CZECH TECHNICAL UNIVERSITY IN PRAGUE

FACULTY OF MECHANICAL ENGINEERING
DEPARTMENT OF PROCESS ENGINEERING

HEAT TRANSFER OF ELECTRONIC DEVICE COOLING

BACHELOR’S THESIS

2023 GIANG TRUONG NGUYEN

ČVUT
ČESKÉ VYSOKÉ UČENÍ TECHNICKÉ V PRAZE

ZADÁNÍ BAKALÁŘSKÉ PRÁCE

I. OSOBNÍ A STUDIJNÍ ÚDAJE

Příjmení: **Nguyen** Jméno: **Giang Truong** Osobní číslo: **462144**
Fakulta/ústav: **Fakulta strojní**
Zadávající katedra/ústav: **Ústav procesní a zpracovatelské techniky**
Studijní program: **Strojírenství**
Studijní obor: **Energetika a procesní technika**

II. ÚDAJE K BAKALÁŘSKÉ PRÁCI

Název bakalářské práce:

Přestup tepla při chlazení elektronické součástky

Název bakalářské práce anglicky:

Heat transfer of electronic device cooling

Pokyny pro vypracování:

Optimal cooling of electronic components enables their proper function and prolongs their service life. The heat transfer is controlled by the thermal resistances of the individual layers and the method of heat dissipation from the electronic component. The work deals with the analysis of the heat flow through a composite wall representing an electronic component and its cooling element. As part of the bachelor thesis, perform:
1) Literature search on the topic of heat flux through the composite wall and estimation of contact thermal resistances between solid walls.
2) Analysis of the influence of thermal paste layers on the heat transfer between solid walls.
3) Simulations and assessment of parameters affecting the heat transfer efficiency when cooling an electronic device.
4) Comparison of heat transfer process using water and air for cooling the electronic device.

Seznam doporučené literatury:

VDI Heat Atlas. Second Edition. Berlin: Springer, [2010]. ISBN 3540778764.
BIRD, R. Byron, Warren E. STEWART a Edwin N. LIGHTFOOT. Transport phenomena. 2nd, rev. ed. New York: John Wiley, c2007. ISBN 978-0470115398.

Jméno a pracoviště vedoucí(ho) bakalářské práce:

doc. Ing. Jan Skočilas, Ph.D. ústav procesní a zpracovatelské techniky FS

Jméno a pracoviště druhé(ho) vedoucí(ho) nebo konzultanta(ky) bakalářské práce:

Datum zadání bakalářské práce: **19.10.2022** Termín odevzdání bakalářské práce: **16.01.2023**

Platnost zadání bakalářské práce: **24.09.2023**

doc. Ing. Jan Skočilas, Ph.D. podpis vedoucí(ho) práce	prof. Ing. Tomáš Jirout, Ph.D. podpis vedoucí(ho) ústavu/katedry	doc. Ing. Miroslav Španiel, CSc. podpis děkana(ky)

III. PŘEVZETÍ ZADÁNÍ

Student bere na vědomí, že je povinen vypracovat bakalářskou práci samostatně, bez cizí pomoci, s výjimkou poskytnutých konzultací.
Seznam použité literatury, jiných pramenů a jmen konzultantů je třeba uvést v bakalářské práci.

.
Datum převzetí zadání

Podpis studenta

I confirm that the diploma bachelor’s work was disposed by myself and independently, under leading of my thesis supervisor. I stated all sources of the documents and literature.

In Prague ……………………

………………………………
Giang Truong Nguyen

Acknowledgement

I would like to say a special thank you to doc. Ing. Jan Skočilas Ph.D. for the immense support, help and patience towards my thesis. I have learnt many great qualities with him not just from the thesis but also as a person. He has given up many of his free time in order for me to understand some of the problems. He has guided me from the very start and to the very finish and without out any hesitation, he would offer his help in no matter the conditions. I am truly grateful for him to be my supervisor and I can't thank him enough.

Annotation sheet

Name: Giang Truong
Surname: Nguyen
Title Czech:
Title English: Heat transfer of electronic device cooling
Scope of work: number of pages: 49
number of figures: 52
number of tables: 19
Academic year: 2022/2023
Language: English
Department: Department of Process Engineering
Specialization: Power Engineering and Process Technology
Supervisor: doc. Ing. Jan Skočilas Ph.D.

Annotation - English: The contents of this work are aimed at the impact of thermal paste parameters on heat transfer during the cooling process for electrical devices. It was suggested to use both water and air matters in two straightforward cooling system situations. The two designed cooling systems are first analysed by calculations of 1D problems to cool down the electrical device (CPU) below 60°C. CFD simulations are then carried out using 3D geometry to compare what different effects it has. Various suggestions for the ideal heat transfer coefficient that correlate with the flow rate of both water and air, along with the ideal thermal conductivity and the thickness of the thermal paste layer were investigated.

Keywords: heat conduction, heat transfer, thermal paste, cooling system

Index

Figures

Table

1. Introduction

The purpose of this research is to analyse the heat transfer from an electronic device to our cooling devices. This will consist of cooling the electronic device in two main ways, which is water cooling and air cooling. We will be exploring the heat transfer through a wall with specific heat resistant in order to find effect of different variables using simulation Computational Fluid Dynamic CFD to help us achieve our goal. The specific heat transfer will be represented by thermal paste. We will be proposing different types of ideas based on our literature research to help us maximise the efficiency of the cooling of the device.

Research about the heat flux through the literature will help us to understand the principle and fundamental of heat travelling through a material. We will be covering heat transfer methods such as convection and conduction in order for us to analyse and maximise our heat flow from the electronic device into our cooling device.

The contact thermal resistance between two walls is one of the key variables that affect heat transfer. Understanding the concept of contact thermal resistance will help us reduce down/increase (in case of cooling) the heat loss and is able to transfer all the heat efficiently to cool it down using multiple methods.

We will cover the use of thermal paste and its effect on heat transfer. This is one of the components that is very critical and important to be added between the source and sink of the heat to reduce contact thermal resistance based on the function it has towards heat transfer and how much more efficient it is to help cool the heat source. There are different types of thermal paste with different properties, which will help us achieve our goal.

Different ways and designs of the cooling device will be tested and compared with each other. We will introduce air cooling versus water cooling. Each of their own geometry will be designed from the start based on the research we have had and then will be simulated on Computational Fluid Dynamics Simulation to test out while changing the right variables to help us achieve cooling down the electronic device at a certain level.

2. Literature search

The literature search focusses mainly on the basic principle of heat transfer. The important role of increasing heat transfer between complex walls devices is heat resistance analysis. Contact heat resistance between two interfering objects is commonly reduced by thermal pastes. The main focus will also be on the effect of thermal pastes properties. Simulation tools such as CFD software will be used for problem analysis; therefore, a literature search will obtain results of new contributions dealing with simulation of heat transfer in the presence of thermal paste.

2.1 Heat transfer

The heat flux is defined by the amount of energy transferred per unit of time and per unit area perpendicular to the heat flow.

$$\dot{q} = \frac{d\dot{Q}}{dA} \tag{1}$$

The energy transferred per unit of time is referred to us heat flow rate.

$$\dot{Q} = \frac{dQ}{dT} \tag{2}$$

Heat is transferred by 3 essential phenomena: conduction, convection, and radiation. The most common are conduction and convection in our surroundings. In our application, we will be focusing on heat transfer through conduction and convection (because the temperature achieved by common electronic devices, e.g. CPU under cooling, is low, therefore the radiation can be neglected).

Heat conduction is the process through which energy is transferred as a result of molecular contacts between nearby molecules brought on by their erratic motion. "With increasing temperature the random motion of molecules is intensified, and with this the kinetic energy on the molecular level. Collisions between neighbouring molecules cause a transfer of energy from those with higher kinetic energy to those with lower kinetic energy” [1]. Therefore, heat conduction can be found in solids, liquids, or gases, but it does not necessarily need any macroscopic motion of flow of molecules.

The heat conduction (figure 1) between the bodies is affected by four main factors. The physical properties of the materials, the cross-section of the body, the path length, and the temperature difference. With these factors, we can control some of the variables and further maximise our efficiency in cooling the heat source.

The properties of the materials, such as those in the composite wall, will deter the outcome of heat conduction. Each material has its own thermal conductivity properties (and therefore creates its own thermal resistance). “Heat conduction coefficient is defined as the amount of heat per unit time per unit area that can be conducted through a plate of unit thickness of a given material, the faces of the plate differing by one unit of temperature” [3].

$$\lambda = \frac{QL}{A\Delta T} \tag{3}$$

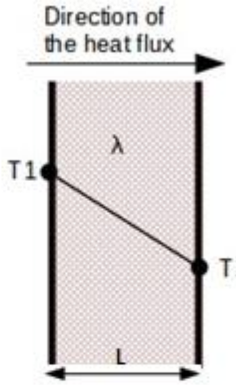


FIGURE 1 - Heat flux through a plane wall of a material of conductivity λ [6]

The cross section and the length of the path are also important factors. The larger the size of the material involved in the transfer, the more heat that is needed to warm it. Additionally, the greater the surface area that is exposed to the open air, the greater the likelihood of heat loss. Therefore, shorter objects with a smaller cross section are the best means of minimising the loss of heat energy, and a cooling device has a large surface area as possible to maximise the heat take-off.

“A temperature gradient is a physical quantity that describes in which direction and at what rate the temperature changes in a specific location. Temperature always flows from the hottest to coldest source, due to the fact that cold is nothing but the absence of heat energy. This transfer between bodies continues until the temperature difference decays, and a state known as thermal equilibrium occurs” [7].

Convection is a process through which energy is transferred as a result of a macroscopically flowing fluid as a heat transport mode, see figure 2. “It is a superposition of conductive heat transport in the fluid and the energy transport due to the macroscopic movement of the fluid, which includes the transport of enthalpy and kinetic energy" [1]. There will be another variable that will affect the convective heat flux other than the material itself, such as the fluid velocity.

The specific heat capacity is measured in J/(kg K) or J/(kg °C), where J is the amount of heat absorbed by the material per unit mass (kg) as its temperature rises by 1 K (or 1 °C) [17]. Calculating the heat flux of convective heat transfer may be very difficult due to turbulent flow, as it can be complex and nonstationary. A simplified equation can be used to describe the relationship of heat flux with the bulk fluid temperature and the wall temperature.

$$\dot{q} = \alpha(T_w - T_F) \tag{4}$$

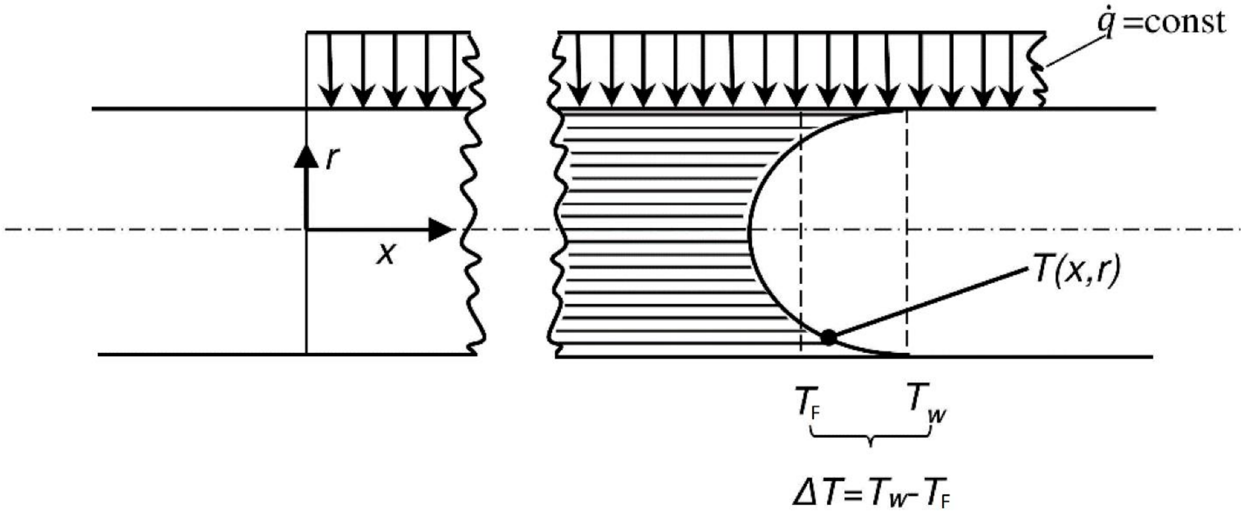


FIGURE 2 – Temperature distribution in a fully developed pipe flow [17]

2.2 Contact thermal resistances between solid walls

"When a junction is formed by pressing two similar or dissimilar metallic materials together, only a small fraction of the nominal surface area is actually in contact because of the non flatness and roughness of the contacting surfaces, see figure 3. If a heat flux is imposed across the junction, the uniform flow of heat is generally restricted to conduction through the contact spots. The limited number and size of the contact spots results in an actual contact area which is significantly smaller than the apparent contact area. This limited contact area causes a thermal resistance, the contact resistance or thermal contact resistance" [8].

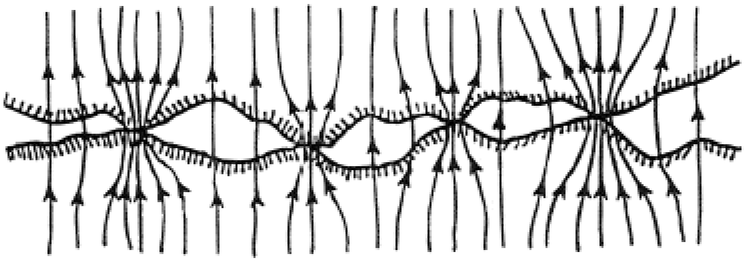


FIGURE 3 - Magnified view of two materials in contact [4]

Depending on the thermal conductivity, thickness, and hardness (in the case of a solid), the existence of a fluid or solid medium between the contact surfaces may contribute to or limit heat transfer at the junction. Heat transfer through radiation may occur across the gaps between the contact surfaces if there is a large temperature difference between the surfaces that make up the junction. Thermal resistance by conduction phenomena is defined by equation

$$R_t = \frac{L}{\lambda A} \tag{5}$$

In the minor contact area between two bodies, the force is applied, providing the apparent contact pressure on the junction. "The mean junction temperature, T_m, is the average of the contacting surface temperatures. The apparent contact pressure and the mean junction temperature, combined with the thermophysical and mechanical properties of the contacting materials and the surface characteristics, are the primary factors in determining the magnitude of the contact resistance" [8]. The contact resistance depends on the loads (pressure acting on the surface) and the temperature, see figure 4, where the low contact resistances come from high junction loads and high temperatures. The opposite can be said; with the same correlation, light junction loads and low temperatures will result in high contact resistances.

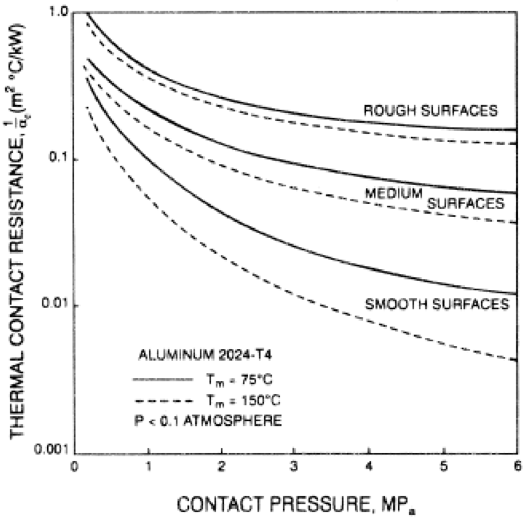


FIGURE 4 – Temperature distribution across flat and cylindrical contacting solids [4}

The significance of the contact resistance is majorly affected by the surface finish, roughness, and the flatness of the contacting surfaces. “If the axial force on the contacting surfaces is increased, the surface roughness peaks or asperities may deform plastically or elastically, depending upon the material properties, leading to increased contact area and decreased contact resistance” [8]. With softer materials, rising of the temperature may affect the physical properties and the surface. Plastic and/or elastic deformation may occur and, therefore, will reduce the contact resistance due to increase of the junction between the two bodies.

2.3 Influence of thermal paste layers

Thermal paste (grease) or thermal compound is a substance that is used to increase the heat conduction between two surfaces and is commonly used between a microprocessor and a heatsink. Most microprocessors will have a rough surface area and not be completely smooth because of the nature of solid materials. “Some have microscopic grooves and others might even have a slight curve, which produces air gaps between the microprocessor and the heatsink and reduces the cooling performance of the heatsink” [5]. The air gaps will be mostly eliminated by filling up a thin layer of thermal paste between the processor and the heat sink, see figure 5. Also temperature distribution at cooled surface is uniformed by using thermal paste.

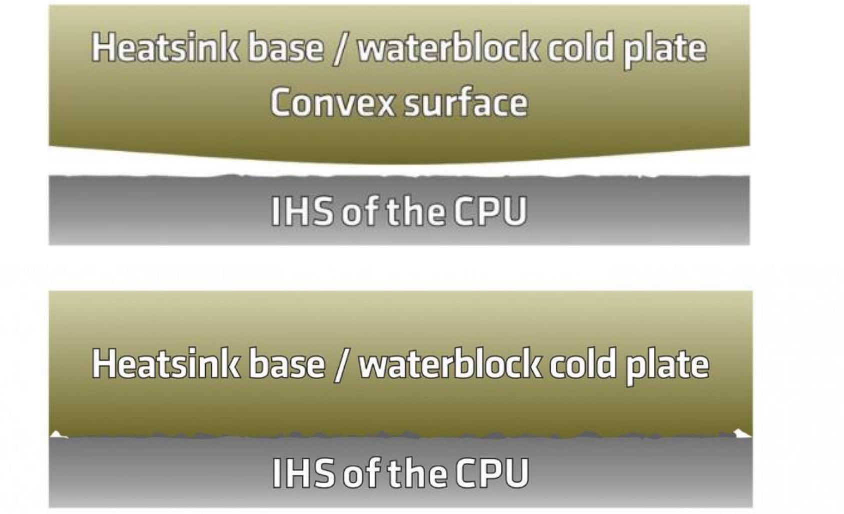


FIGURE 5 – Gap between heatsink and CPU with and without thermal paste [5]

Many of the commercial thermal pastes are made from a mixture of a bonding material and a filler material. “The bonding material could be various types of silicones, urethanes, epoxies and acrylates, while the filler material is the one that has good thermal conductivity. The choice of the filler can be broad – some companies use silver, aluminium, ceramic, and diamonds which are ground to a very fine powder and mixed with the bonding compound” [5]. Thermal paste does not add mechanical strength to the bond between the heat source and the heat sink.

The thermal conductivity of a high-grade thermal paste is around 8-9 W/(mK) [11]. The thermal compound is actually a poor heat conductor, which is why a thin layer (0.0001 mm) of thermal paste is necessary to fill in the micro-gaps between the heat source and the heat sink.

2.4 Ways of cooling down electronic devices

An electronic device due to electric resistance generate heat, and there are different ways of cooling down the device. The model of CPU that is used in the experiment will need to be properly cooled in order to achieve its maximum performance. The CPU contains transistors inside, which then turns electrical energy into thermal energy, therefore raising the CPU temperature [14].

2.4.1 Force Convection Air Cooling

Forced convection air cooling is most commonly used to cool down a CPU. "Heat is removed directly by fan with heat sink collected and placed on the CPU. The air is pushed through the heat sink by the fan. By this method, the heat is removed and transfer directly to the final medium, air" [14]. A geometry of straight fins mounted perpendicular to cooled surface provides the best performance of forced convection air cooling [13], having air passing through the most surface area to extract the heat that is transferred from the CPU to the heat sink, see figure 6.

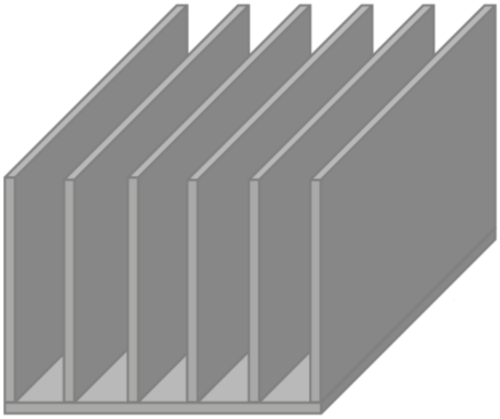


FIGURE 6 – Perpendicularly mounted Air fins design [15]

2.4.2 Liquid Cooling

Liquid cooling is another popular form of cooling system. Water is widely used now in liquid cooling technology, but other mediums are available to be used too. "Water cooling block is a kind of metal block with internal water circuit, which is made of copper or aluminium and has contact with the CPU to absorb CPU heat" [12]. Similar to air cooling, it takes the principle of absorbing heat away from the CPU, but the difference is that, instead of fins, water cooling needs a channel for liquid to pass through and cool down the heat, see figure 7. Circulating liquid performs a similar function to air, except that it absorbs more heat and the temperature varies less. The drawback of the water utilisation is electrical conductivity of water. The water must not be in direct contact with electrical device to eliminate electrical short circuit.

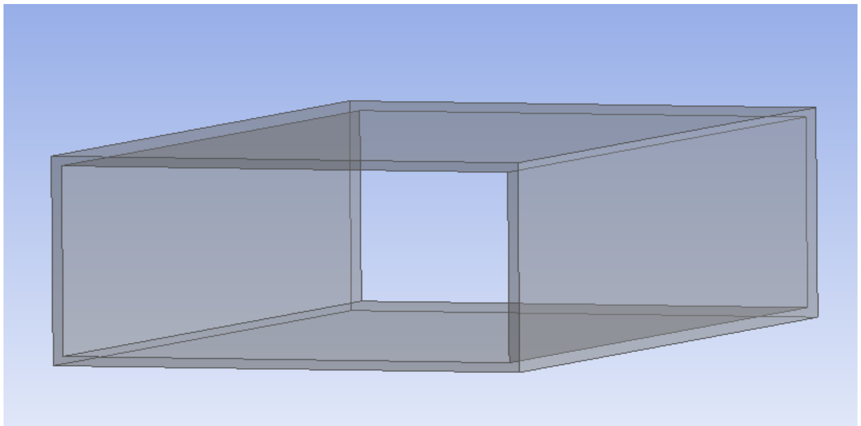


FIGURE 7 – Basic water channel design

2.5 Computational Fluid Dynamics (CFD)

ANSYS Computational Fluid Dynamics Simulation is software that helps us to test our ideas into practical scenarios. “It’s the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena” [16]. CFD solves equations for the conservation of mass, momentum, energy, and others to forecast these occurrences. With the help of the use of CFD, we can get detailed results on the fluid flow behaviour or how heat is transferred through multiple layers of walls.

CFD uses Meshing method to map out discrete cells and the formulated equations are solved at those cell/nodal locations. The discrete cells (meshed) are divided across the surface areas of the body parts where we want to analyse. The meshing requirements will be based on its efficiency and accuracy, along with the quality of the mesh. The more refined cells (element size) there are, the more detailed and accurate the solution will be. The quality of the mesh will depend on how the mesh cells deviate from the ideal shape.

The finite-volume method is used by ANSYS CFD solvers. "Domain is discretized into a finite set of control volumes. General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes"[2]. A system of algebraic equations is created using partial differential equations. The solution field is then produced by numerically solving all algebraic equations. The trade-off between how accurate and quality the result must be vs. time consumed doing the simulation has to be done.

To determine the thickness of the thermal grease layer and modelling it into CFD, a study shows “a calibration procedure that involves two numerical environments (ANSYS Fluent and Simplorer) is proposed” [17]. Simplorer is a circuit simulator that can handle multiple physical properties and allows you to combine electrical, mechanical, hydraulic, and thermal components in a single schematic. The mathematical operations can be described using State Space, block diagrams, State Machines, and scripting algorithms [18]. The model, which has been adjusted for accuracy, is utilized to investigate the impact of the thickness of either the thermal paste layer or the space between the surfaces.

Another study shows another way of modelling thermal paste layer into CFD by modelling the parts as blocks. In order to achieve the correct thermal resistance, the thermal conductivities of the blocks were modified in the same ratio as the cross-sectional areas of the real parts model. [19]

3. Aims and problem description

The aim is to find the best solution to cool down an electronic device. We are given parameters to cool down a CPU through water cooling versus air cooling. An overall structure of how the CPU and other plates are supposed to be mounted has been given as part of the simulation. A design of air cooling fins was designed based on literature research and will be tested against a standard water channel cooling design. Find out how thermal paste affects the heat flux through the structure and which parameters matter the most is also the goal.

3.1 Geometry and analysis conditions

3.1.1 Overall Given Structure

The overall controlled structure will be given for both air and water cooling in the following figure 8.

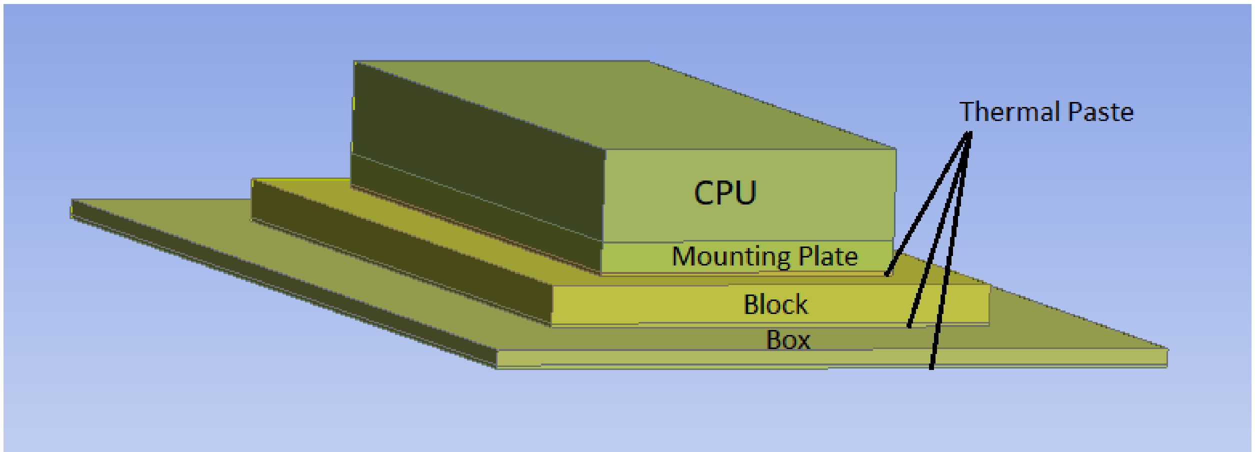


FIGURE 8 – General Given Structure

First, the CPU is mounted with the mounting plate, below that is a layer of thermal paste and a block, followed by another layer of thermal paste. Under that will be mounted a box where either will be connected to a water box channel or air fins, along with another layer of thermal paste.

The material of the CPU, mounting plate, block and the box are made from aluminium in this simulation. The primary material of the thermal paste is made from zinc oxide [11].

3.1.2 Given Boundary Conditions

The CPU has operating power at 500W with heat coefficient transfer of 5 W/(m^{2}K) to the surrounding air and for the rest of the component will also have the same heat coefficient transfer. The surrounding outside temperature of this operation is 40 degrees Celsius.

The dimensions of the controlled structure are presented in the figures 9 and 10.

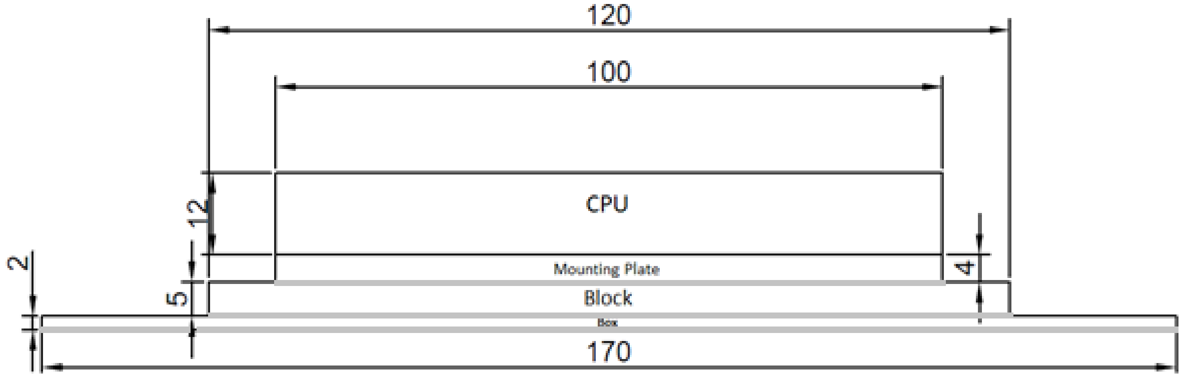


FIGURE 9 - Given Structure Side View

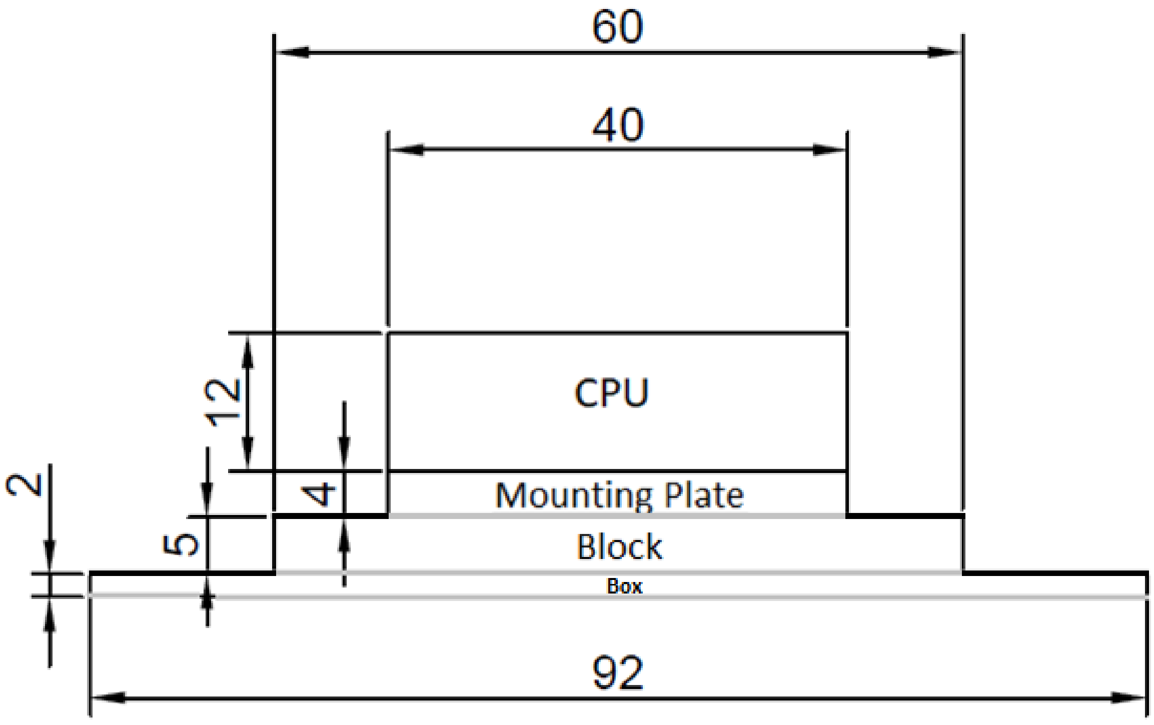


FIGURE 10 – Given Structure Front View

3.1.3 Aim

The goal is to be able to keep the CPU operating at its optimal temperature which is under 60 degrees Celsius. We will be using two different types of cooling method, forced air convection and water cooling. With the given boundaries conditions and parameters, our aim is to design a geometry for each method based on literature research for an optimal cooling system. With the help of CFD simulation, we will be able to change different parameters and will find the optimal mass flow rate for each medium in these boundaries conditions to achieve the operating temperature under 60 degrees.

3.2 Water Channel

The purpose of the design of the water channel box is to find the ideal mass flow rate of water and its heat coefficient of transfer to achieve our goal. A standard water channel is designed on the basis of literature research, which being the simplest to design along with its effectiveness. The main change parameter is the water heat coefficient to obtain our goal. A water channel box is mounted below the overall given structure and the following design was suggested, see figures 11, 12 and 13.

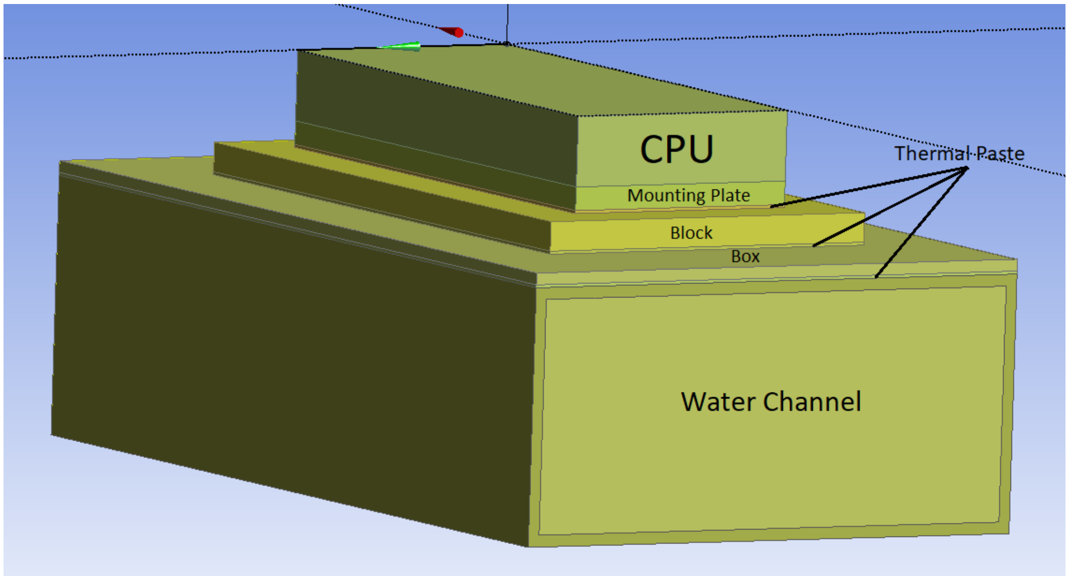


FIGURE 11 - Water Channel Design

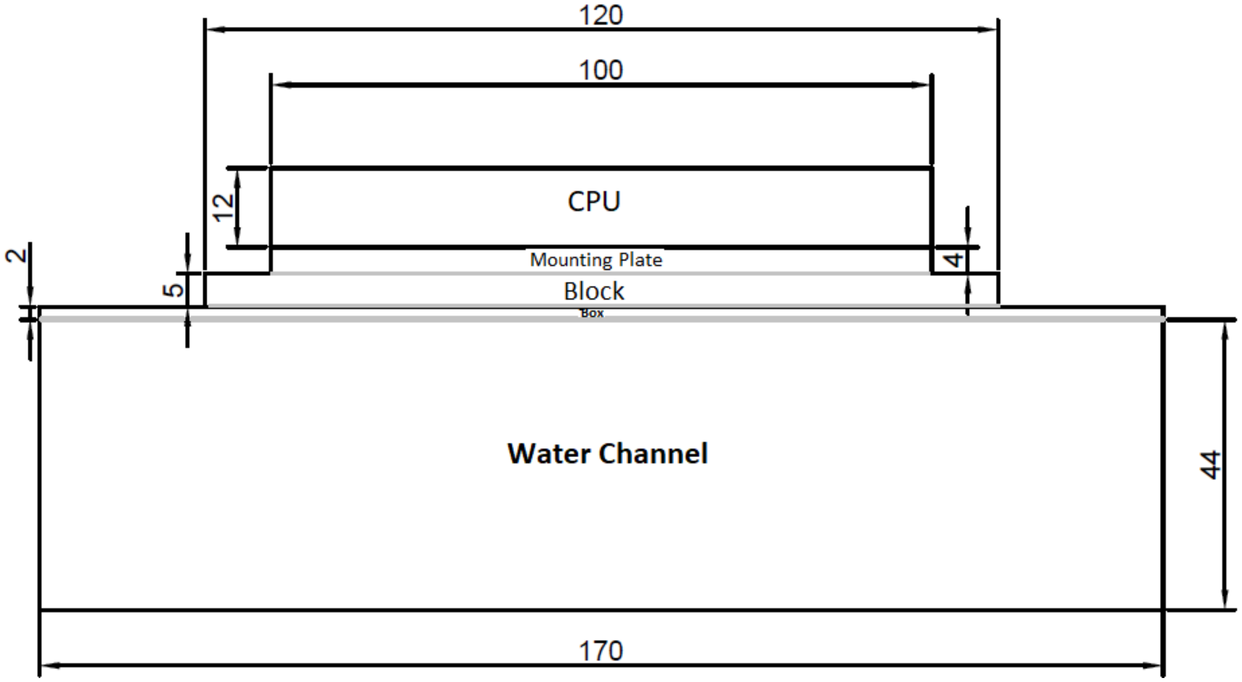


FIGURE 12 – Water Channel Side View

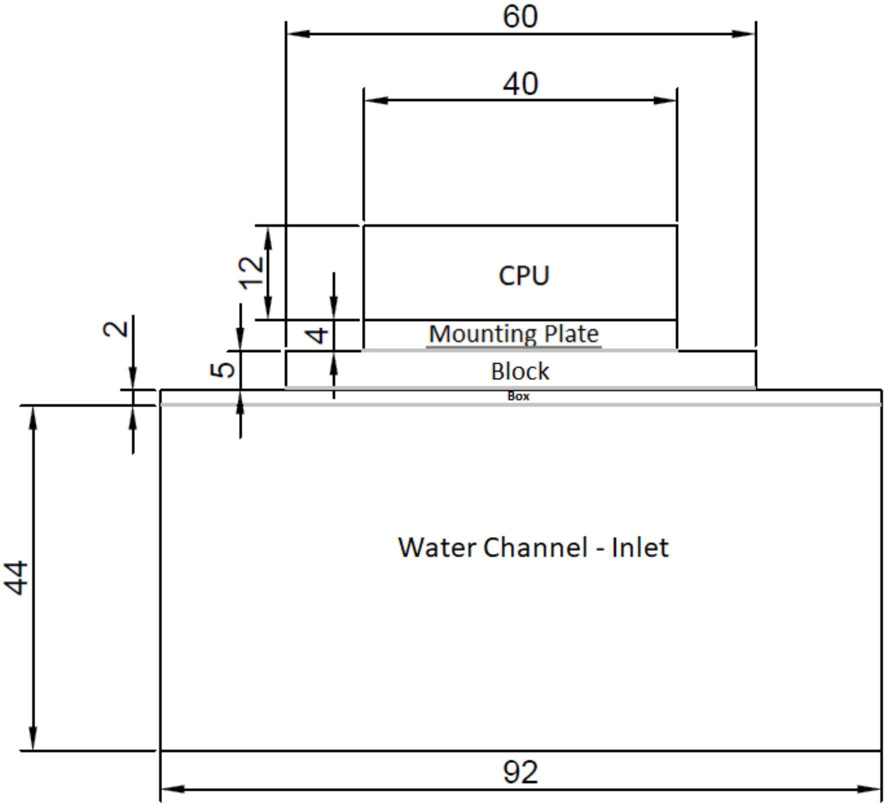


FIGURE 13 – Water Channel Front View

3.3 Air Cooling

The purpose of the fin design is to find the ideal air mass flow rate and its heat coefficient transfer to achieve our goal. A design has been proposed based on research from the literature that states that 90 degree straight fins are the most effective fins for air cooling [13]. The straight fins were designed as follows with various lengths A where our main parameters will change to obtain our goal, see figures 14, 15 and 16:

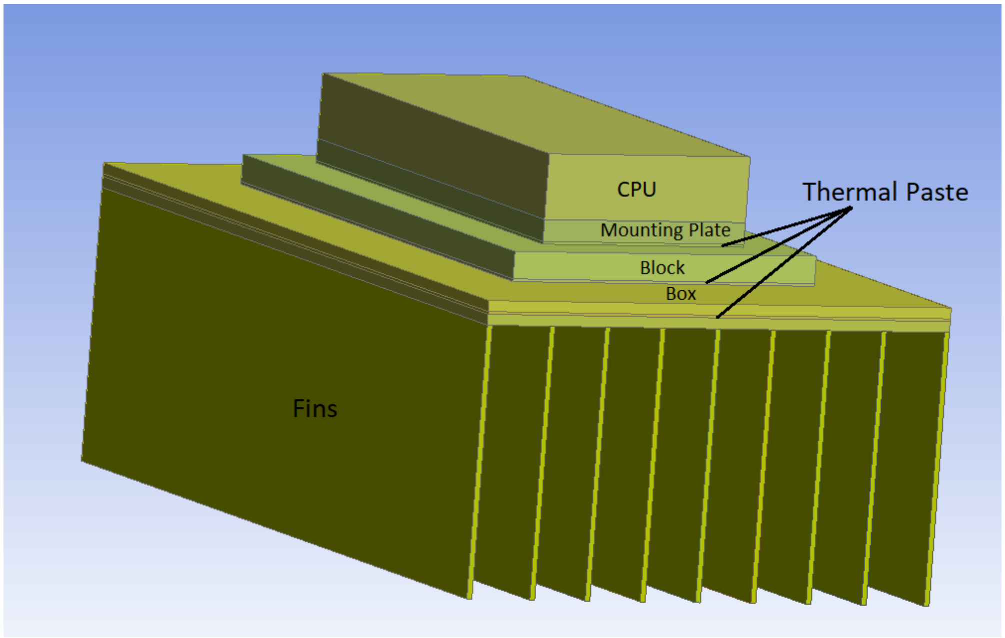


FIGURE 14 – Air Fins Design

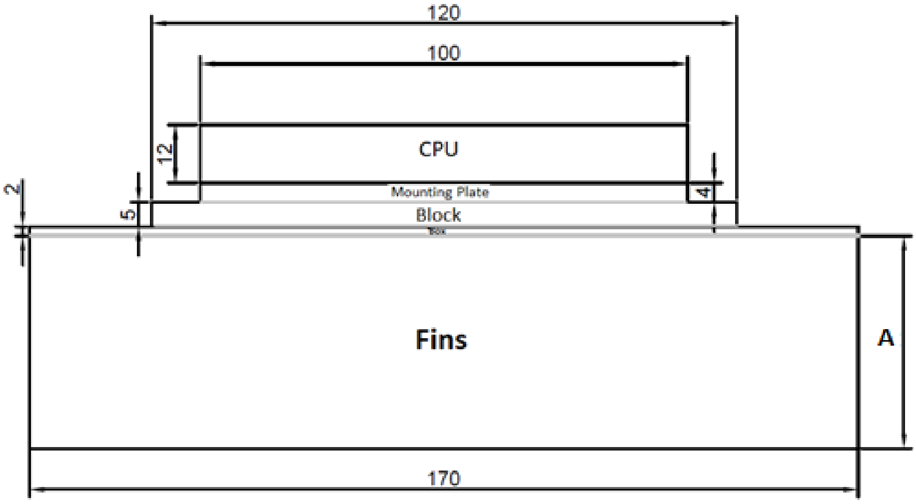


FIGURE 15 – Air Fins Design Side View

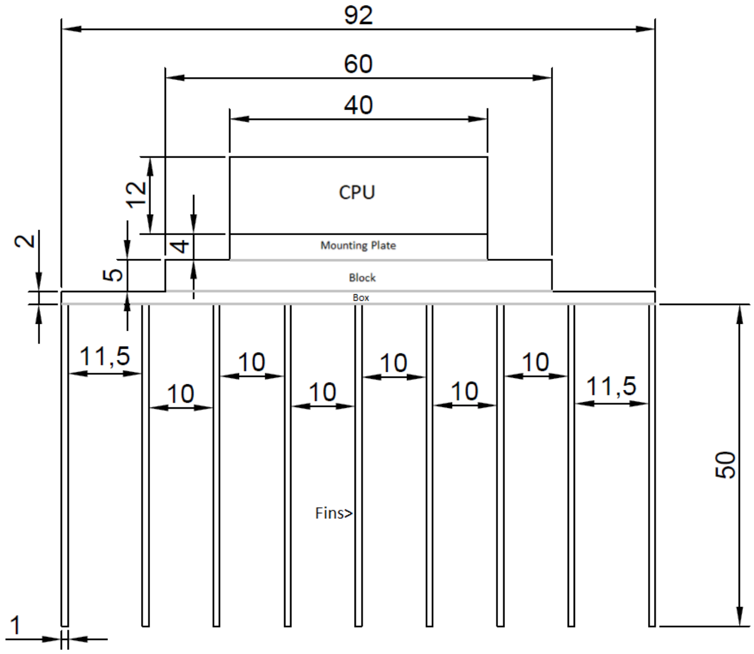


FIGURE 16 – Air Fins Design Front View

3.3.1 Fins Designs

We will compare 3 different fins designs by changing its length and analysing what impact it has on the results. First, we have the fins length closer to our water channel box length to have a similar geometry comparison at 50 mm in length. The design is as follows, see figure 17. And other two length were also investigated A = 10 mm and A = 80 mm.

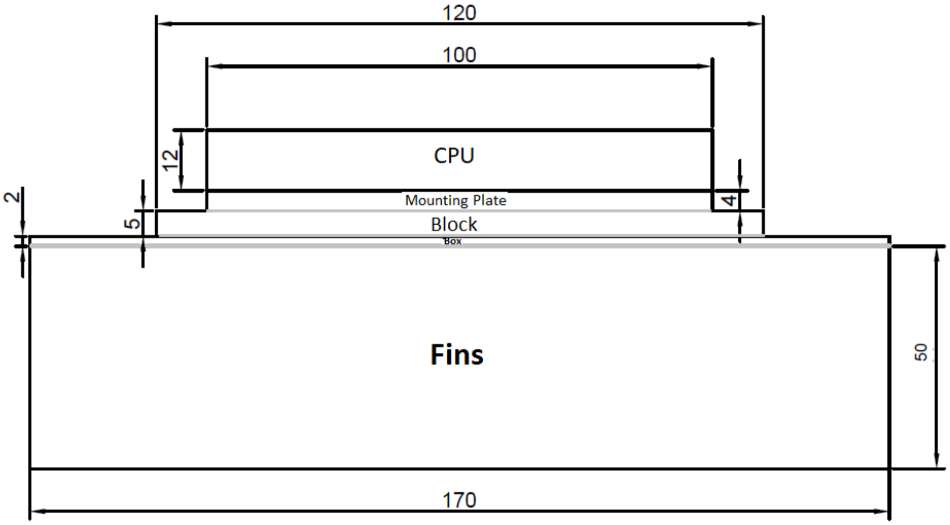


FIGURE 17 – Air Fins Design Side View, Length 50 mm

4. Analysis 1D Problem

From an engineering perspective, we will be suggesting a 1-dimensional problem to solve a similar replication of our simulation and to give us an idea what to expect. We will introduce 2 different 1D cases to compare to our simulation. One case will be a simple symmetrical geometry, and the other case will be involving water cooling based on our given geometry and dimensions.

4.1 Symmetrical 1D Example Problem

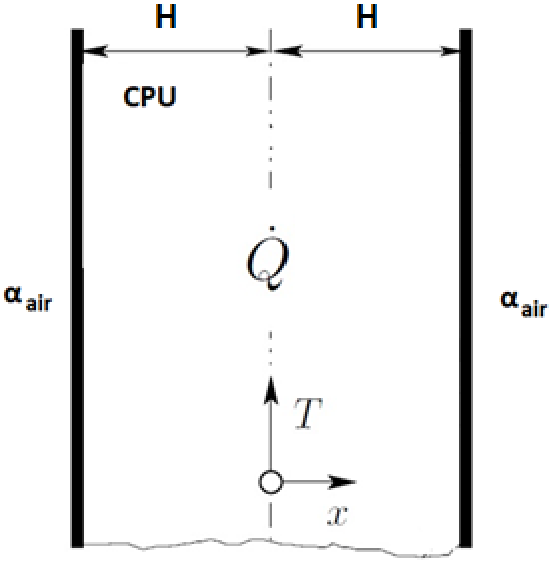


FIGURE 18 – Symmetrical 1D Problem

This case represents absence of the cooling system, and CPU is cooled down only by surrounding air. The general equation to find heat conduction through a flat wall (aluminium), see figure 18 with a heat source is the following:

$$T_w - T_F = \frac{\dot{Q}}{2\lambda_{AL}} H^2 \left[1 + \frac{4}{Bi} - \left(\frac{x}{H}\right)^2\right] \tag{6}$$

From the equation, the Biot Number must be calculated as follows:

$$Bi = \frac{2\alpha_{air} H}{\lambda_{AL}} \tag{7}$$

To calculate the heat flux generated from the CPU, we will be using the following equation:

$$\dot{Q} = \frac{P_{CPU}}{V_{CPU}} \tag{8}$$

With all the equations combined and the following inputs, we obtained the result of the temperature at the surface of the CPU wall and in the CPU core, see table 1.

TABLE 1 - Difference of Temperature on the Surface and Core Of CPU

x [m]	$\dot{Q}\ [\frac{W}{m^3}]$	H [m]	T_{air} [°C]	$\alpha_{air}\ [\frac{W}{(m^2K)}]$	$\lambda_{AL}\ [\frac{W}{(mK)}]$	Bi	T [°C]
0.006	10416666	0.006	40	5	202.4	0.000296	12540
0	10416666	0.006	40	5	202.4	0.000296	12541

As we can see, the CPU core will have the highest temperature of 12541 °C, whereas the CPU surface temperature is 12540 ° C. The alfa 5 W/(m^{2}K) represents the slightly moving or almost stagnant air. To find out what the desired air flow rate will be, we can rearrange the equation to find the heat coefficient transfer α_{air} to satisfy the maximum temperature of the CPU at 60 ° C. The result is shown as follows in table 2. The CPU will be cooled down by forced convection without any heat transfer support mechanical construction.

TABLE 2- Ideal heat coefficient of air at the Core and Surface of CPU

x [m]	$\dot{Q}\ [\frac{W}{m^3}]$	H [m]	T_{air} [°C]	T [°C]	$\lambda_{AL}\ [\frac{W}{(mK)}]$	$\alpha_{air}\ [\frac{W}{(m^2K)}]$
0.006	10416666	0.006	40	60	202.4	3125
0	10416666	0.006	40	60	202.4	3277

The air heat coefficient transfer must be greater than 3277 W/(m^{2}K) to allow the maximum CPU temperature to be lower than 60 ° C. The maximum temperature will be in the core of CPU. The value of alfa is rather high for air, it means, that increase of heat transfer surface will be needed.

4.2 Water Cooling 1D Example Problem

For our next problem, it will mimic more of our overall geometry combined with water cooling with mounting plate, block, box and 3 layers of thermal pastes attached, see figure 19. The case representing the simplified problem, which represents the cooling system installed from both sides of CPU (due to symmetry). In fact, the stagnant air will be in contact at one side of CPU, and the colling system will be installed at opposite side of the CPU. Due to high conductivity of the CPU, it can be expected, that obtained alfa of water will not be so far from reality.

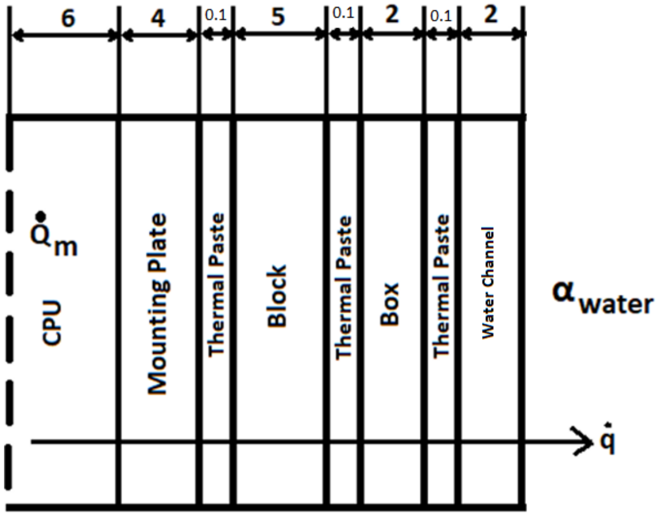


FIGURE 19 – 1D Water Cooling Problem

For this problem, we will replace the convective heat transfer coefficient in the Biot number equation with the total heat transfer coefficient k between all walls and parts corresponding to their thickness and thermal conductivity as follows:

$$Bi = \frac{2kH}{\lambda_{AL}} \tag{9}$$

$$k = \frac{1}{\frac{H_{Mounting\ plate}}{\lambda_{AL}} + \frac{H_{TP}}{\lambda_{TP}} + \frac{H_{Block}}{\lambda_{AL}} + \frac{H_{TP}}{\lambda_{TP}} + \frac{H_{Box}}{\lambda_{AL}} + \frac{H_{TP}}{\lambda_{TP}} + \frac{H_{Water\ Channel}}{\lambda_{AL}} + \frac{1}{\alpha_{water}}} \tag{10}$$

To find the temperature at the rightest wall surface, we will be using the same equation as in the previous problem. The initial results were obtained with the following parameters, see table 3.

TABLE 3 - Water Cooling Problem First Solution

x [m]	$\dot{Q}\ [\frac{W}{m^3}]$	T$_{water}$ [°C]	$\alpha_{water}[\frac{W}{(m^2K)}]$	$\lambda_{TP}\ [\frac{W}{(mK)}]$	Bi	T [°C]
0.006	10416666	40	50	9	0.00295	1296
0	10416666	40	50	9	0.00295	1297

Similar to the previous problem, the highest temperature is in the centre of the CPU 1296°C for this example and at the surface of the CPU, it is at 1297°C. The profile temperature decreases from the centre of the CPU to the surface. To find out what the desired water flow rate will be, we can rearrange the equation to find heat coefficient transfer α_{water} to satisfy the maximum CPU temperature at 60°C. The result is shown as follows:

TABLE 4 - Water Cooling Problem Ideal Heat Coefficient Transfer of Water

x [m]	$\dot{Q}$ [$\frac{W}{m^3}$]	H [m]	T_{water} [°C]	T [°C]	λ_{AL} [$\frac{W}{(mK)}$]	α_{water} [$\frac{W}{(m^2K)}$]
0.006	10416666	0.006	40	60	202.4	4496
0	10416666	0.006	40	60	202.4	4817

The heat coefficient transfer of water must be greater than 4496 W/(m^2K) for the maximum CPU temperature to be lower than 60°C.

We can formulate following conclusions based on physical principles and observation, which can be expectable.

From the formulas, we can see that decreasing thermal conductivity of thermal paste will increase thermal resistance which will decrease the heat transfer and therefore increases the temperature CPU.

Increasing the thickness of the thermal paste will decrease the thermal resistance which will increase the heat transfer and therefore decreases the temperature CPU.

Increasing the heat transfer coefficient of water which correlates to increasing the velocity of water will decrease heat resistance on the side of the water which will increase the heat transfer and therefore decreases the temperature CPU.

Increase the heat transfer coefficient of air which corresponds to increasing the velocity of air will decrease heat resistance on the side of the air which will increase the heat transfer and therefore decreases the temperature CPU. But in case of air, the expected velocity of air is expected to be much higher than water.

Increasing the fin length will increase the surface area which will increase the heat transfer and therefore decreases the temperature CPU.

5. Computational Fluid Dynamics Simulation

We will be using Computational Fluid Dynamics Simulation (CFD) to test our designed water channel box and air cooling fins. The main goal of the simulation is to find out at which water's heat coefficient transfer at which temperature is suitable for the optimal operating temperature of the CPU, which is below 60 degrees. From that we will be able to calculate our mass flow rate using the Nusselt number and Reynold number.

5.1 Geometry description

5.1.1 Geometry

The geometry designs have been drawn up based on the propose technical drawing and put into Design Modeler, see figures 20 and 21.

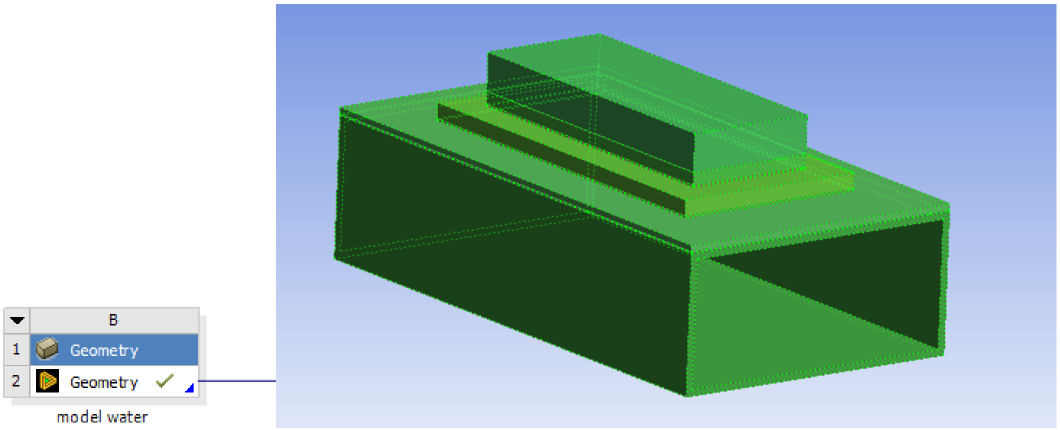


FIGURE 20 – Design Modeler Water Channels

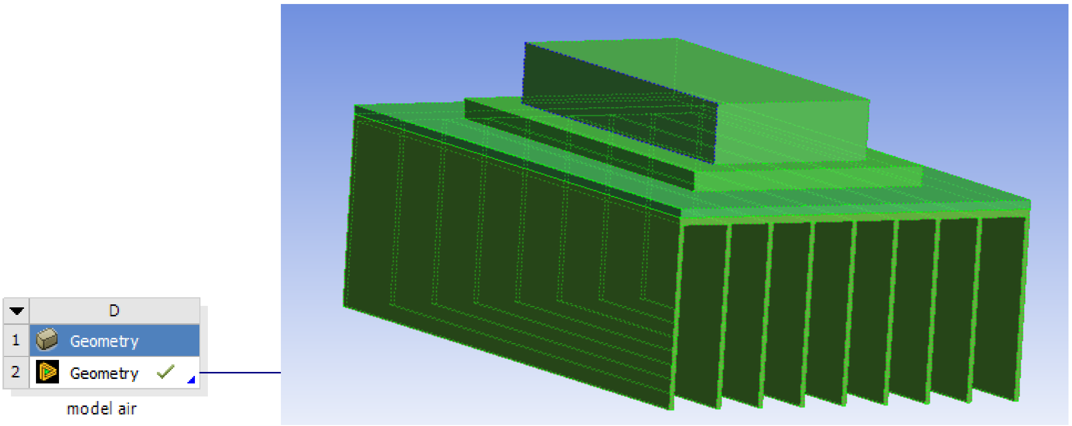
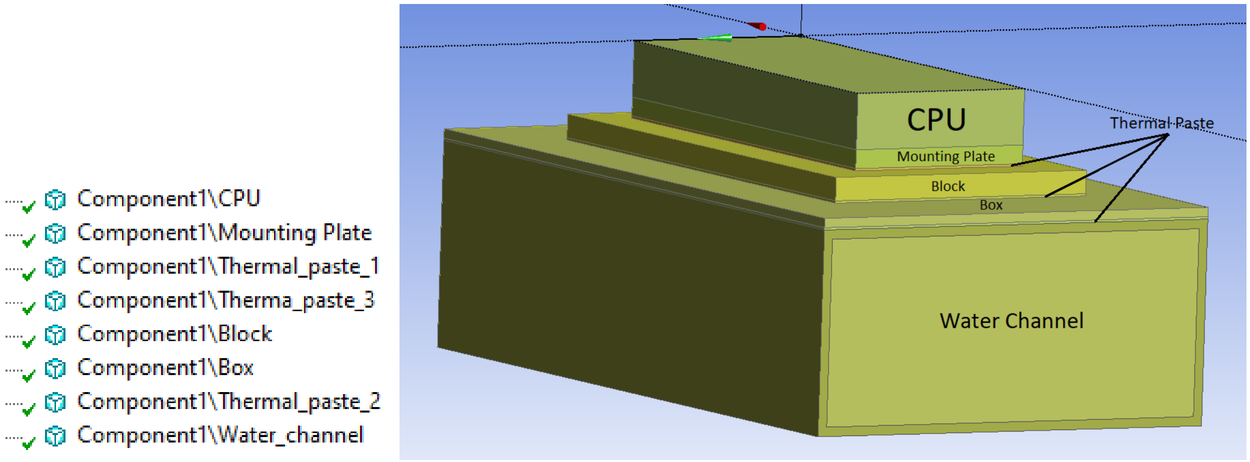


FIGURE 21 – Design Modeler Air Fins

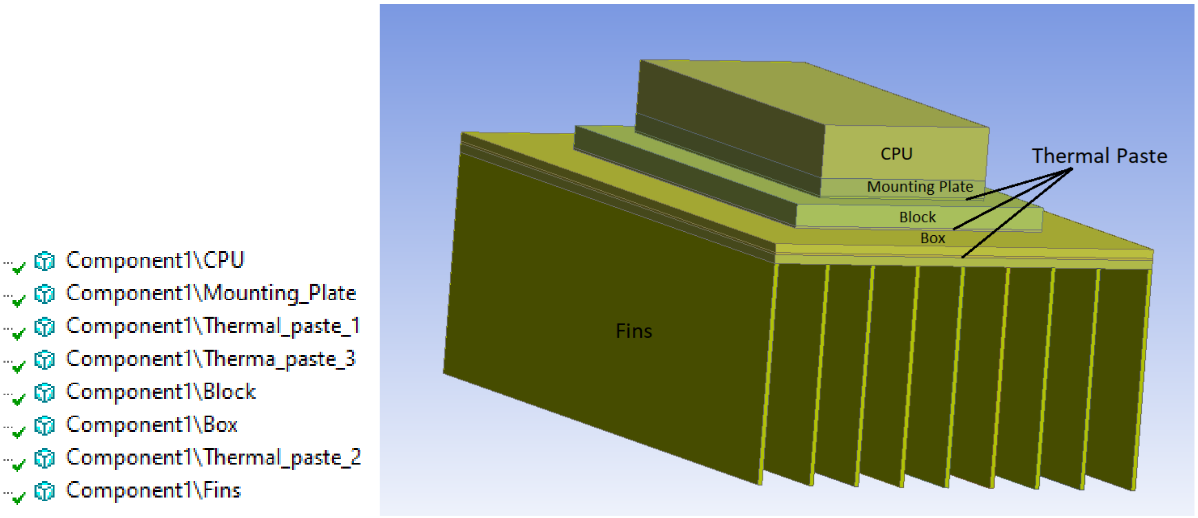
5.1.2 Named Body Parts

Furthermore, each part of the geometry has been named for identification and for setting up boundaries conditions and materials properties, see figures 22 and 23.



- Component1\CPU
- Component1\Mounting Plate
- Component1\Thermal_paste_1
- Component1\Therma_paste_3
- Component1\Block
- Component1\Box
- Component1\Thermal_paste_2
- Component1\Water_channel

FIGURE 22 – Named Water Channels Parts



- Component1\CPU
- Component1\Mounting_Plate
- Component1\Thermal_paste_1
- Component1\Therma_paste_3
- Component1\Block
- Component1\Box
- Component1\Thermal_paste_2
- Component1\Fins

FIGURE 23 – Named Fins Parts

5.2 Meshing

As for meshing, our geometry is connected to the meshing

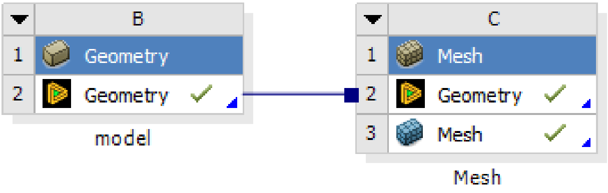


FIGURE 24 – Mesh Connection

A balance of grid density (higher accuracy for denser mesh) and computation time, a proposition of element size that satisfied our needs were set. Only heat conduction will be performed by simulation. The convection will be set up on the particular surfaces in the form of boundary condition. The uniform distribution of the convective heat transfer coefficient is assumed to be at each surface. Therefore, the mesh sensitivity analysis is not necessary to perform. The only limitation for the meshing was the minimal number of element layers across the thickness of the geometry entity of 5 elements. To simplify thermal paste simulation, thermal resistance is set up in the selected surfaces (imaginary thickness and heat conductivity) due to its small thickness and related problems with meshing.

5.2.1 Meshing of Water Channel Design

The water channel mesh generated started with mismatched grids that aligned with different parts of the body, which was not enough for the accuracy of the data as shown in figure 25.

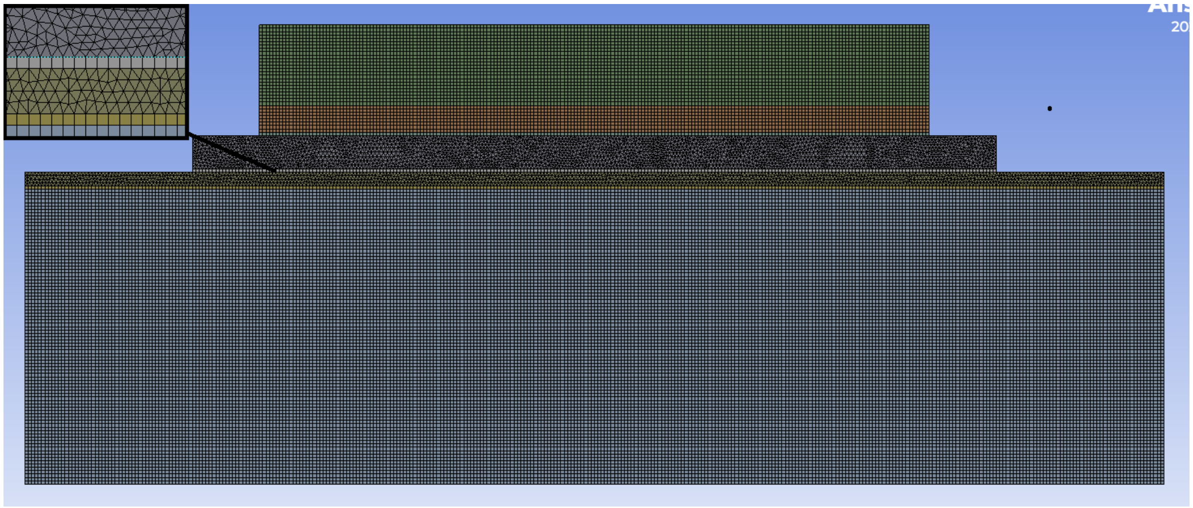


FIGURE 25 – Meshing Water Channel Attempt #1

To make the meshing smoother and well aligned, a separation of parts was recommended, and meshing them individually has helped resolve the issue and have a more accurate result when simulating, see figure 26. The final element size is 0.0005 with the element order “Linear” for the meshing and the number of elements is 1,875,680.

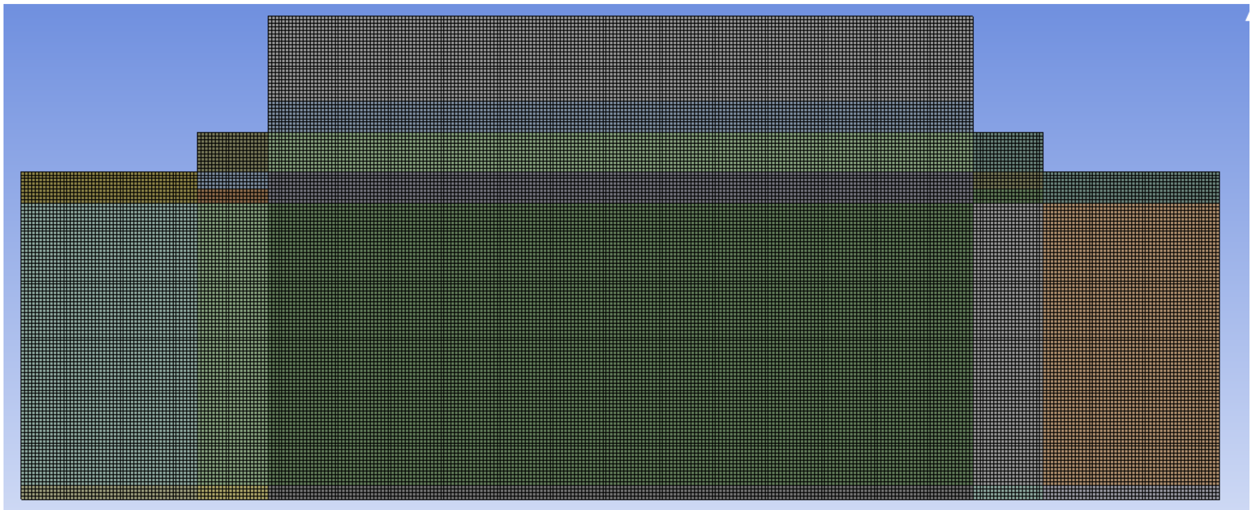


FIGURE 26 – Meshing Water Channel Final

5.2.2 Meshing of Air Fins Design

For generating mesh of air fins design it is tolerable with air medium for such meshing, no parts were needed to be divided into sections. The generated mesh of element size 0.0005 was satisfied with element order “Linear” for the meshing and the number of elements is 5,064,173. The mesh is shown in figure 27.

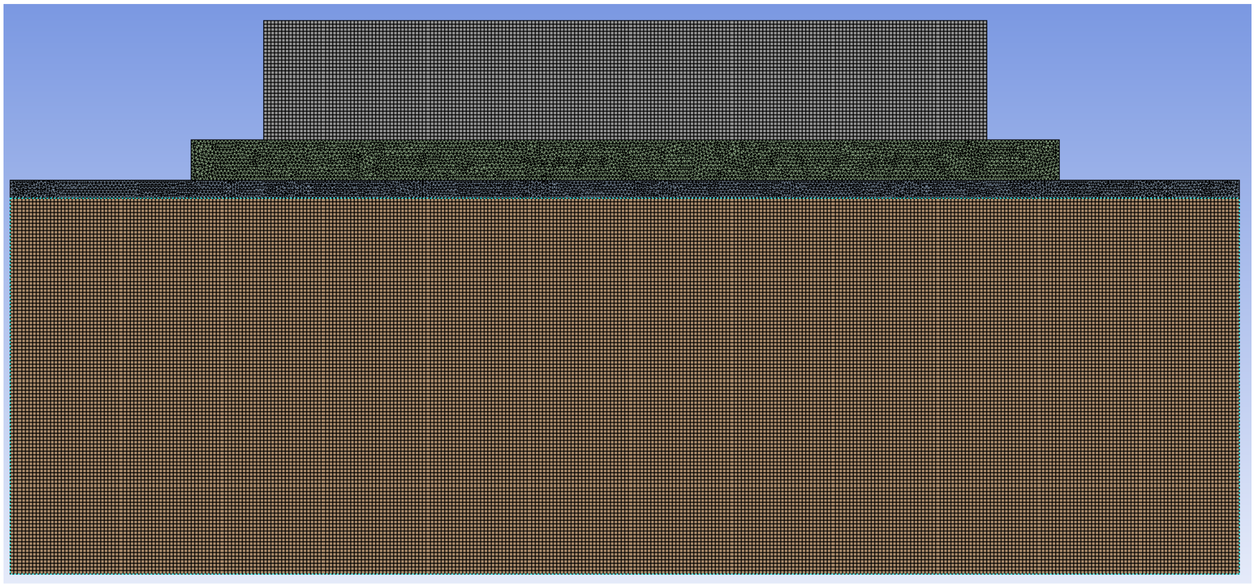


FIGURE 27 – Meshing Air Fins Final

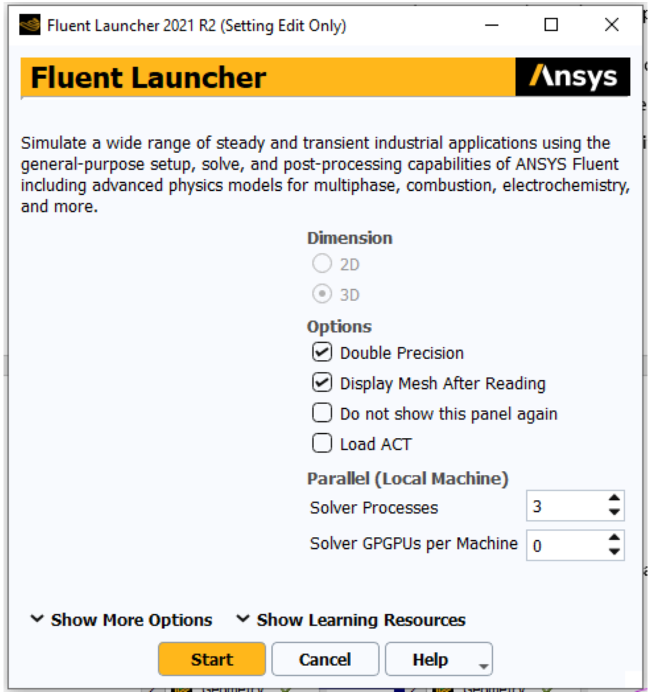
5.3 Fluent Settings

The meshing is then connected to CFD Fluent for final simulation, see figure 28.

FIGURE 28 – Connection to Fluent

5.3.1 General and Models

To achieve a more accurate result, Double Precision is selected before launching CFD Fluent, with the number of CPU cores for the speed of solving the result see figure 29.



Simulate a wide range of steady and transient industrial applications using the general-purpose setup, solve, and post-processing capabilities of ANSYS Fluent including advanced physics models for multiphase, combustion, electrochemistry, and more.

Dimension
○ 2D
● 3D

Options
☑ Double Precision
☑ Display Mesh After Reading
☐ Do not show this panel again
☐ Load ACT

Parallel (Local Machine)
Solver Processes: 3
Solver GPGPUs per Machine: 0

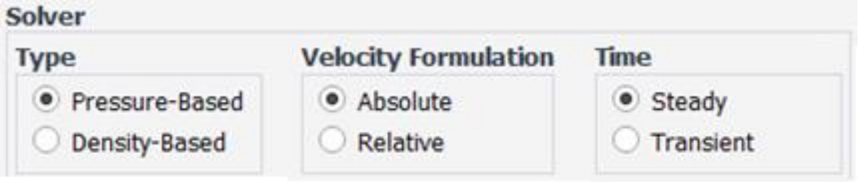
FIGURE 29 – Fluent Launch Settings

Once the software is open, the first thing to do is checking for mesh to see if there are any problems (figure 30). Scale and units settings are checked manually to satisfy our display of results in SI Units.

```
 Checking mesh.................
Done.
```

FIGURE 30 – Mesh Check

Our time is in a steady state as there is no time dependent involve in the simulation. Solving type will be a “Pressure-Based” with the velocity formulation in “Absolute, see figure 31.



Solver

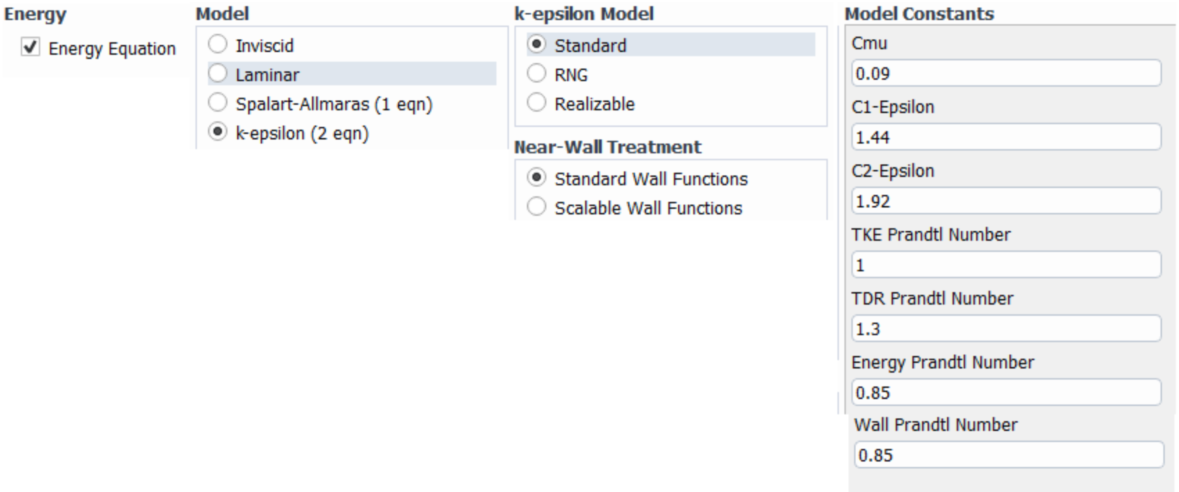
Type
● Pressure-Based
○ Density-Based

Velocity Formulation
● Absolute
○ Relative

Time
● Steady
○ Transient

FIGURE 31 – Solver Settings

We will set energy equation on for us to see the result after. As for viscous model, we will be using k-epsilon with 2 equations for us to control the materials properties and also the result will be accurate for our needs, see 32. We realized, however the case has not any convection issues the model for convection has to be switched on (selected) no matter which one to simulate the conduction.



Energy
☑ Energy Equation

Model
○ Inviscid
○ Laminar
○ Spalart-Allmaras (1 eqn)
● k-epsilon (2 eqn)

k-epsilon Model
● Standard
○ RNG
○ Realizable

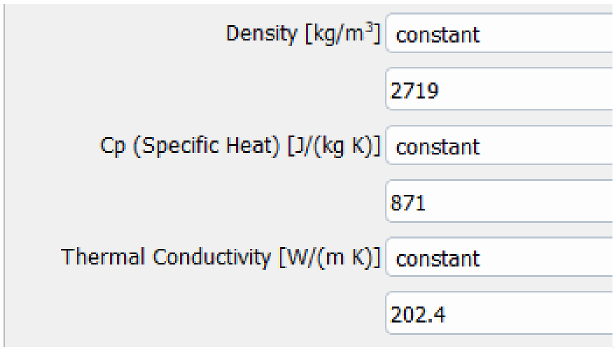
Near-Wall Treatment
● Standard Wall Functions
○ Scalable Wall Functions

Model Constants
Cmu: 0.09
C1-Epsilon: 1.44
C2-Epsilon: 1.92
TKE Prandtl Number: 1
TDR Prandtl Number: 1.3
Energy Prandtl Number: 0.85
Wall Prandtl Number: 0.85

FIGURE 32 – Viscous Model Settings

5.3.2 Materials

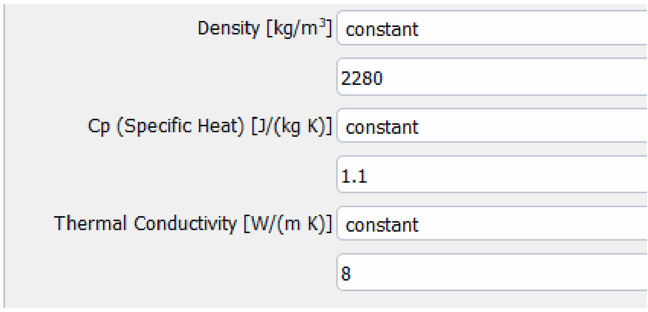
Materials are separated into fluid (default) and solid. As for solid materials, all the body parts are set as aluminium and its standard aluminium properties, see figure 33.



Density [kg/m^3] constant
2719
Cp (Specific Heat) [J/(kg K)] constant
871
Thermal Conductivity [W/(m K)] constant
202.4

FIGURE 33 – Aluminium properties

As for the thermal paste material, the properties have been set up according to [11], see figure 34.

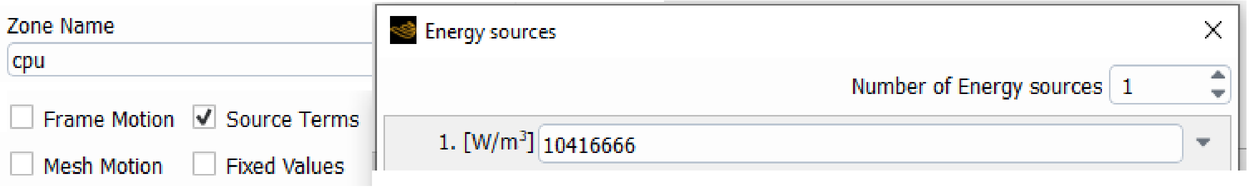


Density [kg/m^3] constant
2280
Cp (Specific Heat) [J/(kg K)] constant
1.1
Thermal Conductivity [W/(m K)] constant
8

FIGURE 34 – Thermal Paste (TG-600-09) properties

5.3.3 Cell Zones Conditions

All solid materials have been set to aluminium and thermal paste properties of paste in form of thermal resistance. In the CPU Solid conditions as it is the energy source, an additional heat generation is added and set to 10,416,666 W/m^3 based on the calculations, see figure 35.



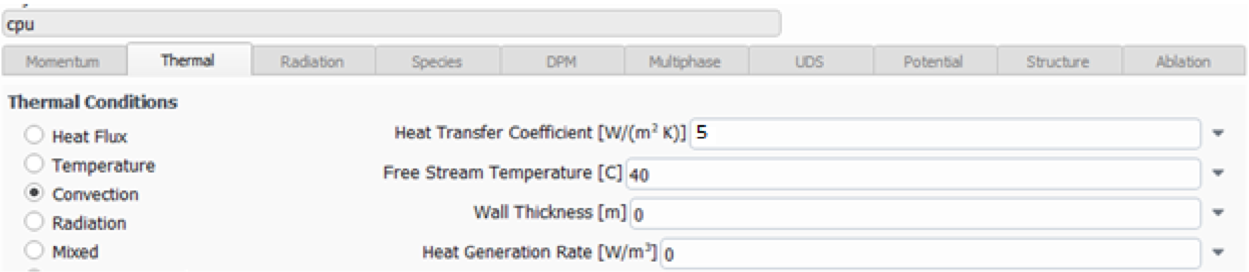
Zone Name
cpu
☐ Frame Motion ☑ Source Terms
☐ Mesh Motion ☐ Fixed Values

Energy sources
Number of Energy sources 1
1. [W/m^3] 10416666

FIGURE 35 – CPU Source Terms

5.3.4 General Boundary Conditions

The boundary conditions for the wall of each part of the body have been set individually. The thermal condition of every part surrounded by stagnant air is set as heat transfer through convection air with Heat Transfer Coefficient being 5 W/(m^{2}K) and the outside temperature being 40°C, see figure 36.



cpu

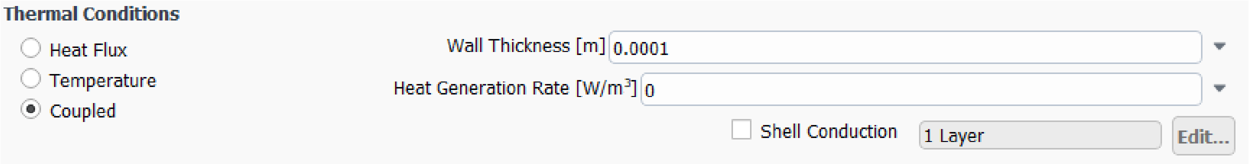
Thermal Conditions

○ Heat Flux
○ Temperature
● Convection
○ Radiation
○ Mixed

Heat Transfer Coefficient [W/(m^2 K)] 5
Free Stream Temperature [C] 40
Wall Thickness [m] 0
Heat Generation Rate [W/m^3] 0

FIGURE 36 – CPU, Mounting plate, Block, and Box Wall Boundary Condition

For the thermal paste, it has been set to 100 µm as the optimal thickness of the layer [5] with the material properties of the thermal paste, see figure 37.



Thermal Conditions

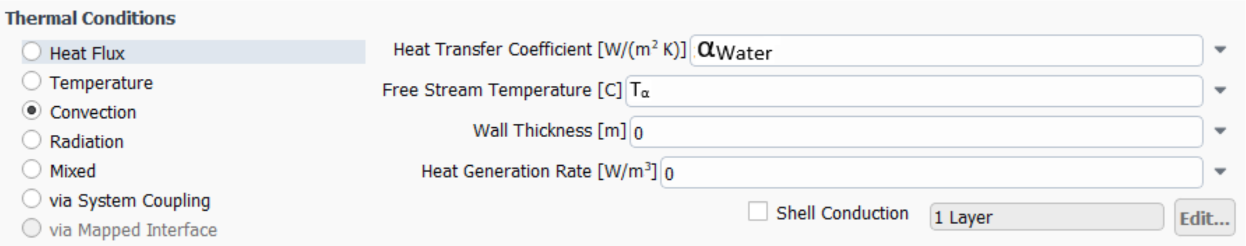
○ Heat Flux
○ Temperature
● Coupled

Wall Thickness [m] 0.0001
Heat Generation Rate [W/m^3] 0
☐ Shell Conduction 1 Layer

FIGURE 37 – Thermal Paste Boundary Condition

5.3.4.1 Boundary Conditions for Water Channel Design

In terms of changes in parameters, the boundary conditions for walls of the channel at contact with water were set up, see 38. Heat Transfer Coefficient α_{Water} is our variable along with Free Stream Temperature T_{α}.



Thermal Conditions

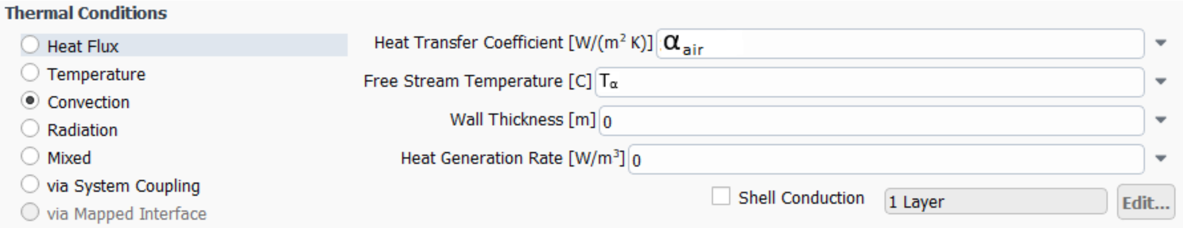
○ Heat Flux
○ Temperature
● Convection
○ Radiation
○ Mixed
○ via System Coupling
○ via Mapped Interface

Heat Transfer Coefficient [W/(m^2 K)] α_{Water}
Free Stream Temperature [C] T_{α}
Wall Thickness [m] 0
Heat Generation Rate [W/m^3] 0
☐ Shell Conduction 1 Layer

FIGURE 38 – Water Channel Surface In Boundary Conditions

5.3.4.2 Boundary Conditions for Air Fins Design

As for changing parameters, the fin walls are changed to find our result, see figure 39. Heat Transfer Coefficient α_{air} is our variable along with Free Stream Temperature T_{α}. The length of fins will be changed through the Design Modeler.



Thermal Conditions

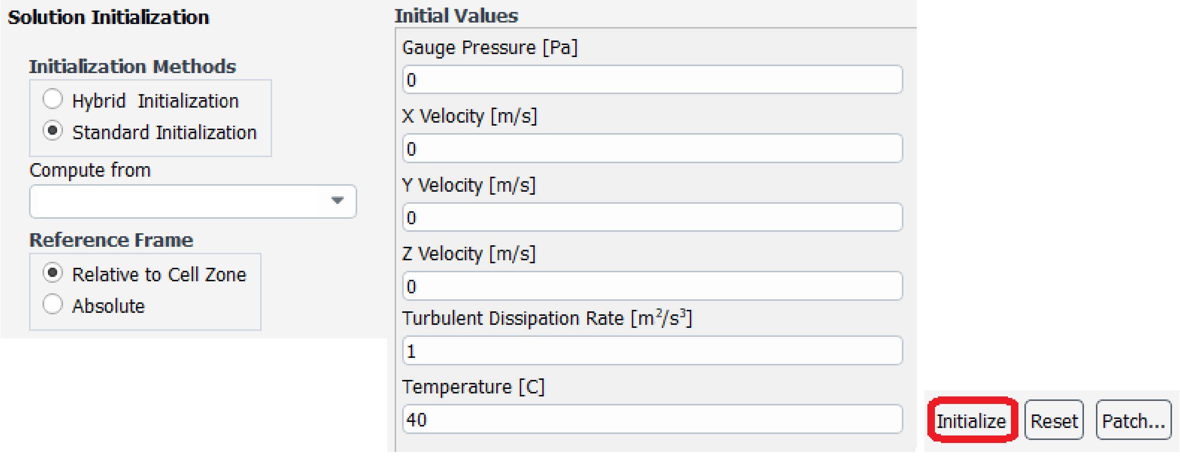
○ Heat Flux
○ Temperature
● Convection
○ Radiation
○ Mixed
○ via System Coupling
○ via Mapped Interface

Heat Transfer Coefficient [W/(m^2 K)] α_{air}
Free Stream Temperature [C] T_{α}
Wall Thickness [m] 0
Heat Generation Rate [W/m^3] 0
☐ Shell Conduction 1 Layer

FIGURE 39 – Air Fins Walls Boundary Conditions

5.3.5 Initialization

The initialisation method will be set to standard with the relative cell zone for the reference frame. The initial value of the temperature of the all parts is set to 40°C as it is the surrounding temperature. The simulation is then being initialised, see figure 40.



Solution Initialization

Initialization Methods

○ Hybrid Initialization
● Standard Initialization

Compute from

Reference Frame

● Relative to Cell Zone
○ Absolute

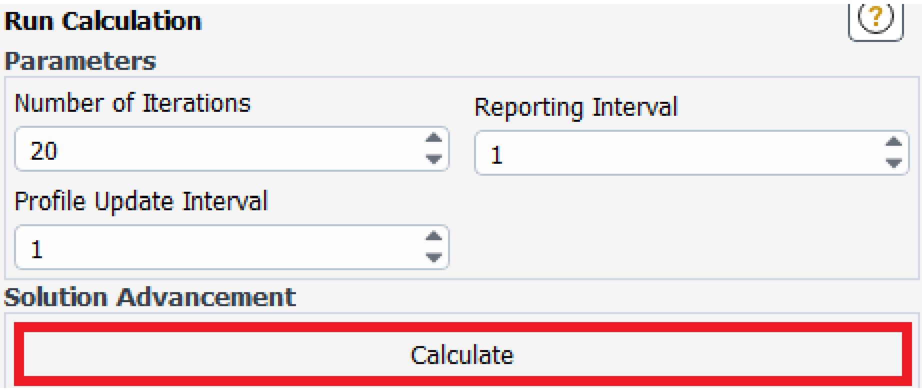
Initial Values

Gauge Pressure [Pa]: 0
X Velocity [m/s]: 0
Y Velocity [m/s]: 0
Z Velocity [m/s]: 0
Turbulent Dissipation Rate [m^2/s^3]: 1
Temperature [C]: 40

FIGURE 40 – Initialization Settings

5.3.6 Run Calculation Settings

After running the initialisation, have everything set up and ready for the final simulation. The 'run calculation' must be set as the final procedure. The number of iterations set for water channel design is 20 as that is where the residuals will stabilise. As for air fins design, it takes a longer while to stabilise to the number of iterations being set to 50. The final step is to press “Calculate” for the result, see figure 41.



Run Calculation

Parameters

Number of Iterations: 20
Reporting Interval: 1
Profile Update Interval: 1

Solution Advancement

FIGURE 41 - Run Calculations

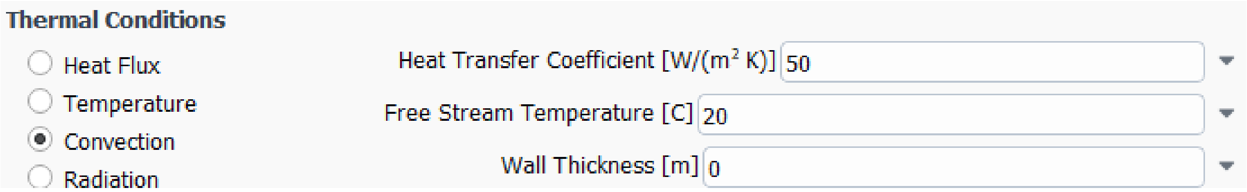
5.4 Results of simulations

5.4.1 Water Channel Design Results

The initial conditions for the design of the water channel were put as follows: water temperature at 20°C with a heat coefficient transfer α of 50 W/(m^{2}K). The regular thermal conductivity of the thermal paste at 2.5 W/(mK) with a thickness of 0.0005 m for testing, see table 5 and figures 42 and 43.

TABLE 5 – Water Channel Design Input #1

T water in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α water in [W/(m^{2}K)]
20	2.5	0.0005	50

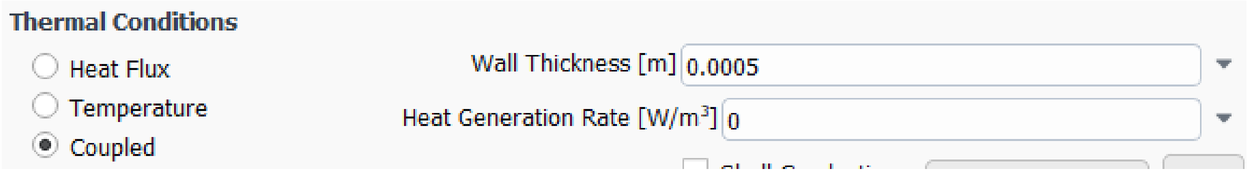


Thermal Conditions

○ Heat Flux
○ Temperature
● Convection
○ Radiation

Heat Transfer Coefficient [W/(m^2 K)] 50
Free Stream Temperature [C] 20
Wall Thickness [m] 0

FIGURE 42 – Water Channel Inlet Input #1



Thermal Conditions

○ Heat Flux
○ Temperature
● Coupled

Wall Thickness [m] 0.0005
Heat Generation Rate [W/m^3] 0

FIGURE 43 – Thermal Paste Thickness Input #1

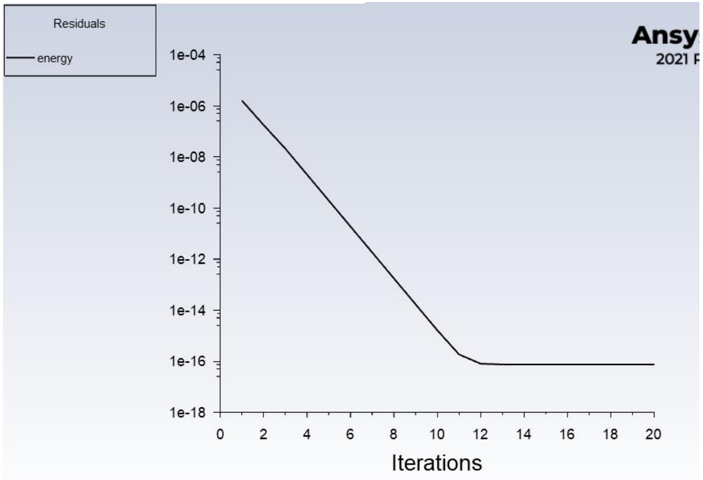


FIGURE 44 - Scaled Residual of First Result

As we can see in figure 44, with 20 iterations, the result has time to consolidate and be stable enough to generate accurate data.

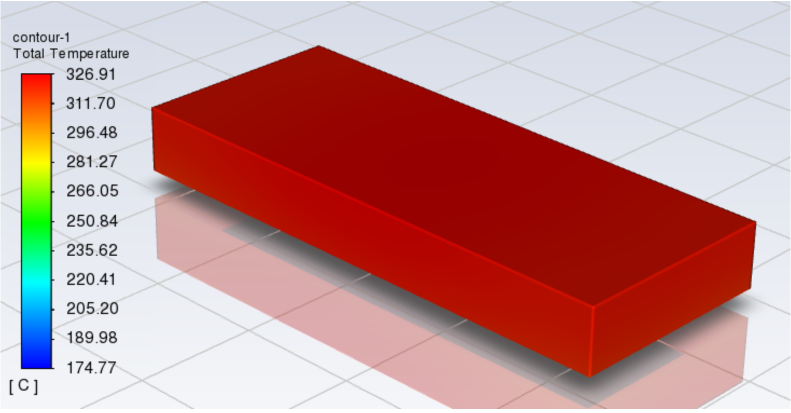


FIGURE 45 – Surface Contours on Total Temperature of CPU

The graphical solution only shows the temperature profile of the CPU as it our main goal. Figure 45 shows that there is a high temperature of 326.9°C in all the CPU grids, which does not satisfy our result of an optimal working temperature below 60°C.

Further input of parameters has been manually changed to generate our desired result of reaching a temperature below 60°C of the CPU, see table 6, also the effect of other parameters was investigated Each row represents one simulation.

TABLE 6 – Water Channel Design Results

T water in [°C]	Thermal conductivity TP [W/(mK)]	TP Thickness [m]	α water in [W/(m^{2}K)]	T max [°C]
20	2.5	0.0005	50	326.9
15	9	0.0005	50	292
15	9	0.0001	50	343
10	9	0.0001	50	338
10	9	0.0005	50	287.68
10	9	0.0001	200	130.16
10	9	0.0001	400	91.71
10	9	0.0001	2000	47.53
40	9	0.0001	2000	77.44
40	9	0.0001	2500	74.15
40	9	0.0001	4000	68.6
40	9	0.0001	6000	65.02
25	9	0.0001	2500	59.19
25	9	0.0001	3000	56.8
25	7	0.0001	3000	57.27
25	7	0,0001	2500	59.59
25	6	0,0001	2500	60.17

The ideal thickness of thermal paste as stated in the literature research has shown that 0.0001m is the most optimal compared to 0.0005m. In the table we can see the comparison between the two thicknesses, since it has made a difference of 51°C (17%), see table 7. However, this is just the effect of the thickness of the thermal paste. The situation without thermal paste with contact thermal resistance will be definitely worst for CPU (overheat).

TABLE 7 – Thermal Paste Thickness Difference

T water in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α water in [W/(m^2K)]	T max [°C]
15	9	0.0005	50	292
15	9	0.0001	50	343

Having the highest thermal conductivity of the thermal paste has helped to reach the optimal working temperature. As shown on Table 8, reducing the thermal conductivity of the thermal paste has increased our temperature in this case by 0.47°C. Thermal conductivity of the thermal paste is related to its “quality” and its price off course. Higher is the thermal conductivity, the better is heat transfer but the higher is price. The results show that thermal paste with lower thermal conductivity will not satisfy the demanded heat loss.

TABLE 8 – Thermal Paste Thermal Conductivity Difference

T water in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α water in [W/(m^2K)]	T max [°C]
25	9	0.0001	3000	56.8
25	7	0.0001	3000	57.27

The main parameters that have the greatest effect on changing the results are the temperature of the water and its heat coefficient transfer. Changing the water temperature, for example, from 10°C to 40°C has risen our Max Temperature of CPU by 29.91°C, see table 9. The demanded temperature of the cooling water is 40 °C, but only by changing the convective heat transfer coefficient, it was not possible to reduce temperature of CPU below 60 °C (with respect to purposeful velocity.

TABLE 9 – Water Inlet Temperature Difference

T water in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α water in [W/(m^2K)]	T max [°C]
10	9	0.0001	2000	47.53
40	9	0.0001	2000	77.44

In terms of changing the heat coefficient transfer of the water, the higher the α the lower the maximum temperature of the CPU. As shown in Table 10, the transfer of the heat coefficient at 200 W/(m^{2}K) gives us 130.16°C, but increasing it to 2000 W/(m^{2}K) gives us a decrease in the maximum temperature difference of 82.63°C.

TABLE 10 – Water Inlet Heat Coefficient Transfer Difference

T water in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α water in [W/(m^{2}K)]	T max [°C]
10	9	0.0001	200	130.16
10	9	0.0001	400	91.71
10	9	0.0001	2000	47.53

Together with all the changes in the parameters, we were able to find our ideal water temperature stream of 25°C (out of the demanded temperature 40 °C), using the highest thermal conductivity of thermal paste with the ideal thickness. The 25 °C is common room temperature with no additional requirements to pre-cooling the water down. The final heat coefficient transfer of water is 2500 W/(m^{2}K) to achieve the maximum temperature of the CPU at 59.19°C, see table 11.

TABLE 11 – Water Channel Design Final Result

T water in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α water in [W/(m^{2}K)]	T max [°C]
25	9	0.0001	2500	59.19

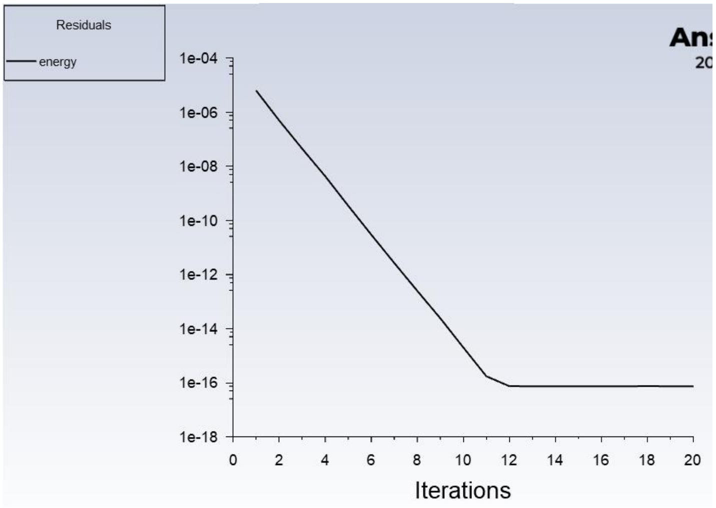


FIGURE 46 – Water Channel Design Final Scaled Residual

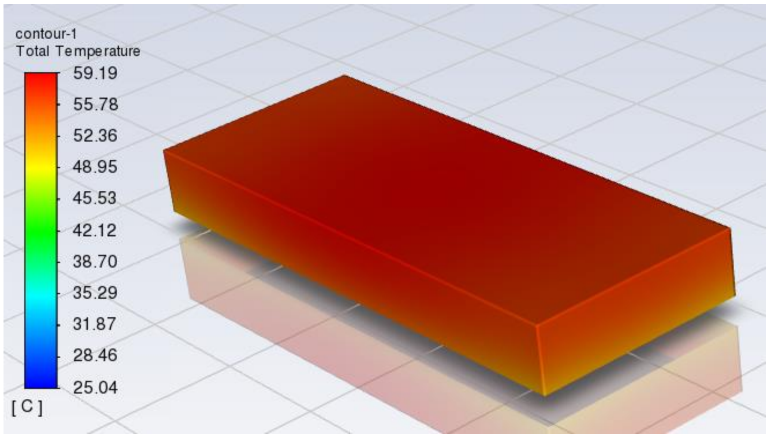


FIGURE 47 – Water Channel Design Final Total Temperature Contour of CPU

From Figure 47, we can see based on the contour of the total temperature of the CPU at 59.19°C right on the top surface of the CPU where the most heat is being generated there due to the cooling system at the bottom at around 54°C. The residual trend is shown in the figure 46.

5.4.2 Air Fins Design Results

The initial conditions for the air fin design were the following: air fin length is 0.05 m and air temperature at 40°C with heat coefficient transfer α of 2000 W/(m^2K). The highest thermal conductivity of thermal paste at 9 W/(mK) with a thickness of 0.0001m was applied. The result was 77°C which does not satisfy our goal, see table 12.

TABLE 12 – Air Fins Design Input #1

Fins Length [m]	T_{air} in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α in air [W/(m^2K)]	T max [°C]
0.05	40	9	0.0001	2000	77

Adjustments have been made, based on the water channel design, we have learnt which parameters have the most effect and will act on it for testing purposes to reach maximum temperature of CPU under 60°C. The total trials for air fins length of 0.05m is shown below, see table 13.

TABLE 13 – Air Fins Design Results

T_{air} in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α in air [W/(m^2K)]	T max [°C]
40	9	0.0001	2000	77
40	9	0.0001	3000	68.9
40	9	0.0001	5000	65
25	9	0.0001	1800	63.07
25	9	0.0001	2500	59.94
25	7	0.0001	2500	60.56
25	7	0.0001	3000	59
25	5	0.0001	3000	60.99
25	5	0.0005	3000	88.11
25	7	0.0001	3000	59
25	5	0.0001	2500	62.54

The difference in air fins length design was also tested, having longer fins will reduce the max temperature of CPU due to an increase in surface area for heat transfer, see table 14.

TABLE 14 – Air Fins Length Difference

Fins Length [m]	T_{air} in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α in air [W/(m^2K)]	T max [°C]
0.01	25	9	0.0001	3000	65.98
0.05	25	9	0.0001	3000	62.2
0.08	25	9	0.0001	3000	53.9

As shown in Table 15, the result for air fin length 0.05 m can only reach the goal by reducing the ambient air temperature to around 25°C with a heat coefficient transfer of 2500 W/(m^2K). It is not possible to have such a high heat coefficient transfer in the real world, with respect of the further analysis in the thesis.

TABLE 15 – Air Fins Design Final Result

Fins Length [m]	T_{air} in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α in air [W/(m^2K)]	T max [°C]
0.05	25	9	0.0001	2500	59.94

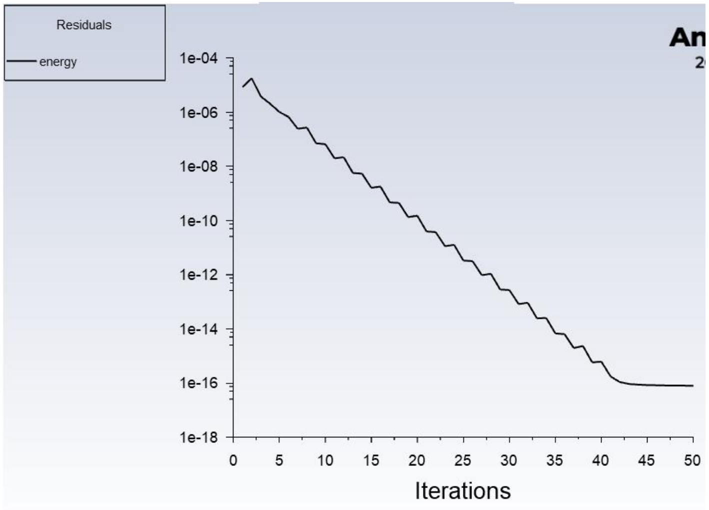


FIGURE 48 – Air Fins 0.05m Length Design Final Scaled Residual

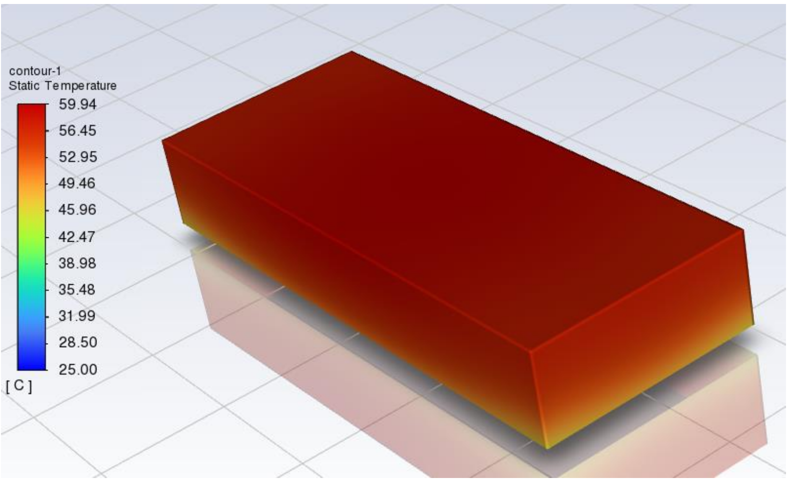


FIGURE 49 – Air fins 0.05m Length Design Final Total Temperature Contour of CPU

From Figure 49, we can see based on the contour of the total temperature of the CPU at 59.94°C right on the top surface of the CPU where the most heat is being generated there due to the cooling system at the bottom at around 54°C. The residual trend is shown in the figure 49.

5.5 Summary of CFD Simulation

For water channel design, we find that the ideal thickness of the thermal paste is 0.0001m with a thermal conductivity of 9 W/(mK). Lowering the thermal conductivity of the thermal paste with the water heat coefficient of 2500 W/(m^{2}K) will exceed our CPU max temperature of 60°C. Therefore, we need to keep the thermal paste at its highest thermal conductivity of 9 W/(mK), with the water temperature at 25°C. Increasing the flow rate of water by 20%, which corresponds to the heat coefficient transfer of water from 2500 W/(m^{2}K) to 3000 W/(m^{2}K), will decrease the maximum temperature by 4.2%.

For air fins design, we find that the ideal heat transfer coefficient for air is 2500 W/(m^{2}K) with a temperature of 25°C in the simulation. Regarding the thermal conductivity of the thermal paste, we can use a lower grade at 7 W/(mK), which will still have the maximum CPU temperature below 25°C. We will need a higher air flow rate to have a lower thermal conductivity; therefore 9 W/(mK) is chosen for its thermal conductivity. Increasing the flow rate of air by 20%, which corresponds to the heat coefficient transfer of water from 2500 W/(m^{2}K) to 3000 W/(m^{2}K), will decrease the maximum temperature by 1.7%. The final results are presented in tables 16 and 17.

TABLE 16 - Water Channel Result

T_{water} in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α water in [W/(m^{2}K)]	T max [°C]
25	9	0.0001	2500	59.19

TABLE 17 - Air Fins Result

Fins Length [m]	T_{air} in [°C]	Thermal conductivity [W/(mK)]	TP Thickness [m]	α in air [W/(m^{2}K)]	T max [°C]
0.05	25	9	0.0001	2500	59.94

6. Design Calculations

The data has been gathered with our result found, now we will be processing the data to find our mass flow rate of water needed to cool down the CPU to its optimal temperature.

6.1 Water Channel Design Calculations

We will be using the result displayed on Table 16 as our input. First, we find characteristic length D_h of the water channel box:

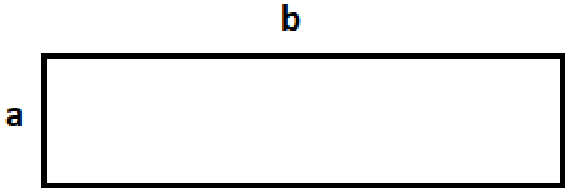


FIGURE 50 – Water Channel Box Geometry

$$D_h = \frac{4ab}{2(a+b)} \tag{11}$$

Upon obtaining characteristic length D_h, we can calculate the Nusselt number using the transfer of the heat coefficient transfer of water of 2500 W/(m^{2}K) taken from the result of our simulation.

$$Nu = \frac{\alpha D_h}{\lambda_f} \tag{12}$$

From Nusselt's number, we can calculate Reynold's number based on Whitaker's correlation.

$$Re = \sqrt[0.83]{\frac{Nu}{0.015Pr^{0.42}}} \tag{13}$$

From Reynold’s number, we can determine whether the water flow is laminar or turbulent. The flow speed will be calculated using the Reynold number equation.

$$\bar{u} = \frac{Re \times \mu}{D_h \rho} = \frac{Re \times \nu}{D_h} \tag{14}$$

Lastly, the flow rate will be calculated with the channel cross-section

$$\dot{V} = S\bar{u} = ab\bar{u} \tag{15}$$

Following results were obtained using these formulas, see table 18.

TABLE 18 - Water Channel Design Flow Rate

Nusselt's Number	Prandtl's Number	Reynold's Number	velocity [m/s]	Vol. Flow rate [m^3/s]
89.58	6.90	13337.96	0.62	0.00075

As we can see, Reynold's number exceeds 2300, which means the flow is turbulent. The water mean velocity is 0.62 m/s with Vol flow rate at 0.00075 m^3/s. The CPU power 500 W corresponds rather to control unit for heavy electrical device or drive of the motor than for computer (30-60W). Therefore the flowrate of the water to cool down the investigated CPU is real.

6.2 Air Fins Design Calculations

Calculating the mass flow rate of air will be a similar procedure as 6.1 Water Channel Design Calculations, with the exception of different medium being Air. We will use the result shown on Table 17, our input of the heat coefficient transfer of 2500 $W/(m^2K)$. As for the calculation of cross section (simplification to two parallel plates), it is calculated as follows:

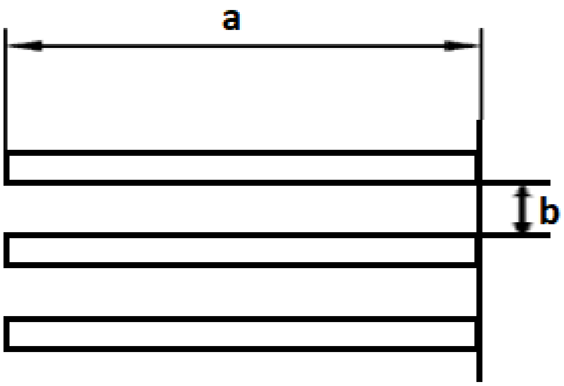


FIGURE 51 – Air Fins Geometry

$$D_h = \frac{4ab}{2a} = 2b \tag{16}$$

The rest of the formula will be following same as 6.1 Water Channel Design Calculations.

TABLE 19 - Air Fins Design Flow Rate

Nusselt's Number	Prandtl's Number	Reynold's Number	velocity [m/s]	Vol. Flow rate [m^3/s]
2286.59	0.0007159	2060518.64	1617.51	1.29401

During the simulation, we are able to get our result at a max temperature of 60°C for the CPU. As the result shows from our calculations, the velocity is in the supersonic region, that is why it is not possible to cool the CPU.

7. Conclusion

The thesis deals with investigation of the effect of the thermal paste parameters on the heat transfer in electronic device cooling process. Two simple cases of the cooling system were proposed, utilizing water and air. The thermal pastes are used for contact thermal resistance reduction.

The simplified 1D symmetrical problem have been created and calculated for both cases, air cooling and water cooling. The heat transfer coefficient of air must be higher than 3277 W/(m^2K) and for water must be higher than 4495 W/(m^2K) to get the maximum CPU temperature to be lower than 60°C.

Compared to the simulation, with air temperature at 40°C is not possible to cool the CPU below 60°C due to additional heat resistances with different thickness of materials even the both cooling media.

Increasing of the fins length has increased the surface area to increase the heat transfer from the CPU to the air fins cooling as proven from laws of physics and then is tested in the simulation. Increasing the length from 0.01m to 0.05m has decreased the temperature from 65.98°C to 62.2°C (a decrease of 6%) with an air heat transfer coefficient of 3000 W/(m^2K).

To cool the CPU below 60°C with air convection, we have simulated in 3D geometry that an air flow rate of 1618 m/s at the air heat coefficient transfer of 2500 W/(m^2K) with an air temperature of 25°C, but this is also not possible due to the air flow rate being in a supersonic state. Therefore, it is not possible to use air fins design to cool down under such conditions.

Regarding water, the heat transfer coefficient of water must be higher than 4495 W/(m^2K) with a water temperature of 40°C to get the maximum CPU temperature to be less than 60°C, calculated from 1D problem. Compared to the simulation, with water temperature at 40°C, it is also not possible to cool the CPU below 60°C due to heat loss in heat transfer. Therefore, we can see that in calculations and in practise, the difference in maximum CPU temperature for heat coefficient transfer 4000 W/(m^2K) is 12.2%. But to cool the CPU below 60°C, we would need a turbulent water flow rate of 0.62 m/s at a water heat coefficient transfer of 2500 W/(m^2K) with a water temperature of 25°C.

Having a higher thermal paste grade of 9 W(mK) will help us decrease in the maximum CPU temperature, as seen in the simulation. Comparing the thermal conductivity of 5 W/(mK) and 9 W/(mK), an increasing of max CPU temperature from 59.94°C to 62.54°C, respectively (4.33%). The quality of the thermal paste is also important.

By changing the thickness of the thermal paste layer, we see how important it is to use a thermal paste solution to eliminate the gap with lower contact resistance and to have a better heat transfer from the CPU to the cooling system, as predicted in the literature research and tested in the simulation. The spatial 3D case with complex geometry was investigated. Increasing the thickness of thermal paste (3 layers) in the design of air fins in the simulation from 0.0001m to 0.0005m increasing thermal resistance $R_t = H/\lambda$ (400%) has also increased the maximum CPU temperature from 60.99°C to 88.11°C (44.4%).

In conclusion, the use of thermal paste is important for better heat transfer from the electronic device to the cooling source. By comparing the calculated example with the simulation, we were able to obtain a similar result in finding the heat coefficient transfer for water. During simulation, we have successfully cooled the CPU under the desired temperature of 60°C but in practice, it is not possible for air fins due to its high flow rate. As for water, it was a successful result to find the optimal heat coefficient transfer of water 2500 W/(m^2K), the volume flow rate 0.00075 m^3/s, the thermal conductivity of the thermal paste 9 W/(mK) and its optimal thickness 0.0001m to achieve our goal of cooling down the CPU below 60°C, but with the water temperature 25°C. If the utilization of the 40°C water is inevitable, the best way to cool down the CPU is to adapt the water channel with inner fins to increase the heat transfer surface but with the cost of higher pressure drop in the water side.

List of symbols

SYMBOL	NAME	MEASURE UNIT
A	Surface Area	$[\mathrm{m}^2]$
D_h	Characteristic length	[m]
H	Half of Length CPU	[m]
L	Thickness of the material	[m]
P	Power	[W]
Q	Heat Flow	$[\mathrm{W}]$
q	Heat Flux	$[\frac{\mathrm{W}}{\mathrm{m}^2}]$
R_t	Thermal Resistance	[K/W]
ΔT	Temperature gradient	[K]
T	Time	[s]
T_F	Bulk Fluid Temperature	[K]
T_m	Mean Junction Temperature	[K]
T_w	Wall Temperature	[K]
$\bar{\mathrm{u}}$	Mean velocity	[m/s]
V	Volume	$[\mathrm{m}^3]$
$\dot{V}$	Volume flow rate	$[\mathrm{m}^3/\mathrm{s}]$
x	Length parameter	[m]

α	Heat Transfer Coefficient	$[\frac{\mathrm{W}}{\mathrm{m^2K}}]$
λ	Thermal Conductivity	$[\frac{\mathrm{W}}{\mathrm{mK}}]$
μ	Dynamic Viscosity	[Pa.s]
ρ	Density	$[\mathrm{kg/m^3}]$
ν	Kinematic Viscosity	$[\mathrm{m^2/s}]$
Bi	Biot Number	-
Pr	Prandtl Number	-
Nu	Nusselt Number	-
Re	Reynolds Number	-

References

[1] *VDI Heat Atlas*. Second Edition. Berlin: Springer, [2010]. ISBN 978-3-540-77876-9.

[2] *Shanmugam, M. (2014). Lecture 2: Introduction to the CFD Methodology. Czech Technical University Fluent Intro 15.0 L02 Intro CFD: https://www.cvut.cz/*

[3] *ENSC 388 engineerimg thermodynamics and Heat transfer. Majid Bahrami. (2015, September 30) - Teaching - ENSC 388. (n.d.). Retrieved from https://www.sfu.ca/~mbahrami/ensc388.html*

[4] *Entrée, E. B. (n.d.). Thermal contact resistance. THERMOPEDIA. Retrieved fromhttps://www.thermopedia.com/content/1188/*

[5] *Gobor, A., & Habeeb, B. (2017, March 23). Thermal compound guide. ekwb.com. Retrieved, from https://www.ekwb.com/blog/thermal-compound-guide/*

[6] *Tools, P. E. (n.d.). News : Process engineer's tools is moving to a new address, www.myengineeringtools.com, click here to discover the site. Conduction through a wall and a composite wall - Equations and calculator - Heat conduction. Retrieved from https://powderprocess.net/Tools_html/Thermodynamics/Heat_Conduction_Wall.html*

[7] *Williams, M. (2014, December 9). What is heat conduction? Phys.org. Retrieved from https://phys.org/news/2014-12-what-is-heat-conduction.html*

[8] *Fletcher, L. S. (1972). A review of thermal control materials for metallic junctions,* A1AA Journal of Spacecraft and Rockets *Retrieved from https://www.thermopedia.com/content/1188*

[9] *Lambert, M. A. and Fletcher, L. S. (1995) Thermal Contact Conductance of Spherical Rough Metals: Theory and Comparison to Experiment, Proceedings of the ASME/JSME Thermal Engineering Joint Conference, Maui, Hawaii Retrieved from https://www.thermopedia.com/content/1188*

[10] *Contributor, T. T. (2011, March 24). What is thermal grease (thermal paste or thermal compound)? - definition from whatis.com. WhatIs.com. Retrieved from https://www.techtarget.com/whatis/definition/thermal-grease-thermal-paste-or-thermal-compound*

[11] *Admin. (2021, October 12). Thermal pastes. Thermal Management. Retrieved, from https://www.thermalmgt.com/thermal-pastes/*

[12] *Baojun Liu 2020 IOP Conf. Ser.: Earth Environ. Sci. 571 012020*

[13] *ZhihaoZhangXuehuiWangYuyingYanPersonEnvelope, (2021, October 29). A review of the state-of-the-art in Electronic Cooling.*

[14] *Jalil, Jalal & Ali, Ekbal & Kurdi, Holdani. (2018). Numerical and Experimental Study of CPU Cooling with Finned Heat Sink and Different P.C. Air Passages Configurations. Al-Nahrain Journal for Engineering Sciences. 21. 99. 10.29194/NJES21010099.*

[15] *Yada, Tomoharu & Katamoto, Yuta & Yamada, Hiroaki & Tanaka, Toshihiko & Okamoto, Masayuki & Hanamoto, Tsuyoshi. (2018). Design and Experimental Verification of 400-WClass LED Driver with Cooperative Control Method for Two-Parallel Connected DC/DC Converters. Energies. 11. 2237. 10.3390/en11092237.*

[16] Shanmugam, M. (2014). Lecture 3: Introduction to ANSYS Meshing Methodology. Czech Technical University Introduction to Ansys Meshing: https://www.cvut.cz/

[17] Herwig H. (2016). What Exactly is the Nusselt Number in Convective Heat Transfer Problems and are There Alternatives?; 18(5):198. https://doi.org/10.3390/e18050198

[18] CHOIŃSKI, Dariusz, Artur WODOŁAŻSKI, Piotr SKUPIN, Andrzej MALCHER a Krzysztof BERNACKI. Modeling and CFD simulation of an isothermal heat flow microcalorimeter. Sensors and Actuators A: Physical [online]. 2021, 331 [cit. 2023-01-14]. ISSN 09244247. Dostupné z: doi:10.1016/j.sna.2021.112999

[19] Introduction to Ansys Simplorer. (n.d.). Retrieved October, 2022, from https://www.ansys.com/

[20] Okcay, M. (2019, July 2). Thermal interface under a Plastic Quad Flat Pack. Electronics Cooling. Retrieved from https://www.electronics-cooling.com/1998/05/thermal-interface-under-a-plastic-quad-flat-pack/