REPUBLIC OF TURKEY

ALTINBAS UNIVERSITY

Institute of Graduate Studies

Mechanical Engineering

NUMERICAL ANALYSIS OF THE INFLUENCE OF MATERIAL AND GEOMETRIC PARAMETERS ON THE THERMAL EFFICIENCY OF HEAT SINK

Muftah, Idris Abdulhadi Emhamed

Master of Science

Supervisor

Asst. Prof. KOC, Ibrahim

Istanbul, 2021

NUMERICAL ANALYSIS OF THE INFLUENCE OF MATERIAL AND GEOMETRIC PARAMETERS ON THE THERMAL EFFICIENCY OF HEAT SINK

by

Muftah, Idris Abdulhadi Emhamed

Mechanical Engineering

Submitted to the Graduate School of Science and Engineering

in partial fulfillment of the requirements for the degree of

Master of Science

ALTINBAŞ UNIVERSITY

2021

The thesis titled “NUMERICAL ANALYSIS OF THE INFLUENCE OF MATERIAL AND GEOMETRIC PARAMETERS ON THE THERMAL EFFICIENCY OF HEAT SINK” prepared and presented by “IDRIS ABDULHADI EMHAMED MUFTAH” **was accepted as** a Master of Science Thesis in Mechanical Engineering.

______________________________ ______________________________

Asst. Prof. Ibrahim KOC

Supervisor

Thesis Defense Jury Members:

Asst. Prof. Haydar UYANIK	College of Applied Sciences, KTO Karatay University	___________________
Asst. Prof. Serdar AY	Faculty of Engineering and Natural Sciences, Altinbas University	___________________
Asst. Prof. Ibrahim KOC	Faculty of Engineering and Natural Sciences, Altinbas University	___________________

I certify that this thesis satisfies all the requirements as a thesis for the degree of Master of Science.

Approval Date of Institute of Graduate Studies:

____/____/____

I hereby declare that all information in this document has been obtained and presented in accordance with academic rules and ethical conduct. I also declare that, as required by these rules and conduct, I have fully cited and referenced all material and results that are not original to this work.

Idris Abdulhadi Emhamed MUFTAH

Signature

Accepted by Altınbaş University Institute of Graduate Studies on the following date:

____/____/____

DEDICATION

I dedicate this humble research to my dear parents who have done everything in their power since my childhood to push me forward for educational attainment, as well as my beloved wife and children who have devoted their efforts to creating the appropriate environment and creating an encouraging atmosphere to continue research and analysis, as well as to every researcher looking forward to collecting information, which I tried as much as possible to collect as much as possible how much can be related to the topic of research.

ACKNOWLEDGEMENTS

I would like to extend my sincere thanks to Assistant Professor Dr. Ibrahim KOÇ, lecturer at ALTİNBAŞ University, Department of Mechanical Engineering, who was my supervisor during the writing of this research and who gave me the right direction by providing much correct advice, instructions, and information. I would also like to thank Assistant Professor Dr. Suleyman BAŞTÜRK, Head of the Mechanical Engineering Department at ALTİNBAŞ University, who in turn assisted immediately to solve all technical problems from laboratories and others. As well as thanks to all the friends who supported, encouraged me and gave me valuable advice and information.

ABSTRACT

NUMERICAL ANALYSIS OF THE INFLUENCE OF MATERIAL AND GEOMETRIC PARAMETERS ON THE THERMAL EFFICIENCY OF HEAT SINK

MUFTAH, Idris Abdulhadi Emhamed

M.Sc., Mechanical Engineering, Altınbaş University,

Supervisor: Asst. Prof .Dr. Ibrahim Koc

Date: 09/2021

Pages: (94)

The challenge for designers is to try to make electronic boards smaller. From this point of view, in this study, the size of a heat sink installed on a central processing unit for a computer that generates 100 W was reduced to 50 % by changing the type of metal as well as adding a modification to the geometric shape of the fin. A numerical investigation was conducted for the results of a practical experiment. Six different types of heat sinks, differing from each other in terms of fin shape, were designed using commercial design software, taking into consideration the uncomplicated design, ease of manufacture, and cost avoidance. The material of the heat sink was copper, all designs were of the same sizes, the comparison was conducted to evaluate the efficiency fairly. The conjugate method is used. After the computational fluid dynamics simulation tool was set under forced convection conditions, and the cooling air flow velocity was set in a laminar form, which was set to 1 m/s. The comparison was based on the pressure difference as well as the thermal resistance, it was shown that separated short angled plate fins were superior to the rest designs as it had the lowest thermal resistance 0.3610 ℃/W, but on the other hand, recorded the highest-pressure difference which was 5.6340 Pa, thus was the worst for the pressure difference. The results showed that the intermittent zigzag fins were superior in terms of pressure difference, which had the lowest value of 3.2040 Pa, but on the other hand, it was the worst in terms of thermal resistance, as it recorded the highest value 0.4720 ℃/W. Through all of the above, it can be concluded that the process of reducing the size of the heat sink to 50% was completed by changing the type of material as well as making some modifications to the design of the fin, and in addition to that, the geometric shape of the fin affects the performance of the heat sink where it was evident in each of the pressure difference as well as of thermal resistance factors.

Keywords: Fin geometry, Thermal resistance, Heat sink, Forced convection, Pressure drop.

TABLE OF CONTENTS

Pages

ABSTRACT .... **vii**

LIST OF TABLES .... **xi**

LIST OF FIGURES .... **xii**

LIST OF ABBREVIATIONS .... **xv**

LIST OF SYMBOLS .... **xvi**

1. INTRODUCTION **1**

1.1 HEAT TRANSFER PROCESS 2

1.1.1. Heat Transfer by Conduction 2

1.1.2. Heat Transfer by Convection 3

1.1.3. Heat Transfer by Radiation 4

1.2 THE COOLING PROCESS OF ELECTRONICS 4

1.3 THE HEATSINK 5

1.3.1 Passive Heat Sink 5

1.3.2 Active Heat Sink 5

1.3.3 Heatsink Material 6

1.3.4 The Plate-Fin Heat Sink 6

1.3.5 The Pin Fin Heat Sink 7

1.4 THE PECULIAR SHAPED HEAT SINK 7

1.5 CFD (COMPUTATIONAL FLUID DYNAMICS) 8

2. LITERATURE REVIEW **9**

.2.1 THERMAL RESISTANCE 10

2.2. MICROCHANNEL HEAT SINK 14

.2.3 HEAT SINK DESIGN CONSIDERATIONS 18

2.4. MAIN PURPOSES OF THE LITERATURE REVIEW 19

2.5. RESEARCH GOALS 19
3. METHODS AND APPROACH 21
3.1. EXTENDED SURFACES 21
3.2. CONTACTED SURFACES 24
3.3. METHODOLOGY 26
3.3.1. Conjugate Method 27
3.3.2. Modeling of The Fluid 27
3.4. THE REYNOLDS NUMBER 28
3.5. EQUATIONS OF GOVERNANCE 29
3.6. THE FLUID FLOW THROUGH THE FINS 31
3.7. VALIDATION CASE 31
3.8. CAD MODEL CREATION 36
3.9. ANSYS FLUENT VALIDATION CASE SIMULATION 37
3.10. THE RESULT OF THE VALIDATION CASE SIMULATION 41
3.11. ASSUMPTIONS OF SIMULATION AND CREATION OF THE HEAT SINK 43
.3.12 SAMPLE TEST DIMENSIONS 44
3.13. COMPARISON METHODOLOGY 45
.4 RESULTS AND DISCUSSION 46
4.1. THE RESULTS OF TEMPERATURE EFFECTS 46
4.2. THE RESULTS OF PRESSURE EFFECTS 47
4.3. THE RESULTS OF SIMULATED SAMPLES 48
4.3.1. The CAD of Ordinary Rectangular Plate Fins Heat Sink and Analyzing Results 48
4.3.2. The CAD of Separated Short Plate Fins Heat Sink and Analyzing Results 52
4.3.3. The CAD of Separated Short Angled Plate Fins Heat Sink and Analyzing Results 56
4.3.4. The CAD of the Separated Short Airfoil Fins Heat Sink and Analyzing Results 60
4.3.5. The CAD of The Continuous Zigzag Fins Heat Sink and Analyzing Results 64
4.3.6. The CAD of The Intermittent Zigzag Fins Heat Sink and Analyzing Results 68
4.4 THE DISCUSSION OF RESULTS 72
5. CONCLUSION 75

FUTURE WORK **77**

REFERENCES **78**

LIST OF TABLES

Pages

Table 3.1: The fluid (Air) properties........ 27

Table 3.2: Dimensions of validation case heat sink[41]........ 34

Table 3.3: Validation case domain physics........ 38

Table 3.4: Validation case mesh information........ 39

Table 3.5: The sufficient mesh number........ 40

Table 3.6: The heat sink material (copper) properties........ 43

Table 3.7: The boundary conditions and initials........ 44

Table 3.8: The dimensions of the sample heat sink........ 45

Table 4.1: The comparison between all heat sink types results........ 73

LIST OF FIGURES

Pages

Figure 2.1: Microchannel heat sink schematic diagram [30] 15

Figure 2.2: Diagram of (a) ordinary plate-fin heat sink and (b) cross fin heat sink [8] 17

Figure 3.1: Normal surface (a) and extended surface (b)[38] 22

Figure 3.2: Illustrates temperature distribution and thermal resistances 25

Figure 3.3: The wind tunnel which is used in the experiment case[42] 33

Figure 3.4: CAD model of the validation case heat sink[42] 34

Figure 3.5: The outcomes of the validation experiment[42] 35

Figure 3.6: Airflow scheme active heat sink[42] 36

Figure 3.7: The validation heat sink surrounded by the fluid 38

Figure 3.8: Validation case boundary conditions 40

Figure 3.9: The comparison between the experimental and numerical studies 41

Figure 3.10: Temperature gradient of the validation sample 42

Figure 3.11: Temperature gradient through heat sink validation case 42

Figure 4.1: The CAD design of rectangular plate fins heat sink 48

Figure 4.2: The result of temperature through the rectangular plate-fin heat sink 49

Figure 4.3: The temperature distribution of rectangular plate fin 50

Figure 4.4: The pressure differences result from the rectangular plate fin 51

Figure 4.5: Velocity path through the rectangular plate-fin heat sink 52

Figure 4.6: The CAD design of separated short plate fins heat sink 53

Figure 4.7: The result of temperature through separated short plate fins 54

Figure 4.8: Temperature distribution through separated short plate fins 54

Figure 4.9: The result of pressure differences through separated short plate fins 55

Figure 4.10: Velocity stream through separated short plate fins 56

Figure 4.11: The CAD design of separated short angled plate fins heat sink 57

Figure 4.12: The result of temperature through the separated short angled plate fins heat sink .. 58

Figure 4.13: Temperature distribution through separated short angled plate fins 58

Figure 4.14: Pressure result through separated short angled plate fins heat sink 59

Figure 4.15: Velocity stream through separated short angled plate fins heat sink 60

Figure 4.16: The CAD design of separated short airfoil fins heat sink 61

Figure 4.17: The result of temperature through the separated short airfoil fins heat sink 62

Figure 4.18: Temperature distribution through separated short airfoil fins 62

Figure 4.19: Pressure result through separated short airfoil fins heat sink 63

Figure 4.20: Velocity stream through separated short airfoil fins heat sink 64

Figure 4.21: The CAD design of continuous zigzag fins heat sink 65

Figure 4.22: The temperature result of continuous zigzag fins 66

Figure 4.23: The temperature distribution through the continuous zigzag fins heat sink............ 66

Figure 4.24: Pressure result through the continuous zigzag fins heat sink.................................. 67

Figure 4.25: The velocity stream through the continuous zigzag fins heat sink.......................... 68

Figure 4.26: The CAD design of intermittent zigzag fins heat sink 69

Figure 4.27: The temperature result of intermittent zigzag fins 70

Figure 4.28: The temperature distribution through the intermittent zigzag fins heat sink 71

Figure 4.29: Pressure result through the intermittent zigzag fins heat sink................................. 71

Figure 4.30: The velocity stream through the intermittent zigzag fins heat sink 72

Figure 4.31: Thermal resistance comparison.. 73

Figure 4.32: Pressure drop comparison 74

LIST OF ABBREVIATIONS

CPU	:	Central processing unit
CFD	:	Computational fluid dynamics
MAPE	:	Mean absolute procedure error standard
CAD	:	Computer-aided design
3D	:	Three dimensional

LIST OF SYMBOLS

q″	:	Heat flux	q_f	:	Fin heat rate
K	:	Thermal conductivity	θ_b	:	Fin base
$\frac{dT}{dX}$	:	Temperature gradient	$\mathcal{E}_f$	:	Fin effectiveness
T_s	:	Surface temperature	$R_{t.f}$	:	Fin thermal resistance
T_∞	:	Fluid temperature	η_f	:	Fin efficiency
Ɛ	:	Emissivity of surface	A_f	:	Fin surface area
σ	:	Stefan Boltzmann constant	ρ	:	Density
T_{surr}	:	Surrounding surface temperature	C_P	:	Specific heat
R_{th}	:	Thermal resistance	µ	:	Viscosity
T_{base}	:	Base temperature	D_h	:	Hydraulic diameter
Q	:	Heat amount	A	:	Area
Re	:	Reynolds number	▽	:	Nabla
V_∞	:	Fluid velocity	H	:	Heat sink height
υ	:	Kinematic viscosity	h	:	Fin height, enthalpy
θ	:	Temperature excess	W	:	Heat sink width
m^2	:	Parameter of fin	L	:	Heat sink length

A_c : Across section area

P : Fin perimeter

t : Fin thickness

b : Base thickness

l : Fin length

N : Fin number

I : Tensor unit

E : Energy total

J : Flux diffusion

S_h : Volumetric heat

ΔP : Pressure drop

P_{in} : Pressure inlet

P_{out} : Pressure outlet

g : Channel width

$\bar{\bar{\tau}}$: Stress tensor

$\overrightarrow{\rho g}$: Gravitational body force

$\overrightarrow{F}$: External forces

$R_{t,b}$: Thermal resistance of fin base

f_{app} : Apparent friction

K_c : Contraction coefficient

K_e : Expansion coefficient

V_{ch} : Velocity of channel

V_{ap} : Approach velocity

A_f : Frontal area

A_{tf} : Fluid flow through area

1. INTRODUCTION

In light of the rapid development in all industries such as food, health, military, and space sciences, the total dependence on the development of these industries is based on electronic devices such as computers and other electronic devices. Most modern devices contain electronic panels including many electronic elements, for example in military industries such as radars, smart weapons, missiles, and aircraft navigation devices. All of these devices have electronic panels with many units that generate heat while working, so it is necessary to cool these pieces to work efficiently. However, in the current development in terms of many designers seeking to design small electronic parts to reduce both size and weight for electronic devices, another challenge has emerged for designers in terms of cooling these parts.

One of the most well-known electronic pieces in the process of generating heat is the computer central processor unit (CPU), where many studies have shown that the heat generated in electronic parts during their work must be within certain limits, since if they rise too much may cause either inaccuracy in the work of the piece or electronic chip, such as the process of reading data and others, or may cause collapse and burning and thus lead to damage to the electronic panel or device completely resulting in material losses[1].

There has been previous research has shown that the number of fins harms the fluid flow rate because it increases the pressure drop, but on the other hand, it increases the surface area of the heat sink which has a positive role in heat dissipation. The study concluded that if the fin height increases, the number of fins decreases[2]. In a previous study focusing on the speed of flow, they found that the rectangular plate-fin heat sink outperformed the cylindrical pin-fin heat sink in terms of the thermal resistance factor, it proved that thermal resistance was 30% less in plate fins comparing to pin-fin[3]. Some previous experiences using computational fluid dynamics (CFD) have extracted abundant and very detailed information about the performance of the heat sink[4]. Another research using CFD (ANSYS FLUENT) compared four different types of a heat sink that were used to cool some computers, and the study concluded that the distance between the fins, or what is known as the pitch, as well as the surface area, have a clear effect on the performance of the heat sink[5]. Also, using CFDs, another research was found through thermal analysis during a comparison between a rectangular plate-fin heat sink and cylindrical pin fin heat sink, both of the same dimensions and under the same conditions, surpassed the rectangular plate fin to the cylindrical pin fin, because it achieved less thermal resistance, and that is clearly because of its large area[6].

There are many methods to cool electronics, one of these methods is using a heat sink. The process of cooling electronic parts depends on the principle of two ideas, either free cooling or forced cooling, through the use of a heatsink. To understand the working principle of the heatsink, it is necessary to touch upon the basic idea of its working principle, which is the heat transfer process.

1.1 HEAT TRANSFER PROCESS

The heat transfer process is a phenomenon that occurs when there is a difference in temperature between two mediums so that the heat transfers from the higher temperature to the less hot medium through three methods of the heat transfer process, which are conduction, convection, and radiation.

1.1.1. Heat Transfer by Conduction

In this process, heat is transferred between the solid bodies from the higher temperature body to the lower temperature body by the vibration of the solid body atoms, so the heat passes through the body according to the temperature gradient, until it is transferred to the adjacent body by the same principle. But this process depends mainly on the thermal conductivity of the body, so if the thermal conductivity is high, the heat transfers inside the body very fast and effectively. Otherwise, if the conductivity is low, then the heat transfer process is negatively affected, and thus the thermal insulation appears. All calculations for this process are governed by Fourier's law, by the following equation[7]:

$$q^{"} = -k\frac{dT}{dx} \tag{1.1}$$

Where $q^{"}$ is the heat flux (W/m^2), k is the thermal conductivity of the body (W/m·K) and $\frac{dT}{dX}$ is the temperature gradient.

The method of heat transfer by conduction plays an important role in the work of the heat sink, where the heat is transferred by conduction from the electronic part to be cooled to the base of the heat sink adjacent to it as a first stage, then the heat is transferred by the same principle from the base to the fins as a second stage.

1.1.2. Heat Transfer by Convection

In this case, the heat transfer is between the fluids, such as between liquids, gases, or a liquid and a solid body or gas and a solid body, where the heat is transferred from the higher temperature medium to the lower temperature medium, through the density (density variation) or in other meaning by the movement of the fluid molecules. Calculations for this process are governed by Newton's law of cooling and through the following equation[7]:

$$q^{"} = h(T_s - T_\infty) \tag{1.2}$$

Where $q^{"}$ is convection heat flux (W/m^2), h is convection heat transfer coefficient (W/m^2·°C), T_s is surface temperature (°C or K) and T_∞ is fluid temperature (°C or K).

The method of heat transfer by convection is shown in two forms, either forced convection heat transfer, which requires the presence of an external influence such as a pump or fan to direct the flow of liquid or air directly to the fins of the heat sink or the other form is free convection heat transfer, as this type does not require the presence of an external influence like the previous one. Moreover, the convection heat transfer process has an effective and important role in the work of the heat sink too, so after the heat is transferred by conduction from the base of the heat sink to the fins, the role of the fins comes in dissipating the heat to the surrounding medium, whether it is liquid or gas, where the principle of heat transfer by convection is evident.

1.1.3. Heat Transfer by Radiation

In this case, the heat transfer is done, but the presence of a medium is not required, which is known as the transfer of heat by radiation, as this is done through electromagnetic waves. This phenomenon is subject to Stephen-Boltzmann's law in terms of calculations and through the following equation[7]:

$$q^{"} = \varepsilon\sigma(T_s^4 - T_{sur}^4) \tag{1.3}$$

Where $q^{"}$ is the heat flux (W/m^2), $\mathcal{E}$ is the emissivity of the surface (no units), σ is Stefan-Boltzmann constant = 5.67×10^{-8} (W·m^{-2}·K^{-4}), T_s is surface temperature (K) and T_{sur} is surrounding surfaces temperature (K).

1.2 THE COOLING PROCESS OF ELECTRONICS

Most electronic devices generate heat working through some of the parts inside them, and thus this heat poses a great threat to electronic parts, which may affect their performance. From this standpoint, designers have devoted their efforts to get rid of this generated heat by innovating some technologies for cooling, so that these devices can work safely and effectively, so the heat sink is the most famous and most used of these technologies. It can be classified according to the way it works and the idea of heat transfer through it, either the heat transfer through it is free or forced.

1.3 THE HEATSINK

It is a device used to cool the electronics that generate heat while working, to protect them, to operate safely and efficiently. Where the principle of its work is based on absorbing heat from the part to be cooled and spreading it to the surrounding medium. There are two types of the heatsink:

1.3.1 Passive Heat Sink

In this type, the cooling process is without the presence of any external influences such as the presence of a fan or a pump (which is known as free cooling), so the principle of heat transfer is among the three methods mentioned above.

1.3.2 Active Heat Sink

In this type, the cooling process requires an external influence such as the presence of a fan or a pump (which is known as forced cooling). Here, two phenomena of heat transfer processes are evident, which are heat transfer by convection and heat transfer by conduction. Where previous researches have shown that when forced cooling is present, the transfer of heat by radiation is of little value so that it can be neglected[4].

1.3.3 Heatsink Material

As mentioned earlier, the idea of the working principle of the heat sink depends on the thermal conductivity of the material, as most of the widely used and famous heat sinks are made of aluminum and copper.

When coming to aluminum, it is cheap, light in weight, easy to shape, and has good thermal conductivity reaching k = 237 W/m·K. While copper is more expensive in terms of price, heavier in weight, also easy to form, but it is superior to aluminum in terms of thermal conductivity where k = 401 W/m·K.

One of the most popular and widely used forms of the heat sink is the plate-fin heat sink and the pin fin heat sink, due to their ease of manufacture and efficiency in cooling.

1.3.4 The Plate-Fin Heat Sink

It is a form of a heat sink and its name is inspired by its shape, as it is a flat base of thickness as required, and the fins emerge from it upwards, so that they are perpendicular to this base and the fins are in the form of sheets. These plates can also be of various geometric shapes, such as the rectangular, which is the most popular being, the trapezoidal, and the triangular. The plate-fin heat sink is the most widely used in cooling electronic parts due to several factors, including its cheapness, its ease of manufacture, and its efficiency in cooling, as it covers the largest area of the part to be cooled from the direction of the base, and it can dissipate heat efficiently, due to its large surface area, and that by having a large area of contact between it and the surrounding medium. Taking into account concerning the direction flow of the refrigerant fluid (inlet and outlet) in the case of forced consideration cooling[8].

1.3.5 The Pin Fin Heat Sink

It is also widely used and it is a type of heat sink consisting of a base and fins, but it differs from the previous type through the fins, as the fins emerge from the base in the form of pins, and in certain numbers, so that increasing the number of these pins increases the surface area of the heat sink. And where the location of these pins on the base is coordinated in two forms, either in a regular alignment or in a staggered form, and the process of choosing the coordination of the location of the pins is due to the design process to enhance the fluid flow process. These pins can take many different geometric shapes, such as square, cylindrical, triangle, etc., to increase the cooling efficiency. What distinguishes this type is the lack of compliance with the direction of the coolant flow, except in some special designs of the pin shape to increase the performance efficiency[9].

1.4 THE PECULIAR SHAPED HEAT SINK

The technological progress in the various fields of the industry and the latest development through modern innovative technologies, has helped many designers as well as manufacturers to create some strange geometric shapes and may be complex at other times, to push the research and development process to solve the problems and obstacles facing process to improve heat sink performance. Through the results of previous experiments and research, it has been proven that the geometry of the heat sink, which is the shape of the fin, affects its thermal performance.

In this research, several different geometric shapes of the heat sink are studied and analyzed under the same conditions, so that their results are compared with each other to find out their thermal performance to determine the most efficient among them. Drawing on the knowledge of the heat transfer process and fluid dynamics, peculiarly shaped heat sinks were designed in this study to improve heat sink performance.

1.5 CFD (COMPUTATIONAL FLUID DYNAMICS)

It is a software package used by a computer to simulate the experiments to be worked on or developed. Like (ANSYS FLUENT), and other programs, it is efficient and quick to show results if used correctly. The extensive use of CFD for experiments as well as the required development processes made it the most suitable tool for experimenting and simulation as it is fast, cheap, time-saving, and cost-effective. In addition to its ability to analyze fluid movement in terms of flow speed and friction as well as the type of flow being laminar, transition, or turbulent, also to its ability to analyze strains and stresses in solid objects, the (ANSYS FLUENT) package is also highly efficient in thermal analysis, both solid and fluid. That's why (ANSYS FLUENT) was selected to conduct the necessary investigation to develop and enhance the thermal capabilities of the heat sink in this research.

2. LITERATURE REVIEW

In this chapter, some of the previous relevant researches are reviewed in-depth to the study and analysis of the possibility of improving the performance of the heat sink, as well as all the results obtained on the subject of heat transfer.

Our almost dependent on electronic devices in performing a lot of our tasks has made it a permanent goal of development, meeting all needs, which has led designers to harness their efforts to operate safely and effectively[6]. One of the most common methods used to cool electronic parts is a heat sink and it is for several reasons, that it is cheap, easy to manufacture, and efficient in cooling.

An experiment was conducted with the use of the simulation method to evaluate the heat sink performance installed on the CPU of a computer, to dissipate 115 W. Copper oxide (CuO) nanofluid and water has been used as a coolant. The results showed that the experiment and the simulation method were identical. Also, water is considered to be very sufficient if used in the cooling process[10].

The heat sink's cooling efficiency was studied through a mini channel. Air and water were used together as a coolant in this channel. The idea was to make this channel so that the air passes from the outside and the fins are interspersed, while the water flows inside it. They found that the *Nu* decreases as the ratio between the height and width of the channel decreases. But on the other hand, the pressure drop increases. It is known that any increase in *Re* is also accompanied by an increase in *Nu*[11]. Also, through previous research similar but with the hypothesis that the surface temperature is as constant as the boundary conditions, intending to enhance the best performance evaluation of heat sink. However, the results were not satisfactory or unpromising, due to the difficulty of the hypothesis that the surface temperature was fixed[12].

Another study was conducted using a CFD computer software program to evaluate and analyze the performance of four different types of heat sink, installed on four different types of computers. The simulation results were verified by comparing them to the results of the experiment. The temperature for the fields as well as the flow of the heat sink, and at different values of *Re* have been analyzed. They have found that Pentium IV is the best and that when the big pitch gives preference to heat sink. Furthermore, the appropriate number of fins and the larger area give preference to the heat sink. As for the thermal resistance, it decreases as the *Re* increases[5].

2.1. THERMAL RESISTANCE

Thermal resistance (R_{th}) is the difference in temperature between both the temperature of the heat sink base and the temperature of the surrounding medium, divided by the amount of heat. Equation (2.1) below illustrates how to calculate thermal resistance[13]:

$$R_{th} = \frac{T_{base} - T_{\infty}}{Q} \tag{2.1}$$

Where R_{th} is the thermal resistance, T_{base} is the temperature of the heat sink base and Q is the amount of heat.

Equation (2.2) below shows how to calculate Reynolds number[7]:

$$Re = \frac{V_{\infty} L}{\mathcal{V}} \tag{2.2}$$

Where Re is Reynolds number, V_{∞} is the speed of the fluid flow, L is characteristic length and $\mathcal{V}$ is kinematic viscosity.

There was a previous study a CFD is used in their research, which two heat sinks were chosen, cylindrical pin fin and rectangular plate fin. Thermally analyzed with respect of heat flux, and the temperature at the same boundary conditions, also they were the same in dimensions. It was proven that the amount of heat transfer is bigger in rectangular plate-fin, in another word, rectangular plate fin is more efficient than cylindrical pin fin heat sink[6].

In another study, also using CFD, was used in the design development process for a heat sink. Once again, using CFD properly saves time and effort during making experiments and development processes, as well as predictability of every detail that is expected for each design or experiment that is worked on in detail and effectively[4] This is the motivation that encouraged the selection of CFD as a tool in the process of analysis and showing results in this thesis.

In another research to develop the efficiency of the heat sink, all the factors affecting, such as cost, were taken into account, as well as the process of developing pin fins. It has been concluded from experience that for each specific design process to increase the efficiency of the heat sink, it requires a suitable and specific length of the fin to suit that design[14].

In previous research, an investigation was conducted on the efficiency of the heat sink, considering that the process of heat transfer is in the manner of free convection, where the experiment was to use a horizontal channel and a plate-fin heat sink at the same time, by placing the heat sink inside the channel. The experiment concluded that the ideal value of the pitch is between 8 and 12 mm while increasing the number of fins increases the surface area of the heat sink and thus increases the possibility of dissipating as much heat as possible, but in return increases the pressure drop and also decreases the speed of the fluid flow[15].

The possibility of developing the design was studied in another research, in which the relationship between pressure drop and heat transfer was studied, in a pin fin heat sink. The research revealed that the development processes in the design gave different results according to the shape of the fin, as well as the fluid flow varies according to the design[16]. Three different types of heat sink fin shapes were studied, triangular, trapezoidal, and rectangular. The experiment was to perform a comparison between a fin array heat sink and a single fin heat sink to determine efficiency. The study found that for the fin array type, the trapezoidal fins were the most efficient. As for the single fin type, the triangular fin was the most efficient[17].

Previous research was conducted, by using a digital method that deals in three-dimensional (3D), to investigate the efficiency of the heat sink, with using a mini channel, in which water and airflow through it. The experiment has proven that the effectiveness and efficiency of these mini channels in enhancing the heat sink's cooling capabilities[11].

There are other methods used in cooling operations, such as impinging jets, as they are used just like using a heat sink, but they are used when the transfer of heat and mass is required in large quantities. They appear in various forms in many applications, such as in turbine blades as well as missile platforms[18].

Many fluids are used in the cooling process that can be used to enhance the efficiency of the heat sink. Among the additives that have been used to improve the performance of the heat sink are fans, which greatly contribute to the heat transfer process from the heat sink to the ambient air. Air is one of the most prominent refrigerants used to cool panels and electronic devices. This is due to the properties that encourage designers to use it in cooling operations. As it is characterized by being safe on devices, it does not cause any kind of damage or pollution and besides, it is available and free[19].

There was a previous study on enhancing the efficiency of the heat sink, through using it in a cooling process, by using air as a coolant, and taking into account that the cooling process took place under conditions of forced convection. The study showed encouraging results[20].

A previous study was conducted, using a simulation system to analyze and know the effect of the surrounding environment on the efficiency of the heat sink, especially if it is inside an area that is crowded with parts that generate heat, and the effect of this type of thermal load on the performance of the heat sink. The composition of the heat sink material was modified, which was of a pin-fin type and the surrounding environment was adopted from a computer chassis. The heat sink design has also been changed several times, keeping the parts inside the chassis in place unchanged. They concluded from their research that the performance of the heat sink is affected by the surrounding environment, as it generated heat that warms the surrounding air, and also affected the velocity of flow through the heat sink. The study also concluded that the use of a plate-fin is better than the pin fin-type to reduce the hot air generated inside the computer chassis, which negatively affected the performance of the heat sink[21].

One of the salient factors, which is a clear challenge for designers when designing a heat sink, is weight and thus designers devote their efforts to reduce the weight of the heat sink, as much as possible, as this reduction positively contributes to reducing the weight of the electronic board or device, therefore, light devices are desired by everyone. From this standpoint, a study was conducted that aimed to obtain an ideal weight for the heat sink, as two different types of heat sink were used for the shape of the fin, the first type was a heat sink with continuous rectangular plate fins in length, and the other was with non-continuous rectangular plate fins, in terms of the shape of the fin, and two other models of the same previous two types were used, but with the creation of holes in the fins. They found that the perforated fins contributed to this weight reduction on the one hand, as well as the airflow through these holes enhanced the heat transfer process. Moreover, the performance of the heat sink with rectangular plate fins that were interrupted and perforated was the best[22].

In order to improve the heat sink design, several methods can be applied to obtain a design that effectively serves the requirements. One of these methods is the topological method, and this method is characterized by a broad physical, and thermal analysis, and the topology is somewhat complex, but it helps in the process of speeding up and improving the design, especially for complex design forms, but taking into account the inclusion of factors to be improved during the design process, such as the inclusion of pressure drop, weight and thermal resistance[23].

One of the modern manufacturing methods used to further improve heat sink designs is the 3D printing method. It is a method that is controlled digitally through computers, where the idea of creating any design in this method depends on building the design in the form of continuous layers. This method facilitated the creation of many engineering designs for the heat sink, especially the most complex ones[24].

2.2. MICROCHANNEL HEAT SINK

One of the most important methods used in the process of cooling electronic parts is the microchannel. It is a device used to dissipate heat, but in larger quantities just like a heat exchanger does, but it is small in size. Furthermore, it is the product of the development of the heat sink design, but the idea of its operation in most designs depends on the flow of the coolant inside it, and thus the liquid inlet and outlet are determined in the design process. Extra more, is considered an effective tool in the cooling process, however, its performance is affected by the shape of the design as well as the size[25].

A numerical investigation was performed on the micro-channel module and hybrid jet, to determine the effect of the sectional channel shape on the cooling performance. Three microchannels, different in shape, but with the same cross-sectional area, were selected, trapezoidal channel module, circular channel module, and rectangular channel module. Firstly, they found that at the same flow rate, the cooling performance was affected by the duct shape. Secondly, they found that at the same flow rate, the cooling performance was affected by the duct size. Thirdly, they also found that the trapezoidal channel is the best in terms of cooling performance and finally, they found that the highest pressure drop value was in the circular-shaped channel[26].

A new study was conducted on how to improve the performance of the micro-channel, with the addition of a jet-injection technique, which in turn reduced the flow path inside the channel. This technique has proven its effectiveness in improving the cooling performance of the mini-channel, and as this experiment opened the way in the process of improving the channel design, in terms of the presence of curves, the inlet speed, the channel height, the channel distribution, and other matters related to the design in the first place, as well as everything that leads to improving the cooling performance concerning the duct to obtain an efficient heat sink[27].

Research aimed at fluid flow analysis as well as thermal analysis by a numerical method based on the 3D simulation system on a semi-circular microchannel. Where the idea of the experiment states the change in the geometry of the channel as affecting the cooling performance. Thus, three channels were created with different shapes, rectangular, triangular, and trapezoidal. From the study, the superiority of the triangular channel over the other two forms was extracted in terms of quality of performance[28]. Once again, this experiment demonstrated the effect of shape on channel performance.

An experiment was performed on a microchannel with a rectangular shape of various dimensions for heat transfer analysis. The experiment resulted in a decrease in the thermal resistance compared to the high aspect ratio of the microchannel[29]. And in (Figure 2.1) is an illustration of one of the microchannels used frequently in cooling operations in order to clarify the idea regarding the shape and design:

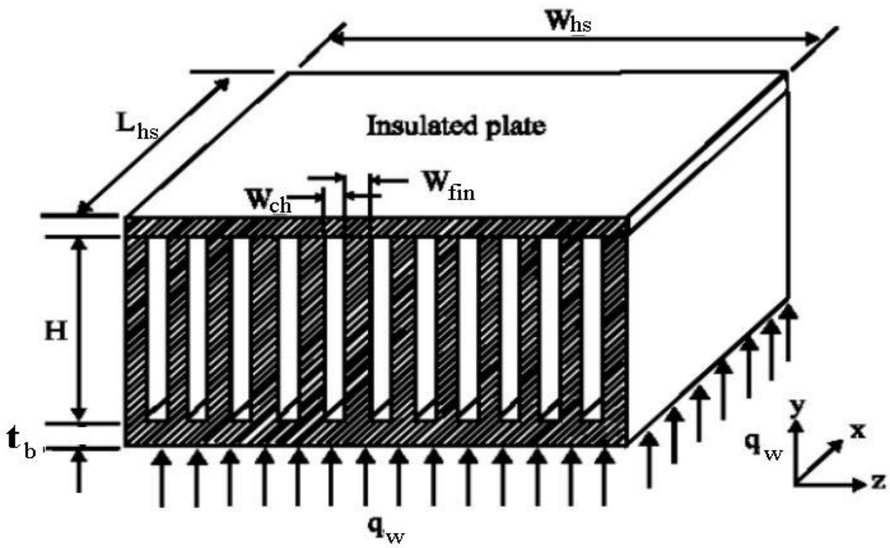

Figure 2.1: Microchannel heat sink schematic diagram [30]

Through the experiments and researches conducted by researchers on developing, and enhancing the efficiency of the heat sink, until they reached the manufacture of microchannels, which contributed to enhancing the process of heat transfer, and dissipation as they made many modifications to the design, such as the channel shape, flow rate, and other developments which resulted in the quality of its performance.

A study was carried out for the purpose of enhancing the cooling performance of a microchannel heat sink designed with dimples and a jet injector, where several additions were used to the design of the microchannel, so a comparison was made between channels that do not contain dimples, others that contain dimples, including convex and concave, and others mixed with two types of dimples. They prove that the lowest cooling performance was in the channels that included the concave dimples, and the performance of the channels containing the mixture of dimples was slightly better than the previous one, while the best cooling performance was in the channels that included the convex dimples[31].

A numerical study was performed in a three-dimensional domain on a rectangular channel to evaluate the performance, by using several forms of protrusions on the surface, trapezoidal, rectangular, and triangular, also by adopting the flow using an air jet. The study concluded that channels that include protrusions require a higher pump capacity compared to channels that do not contain protrusions, furthermore, channels that contain protrusions in a triangular shape are the best in performance[32].

Through numerous researches aimed to improve the performance of the heat sink, from several aspects, such as the material used in manufacturing, as well as the engineering design for the plate-fin heat sink. An experiment was conducted on the heat sink, but through the development of the design by adding cross fins in addition to the longitudinal fins and considering the process of heat transfer by free convection to know the effect of performance concerning radiation and convection[8].

(Figure 2.2) illustrates the comparison between the ordinary plate-fin heat sink and the developed sample:

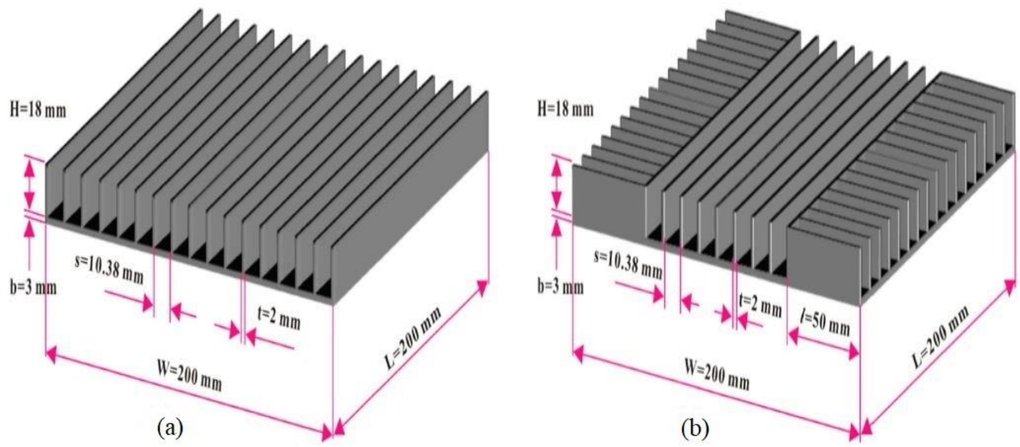

Figure 2.2: Diagram of (a) ordinary plate-fin heat sink and (b) cross fin heat sink [8]

It was found that developing the design by adding the cross fins enhanced the radiation heat transfer coefficient by an increase of 15%, as well as for convection by 11%[8].

A numerical experiment and a study were performed under forced convection and another condition under free convection on the plate-fin heat sink. The fins were modified in such a way that they were designed diagonally to see their effect on cooling performance. The study summarized through its results that the fins with an inclined top contributed to enhancing the efficiency concerning cooling, as well as the amount of flow when switching to forced convection[33].

The research was performed under free convection conditions to analyze the thermal performance of the plate-fin heat sink. The idea of the experiment was to study changing the thickness of the fin and in the direction of normal flow. The study showed that changing the multiple fin thickness contributed to enhancing the performance. Also, when the switch was made to air cooling, the thermal resistance decreased to 10% for the multi-thickness fin heat sink[34].

An experiment was conducted under free convection to study the effect of the fin shape on the performance of the heat sink. which a hollow cylindrical fin and a solid cylindrical fin heat sinks were selected, so the study aimed to analyze the heat flux in each of the two models. The study proved the superiority of the hollow finned heat sink in terms of performance compared to the other solid fin model[35].

2.3. HEAT SINK DESIGN CONSIDERATIONS

There are many factors that directly or indirectly affect the performance of the heat sink, and therefore the methods of evaluating these factors differ in terms of their impact on performance efficiency. A study was conducted on several factors related to the performance impact and it was summarized as follows:

i. The thermal resistance factor of the heat sink.
ii. Pressure drop factor and maximum temperature measurement of the heat sink base.
iii. Fan rotation speed and its relationship to pressure drop in the active heat sink[36].

Some of the variables that can affect heat sink performance such as fin geometry, number of fins, location of heat source, size of fin array, and power of the fan or pump used[19].

Some problems are facing the development process of improving the performance of the heat sink, and its solution may be somewhat complicated, but with some effective methods to solve these problems, which opened the way for improving performance[17].

The fact that many researchers use numerical methods in analyzing all the problems related to improving the performance of the heat sink, has helped a lot in the development process, as the possibilities of numerical methods are wide, which includes saving time and effort especially in complex problems, and designs that are an obstacle to the capabilities of the human mind.

2.4. MAIN PURPOSES OF THE LITERATURE REVIEW

Through reviewing previous studies and researches, whose results are proven by experiments, some points can be drawn which are related to the research topic, so that they act as a guide, as follows:

a. Heat sink performance is affected by fin size and shape.
b. The possibility of reaching the optimum performance of the heat sink is obtained in a moderate region.
c. For evaluating the performance of the heat sink both pressure drop and thermal resistance are effective methods.
d. The performance of a heat sink is affected by the fluids used in its operation.
e. The material of the heat sink affects performance.
f. Numerical methods are considered one of the effective tools used in evaluating the performance of a heat sink, in order to save time and effort.

2.5. RESEARCH GOALS

This thesis aims to add something new, and useful about enhancing the efficiency of the heat sink, through researches, summaries, and results of previous experiments, and to try to find some solutions to problems not previously solved, as well as to add new knowledge based on scientific research and analysis. By reviewing previous experiences, and researches, which have saved a lot of time and effort on many researchers, and also has enriched the field of improving the performance of the heat sink with much valuable information that would reduce errors as well as push forward development, also, these experiments especially the practical ones, have become the real, and realistic reference in any new development process, and from this using of numerical methods appeared widely as they are an appropriate, and effective tool in development processes, especially complex ones. However, they need an initial setup, so it is the most important step in them, which requires the compatibility of the outputs of its results with the results of the real practical experiment, and this is referred to previously, which is known as the validation case. From this standpoint, the main goal of this dissertation was to use the numerical CFD methods in the correct manner after performing the validation process on the practical experience of the previous researchers, and then go to the simulation system through, which a development process is made on the design of the heat sink to reduce the size 50% and also to evaluate the performance, by designing a different set of shapes on the one hand, and by changing the heat sink material on the other hand. The targeted improvement process is by comparing the results of the new designs with the results of the traditional, popular, and most commonly used plate-fin heat sink, this is in an attempt to enrich or add new information to the process of improving performance or to provide some answers to the unsolved problems of the heat sink.

3. METHODS AND APPROACH

In this chapter, computer-aided design (CAD), CFD analysis methods, and techniques employed in generating, and simulating heat sink models are discussed. This chapter also discusses best practices and related parameters used to simulate different geometry heat sinks via the (ANSYS FLUENT 19.2 R) CFD tool[37]. The models selected for the type of heat sink were analyzed to explain the physics behind models of real-world results accuracy.

Also, in this part of the research, and before going into details of the methods used in numerical studies and analysis processes, it is necessary to review some special concepts about the transfer of heat between objects, and adjacent surfaces to facilitate understanding of the theory of the working of the heat sink, as well as the possibility of enhancing its capabilities in cooling operations.

3.1. EXTENDED SURFACES

It is about enhancing the process of heat transfer from the bodies to be cooled so that the idea lies in the process of mixing conduction and convection in the method of heat transfer, so this is evident in the idea of the fins working, which therefore is based on the principle of its work on the principle of increasing the surface area, which is an important factor in the process of heat loss. (Figure 3.1) illustrates two surfaces which (a) is a normal surface and (b) after extended by the extrusion of fins, so it can be seen obviously how the surface area is maximized, which is considered a positive thing in enhancing the heat loss[38].

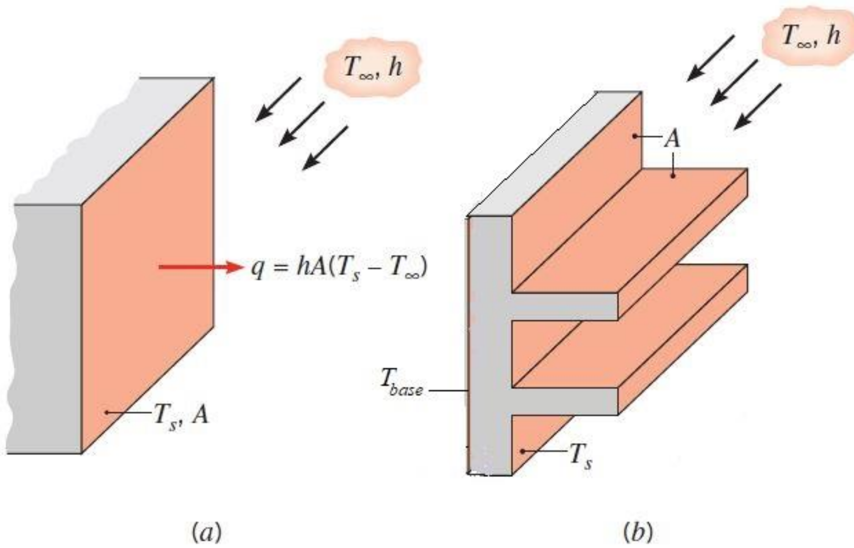

Figure 3.1: Normal surface (a) and extended surface (b)[38]

Some equations can be used to calculate everything about fins, like the heat rate at which the fin can dissipate to the surrounding. Furthermore, the thermal resistance and the efficiency of the fin.

$$\theta(x) = T(x) - T_\infty \tag{3.1}$$

Where θ is the temperature excess and T_∞ is the fluid temperature.

$$T = T_\infty + (T_b - T_\infty)e^{-mx} \tag{3.2}$$

Where T_b is the fin base temperature.

$$m^2 = \frac{hp}{kA_c} \tag{3.3}$$

Where m^2 is parameter of the fin, h is the convection heat transfer coefficient, A_c is fin across section area, p is perimeter of the local fin and k is the thermal conductivity of fin.

$$q_f = \sqrt{hPkA_c\theta_b} \tag{3.4}$$

Where q_f is fin heat rate and θ_b is the fin base.

$$\varepsilon_f = \frac{q_f}{hA_{c,b}\theta_b} \tag{3.5}$$

Where ε_f is fin effectiveness.

$$R_{t,f} = \frac{\theta_b}{q_f} \tag{3.6}$$

Where $R_{t,f}$ is fin thermal resistance.

$$\varepsilon_f = \frac{R_{t,b}}{R_{t,f}} \tag{3.7}$$

Where $R_{t,b}$ is the thermal resistance of the fin base.

$$\eta_f = \frac{q_f}{q_{max}} = \frac{q_f}{hA_f\theta_b} \tag{3.8}$$

Where η_f is fin efficiency and A_f is fin surface area.

Extra more, it can be seen obviously that the efficiency of the fin is a very important factor for choosing the prober shape of fins in heat dissipation. Since fins have many different shapes, and as a result, each fin shape has its formulas for thermal calculations.

Rectangular fin efficiency can be calculated as follows[7]:

$$\eta_f = \frac{\tanh mL_c}{mL_c} \tag{3.9}$$

$$A_f = 2wL_c \tag{3.10}$$

$$L_c = L + \left(\frac{t}{2}\right) \tag{3.11}$$

$$A_p = tL \tag{3.12}$$

Where L is fin length or height, w is fin width, t is fin thickness and A_p is profile area.

3.2. CONTACTED SURFACES

What is meant by adjacent or contiguous surfaces or objects, is a description of the heat transfer process through several different or similar surfaces in terms of type, as well as taking into account the method of heat transfer, whether by conduction, convection, or radiation. Hence, it can be said that the heat transfer inside the body or one surface is by conduction as well as between adjacent or contiguous objects and surfaces. But there may be surfaces contiguous to each other, one of which is exposed to a fluid, where the process of heat transfer from the solid surface to the fluid through convection is clearly visible.

Moreover, the investigation main idea is depending on heat transfer from the electronic piece which produces or generates heat by the concept of contacted bodies, so the heat flows from the hottest body to the cooler adjacent one, which is the heat sink by conduction method, after that the heat sink dissipates the heat to the surrounding by convection method.

Extra more, if considered that there are two bodies (A) and (B) are in contact with each other, and no gaps of air or high spots between them, and to explain in detail the hottest body is (A) which is the CPU which is to be cooled, and (B) is the heat sink. As mentioned earlier, the heat flow through the heat sink faces thermal resistance, which is made by the thermal conductivity of the heat sink material. Furthermore, the thermal conductivity k is one of the most important factors which is to be taken into considerations in the conduction method, so if the thermal conductivity is high so this is a positive thing, and the heat flows smoothly as well as a high amount of heat loss from the body. However, if the thermal conductivity is very low that is a negative thing, and the heat flow interrupts and of course, it is considered as an insulation material.

(Figure 3.2) illustrates the previous explains to make a very clear understanding of the technique which is followed in the thesis procedure.

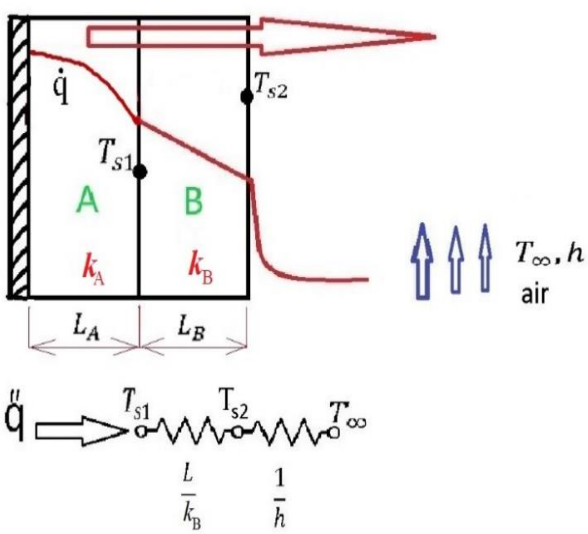

Figure 3.2: Illustrates temperature distribution and thermal resistances

In (Figure 3.2) it can be seen obviously the direction of the heat flow, but it must be mentioned that there are a few assumptions that are taken in explaining the heat flow concept. Firstly, one side of the (A) body which is the (CPU) which generates heat is considered as insulated part, so to focus on calculation in one direction which is the heat sink direction. Secondly, the radiation heat transfer is negligible because of forced convection. The temperature distribution which is mentioned by the red line shows a parabolic diagram in the body (A) the (CPU) because of the heat generation, and shows a linear diagram in the body (B) the heat sink, because of the conduction method, in which the thermal conductivity k plays the main role. Finally, the diagram significantly changed from linear to make a mixture of linear, and parabolic, because of the convection method which is the velocity of the fluid which is here is air, and convection thermal coefficient h plays the main role.

$$q^{"} = \frac{q^{\bullet}}{L_A^2} - \frac{T_{s1} - T_{\infty}}{R_{tot}} \qquad (3.13)$$

$$R_{tot} = \frac{L_B}{k_B} + \frac{1}{h} \qquad (3.14)$$

3.3. METHODOLOGY

The strategy which is followed here in this research is to move from the traditional experiments inside the labs to the numerical analysis, and investigations in terms to save time and money. So (SOLIDWORKS 2020)[39], is used in designing the various geometries of the heat sink as well as (ANSYS FLUENT 19.2R)[37], is used to simulate and validate the experiments.

3.3.1. Conjugate Method

In this study, it was important to apply the combined heat transfer methods in the simulation of models to a solid and fluid region**.** This approach enables the modeling of the physics of the heat sink (solid) as well as of the air (fluid) at their interface. In the passive heat sink system, the heat transfer mechanism dominates as a consequence of the interaction or clash between inner particles within the material. The heat is transmitted firstly from the warmer bottom towards the other areas of the heat sink (copper). Also, in the fluid region convection prevails. The effect of heat transmission seen between the heat sink, as well as its environment (ambient air) is essential for this process simulation. So conjugate method means the unification of the two methods of heat transfer convection, and conduction at the same time.

3.3.2. Modeling of The Fluid

In order to make sure that the results are valid especially in comparison to the various simulations of heat sink shape, assumptions have been made in order to replicate the fluid flow behavior around heat sinks. The combined heat transfer method, thus the existence of the solid or fluid regime, was used in this research. With all heat sink data sets, the fluid characteristics, and therefore its quantity was constant, and the air properties are shown in Table 3.1.

Table 3.1: The fluid (Air) properties

Viscosity (μ)	$1.7891 \cdot 10^{-5}$ kg/m·s
Thermal conductivity (k)	0.0242 W/m·K
Specific heat (C_P)	1006.430J/kg·K
Density (ρ)	1.225 kg/m^3

The fluid stream has been designed to remain in the laminar region. When refers to laminar flow that means smooth fluid or fluid flow with little or no disturbance corresponds. This kind of flow is moving on a normal path and is supposed to have a much steadier speed and pressure drop. Then it is presumed that laminar flow behavior is stable all over its flow path. The laminar flow model was considered appropriate because the current flow was extremely slow, and even the flow channel was rather small due to weather conditions, in which the heat sink analyzed was to be used in this study. In the scenarios when fluid temperature is regarded as separate from the solid under a conjugal heat transfer mechanism, the segregated fluid temperature concept is being used. The energy equation with a different temperature is fixed by this model.

3.4. THE REYNOLDS NUMBER

To characterize the flow of any fluid, and its type as it is laminar regular or random turbulent, this is defined by the Reynolds number, which is a dimensionless quantity. Where the flow type of the fluid is described by calculating the value of the Reynolds number through some equations[40]:

$$R_e = \frac{\rho V D_h}{\mu} \tag{3.15}$$

Where R_e is Reynolds number, ρ is fluid density, V is fluid velocity, D_h is hydraulic diameter and μ is fluid viscosity.

$$D_h = \frac{4A}{P} \tag{3.16}$$

Where A is sectional area and P is wetted perimeter.

The equation mentioned calculating the Reynolds number probably represents the inertial-viscous-potential proportion, and forecasts flow characteristics due to the two specified fluid flows laminar and turbulent. So, at a low Reynold number, less than 2,300 the laminar flow is slower and steadier. The transition between laminar and turbulence starts between a Reynolds number range from 2,300 to 4,000. In this condition either the laminar and turbulence flow environments coexist, and thus a fluid has laminar and turbulent flow attributes in this state. The turbulent regime is achieved after just a Reynolds number of 4,000. Within that regime the flow is chaotic as the inertia force is more powerful, causing more disruption inside the flow. Equation (3.15) shows that the fluid types in an analysis of this kind have been constantly maintained, for the various models of heat sink analysis, the variables ρ, V and μ probably remain the same. The hydraulic diameter D_h remains constant, as with the geometry examined so that the Reynolds number changes in speed. Therefore, a continuous speed of 1 m/s, is selected as approaching airflow speed all around the heat sinks. As already indicated, the simulation flow was all maintained no more than 1257 of Reynolds number, and within the laminar flow area. This induces quite practical results to become achievable even though, most of the fans used to perform airflow speed via heat sinks are typically unable to generate turbulent flow at the needed higher ventilation speed.

3.5. EQUATIONS OF GOVERNANCE

The control formulas used for fluid flow dynamics are computational algorithms for the simulation of fluid behavior across the heat sinks. Fluid motion calculations are generally extremely complicated and require computing capabilities, and expertise to achieve better and precise results. In such cases where fluid behavior is required around and inside a three-dimensional object it becomes a great challenge to solve the continuity equations for fluids due to various dimensions. Navier-Stokes's formula is incorporated into the analytical program[37] used in this analysis. As related to the flow of fluids under pressure, this equation can be derived from Newton's 2nd law. As a consequence of both the conjunction of propagation viscous term and force, the pressures through which the Navier-Stokes formula is obtained. The following is a differential version of the Navier equation[40]:

$$\frac{\partial \rho}{\partial t} + \nabla \bullet \left(\rho \vec{V}\right) = 0 \tag{3.17}$$

Where ∇ is Nabla which represents the quantity partial derivative in all directions in the coordinate system, and it can be found as:

$$\nabla = \frac{\partial}{\partial x}\vec{\imath} + \frac{\partial}{\partial y}\vec{\jmath} + \frac{\partial}{\partial z}\vec{k} \tag{3.18}$$

$$\frac{\partial}{\partial t}\left(\rho \vec{V}\right) + \nabla \bullet \left(\rho \vec{V}\vec{V}\right) = -\nabla P + \nabla \bullet (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \tag{3.19}$$

Where ρ is density, P is static pressure, $\bar{\bar{\tau}}$ is stress tensor, $\rho \vec{g}$ is the gravitational body force, $\vec{V}$ is velocity vector and $\vec{F}$ external forces of the body.

$$\bar{\bar{\tau}} = \mu \left[\left(\nabla \vec{V} + \nabla \vec{V}^T\right) - \frac{2}{3}\nabla \bullet \vec{V} I\right] \tag{3.20}$$

Where μ is the viscosity of molecular and I is the tensor unit.

$$\frac{\partial}{\partial t}(\rho E) + \nabla \bullet \left(\vec{V}(\rho E + P)\right) = -\nabla \bullet \left(\sum_j h_j J_j\right) + S_h \tag{3.21}$$

Where E is the energy total, J is the flux diffusion, h is enthalpy and S_h is volumetric heat.

3.6. THE FLUID FLOW THROUGH THE FINS

In removing heat along its surface, the fluid flow via the extrusion fins is a major function. With both pin, and plate fins of the heat sink, various fin arranges can be created to increase heat transfer. Because of the flow across fins space amongst fins, height and the width of the fins are essential characteristics. This study took steps to optimize flow through the fins of the heat sink to create an efficient flow of energy, from the heat sink to the operating fluid.

3.7. VALIDATION CASE

For the beginning of this research, an evaluation case was built based on a review of the literature to ensure correct simulation outcomes, and the validity of the hypotheses as well as models used. An experimental study was developed using to compare heat sink pressure loss measurement through methodological approaches. So, in their research, a mathematical, test, and computational analysis was carried out and compared for validation. In the mathematical method, the equations were derived through the equilibrium force on the heat sink, which is three equations for each of Flemings and Darcy to calculate the pressure loss[42] which they are:

$$\Delta P = \left(\frac{f_{app} N(2HL + bL)}{HL} + K_c + K_e\right)\left(\frac{1}{2}\rho V_{ch}^2\right) \tag{3.22}$$

Where ΔP is the heat sink pressure drop, f_{app} is the apparent friction, N is fins number, H is fin height, L is fin length, b is the channel width of the heat sink, K_c is contraction coefficient, K_e is expansion coefficient, W is heat sink width, ρ is the density and V_{ch} is the velocity of the heat sink channel.

$$\Delta P = \left(4f_{app}x^{+} + K_c + K_e\right)\left(\frac{1}{2}\rho V_{ch}^{2}\right) \tag{3.23}$$

$$\Delta P = 4(f_{app}x^{+} + K_c + K_e)(\frac{1}{2}\rho V_{ch}^{2}) \tag{3.24}$$

Two separate channel speed formulas (V_{ch}) were inserted in and compared with the pressure drop equations above.

$$V_{ch} = V_{ap}\left(1 + \frac{t_f}{b}\right) \tag{3.25}$$

Where V_{ch} is the velocity of the heat sink channel, V_{ap} is velocity approach, t_f is the fin thickness and b is the channel width of the heat sink.

$$V_{ch} = V_{ap}\left(\frac{A_f}{A_{tf}}\right) \tag{3.26}$$

Where A_f is the frontal area with the base of the heat sink and A_{tf} is the area which fluid flow through.

They reveal that the most precise calculation of the mathematical pressure drop was found by a comparison of the outcomes of equations (3.23) and (3.26)[42].

In this thesis, the simulation system as defined has been used as a foundation for the validation scenario. They divided the research into three parts, which was first the theoretical part, in which the results were extracted by using five mathematical equations to calculate the pressure difference, and air flow velocity through the heat sink. Secondly, the practical part, in which four types of heat sink were used, differing in sizes as well as the type of metal, by placing them inside the wind tunnel as shown in (Figure 3.3), and also sensors were placed inside the tunnel to measure pressure, and temperature as well as air velocity, also a heater was placed under each of the four models.

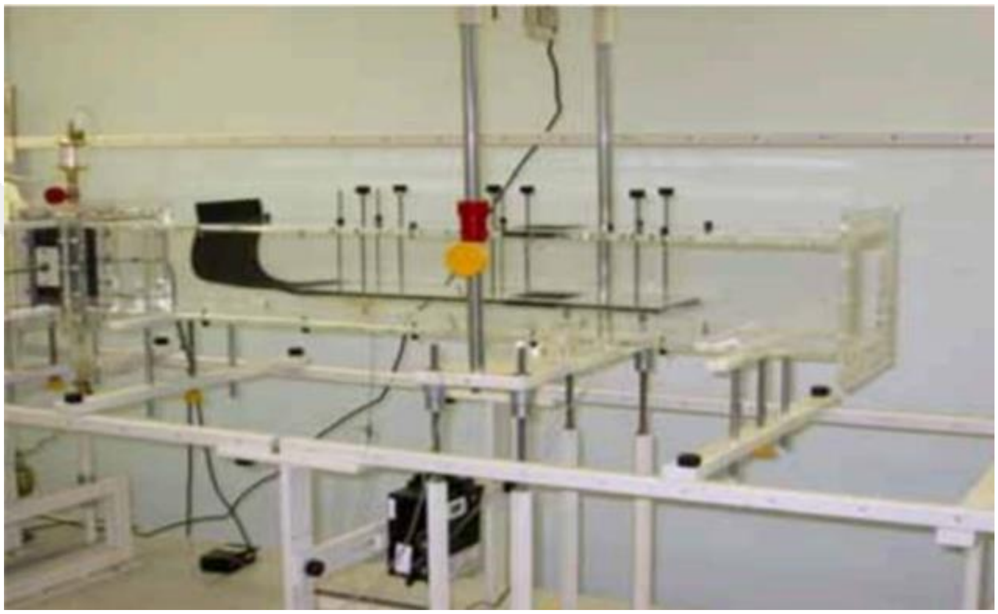

Figure 3.3: The wind tunnel which is used in the experiment case[42]

Thirdly, a (FLOTHERM)[43], a simulation program was used to apply the same experiment to make a comparison between the results of the theoretical part, the practical part, and the numerical part. The tested heat sink was a rectangular plate fin which is illustrated in (Figure 3.4), as well as its dimensions, which are shown in Table 3.2.

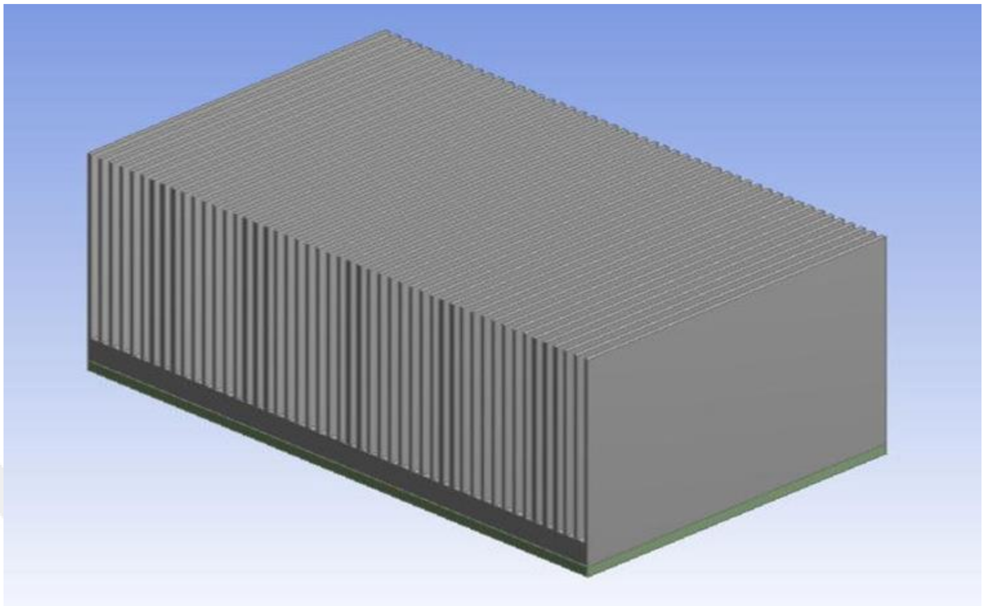

Figure 3.4: CAD model of the validation case heat sink[42]

Table 3.2: Dimensions of validation case heat sink[42]

Base thickness (b)	7.0 mm
Channel width (g)	1.55 mm
Fin thickness (t)	1.0 mm
Height (H)	63.0 mm
Length (L)	75.0 mm
Width (W)	126.0 mm

The heat sink sample was surrounded by the duct to avoid any leak of flow, this allowed the exact higher pressure drop to be measured via the extrusion heat sink fins. The test was done with an air-speed level from 1 m/s to 4 m/s with 0.5 intervals. The experiment[42], computational (FLOTHERM) is a commercial simulation program that is used to conduct thermal analysis as well as to characterize fluid motion in reference experiments, and digital investigations [43], and measurement findings at different air velocities are shown in (Figure 3.5).

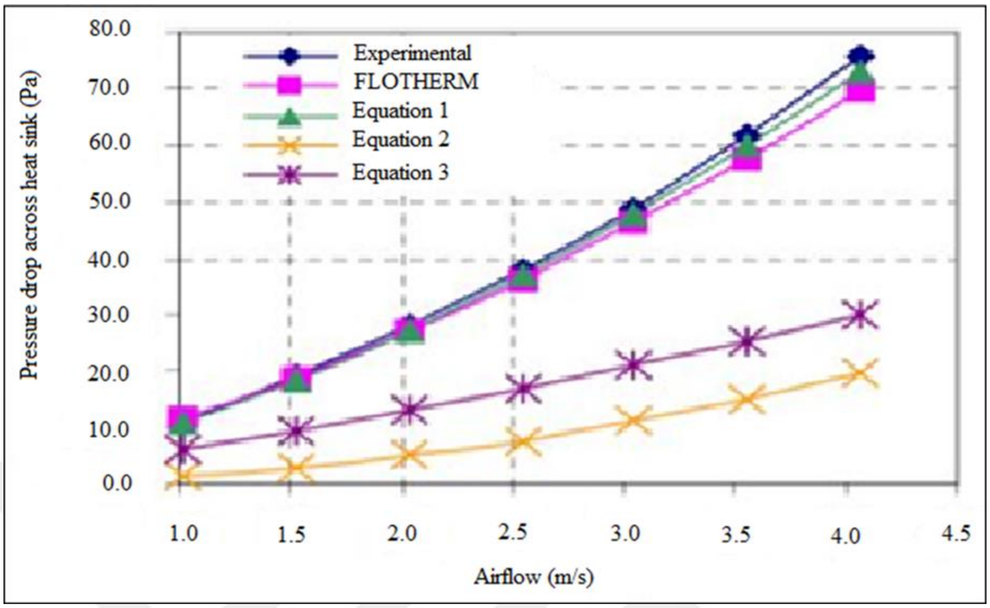

Figure 3.5: The outcomes of the validation experiment[42]

(Figure 3.5) represents a comparison of the results of the experiment referred to previously, where the comparison between the three parts of the experiment. By comparing the results of the three parts of the aforementioned experiment, it can be concluded that some of the mathematical equations used in theoretical studies may achieve results very close to reality, which is exactly what happened when using equation (3.23), but others may not achieve similar results, which is what actually happened with equations (3.24), (3.25), and (3.26) which confirms the importance of the correct choice of equations that serve the experiment correctly, and also it can be confirmed that the simulation results of computer programs are very close to reality, and can be relied upon in showing as well as predicting the expected results.

As described in the previous chapters, active heat sinks are supported by a fan or a fluid pushing unit. All test samples are active heat sinks with a particular direction of the inlet, as in (Figure 3.6) since a very well airflow path (inlet) is modelized in this analysis.

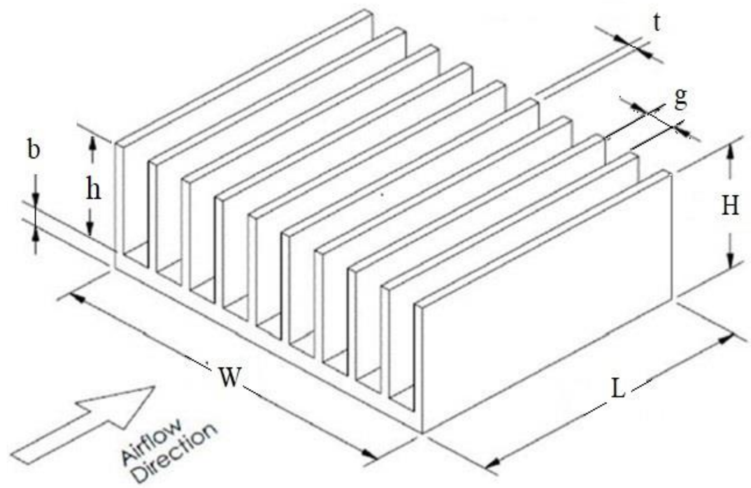

Figure 3.6: Airflow scheme active heat sink[44]

Where all symbols shown in (Figure 3.6) are defined as follows:

b is the thickness of the base, *h* is fin's height, *W* is width of heat sink, *L* is the length of heat sink, *H* is heat sink's height, *g* is spacing of fins and *t* is fin's thickness.

3.8. CAD MODEL CREATION

In this thesis, the (SOLIDWORKS 2020) program was chosen to design, and develop the shapes used in numerical analysis, because of the many features, and options that this program has,66 that make it superior to other programs in terms of accuracy, speed of completion, and saving time[39]. The process of designing a heat sink using (SOLIDWORKS) can be divided into two stages so that in the first stage, the base is drawn in the required dimensions, and then the fins are made. First of all, from the part option, a file is opened, and from the sketch option, the surface to be drawn on is selected, then the top plane is selected to draw the base, after that the dimensions are adjusted, then extrude command is given to obtain the required thickness. After that, the fins are drawn, and from the linear sketch pattern option, the fins array is distributed in both vertical, and horizontal directions to save time. And finally, from the extrude option again to give the desired height for the fins. In this way, the heat sink was created, and thus the file must be saved according to the required format.

3.9. ANSYS FLUENT VALIDATION CASE SIMULATION

The simulation process in this study, as mentioned previously, is by using the forced convection heat transfer method with a focus also on conduction heat transfer, and this method is known as a conjugate, which is the unification of both methods of heat transfer. The heat sink that was used in the experiment as found in (Figure 3.4), and according to the measurements in Table 3.2, was drawn using (SOLIDWORKS), and then it was loaded into the (ANSYS FLUENT) program. Likewise, the same procedure was followed with the rest of the targeted designs in the study subject. To create the simulation, the (ANSYS) program deals with the model to be analyzed based on two parts. One part is solid, which is the heat sink, and the other part is fluid, which simulates the movement of air around the heat sink, and also covers all sides of the heat sink. (Figure 3.7) illustrates the sample which is surrounded by the fluid.

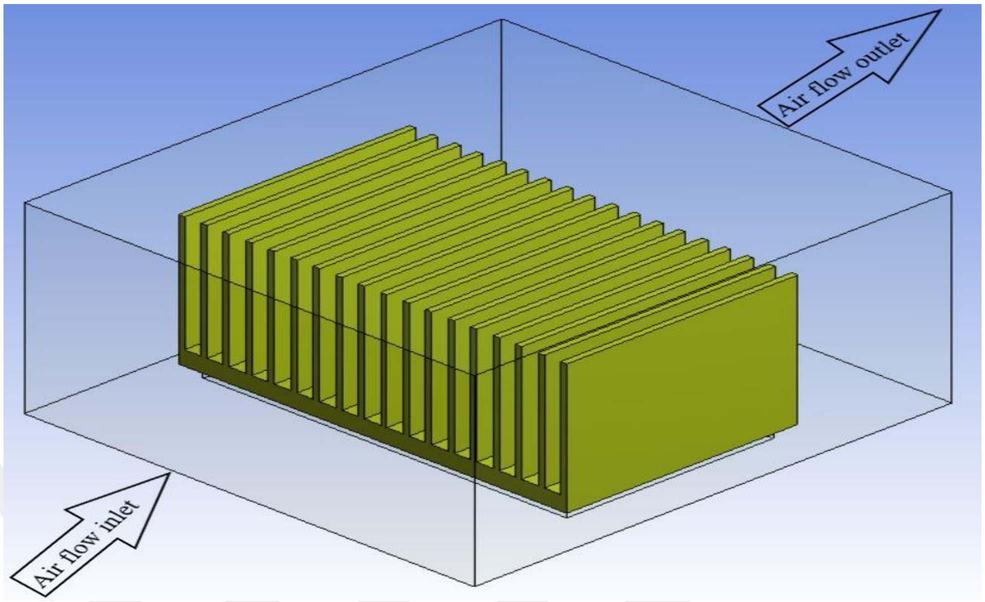

Figure 3.7: The validation heat sink surrounded by the fluid

Then, the mesh is designed for both the fluid, and the steel. Hexahedron mesh is chosen for the solid for two reasons, firstly, to decrease the number of the mesh in one hand, and also to decrease the time of calculations in another hand, and also the fluid tetrahedron is chosen as the general form of the mesh because it is the most suitable type to describe the fluid motion. Besides, the mesh size was chosen through the size option, which was set to 1 mm. Then, inflation was added, which was set on 10 layers. After this adjustment process, the number of mesh elements reached 6532031, while the number of nodes reached 2557371, which is the total number of the whole simulation sample. All these measures have been taken in terms of the physical properties of the fluid, and the solid as well as controlling the simulation to obtain a real environment that is very close to reality. Table 3.3, Table 3.4, Table 3.5, and (Figure 3.8) show domain physics, mesh, sufficient mesh, and boundary conditions respectively.

Table 3.3: Validation case domain physics

Domain	Type
Chip	Solid
Fluid domain	Cell
Heat sink	Solid

It should be noted that the number of mesh for both the heat sink as well as the chip has changed several times by increasing their number each time, and the effect of the increase on the results, especially the temperature, to obtain somewhat accurate results, and the best suitable number of mesh was chosen as shown in Table 3.4, to accurately describe the flow of heat within the two components, taking into account that the number of mesh is very appropriate on the one hand, as well as to avoid the large number, which in turn increases the calculation process and time.

Table 3.4: Validation case mesh information

Domain	**Nodes**	**Elements**
Chip	208	84
Heat sink	5488	2394
Fluid domain	2548239	6524578
All domains	2553935	6527056

In order to choose the appropriate and, sufficient number of meshes to obtain an accurate thermal analysis of both the heat sink, as well as the chip to avoid a large number of meshes on the one hand, which in turn increases the time, and operations of the computer solution, and on the other hand, to obtain an accurate sensitivity to the heat distribution. Table 3.5, shows the change of the mesh size several times to calculate the temperature at the point (X = -0.00175 m, Y = 0.017 m, Z = 0 m) taken on the heat sink, and the result was that the temperature change is very small, and can be neglected.

Table 3.5: The sufficient mesh number

Domain	Nodes	Elements	Temperature at (-0.00175, 0.017, 0)m
Chip	8160	5841	337.705 K
Heat sink	51224	28564	337.705 K
Chip	986	448	337.705 K
Heat sink	6147	2856	337.705 K
Chip	504	220	337.702 K
Heat sink	25100	14568	337.702 K
Chip	270	112	337.697 K
Heat sink	11797	7284	337.697 K
Chip	208	84	337.695 K
Heat sink	9084	5609	337.695 K

Domain	Boundaries	
Chip	Boundary-contact_region contact_region_3 contact_region_2 contact_region-3 scr chip	
	Type	Interface
	Boundary – wall chip	
	Type	wall
Fluid domain	Boundary-contact_region contact_region_3 contact_region_2 contact_region_3 trg	
	Type	Interface
	Boundary-inlet	
	Type	Velocity-inlet
	Boundary-outlet	
	Type	Pressure-outlet
	Boundary-wall fluid_domain	
	Type	Wall
Heat sink	Boundary-base contact_region contact_region_3 contact_region_2 contact_region_3 trg	
	Type	Interface
	Boundary-base contact_region contact_region_3 contact_region_2 contact_region_3 str heat_sink	
	Type	Interface

Figure 3.8: Validation case boundary conditions

3.10. THE RESULT OF THE VALIDATION CASE SIMULATION

By applying all the previously mentioned set steps to conduct a simulation very close to reality, the simulation results were very encouraging and satisfactory. The simulation results were compared with the results of the experiment referred to in the presentation of previous studies, and this is evident from (Figure 3.9). The mean absolute percentage error (MAPE) was 7.4%, which is considered a very good percentage, through the use of (ANSYS FLUENT 19.2R).

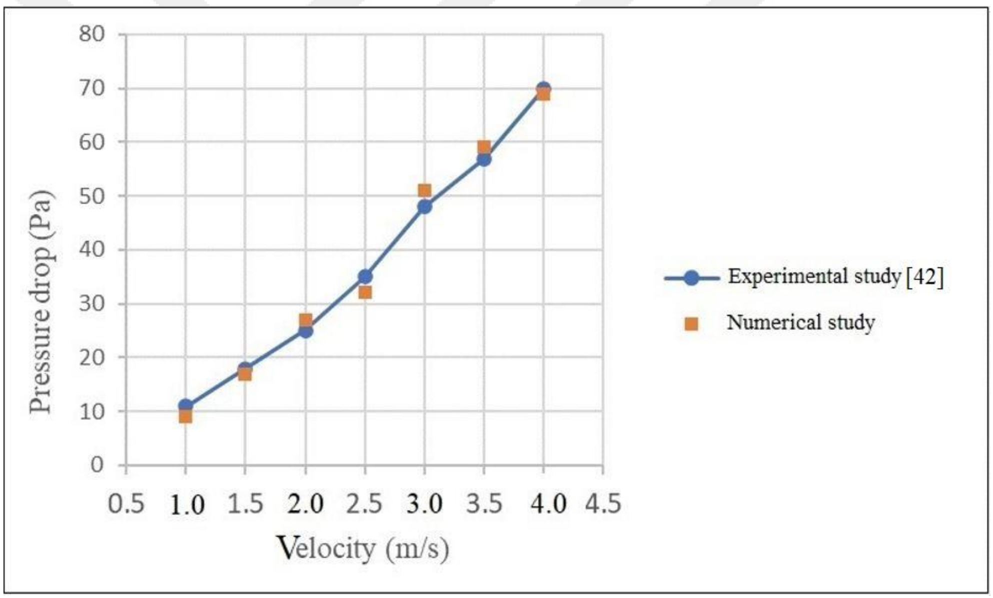

Figure 3.9: The comparison between the experimental and numerical studies

(Figure 3.10) and, (Figure 3.11) illustrate the temperature gradient through the heat sink efficiently, and the air is played a very well role by flowing around the fins to help in spreading the heat. Furthermore, it can be seen clearly that from the gradient color over the fins.

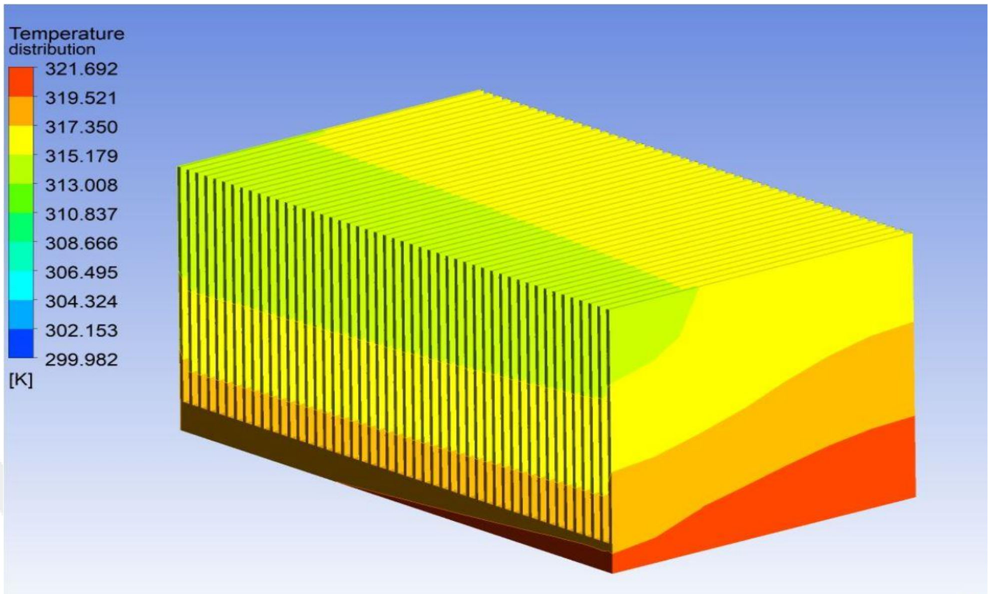

Figure 3.10: Temperature gradient of the validation sample

(Figure 3.11) shows the temperature distribution through the validation case of the heat sink.

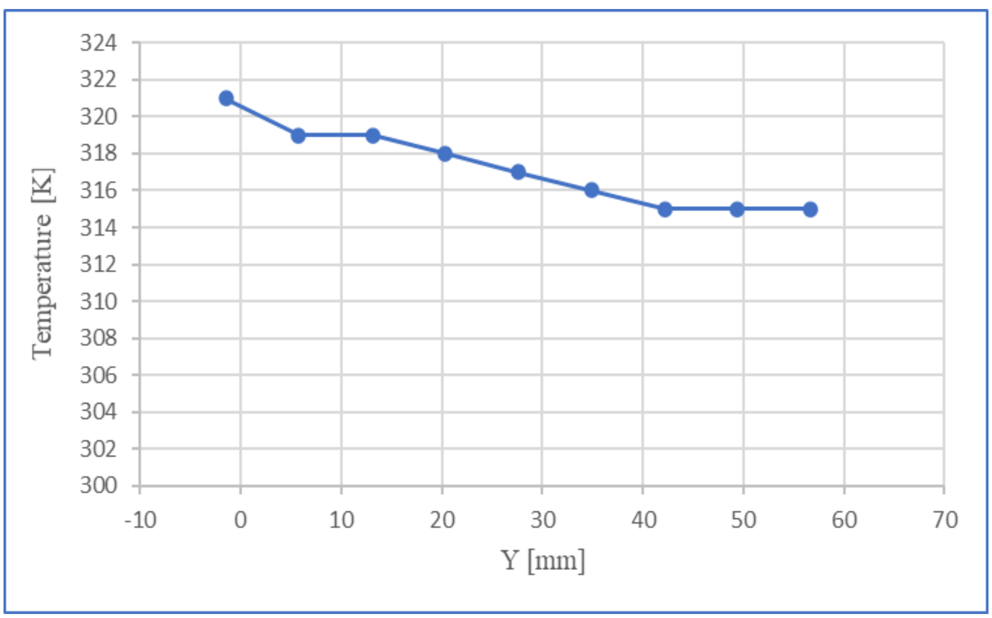

Figure 3.11: Temperature gradient through heat sink validation case

3.11. ASSUMPTIONS OF SIMULATION AND CREATION OF THE HEAT SINK

All conditions that were set to achieve a real environment for the simulation process to obtain results very close to the results of the experiment referred to in the literature review, in terms of fluid specifications, as well as the type of mesh used in both parts, whether solid or fluid, are the same that were used in order to perform thermal analysis of different types of the heat sink. Whereas, the fluid used is air, with the previously mentioned specifications. The air entering speed was 1 m/s, the temperature inlet was 300 K, and the pressure inlet and outlet was 101,325 Pa. The simulation calculation has also been adjusted for 1000 iteration. To compare the performance of the different forms of the heat sink, both the thermal resistance and pressure drop factors were selected. The material which is chosen for the different designs of the heat sink was red copper, and the properties are given in Table 3.6.

Table 3.6: The heat sink material (copper) properties

Specific heat (C_P)	381 J/kg·K
Thermal conductivity (k)	387.6 W/m·K
Density (ρ)	8978 kg/m^3

There was a process of changing the design of the heat sink, in terms of material as well as by changing the design of the fin shape, to obtain several different models to evaluate the performance of each and the extent of the effect of the fin shape. These designs are as follows:

a. Traditional rectangular plate-fins heat sink.
b. Separated short plate fins heat sink.
c. Separated short angled plates fin heat sink.
d. Separated short airfoil fins heat sink.
e. Continuous zigzag fins heat sink.
f. Intermittent zigzag fins heat sink.

The following equation is used to determine the number of fins[45]:

$$N = \frac{W}{t + g} \tag{3.27}$$

Where W is the width of heat sink, N is the number of fins, g is the width of channel and t is the thickness of fin.

Table 3.7, represents all the boundary conditions, and initials that are used in the numerical analysis in this dissertation.

Table 3.7: The boundary conditions and initials

Inlet velocity	1 m/s
Inlet pressure	101325 Pa
Inlet temperature	300 K
Turbulence intensity	5 %
Quantity of heat	100 W
Outlet pressure	101325 Pa

3.12. SAMPLE TEST DIMENSIONS

Among the steps that have been taken in this research, especially in the design process, the dimensions of the model used in the validation case have been reduced to 50% to reduce the cost of the material on the one hand, as well as reduce the volume of the heat sink on the other hand. As a result of that simulation process, and the time it takes in the calculations is also reduced. Moreover, the number of fins was 18, and also for heat sinks contain arrays of fins. The array was designed to be a 4X18 matrix. Furthermore, the modified design of the heat sink, and the target in the analysis in this research, and its dimensions are shown in Table 3.8. And it should also be noted that the thickness of the fin, as well as the distance between the fins, are constant for all types of a heat sink used in the simulation, until the comparison between their performance is done fairly, and also to obtain results that enhance heat sink performance.

Table 3.8: The dimensions of the sample heat sink

Width (W)	63.0 mm
Length (L)	37.5 mm
Height (H)	31.5 mm
Fin thickness (t)	1.0 mm
Channel width (g)	2.5 mm
Base thickness (b)	3.5 mm

3.13. COMPARISON METHODOLOGY

The performance evaluation of the different types of heat sinks used in this thesis was based on the pressure drop and thermal resistance parameters. Through a table, and a bar chart, the comparison was represented to give a clear, and simple idea of the performance evaluation that was analyzed through the numerical studies that were conducted.

4. RESULTS AND DISCUSSION

In this section, the results of the numerical studies that were performed computationally on all the different models of the heat sink used in this research were collected, analyzed, discussed, and their performance was compared as previously indicated to determine their efficiency. Furthermore, the process of reducing the size of the validation case heat sink to 50%, which was applied to the simulation design, has achieved its goals, and this is evident by comparing the weight of the two models, so the weight of the heat sink before the size was reduced was 734.27 g, but after miniaturization, it became 241.8 g.

4.1. THE RESULTS OF TEMPERATURE EFFECTS

In order to know the performance of each design in the cooling process of the resulting heat, the calculations were based on the thermal resistance factor R_{th}, which can be calculated using equation (4.1). The highest temperature is at the base of the heat sink T_{base}, as it is in direct contact with the part to be cooled, then the temperature gradually decreases to the last point in the fin, which in turn is in contact with the cooling air T_{∞}, and thus the heat is transferred by the convection method between the fin, and the air as explained previously. It is important to note that the temperature is not equal in all parts of the heat sink, for several reasons, including the fact that the heat sink is in good contact with the heat source, and there are no spaces between them or high spots that prevent the contact process well, as well as the process of airflow through, and around the fins, and the location of each fin from the cooling airflow. Through the results obtained from numerical studies after performing the simulation calculations of 1000 iterations, and by representing the temperature distribution in each heat sink design, it can be said that as expected, the highest temperature is at the base of the heat sink, and the temperature gradient varies across the rest of the parts of the heat sink. Where it is obvious that the lowest temperature on the front ends of the heat sink, because comes first to contact the cooling air through the fan, and then the temperature increase starts on the back because the cooling air became hotter after touching the front group since the air has absorbed heat from the front ends of the sink.

The thermal resistance is given in equation (4.1)[46]:

$$R_{th} = \frac{T_{base} - T_{\infty}}{Q} \quad (4.1)$$

Where R_{th} is thermal resistance, T_{base} is base temperature of the heat sink, T_{∞} is temperature of fluid and Q is heat rate.

4.2. THE RESULTS OF PRESSURE EFFECTS

The process of calculating the pressure drop in the heat sink is done by finding the difference between the pressure when the fluid enters the heat sink, and the pressure after the fluid leaves it, as the pressure drop is given in equation (4.2):

$$\Delta P = P_{in} - P_{out} \quad (4.2)$$

Where ΔP is pressure drop, P_{in} is pressure inlet and P_{out} is pressure outlet.

Moreover, the pressure drop calculation has an important, and effective role in choosing the right fan for cooling. The higher-pressure difference leads to a greater capacity of the fan motor, and consequently the increase in its size, which is undesirable in terms of weight gain, as well as the increase in electricity consumption, and also its price in terms of cost. The extra more, smaller pressure difference, leads to smaller fan size as well as cheaper in price, also as a result of that weight of the product, become lighter, which is required in terms of design.

4.3. THE RESULTS OF SIMULATED SAMPLES

Below is a review of each of the designs used in this research in terms of showing the shape of the fin design, the distribution of heat, the pressure difference, and airstream across them illustrated in some figures.

4.3.1. The CAD of Ordinary Rectangular Plate Fins Heat Sink and Analyzing Results

As it was previously explained, using a computer program, the heat sink whose fins are rectangular plate, and as shown in (Figure 4.1), was designed through several steps, so that the base was first drawn, given the required size, and then from the extruded option to give it the required thickness. After that, the shape of the fin was drawn, and from the linear sketch pattern command, which is a feature in the program to save time, and effort to give the required number of fins, and then through the extruder also to obtain the required fin height.

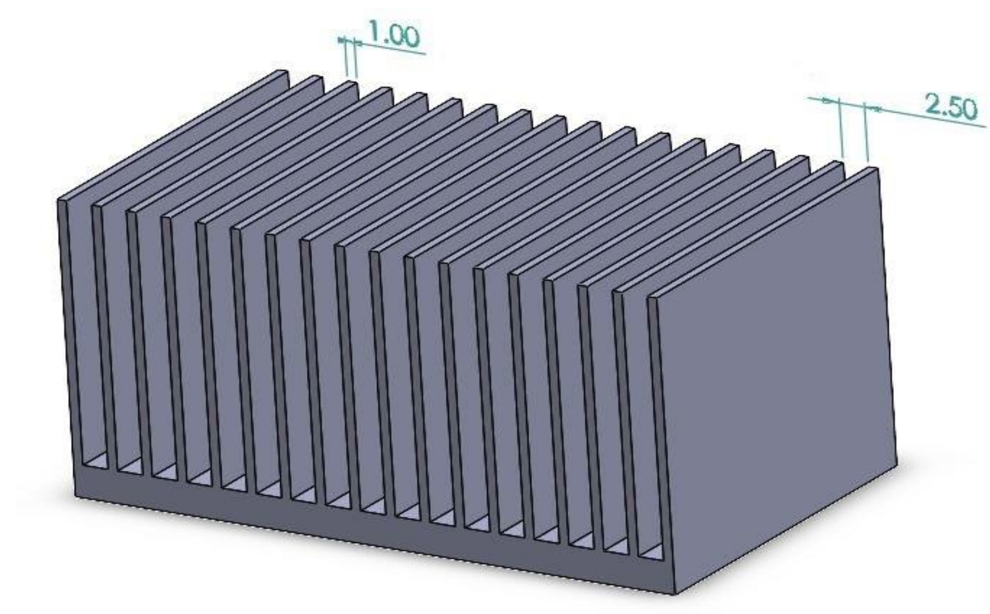

Figure 4.1: The CAD design of rectangular plate fins heat sink

(Figure 4.2) shows the heat spread across the rectangular plate fins heat sink in different degrees, through the gradation of colors on the figure, where the red color indicates the highest temperature, was 339.734 K which appears clearly at the base, due to its direct adhesion to the heat source, and then the temperature gradually decreases until it reaches the furthest point from the heat source, which is the end of the fin tip with the blue color, was 336.082 K which in turn is less hot.

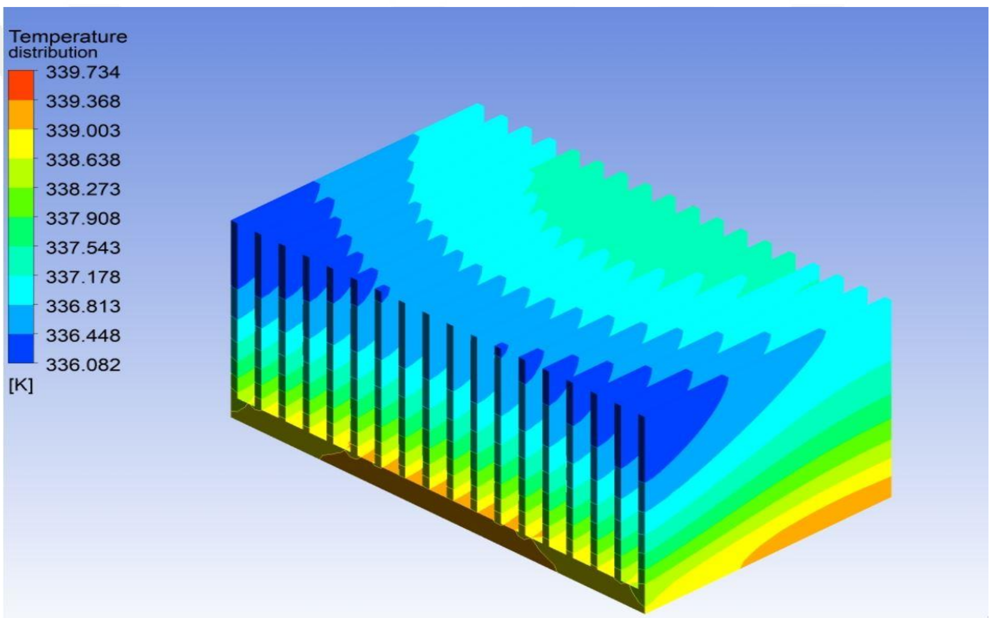

Figure 4.2: The result of temperature through the rectangular plate-fin heat sink

(Figure 4.3) shows the temperature difference across the rectangular plate fins heat sink, where a straight line has been drawn starting from the base, and ending after the end of the fin height to reach the surrounding air in order to collect points across this line to represent them as shown.

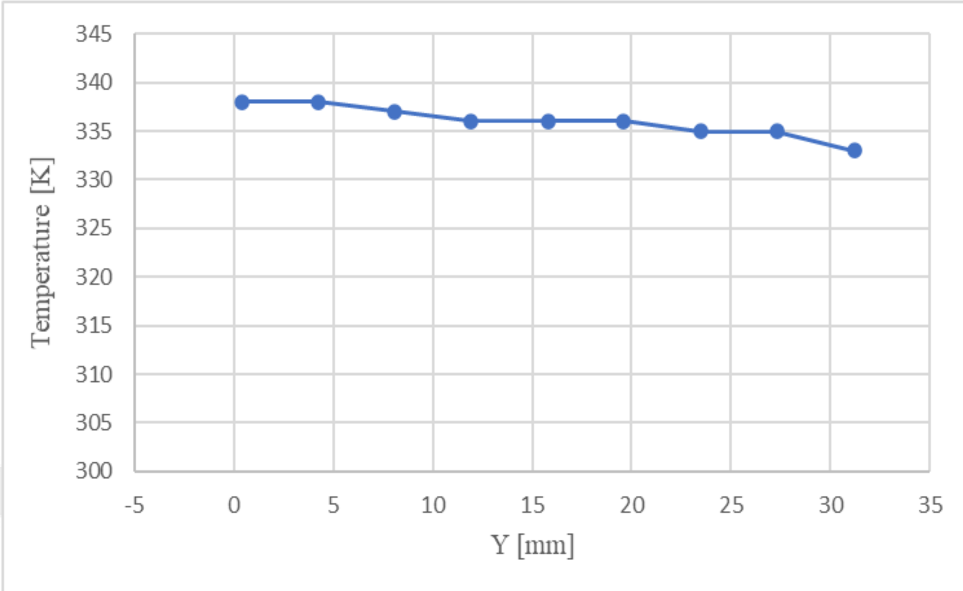

Figure 4.3: The temperature distribution of rectangular plate fin

(Figure 4.4) shows the gradient of the pressure difference across the rectangular plate fins heat sink through the gradation of colors, where the highest pressure difference appears in red, which was 2.941 Pa, which appears on the front face of the heat sink from the base to the end of the height of the fins, due to its geometric shape, and facing the air stream directly, while the lowest pressure difference appears in blue, which was -1.035 Pa and, the negative sign (-) means that it is less than zero, as the program considers the normal atmospheric pressure of 101,325 Pa as zero, and accordingly shows the results.

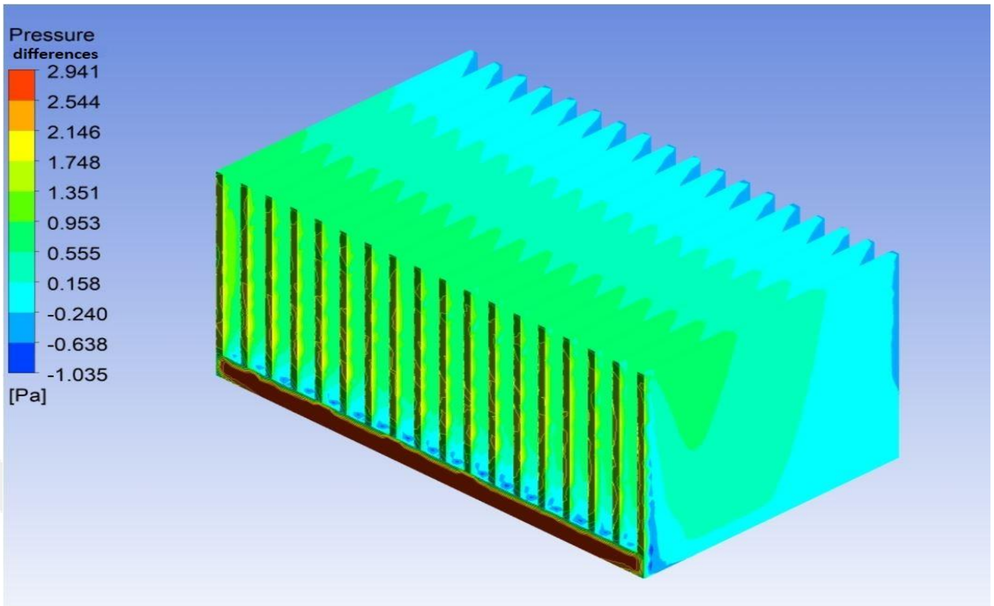

Figure 4.4: The pressure differences result from the rectangular plate fin

(Figure 4.5) shows the airflow through the rectangular plate fins heat sink with a group of lines in different colors to show the places where the maximum speed is shown in red, which was 2.174 m/s, while the places where the flow speed is least are shown in blue.

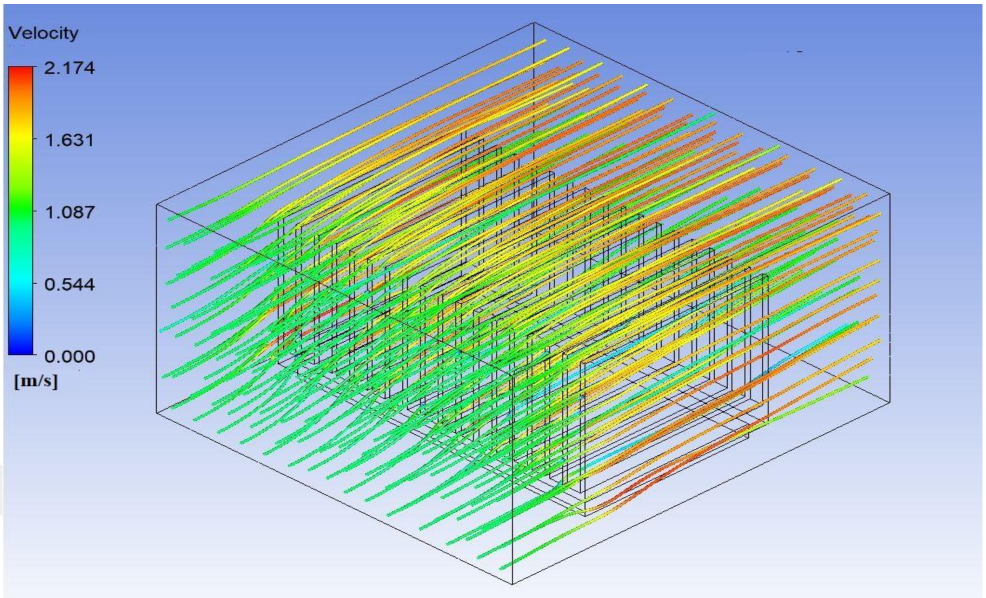

Figure 4.5: Velocity path through the rectangular plate-fin heat sink

4.3.2. The CAD of Separated Short Plate Fins Heat Sink and Analyzing Results

The heat sink with fins, which are in the form of separated short plates as shown in (Figure 4.6), was designed using the previously mentioned computer program in two stages. In the first stage, the base was designed, its dimensions were adjusted, given the required thickness, and in the second stage, the fins were designed as well as they were adjusted in terms of measurements.

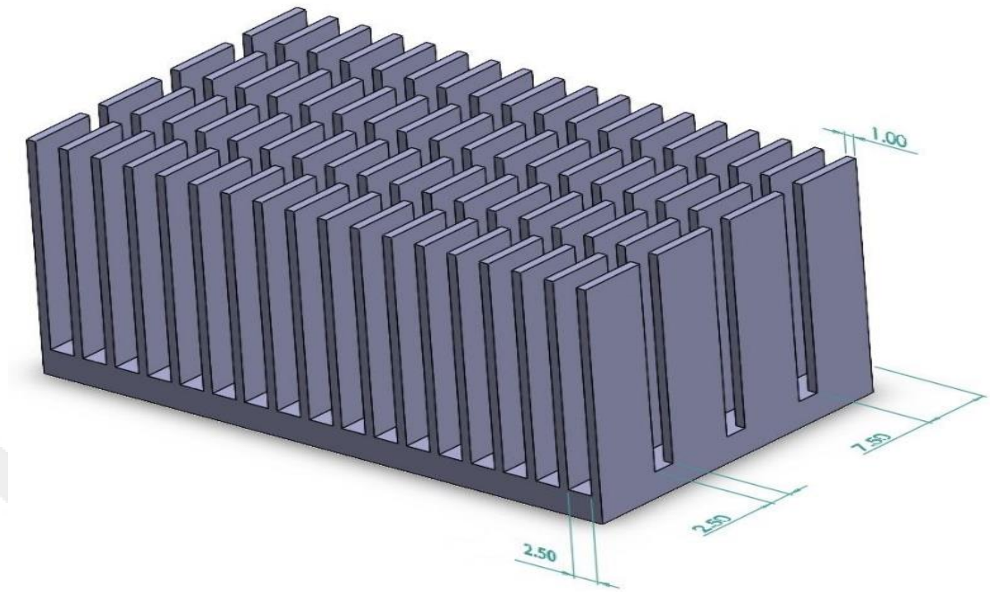

Figure 4.6: The CAD design of separated short plate fins heat sink

The results showed the distribution of heat across the separated short plate heat sink as shown in (Figure 4.7) through the gradation of colors, and the highest temperature was 339.281 K appeared distributed on the base, which was represented by red color, while the blue color indicated the lowest temperatures, which appeared at the top of the fins of the first row, due to the heat exchange between them, and the cooling air, then the temperature began to gradually rise for the rear rows because the cooling air had passed over them after gaining heat from the first row of fins.

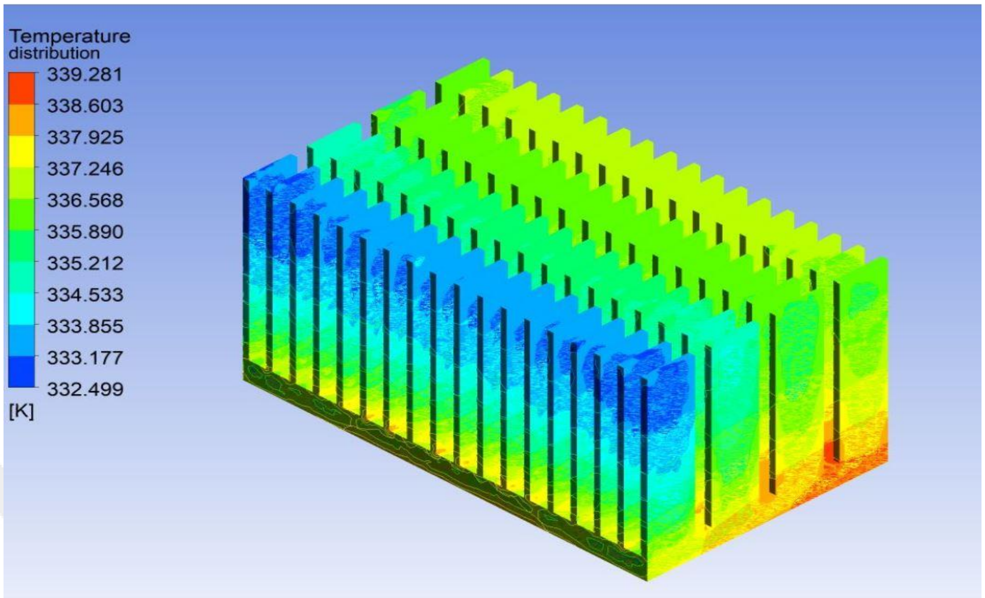

Figure 4.7: The result of temperature through separated short plate fins

(Figure 4.8) shows through several points, the spread of temperature across the separated short plate fins heat sink starting from the base, where the temperature was the highest, and then began to gradually decrease until the last point in the fin, which indicates that the heat sink has played an active role in the cooling process.

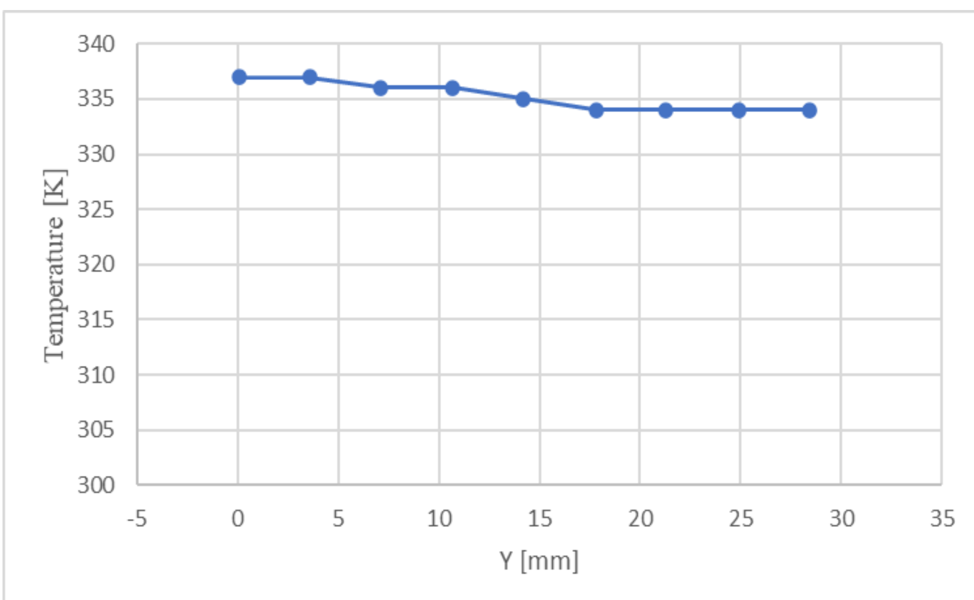

Figure 4.8: Temperature distribution through separated short plate fins

The distribution of the pressure difference across the separated short plate fins heat sink, and as shown in (Figure 4.9) across the color gradient, showing the highest value of the pressure difference, which was 3.093 Pa, indicated in red, as well as highlighting the region where the pressure difference is less in blue.

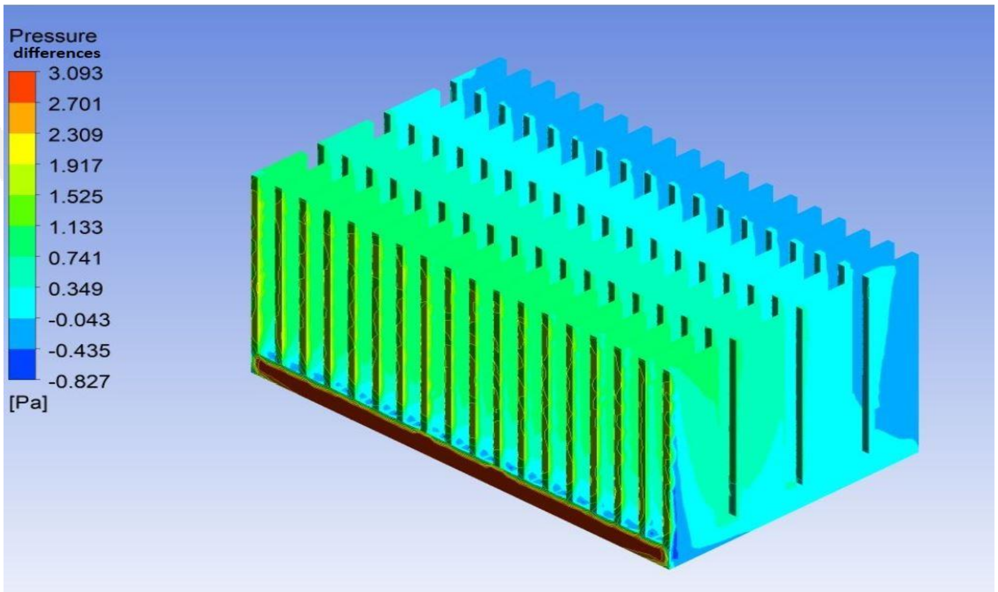

Figure 4.9: The result of pressure differences through separated short plate fins

The airflow through the separated short plate fins heat sink has been described through several lines of different colors showing the difference in velocity at each place. The highest flow velocity was 2.2 m/s, which was indicated in red, while the rest of the colors indicated the lower velocities, as shown in (Figure 4.10).

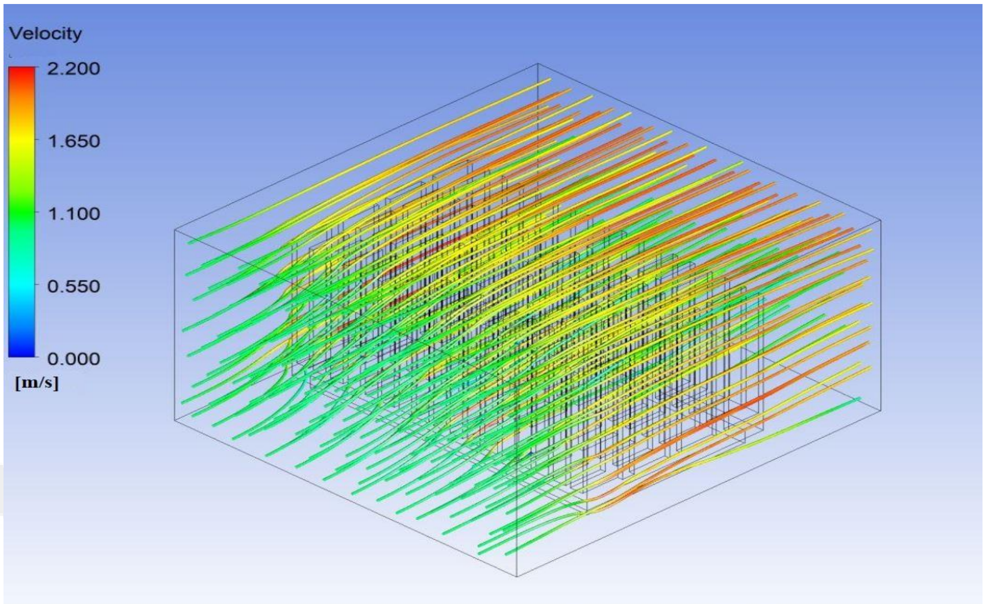

Figure 4.10: Velocity stream through separated short plate fins

4.3.3. The CAD of Separated Short Angled Plate Fins Heat Sink and Analyzing Results

As it was previously detailed that changing the design of the heat sink has an impact on the efficiency of the heat sink, the heat sink was designed with fins in the form of separated short angled plate fins as shown in (Figure 4.11), using the numerical design program, so that the model was designed with two steps base as a first step, and fins as a second step.

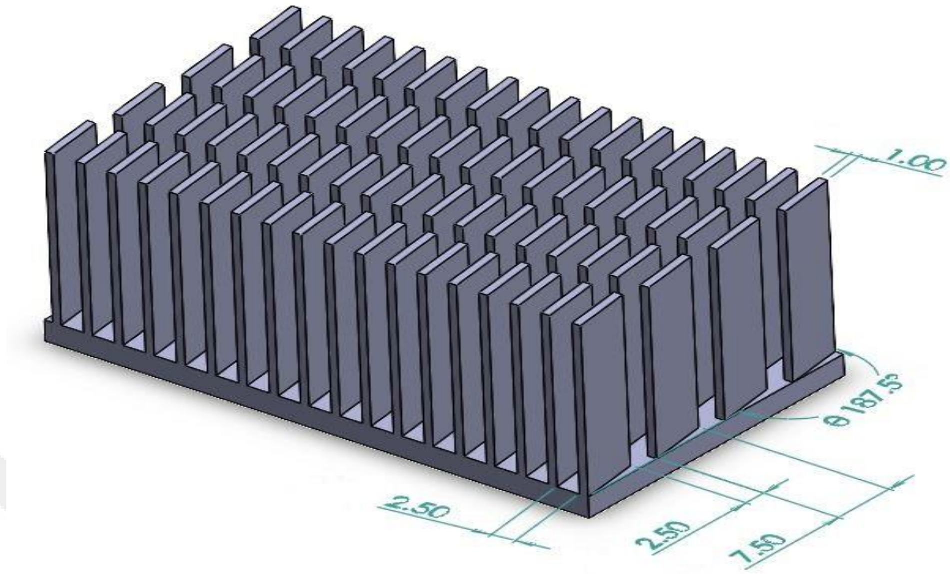

Figure 4.11: The CAD design of separated short angled plate fins heat sink

The results showed the temperature distribution across the separated short angled fins heat sink as shown in (Figure 4.12), by representing the colors for each temperature, where the results indicated the highest temperature in red, which was 335.979 K, while the lowest temperature was represented in blue, and the rest of the colors indicate the temperature between maximum and minimum.

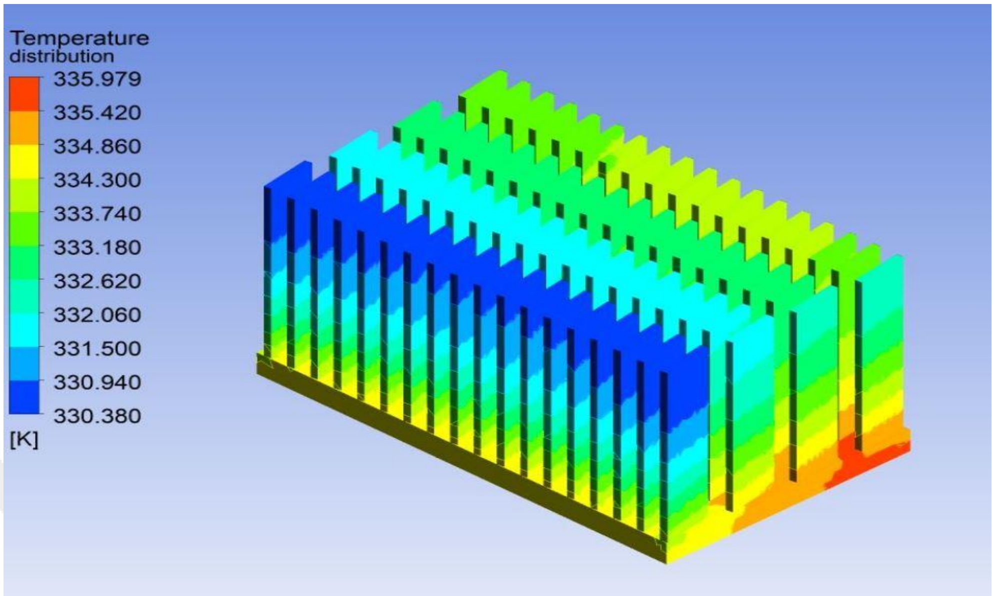

Figure 4.12: The result of temperature through the separated short angled plate fins heat sink

(Figure 4.13) illustrates the temperature distribution through the separated short angled plate fins heat sink from the highest temperature which was across the base to the lowest temperature which was in the top of the fin.

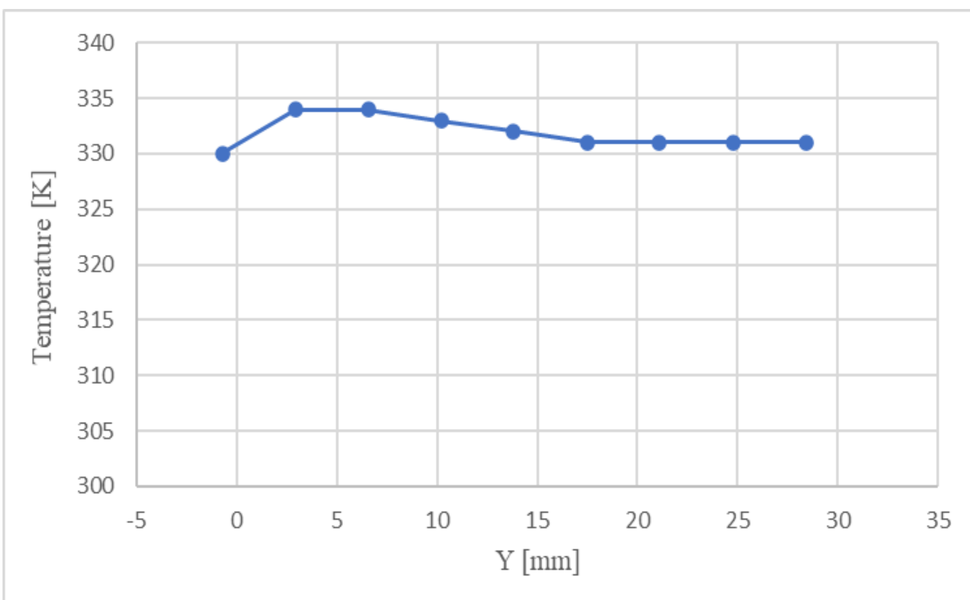

Figure 4.13: Temperature distribution through separated short angled plate fins

The distribution of the pressure difference in the separated short angled plate fins heat sink through the results, was represented based on the maximum pressure difference, and the lowest pressure difference by color gradation, and as shown in (Figure 4.14), where the highest-pressure difference was 4.100 Pa represented by the red color, which appeared on the front end of the heat sink, while the rest of the values were represented by several colors to the other values, where the blue color was representing the smallest value.

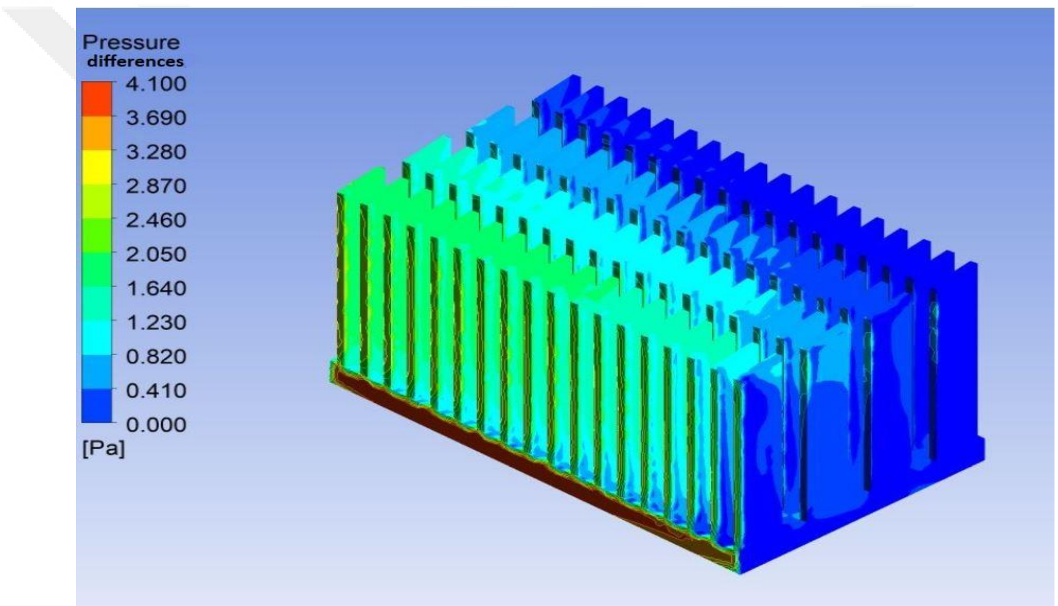

Figure 4.14: Pressure result through separated short angled plate fins heat sink

The results of the velocity of cooling airflow through the separated short angled plate fins heat sink as shown in (Figure 4.15), by clarifying the places of the maximum speed, which were represented in red color, where it was 2.561 m/s, then the colors were gradually inserted to the lowest speed.

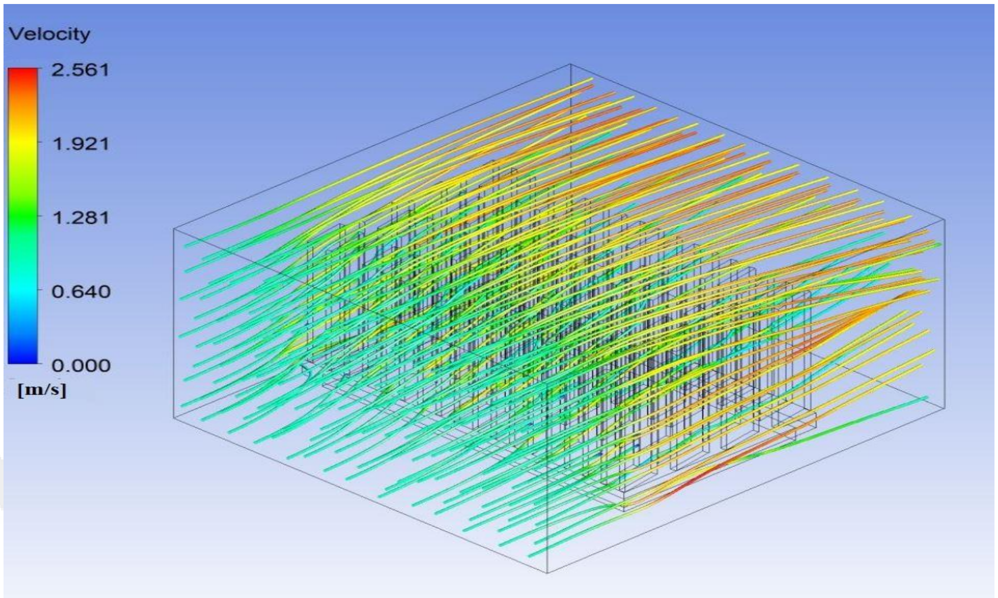

Figure 4.15: Velocity stream through separated short angled plate fins heat sink

4.3.4. The CAD of the Separated Short Airfoil Fins Heat Sink and Analyzing Results

Among the designs that were selected in this study, the separated short airfoil fins heat sink as shown in (Figure 4.16) was designed using the design program adopted in this research, by designing the base firstly, and then after that designing the fins.

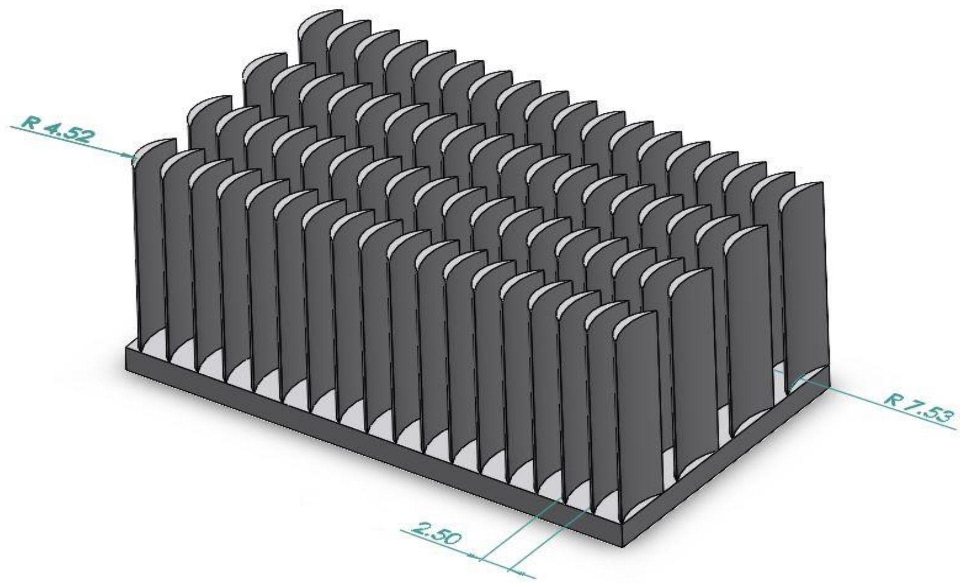

Figure 4.16: The CAD design of separated short airfoil fins heat sink

Through the results, and as shown in (Figure 4.17), which shows the distribution of temperature across the heat sink in several different colors, where the maximum temperature was 339.577 K, and represented in red, while the rest of the lower temperature was represented by other different colors where the blue color represented the lowest temperature.

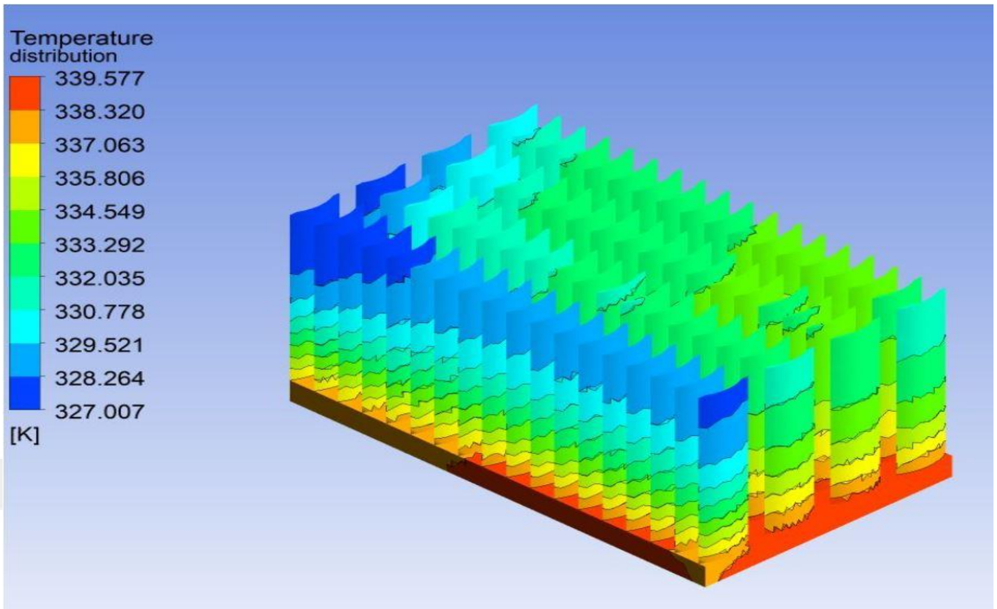

Figure 4.17: The result of temperature through the separated short airfoil fins heat sink

(Figure 4.18) shows the distribution of heat across the separated short airfoil fins heat sink, with the highest temperature at the base, and the lowest temperature at the top of the fin.

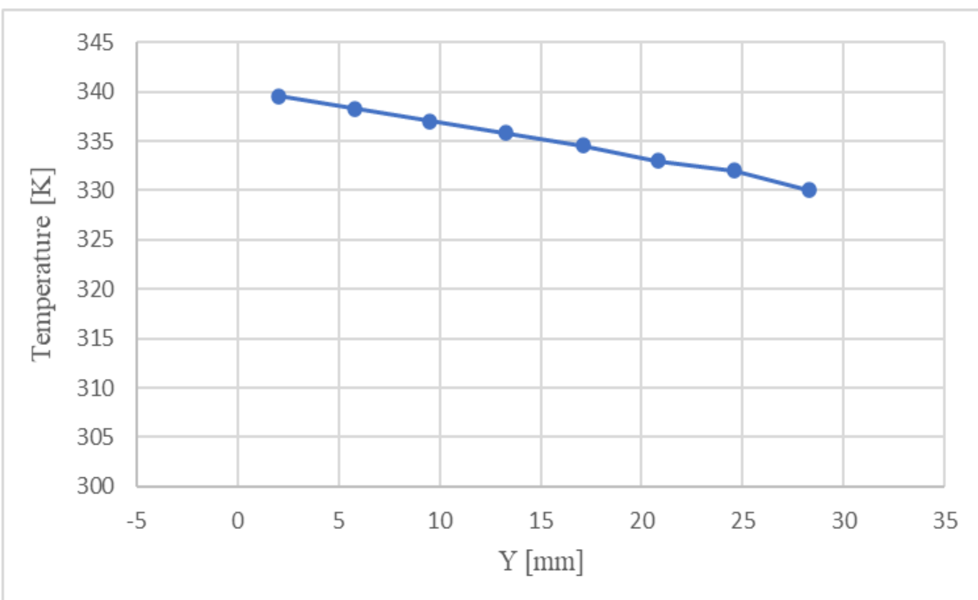

Figure 4.18: Temperature distribution through separated short airfoil fins

The results of the pressure difference across the separated short airfoil fins heat sink as in (Figure 4.19), and as shown in colors to represent the largest pressure difference which was 2.720 Pa indicated in red, as well as the minimum pressure difference indicated in blue, and also indicating the rest of the pressure difference values in different colors between the largest, and smallest value.

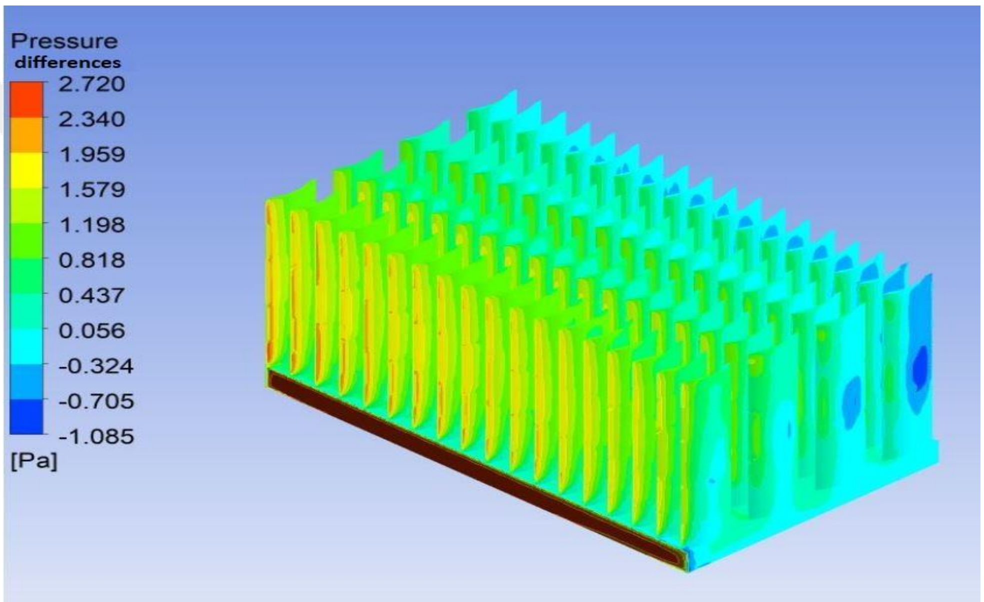

Figure 4.19: Pressure result through separated short airfoil fins heat sink

(Figure 4.20) shows the speed of the cooling air through the separated short airfoil fins heat sink, which was represented in several colors to show the airspeed at each part of the heat sink, where the maximum airspeed was 2.129 m/s, and represented in red, while the rest of the colors show the lower and lower speeds.

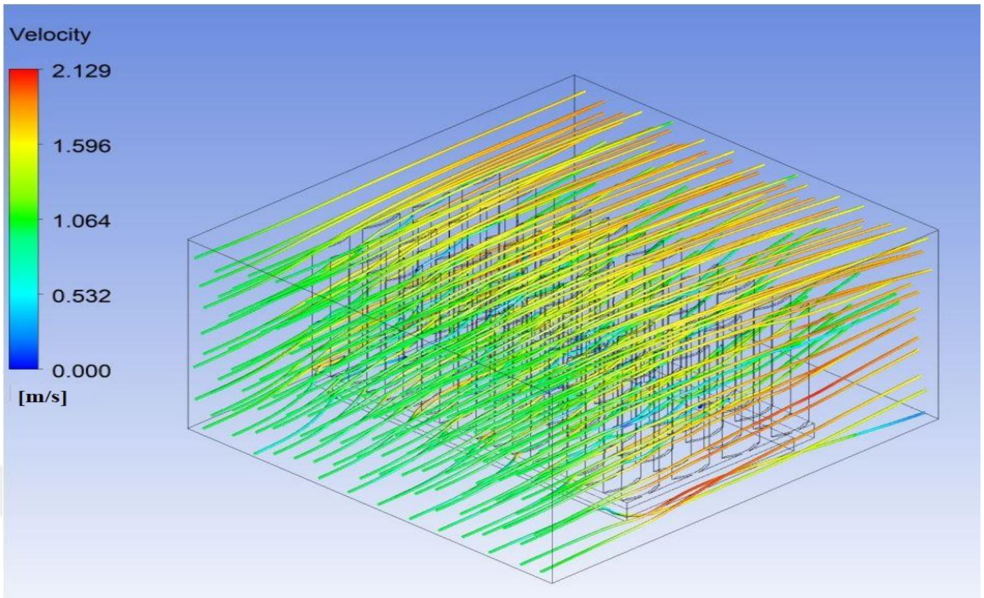

Figure 4.20: Velocity stream through separated short airfoil fins heat sink

4.3.5. The CAD of The Continuous Zigzag Fins Heat Sink and Analyzing Results

The heat sink, which has fins in the form of a continuous zigzag, was designed as shown in (Figure 4.21) using the same design program, and the same steps were followed for the rest of the previous designs.

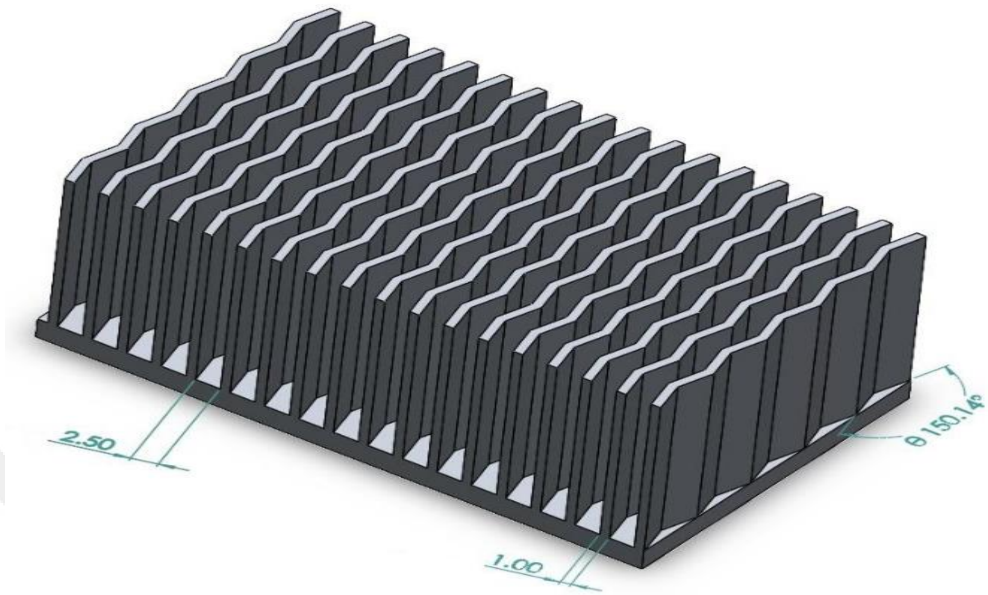

Figure 4.21: The CAD design of continuous zigzag fins heat sink

The results showed in (Figure 4.22), the distribution of heat that transmitted from the heat source through the continuous zigzag fins heat sink, where it was represented by several different colors, so the maximum temperature was 343.818 K, which was represented in red color, and most of it was in the base area, while the temperature began to gradually decrease until it reached the last point in the fin in which the temperature was the lowest, which is indicated in blue.

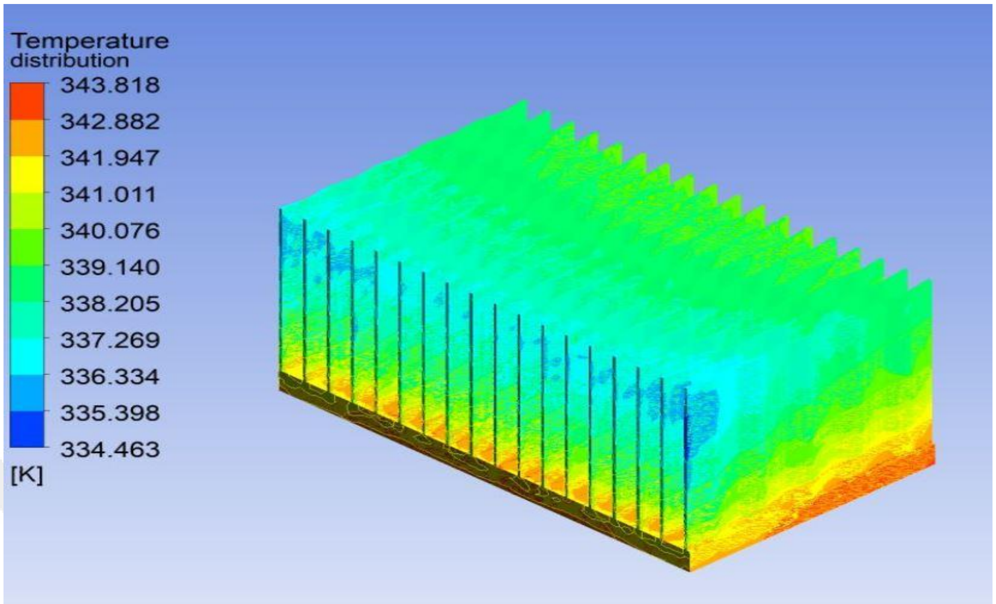

Figure 4.22: The temperature result of continuous zigzag fins

(Figure 4.23) shows that the highest temperature in the continuous zigzag fins heat sink was at the base 344 K, and then continued to decrease gradually across the fin due to heat flow from the base to the fins, and also due to the heat exchange between the fins and the cooling air.

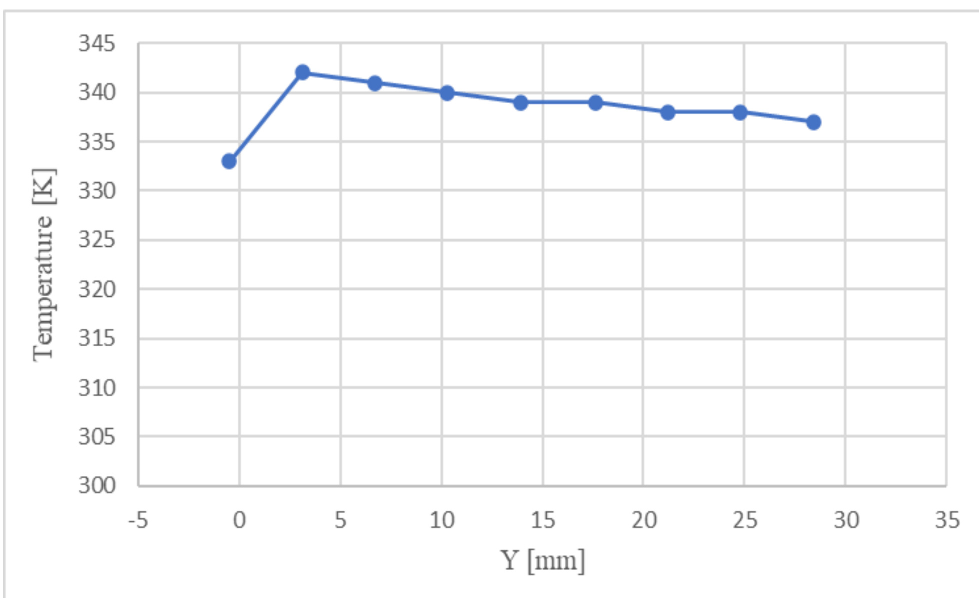

Figure 4.23: The temperature distribution through the continuous zigzag fins heat sink

(Figure 4.24) represents the pressure differences across the continuous zigzag fins heat sink, where the maximum pressure difference was indicated in red, which was 2.439 Pa, and the rest of the lowest pressure difference values were indicated by several colors, up to the lowest pressure difference represented in blue.

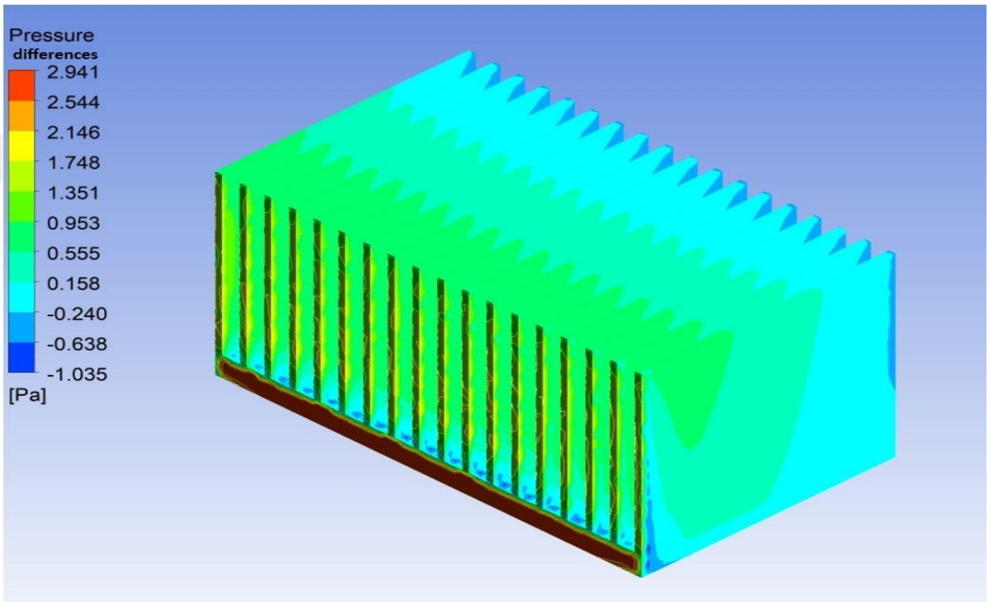

Figure 4.24: Pressure result through the continuous zigzag fins heat sink

(Figure 4.25) represents the path of the cooling airflow velocity through the continuous zigzag fins heat sink by several colors, where the maximum speed was 2.028 m/s, and represented in red color, while the airspeed at each area of the heat sink was illustrated through the color gradation, which reflects the lower speeds.

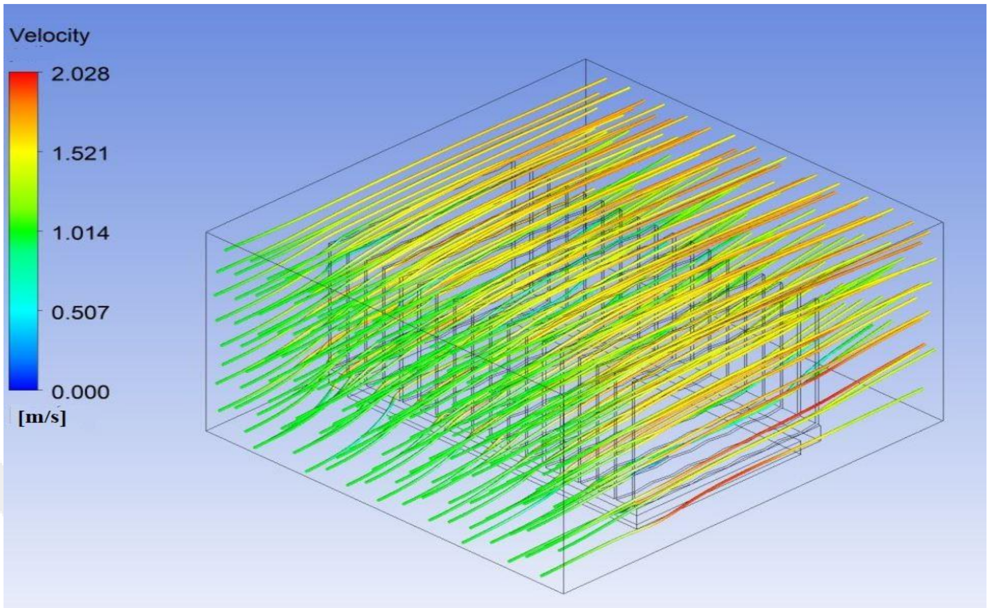

Figure 4.25: The velocity stream through the continuous zigzag fins heat sink

4.3.6. The CAD of The Intermittent Zigzag Fins Heat Sink and Analyzing Results

A supplement to the rest of the designs used in this study, in addition to choosing some strange designs for the fin shape, to improve the performance of the heat sink, and as shown in (Figure 4.26), the intermittent zigzag fins heat sink was designed using the design program, and the same previous steps followed but taking into account the difference in the shape of the fin.

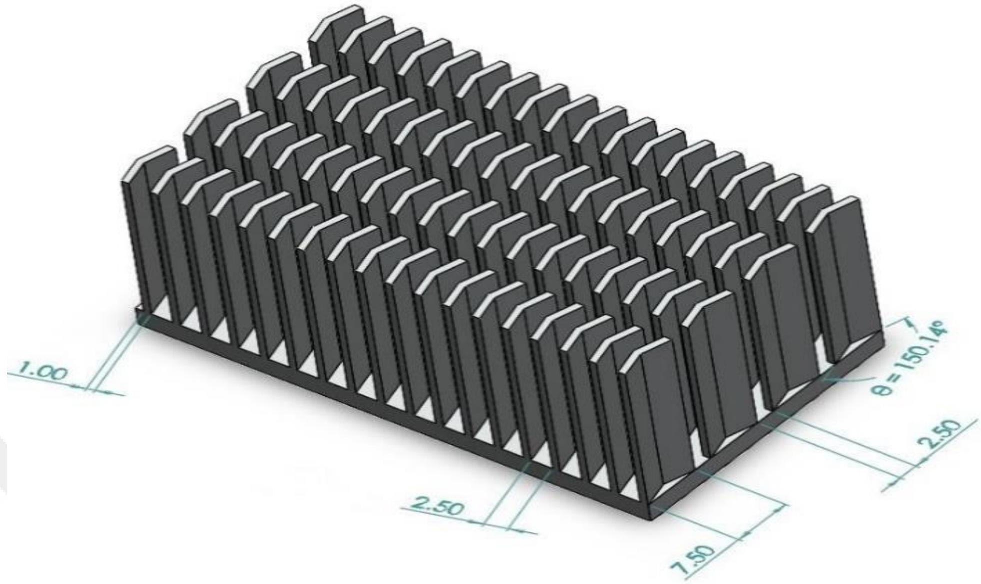

Figure 4.26: The CAD design of intermittent zigzag fins heat sink

(Figure 4.27) shows the temperature distribution across the intermittent zigzag fins heat sink, where the maximum temperature was represented in red, which was 347.224 K, and the minimum temperature was 333.591 K which is represented in blue, while the rest of the colors represented the temperatures between the maximum and minimum.

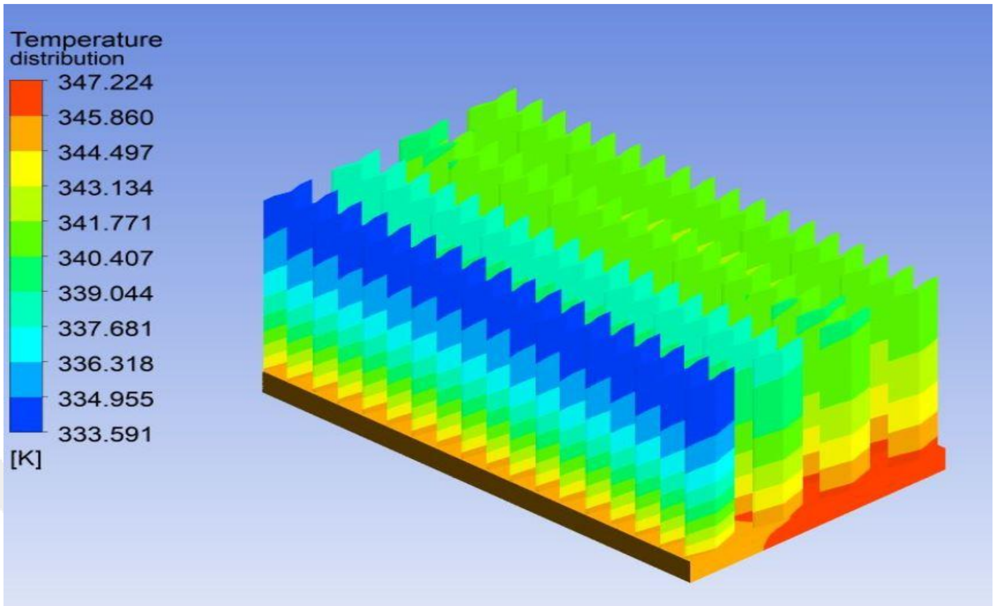

Figure 4.27: The temperature result of intermittent zigzag fins

(Figure 4.28) represents the temperature distribution across the intermittent zigzag fins heat sink from the base, where the maximum temperature is 347 K, and then it started declining through the fins until it reached 336 K, as shown in the figure.

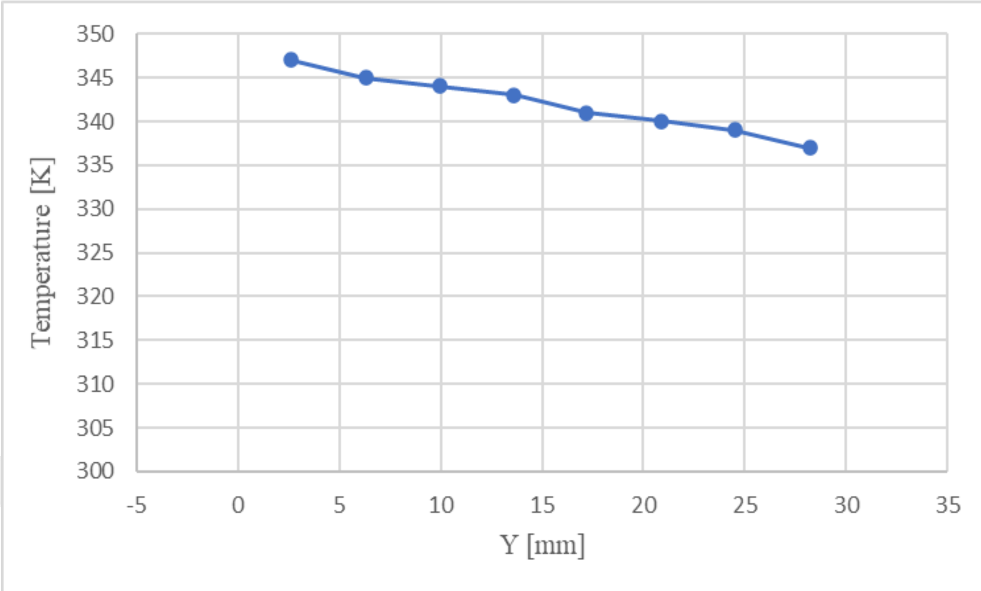

Figure 4.28: The temperature distribution through the intermittent zigzag fins heat sink

Through the results of the pressure differences across the intermittent zigzag fins heat sink shown in (Figure 4.29), the maximum pressure difference was 2.475 Pa indicated in red, while the rest of the pressure differences were gradually represented by several colors to distinguish each region in the heat sink with a color that reflects the value of the pressure difference.

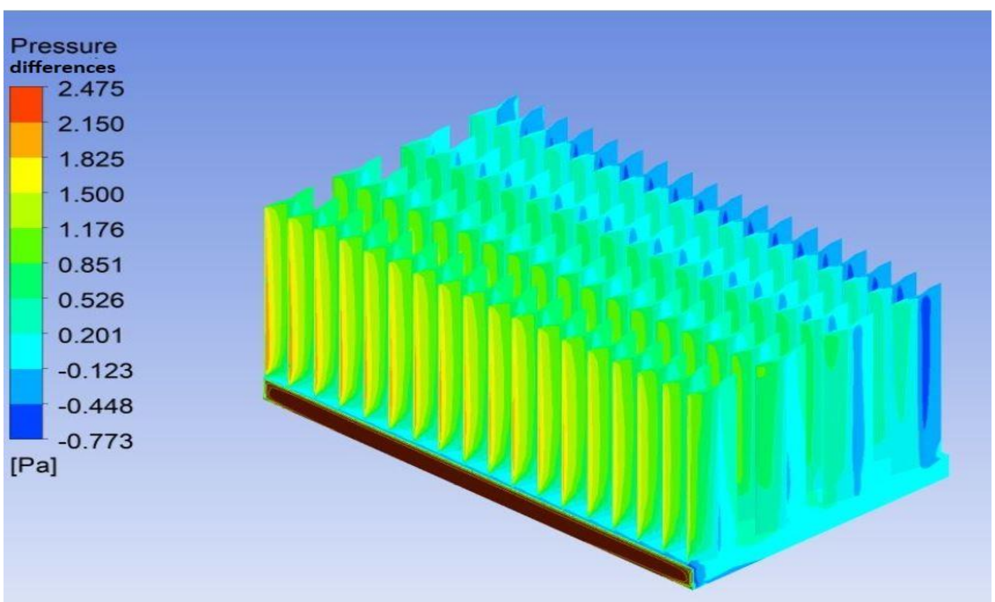

Figure 4.29: Pressure result through the intermittent zigzag fins heat sink

(Figure 4.30) represents the results of the cooling airflow velocity through the intermittent zigzag fins heat sink, showing the maximum speed in red, which was 2.033 m/s, and the rest of the velocities at each area of the heat sink were represented in a different color.

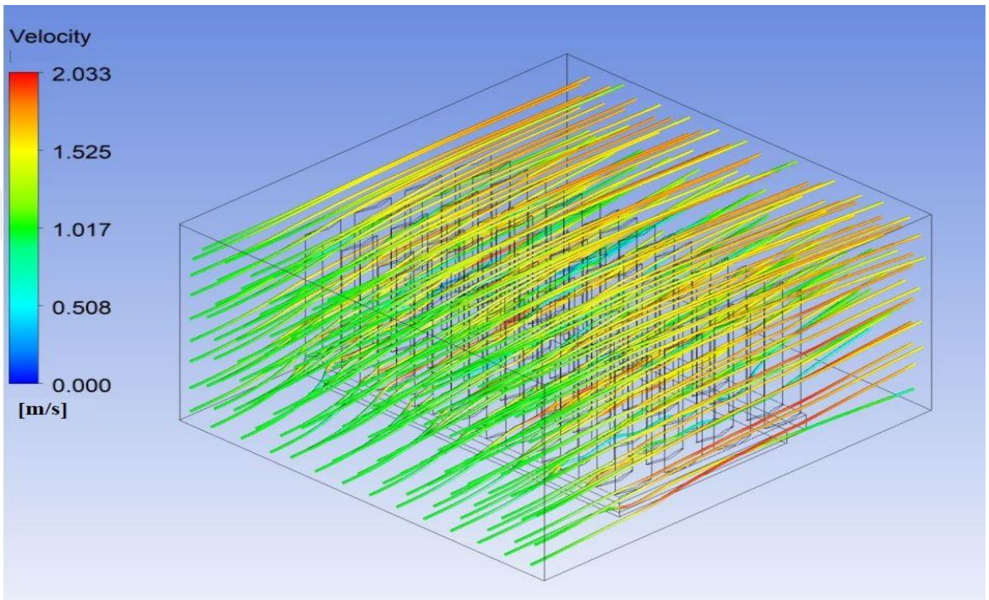

Figure 4.30: The velocity stream through the intermittent zigzag fins heat sink

4.4 THE DISCUSSION OF RESULTS

The results obtained from the simulation process for each of the targeted designs in the study were compared, by comparing the pressure difference as well as the thermal resistance and collecting these results in Table 4.1.

Table 4.1: The comparison between all heat sink types results

Heat sink type	Thermal resistance (R_{th}) W/K	Pressure drop (ΔP) Pa
Ordinary rectangular plate fins	0.397	3.9730
Separated short plate fins	0.394	3.9198
Separated short angled plate fins	0.361	5.6340
Separated short airfoil fins	0.386	3.8050
Continuous zigzag fins	0.440	3.2588
Intermittent zigzag fins	0.472	3.2040

These results were also compared, and clarified in (Figure 4.31), and (Figure 4.32) to give a simplified idea of the comparison process as well as to facilitate the distinction between designs in terms of efficiency.

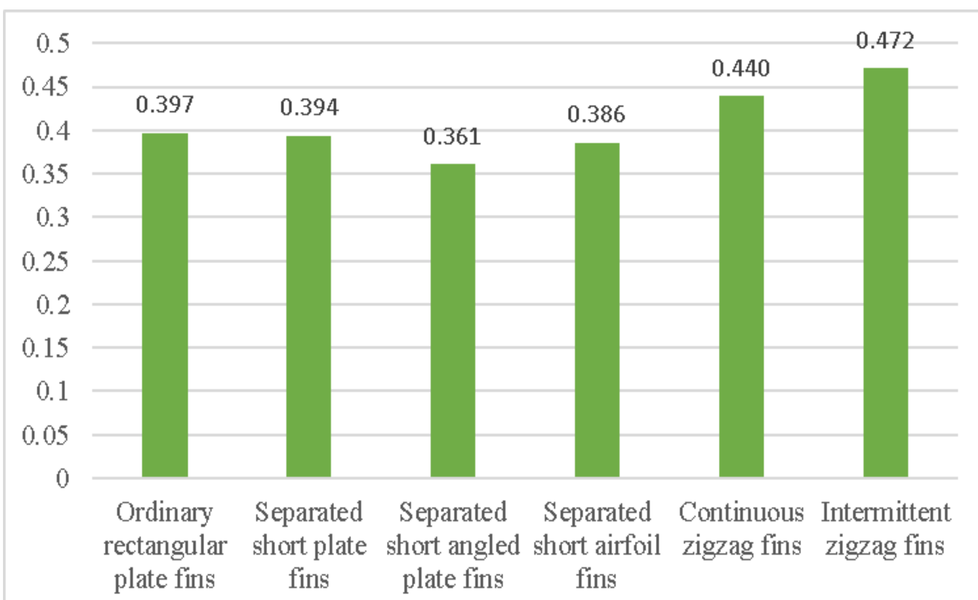

Figure 4.31: Thermal resistance comparison

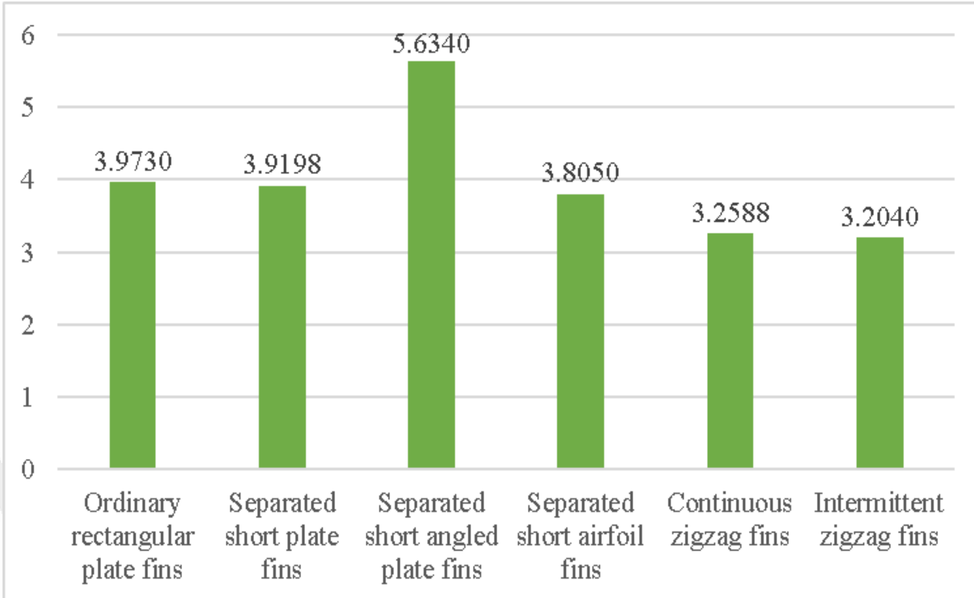

Figure 4.32: Pressure drop comparison

Through the process of comparing the results of the targeted models in this study, it is clear through the evaluation process that the separated short angled plate fins heat sink indicated in (Figure 4.12) is superior to the rest of the designs in terms of thermal resistance, as it has the lowest resistance. Thus, the heat flows from the heat source through the heat sink adjacent to it smoothly, and without hindrance, so that the heat sink later dissipates it through the surrounding medium, thus fulfilling its role satisfactorily. While the results in terms of pressure difference comparison showed that the heat sink with intermittent zigzag fins shown in (Figure 4.29) is the lowest in the pressure difference, it outperformed the rest of the designs by recording the least resistance to the flow of cooling fluid through it. Thus, it gave an important indication in the process of selecting the fan as it is a major factor in terms of economic cost. The larger the pressure difference, which leads to choosing a larger fan with higher capacity, and thus a greater cost, which is not desirable.

5. CONCLUSION

In this research, reducing 50% of the heat sink size by changing the fin shape as well as changing the material is done successfully, and the performance of six different heat sink models in terms of fin shape was studied, and analyzed. Where a commercial program was used in the design of these models, taking into account that the design is not complicated, as well as easy to manufacture, and inexpensive. A CFD digital analysis tool was used, and it was set to give a simulation model to be as close as possible to the real conditions, by comparison with the results of the previously mentioned experiment. The designs were analyzed under conditions of forced convection, so that the uniformity of both the conduction heat transfer phenomenon that appears between the heat source, and the heat sink on the one hand, and the convective heat transfer phenomenon that appears between the heat sink, and the surrounding medium, on the other hand, was studied, which is known as the conjugate. As indicated earlier in the objectives of the thesis, taking into account both the thermal resistance parameters as well as the pressure drop are the basis for the principle of analysis, and the study of all models. Through the results, the heat sink with fins that are in the form of separated short angled plates was superior to the rest of the models, as it showed the lowest thermal resistance, which amounted to 0.3610 K/W, but on the other hand, possessed the highest pressure drop which was 5.6340 Pa. As for the pressure drop parameter, the heat sink with fins in the form of intermittent zigzag was distinguished over the rest of the designs, as it had the lowest pressure drop, which was 3.2040 Pa, but also had owned the highest thermal resistance which was 0.4720 K/W. As for the rest of the designs, the results were different. In the ordinary rectangular plate fins heat sink, the thermal resistance was 0.3970 K/W, and the pressure difference was 3.9730 Pa. The heat sink in the fin form of the separated short plates was a thermal resistance of 0.3940 K/W, while the pressure difference was 3.9198 Pa. The airfoil fins heat sink has a thermal resistance value of 0.3860 K/W, and the pressure difference was 3.8050 Pa. Also, the continuous zigzag fins heat sink has a thermal resistance of 0.440 K/W, and the pressure difference was 3.2588 Pa. It can be said from the above results of this research that the digital CFD instrument is an effective tool that can be relied upon in the analysis, and study as it saves time, effort as well as cost, and also in extracting results that are considered accurate, so that if they are set correctly from the beginning in any research. In addition to this, the geometry of the fin, as well as the distance between the fins or what is known as the fin duct, affect the performance of the heat sink in terms of heat dissipation on one hand and the pressure difference on the other hand.

FUTURE WORK

In order to strive to increase the improvement, and enhancement of the efficiency of the heat sink so that it performs its tasks in the cooling process at the lowest possible cost, as well as reducing its size, on the one hand, to comply with the requirements of advanced electronic manufacturing, and also to ensure that it meets the design requirements. In the next research, work will be done to enhance the efficiency of the heat sink through several steps, including:

i. Working to strengthen the material from which the heat sink is made by adding some materials (composite material) to it that would enhance the conductivity of the heat sink.
ii. Designing the fins with shapes that aim to increase the surface area of the heat sink on the one hand and the other hand, not to generate a high-pressure difference.
iii. It is possible to work on the used cooling fluid to obtain higher efficiency by changing the type or using more than one fluid in a single cooling process.
iv. Also, the type of cooling fluid flow can be worked on, being laminar or turbulent or a mixture between the two to obtain the best results.

REFERENCES

[1] S. Lee, M. Early, and M. Pellilo, "Thermal interface material performance in microelectronics packaging applications," *Microelectronics Journal*, vol. 28, no. 1. Elsevier Ltd, pp. xiii–xx, Jan. 01, 1997. doi: 10.1016/s0026-2692(97)87871-8.

[2] R. Mohan and P. Govindarajan, "Experimental and CFD analysis of heat sinks with base plate for CPU cooling †," *Journal of Mechanical Science and Technology*, vol. 25, no. 8, pp. 2003–2012, 2011, doi: 10.1007/s12206-011-0531-8.

[3] X. Yu, J. Feng, Q. Feng, and Q. Wang, "Development of a plate-pin fin heat sink and its performance comparisons with a plate fin heat sink," *Applied Thermal Engineering*, vol. 25, no. 2–3, pp. 173–182, Feb. 2005, doi: 10.1016/j.applthermaleng.2004.06.016.

[4] S. Subramanyam and K. E. Crowe, "Rapid design of heat sinks for electronic cooling using computational and experimental tools," *Annual IEEE Semiconductor Thermal Measurement and Management Symposium*, pp. 243–251, 2000, doi: 10.1109/stherm.2000.837090.

[5] M. A. Ismail, M. Z. Abdullah, and M. A. Mujeebu, "A CFD-based experimental analysis on the effect of free stream cooling on the performance of micro processor heat sinks," *International Communications in Heat and Mass Transfer*, vol. 35, no. 6, pp. 771–778, Jul. 2008, doi: 10.1016/j.icheatmasstransfer.2008.02.012.

[6] W. : Www, D. Gupta, V. Venkataraman, and R. Nimje, "International Journal of Emerging Technology and Advanced Engineering CFD & Thermal Analysis of Heat Sink and its Application in CPU," 2008.

[7] Y. CENGEL, "HEAT TRANSFER Apractical Approach," *Applied Energy*, vol. 39, no. 4, pp. 341–344, 1991, doi: 10.1016/0306-2619(91)90004-h.

[8] S. Feng, M. Shi, H. Yan, S. Sun, F. Li, and T. J. Lu, “Natural convection in a cross-fin heat sink,” *Applied Thermal Engineering*, vol. 132, pp. 30–37, Mar. 2018, doi: 10.1016/j.applthermaleng.2017.12.049.

[9] “Thermal Management of PCIe Cards.” https://www.qats.com/Applications/Thermal-Management-PCIe (accessed Apr. 28, 2021).

[10] M. Korpyś, G. Dzido, and J. Wójcik, “Experimental and CFD investigation of commercial PC heat sink performance using water and nanofluids,” 2012.

[11] S. Majumder, A. Majumder, and S. Bhaumik, “3-Dimensional numerical study of cooling performance of a heat sink with air-water flow through mini-channel,” in *AIP Conference Proceedings*, Jul. 2016, vol. 1754, no. 1, p. 050010. doi: 10.1063/1.4958401.

[12] M. Bronson De Stadler, “Optimization of the Geometry of a Heat Sink,” 2007.

[13] K. Yakut, N. Alemdaroglu, I. Kotcioglu, and C. Celik, “Experimental investigation of thermal resistance of a heat sink with hexagonal fins,” 2006, doi: 10.1016/j.applthermaleng.2006.03.008.

[14] S. Lee, “Optimum Design and Selection of Heat Sinks,” *IEEE Transactions on Components Packaging and Manufacturing Technology Part A*, vol. 18, no. 4, pp. 812–817, 1995, doi: 10.1109/95.477468.

[15] M. Dogan and M. Sivrioglu, “Experimental investigation of mixed convection heat transfer from longitudinal fins in a horizontal rectangular channel: In natural convection dominated flow regimes,” *Energy Conversion and Management*, vol. 50, no. 10, pp. 2513–2521, Oct. 2009, doi: 10.1016/j.enconman.2009.05.027.

[16] N. Zheng and R. A. Wirtz, “Cylindrical pin-fin fan-sink heat transfer and pressure drop correlations,” *IEEE Transactions on Components and Packaging Technologies*, vol. 25, no. 1, pp. 15–22, Mar. 2002, doi: 10.1109/6144.991170.

[17] M. Lindstedt, K. Lampio, and R. Karvinen, “Optimal Shapes of Straight Fins and Finned Heat Sinks,” *Journal of Heat Transfer*, vol. 137, no. 6, Jun. 2015, doi: 10.1115/1.4029854.

[18] D. Singh and J. Doom, “Simulations of impinging jets with a range of configuration,” 2017. doi: 10.2514/6.2017-0751.

[19] A. Soni, “Study of Thermal Performance between Plate-fin, Pin-fin and Elliptical Fin Heat Sinks in Closed Enclosure under Natural Convection,” *International Advanced Research Journal in Science, Engineering and Technology ISO*, vol. 3297, no. 11, 2007, doi: 10.17148/IARJSET.2016.31126.

[20] K. Lampio and R. Karvinen, “Optimization of convectively cooled heat sinks,” *Microelectronics Reliability*, vol. 79, pp. 473–479, Dec. 2017, doi: 10.1016/j.microrel.2017.06.011.

[21] R. Mohan and P. Govindarajan, “THERMAL ANALYSIS OF CPU WITH COMPOSITE PIN FIN HEAT SINKS,” 2010.

[22] R. S. Sukumar, R. S. Sukumar, G. Sriharsha, S. B. Arun, P. D. Kumar, and Ch. Sanyasi, “Modelling And Analysis Of Heat Sink With Rectangular Fins Having Through Holes,” *CiteSeerX logo*.

[23] E. M. Dede, S. N. Joshi, and F. Zhou, “Topology Optimization, Additive Layer Manufacturing, and Experimental Testing of an Air-Cooled Heat Sink,” *Journal of Mechanical Design, Transactions of the ASME*, vol. 137, no. 11, Nov. 2015, doi: 10.1115/1.4030989.

[24] W. W. Wits, S. J. Weitkamp, and J. Van Es, “Metal additive manufacturing of a high-pressure micro-pump,” in *Procedia CIRP*, Jan. 2013, vol. 7, pp. 252–257. doi: 10.1016/j.procir.2013.05.043.

[25] D. B. Tuckerman and R. F. W. Pease, "High-Performance Heat Sinking for VLSI," *IEEE Electron Device Letters*, vol. EDL-2, no. 5, pp. 126–129, 1981, doi: 10.1109/EDL.1981.25367.

[26] Y. Zhang, S. Wang, and P. Ding, "Effects of channel shape on the cooling performance of hybrid micro-channel and slot-jet module," *International Journal of Heat and Mass Transfer*, vol. 113, pp. 295–309, Oct. 2017, doi: 10.1016/j.ijheatmasstransfer.2017.05.092.

[27] J. Barrau, D. Chemisana, J. Rosell, L. Tadrist, and M. Ibañez, "An experimental study of a new hybrid jet impingement/micro-channel cooling scheme," *Applied Thermal Engineering*, vol. 30, no. 14–15, pp. 2058–2066, Oct. 2010, doi: 10.1016/j.applthermaleng.2010.05.013.

[28] Y. Chen, C. Zhang, M. Shi, and J. Wu, "Three-dimensional numerical simulation of heat and fluid flow in noncircular microchannel heat sinks," *International Communications in Heat and Mass Transfer*, vol. 36, no. 9, pp. 917–920, Nov. 2009, doi: 10.1016/j.icheatmasstransfer.2009.06.004.

[29] P. S. Lee and S. v. Garimella, "Thermally developing flow and heat transfer in rectangular microchannels of different aspect ratios," *International Journal of Heat and Mass Transfer*, vol. 49, no. 17–18, pp. 3060–3067, Aug. 2006, doi: 10.1016/j.ijheatmasstransfer.2006.02.011.

[30] F. Y. Lim, S. Abdullah, and I. Ahmad, "Numerical study of fluid flow and heat transfer in microchannel heat sinks using anisotropic porous media approximation," 2010.

[31] X. Huang, W. Yang, T. Ming, W. Shen, and X. Yu, "Heat transfer enhancement on a microchannel heat sink with impinging jets and dimples," *International Journal of Heat and Mass Transfer*, vol. 112, pp. 113–124, Sep. 2017, doi: 10.1016/j.ijheatmasstransfer.2017.04.078.

[32] A. K. Barik, A. Mukherjee, and P. Patro, “Heat transfer enhancement from a small rectangular channel with different surface protrusions by a turbulent cross flow jet,” *International Journal of Thermal Sciences*, vol. 98, pp. 32–41, Jul. 2015, doi: 10.1016/j.ijthermalsci.2015.07.003.

[33] G. Ledezma and A. Bejan, “Heat sinks with sloped plate fins in natural and forced convection,” *International Journal of Heat and Mass Transfer*, vol. 39, no. 9, pp. 1773–1783, Jun. 1996, doi: 10.1016/0017-9310(95)00297-9.

[34] D. K. Kim, “Thermal optimization of plate-fin heat sinks with fins of variable thickness under natural convection,” *International Journal of Heat and Mass Transfer*, vol. 55, no. 4, pp. 752–761, Jan. 2012, doi: 10.1016/j.ijheatmasstransfer.2011.10.034.

[35] E. A. M. Elshafei, “Natural convection heat transfer from a heat sink with hollow/perforated circular pin fins,” in *2010 3rd International Conference on Thermal Issues in Emerging Technologies, Theory and Applications - Proceedings, ThETA3 2010*, 2010, pp. 185–193. doi: 10.1109/THETA.2010.5766397.

[36] B. Karimpourian and J. Mahmoudi, “Some important considerations in heatsink design,” in *Proceedings of the 6th International Conference on Thermal, Mechanical and Multi-Physics Simulation and Experiments in Micro-Electronics and Micro-Systems - EuroSimE 2005*, 2005, vol. 2005, pp. 406–413. doi: 10.1109/ESIME.2005.1502838.

[37] “Ansys Student Versions | Free Student Software Downloads.” https://www.ansys.com/academic/students (accessed Apr. 29, 2021).

[38] T. L. BERGMAN, A. S. LAVINE, F. P. INCROPERA, and D. DEWITT, *Fundamentals of Heat and Mass Transfer*. 2005.

[39] “SOLIDWORKS.” https://www.solidworks.com/ (accessed Apr. 28, 2021).

[40] A. Husain and K. Y. Kim, “Shape optimization of micro-channel heat sink for micro-electronic cooling,” *IEEE Transactions on Components and Packaging Technologies*, vol. 31, no. 2 SPEC. ISS., pp. 322–330, 2008, doi: 10.1109/TCAPT.2008.916791.

[41] “ANSYS Fluent Theory Guide,” vol. 15317, no. November, pp. 724–746, 2013.

[42] C. K. Loh and D. J. Chou, “Comparative analysis of heat sink pressure drop using different methodologies,” in *Annual IEEE Semiconductor Thermal Measurement and Management Symposium*, 2004, vol. 20, pp. 148–153. doi: 10.1109/stherm.2004.1291317.

[43] “Simcenter Flotherm | Siemens Digital Industries Software.” https://www.plm.automation.siemens.com/global/en/products/simcenter/flotherm.html (accessed Apr. 28, 2021).

[44] “Thermal Resistance Calculator for Plate Fin Heat Sink.” https://myheatsinks.com/calculate/thermal-resistance-plate-fin/ (accessed Apr. 28, 2021).

[45] “Thermal Design: Heat Sinks, Thermoelectrics, Heat Pipes, Compact Heat ... - H. S. Lee - Google.” https://books.google.com.tr/books?id=tAxzqKuqHaoC&printsec=frontcover&dq=heat+sink+design&hl=ar&sa=X&redir_esc=y#v=onepage&q=heat%20sink%20design&f=false (accessed Sep. 26, 2021).

[46] K. T. Chiang, “Optimization of the design parameters of Parallel-Plain Fin heat sink module cooling phenomenon based on the Taguchi method,” *International Communications in Heat and Mass Transfer*, vol. 32, no. 9, pp. 1193–1201, Oct. 2005, doi: 10.1016/J.ICHEATMASSTRANSFER.2005.05.015.