$
1	68.51	2.5
2	70.55	2.2
3	57.51	2
4	116.47	1.6

Results

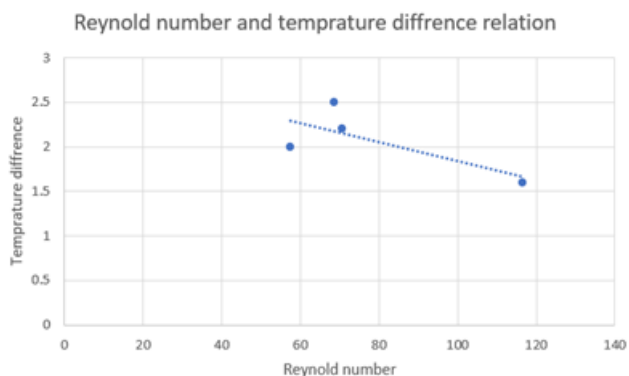


Figure 6.22: Reynolds number and temperature difference relation

6.4.2 Discussion

After we analyzed the data collected, we were able to calculate the Reynolds number for each reading, by first finding the mass flow rate of the water, and then using the calculated mass flow rate for each reading we were able to calculate the velocity for each reading, and from the velocity we were able to calculate the Reynolds number for each reading, also we calculated the temperature difference for each reading, and Table 6.2 is showing the r Reynolds number for each reading, and also showing the temperature difference for each reading. Moreover, using the data in Table 6.2 we were able to generate the graph shown in Figure 6.22, where this graph is showing the change of the temperature difference with the Reynolds number, and the graph is clearly showing as the Reynolds number increases then the temperature difference will decrease, and that's expected because as Reynolds number increases then the velocity of the water increases as well, and that will result in the decrease in the outlet temperature, which means that the temperature difference between the inlet temperature and the outlet temperature is decreasing, all in all having a higher Reynolds number will result in the more efficient cooling process than lower Reynolds number.

6.5 Sources of error and improvements

6.5.1 Sources of error

There is no experiment in this world that can be done with zero experimental error, however in every experiment we try to eliminate the error as much as possible, and that's what we did in our experiment, moreover the most important source of error in our experiment and played a big role in increasing the error was water leakage, although we tried to decrease the water leakage as much as possible still there was a small amount of water leakage which result in increasing the error, in addition to that the room temperature was not controlled, which means that the room temperature was different than the heat sink temperature, which will result in a significant change in the outlet temperature of the water, furthermore the power source of the experiment were not stable, which means that the voltage and current were slightly changing, and this will result in calculations error.

6.5.2 Improvement

After we have done the experiment, we realized that we need to modify our heat sink CAD design, in order to decrease the water leakage, moreover the modification that needs to be done are, the hight of the bottom and top parts were 5 mm,

so we should increase it to 10 mm in order to make a 6 mm hole in diameter for the inlet and outlet to fit our pipes, and the other modification that needs to be done for our CAD design is to increase the number of holes for the bolts, in this way we will prevent the water leakage from the sides.

7. Resources used during the project

Any project should be supported and done by using different resources because it provides an overview of everyone and everything involved in your project and to ensure that the project is done accurately and fully. Various of resources was used during the project in the whole period of the graduation projected and are listed below:

- Scholar articles and documents from the internet and online library.
- Cad model and different types of heat exchanger channels was generated using CATIA software.
- ANSYS work bench and MATLAB were used to simulate the heat exchanger and collect the data.
- Microsoft Teams was used to communicate with team members and social media applications.
- Microsoft office (Word, Excel, Power point, Outlook) was used to write the report.

References

- [1] Reddy, S. R., Ebadian, M. A., & Lin, C. (2015). A review of PV–T systems: Thermal management and efficiency with single phase cooling. *International Journal of Heat and Mass Transfer*.
- [2] R Zakharchenko (2004) Photovoltaic solar panel for a hybrid PV/thermal system
- [3] Russell, C. R., 1981. Optical concentrator and cooling system for photovoltaic cells. In: US.
- [4] Sahay Amit, V. K. Sethi, A. C. Tiwari Fabrication scheme, instrumentation scheme and testing of ground coupled central panel cooling system (GC - CPCS)
- [5] Alnaimat, F., AlHamad, I. M., & Mathew, B. (2021). Heat transfer intensification in MEMS two - fluid parallel flow heat exchangers by embedding pin fins in microchannels. *International Journal of Thermofluids*
- [6] J. A. Gotmare, D. S. Borkar, P. R. Hatwar, *Int. J. Adv. Manuf. Tech.*, (2015).
- [7] T. Nehari, M. Benlakam, D. Nehari, 60 (2016).
- [8] M. Chandrasekar, T. Senthilkumar, *Heat Mass Transfer*, (2016).
- [9] Du, D., Darkwa, J., & Kokogiannakis, G. (2013). Thermal management systems for Photovoltaics (PV) installations: A critical review. *Solar Energy*.
- [10] Al - Waeli, A. H., Sopian, K., Kazem, H. A., & Chaichan, M. T. (2017). Photovoltaic/Thermal (PV/T) systems: Status and future prospects. *Renewable and Sustainable Energy Reviews*.
- [11] Hamid, S. A., Othman, M. Y., Sopian, K., & Zaidi, S. H. (2014). An overview of photovoltaic thermal combination (PV/T combi) technology. *Renewable and Sustainable Energy Reviews*.
- [12] E. C. Kern Jr., M. C. Russel, Combined photovoltaic and thermal hybrid collector systems, in: *Proceedings of the 13th ISES Photovoltaic Specialists*, Washington, DC, USA, 1978, pp.1153 - 1157.
- [13] L. W. Florschuetz, *Sol. Energy* 22 (1979) 361 - 366.
- [14] P. Raghuraman, *J. Sol. Energy Eng.* 103 (1981) 291 - 298.
- [15] H. P. Thomas, S. I. Hayter, R. I. Martin, L. K. Pierce, PV and PV/Hybrid products for buildings, in: *Proceedings of the 16th European PV Solar Energy Conference*, Glasgow, UK, Vol. II, 2000, pp.1894 - 1897.
- [16] A. K. Bharava, H. P. Garg, R. K. Agrawal, *Energy Convers. Manage.* 31 (1991) 471 - 479.
- [17] Y. Tripanagnostopoulos, Th. Nousia, M. Souliotis, P. Yanoulis, *Sol. Energy* 72 (2002) 217 - 234.
- [18] H. A. Zondag, D. W. De Vries, W. G. Van Helden, et al., *Sol. Energy* 72 (2002) 113 - 128.
- [19] J. Gonzalez - Hernandez, J. F. Perez - Robles, A. Manzano - Ramirez, et al., *Appl. Phys. Lett.* 75 (1999) 3192 - 3194.
- [20] D. R. Lide (Ed.), *Handbook of Chemistry and Physics*, 80th ed., 1999 - 2000, CRC Press, Boca Raton, London.
- [21] T. V. Torchynska, *Optoelectron. Rev.* 6 (2) (1998) 121 - 130.
- [22] S. A. Kalogirou, Y. Tripanagnostopoulos, Hybrid PV/T solar systems for domestic hot water and electricity production, *Energy Convers. Manage.* 47 (2006) 3368–3382.
- [23] V. V. Tyagi, S. C. Kaushik, S. K. Tyagi, Advancement in solar photovoltaic/thermal (PV/T) hybrid collector technology, *Renewable Sustainable Energy Rev.* 16 (2012) 1383–1398.
- [24] T. T. Chow, G. N. Tiwari, C. Menezo, Hybrid solar: a review of photovoltaic and thermal power integration, *Int. J. Photoenergy* (2012) 307287.
- [25] K. Moradi, M. Ali Ebadian, C. Lin, A review of PV/T technologies: effects of control parameters, *Int. J. Heat Mass Transfer* 64 (2013) 483–500.
- [26] S. B. Riffat, E. Cuce, A review of hybrid photovoltaic/thermal collectors and systems, *Int. J. Low Carbon Technol.* 6 (2011) 212–241.
- [27] D. Pellicone, A. Ortega, M. del Valle, S. Schon, Simulation of two - phase flow and heat transfer in mini and micro - channels for concentrating photovoltaic cooling, in: *Proceedings of the ASME 5th International Conference on Energy Sustainability ES2011*, Washington, DC, USA, August 7–10, 2011.
- [28] A. Reeser, P. Wang, G. Hetsroni, A. Bar - Cohen, Energy efficient two - phase microcooler design for a concentrated photovoltaic triple junction cell, *J. Sol. Energy Eng.* 136 (2014).
- [29] H. G. Teo, P. S. Lee, M. N. A. Hawlader, An active cooling system for photovoltaic modules, *Appl. Energy* 90 (2012) 309–315.
- [30] Y. Tripanagnostopoulos, Th. Nousia, M. Souliotis, Test results for air cooled modified PV modules, in: *Proceedings of the 17th European PV Solar Energy Conference*, Munich, Germany, 2001, pp.2519–2522.