Steady State Thermal Analysis of a Heat Sink

Harish Jayaraj P

Contents

This Project Report is complied in LaTeX

1 Problem Statement

Heat sinks find its application in so many places from tiny electronic circuits to huge power plants. A heat sink block is to be designed for a low temperature furnace wall with given boundary conditions. Choose materials appropriately and analyse the steady state heat conduction and convection through the heat sink and provide supporting results with colour maps and contour plots.

2 Abstract

A heat sink 3D geometry was designed in ANSYS Design-modeler, Meshed in ANSYS Meshing module, boundary conditions were given and solved using ANSYS Fluent using the energy equation for Steady-state head conduction and convection. The solution was then post-processed and was contour plotted with color maps (jet) and temperature vs. position graphs were plotted at various walls of the geometry.

3 Introduction

This project aims to design, and analyse the heat transfer produced by a heat sink. Heat sinks are used to absorb and dissipate the high temperatures created by many different sorts of electronic and mechanical devices. Its main applications are in industrial facilities, power plants, solar thermal water systems, HVAC systems, gas water heaters, forced air heating and cooling systems, geothermal heating and cooling, and electronic systems.

Heat sinks are made of aluminum or copper, which both have excellent thermal conductivity and low thermal resistance. A heat sink is designed to maximize its surface area in contact with the cooling medium surrounding it, such as the air. Air velocity, choice of material, protrusion design and surface treatment are factors that affect the performance of a heat sink.

4 3D-Model (Computer Aided Design - CAD)

The 3D-Geometry is a combination of 2 bodies with a common interface. The body in contact with the source temperature is *body* 1 and the body in contact with the ambience is *body* 2.

Body 1 is a cuboid with its height and length measuring 0.02 m. This surface comes in contact with the heat source and is named as *Source*. The thickness of this body is 0.002 meter and the surface areas are named as $Wall$ 1. The opposite face to the *Source* is named as $Interface$ 1

Body 2 is a cuboid with its height and length measuring 0.02 m. This surface comes in contact with $Interface$ 1 and is named as $Interface$ 2. The thickness of this plane is 0.002 m. The fins are extruded upon this surface with a dimension of 0.003 875 m in length and 0.0011 m in width. The height of the fins are 0.01 m. The fins are designed in a 4×10 arrangement. The surface area of fins are named as $Wall$ 2.

4.1 Dimensions

In the axis format $x \times y \times z$ and all dimensions are in $meters$ (m)
The dimensions are taken arbitrarily for a furnace wall application. This heat sink is a single block. Multiple such blocks need to be installed to make a full heat sink in a furnace wall.

$$\begin{aligned}
\text{Body 1} &\implies 0.02 \times 0.02 \times 0.002 && meters(m) \\
\text{Body 2 (Surface)} &\implies 0.02 \times 0.02 \times 0.002 && meters(m) \\
\text{Fins (40 Nos.)} &\implies 0.003875 \times 0.0011 \times 0.01 && meters(m) \\
\text{Fins offset (X)} &\implies 0.005375 && meters(m) \\
\text{Fins offset (Y)} &\implies 0.0021 && meters(m)
\end{aligned}$$

4.2 CAD models

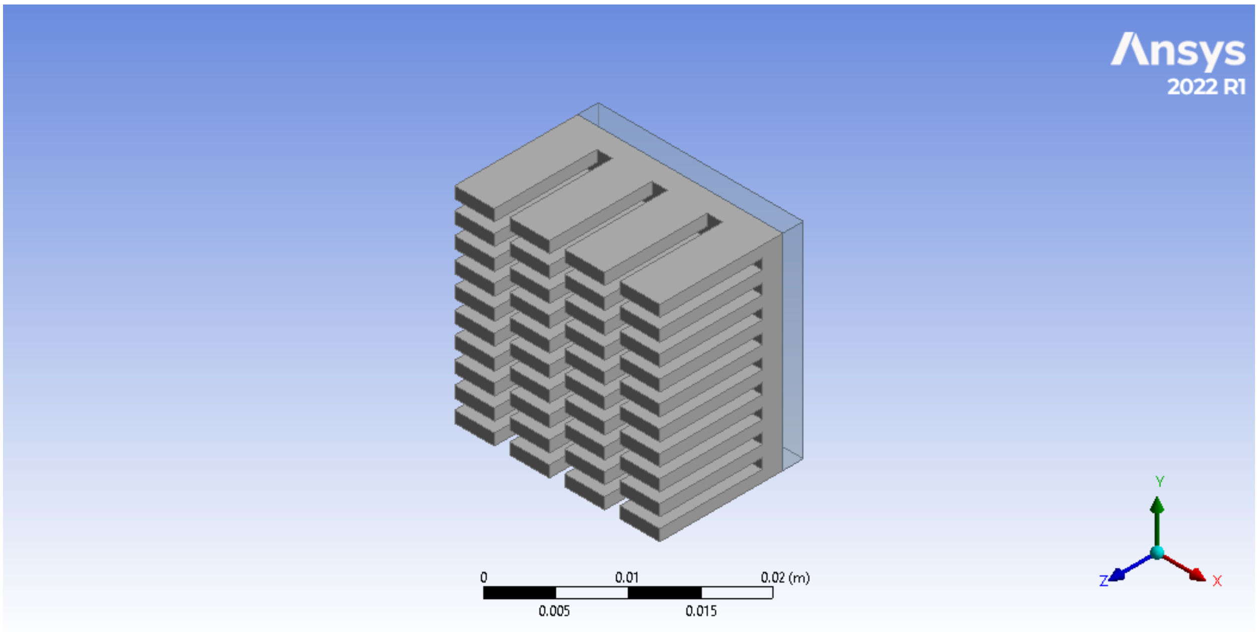

Figure 1: Isometric View (Front)

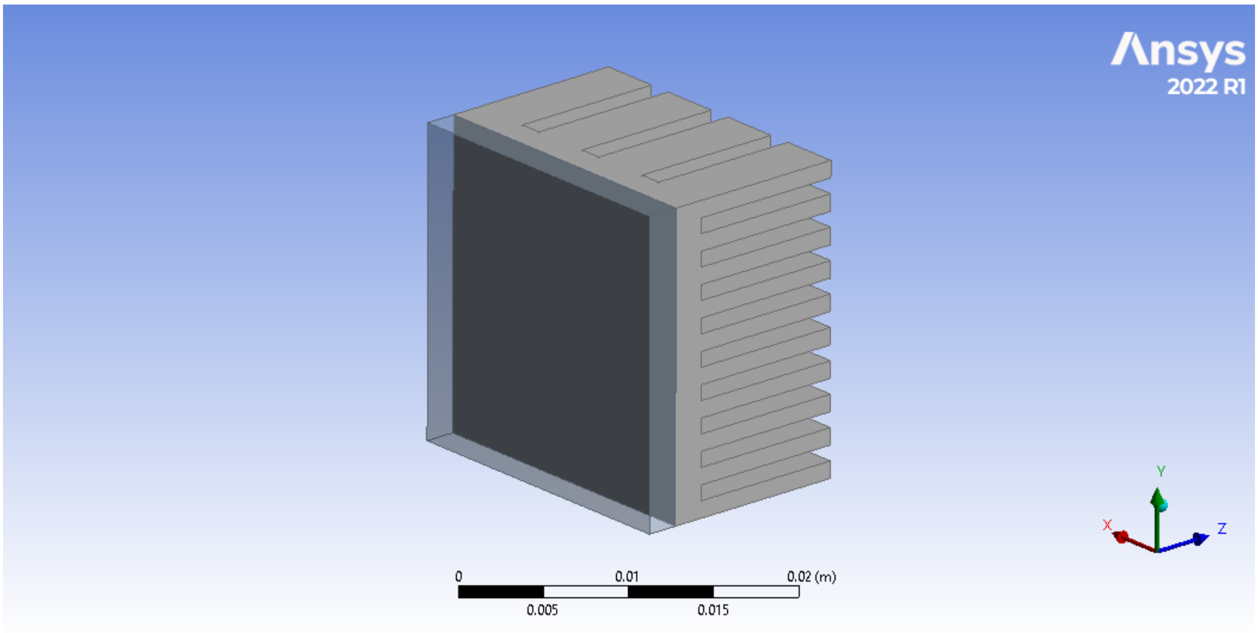

Figure 2: Isometric View (Back)

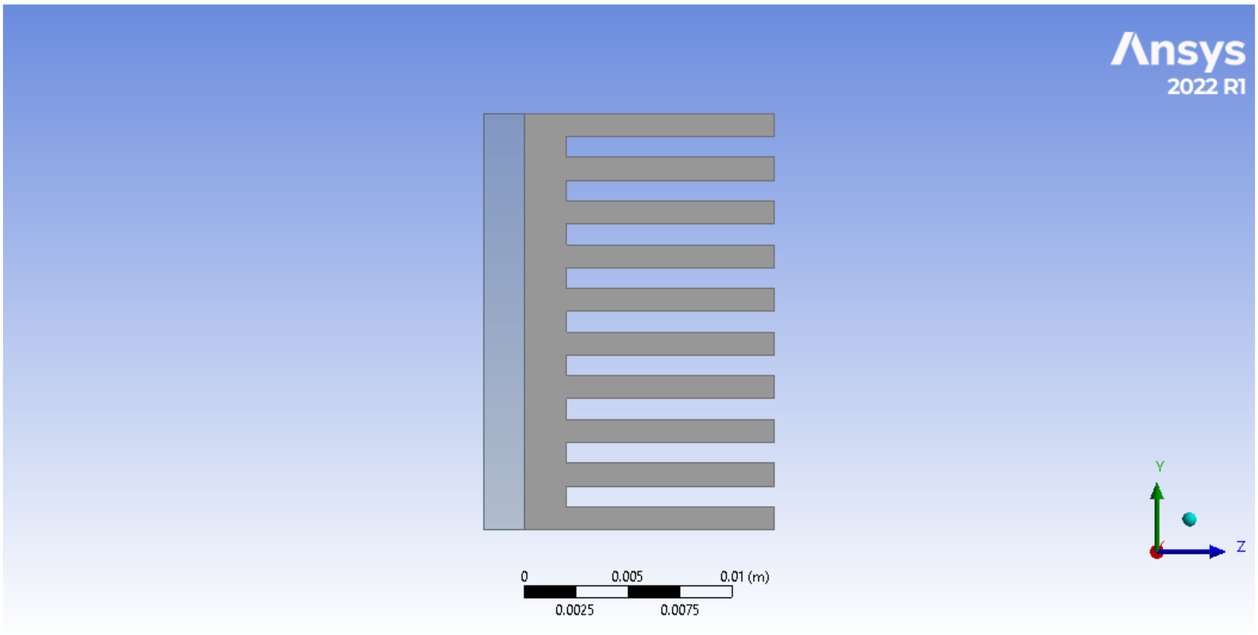

Figure 3: Left View

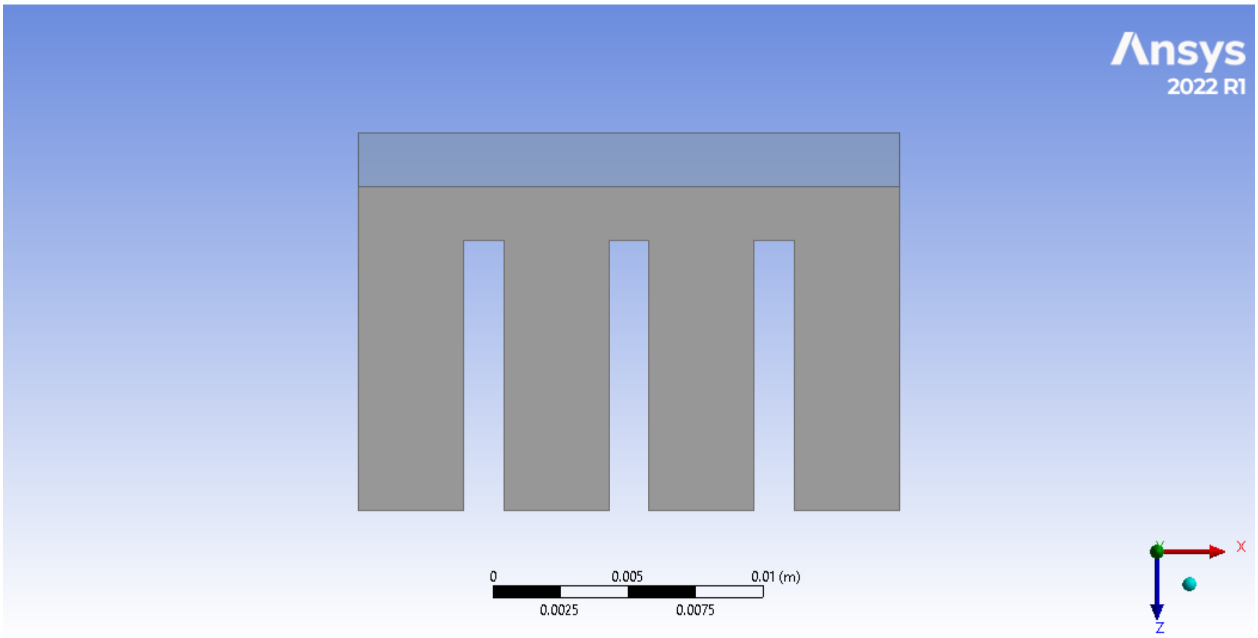

Figure 4: Top View

The 3D-Geometry was sketched and designed in ANSYS design-modeler environment

5 Method of Solution

Fourier's law of heat conduction: For heat transfer by conduction q_k, thermal conductivity k, cross-sectional area A, temperature T and distance x.

$$q_k = -kA\frac{dT}{dx}$$

Newton's law of cooling: For total heat transfer $\dot{Q}$, mass flow rate $\dot{m}$, specific heat C_p, temperature T.

$$\dot{Q} = \dot{m}C_p\Delta T$$

Finite Element Energy Equation (Governing Equation):

$$\begin{aligned}
\rho\frac{d\hat{u}}{dt} + p(\nabla \cdot V) &= \nabla \cdot (k\nabla T) + \phi \\
\rho\frac{d\hat{v}}{dt} + p(\nabla \cdot V) &= \nabla \cdot (k\nabla T) + \phi \\
\rho\frac{d\hat{w}}{dt} + p(\nabla \cdot V) &= \nabla \cdot (k\nabla T) + \phi
\end{aligned}$$

Where $\frac{d\hat{u}}{dt}, \frac{d\hat{v}}{dt}\ and\ \frac{d\hat{w}}{dt} \approx C_v dT$ in $x, y\ and\ z$ directions respectively. ϕ is the governing physical parameter. In case of energy equation it is h, heat transfer coefficient.

The above equation was solved to get the finite element solution to the given problem. The discretization (mesh) parameters are explained in the next section.

Procedure for solving the problem:

1. Define Meshing Parameters
2. Mesh the geometry
3. Create named selections
4. Turn on Energy equation
5. Define solid and fluid materials
6. Define Boundary Conditions
7. Define method of solving
8. Define no. of iterations and convergence order
9. Solve the problem
10. Analyse the contour plots with color maps
11. Plot essential graphs for analysing

6 Mesh Parameters

Mesh Images:

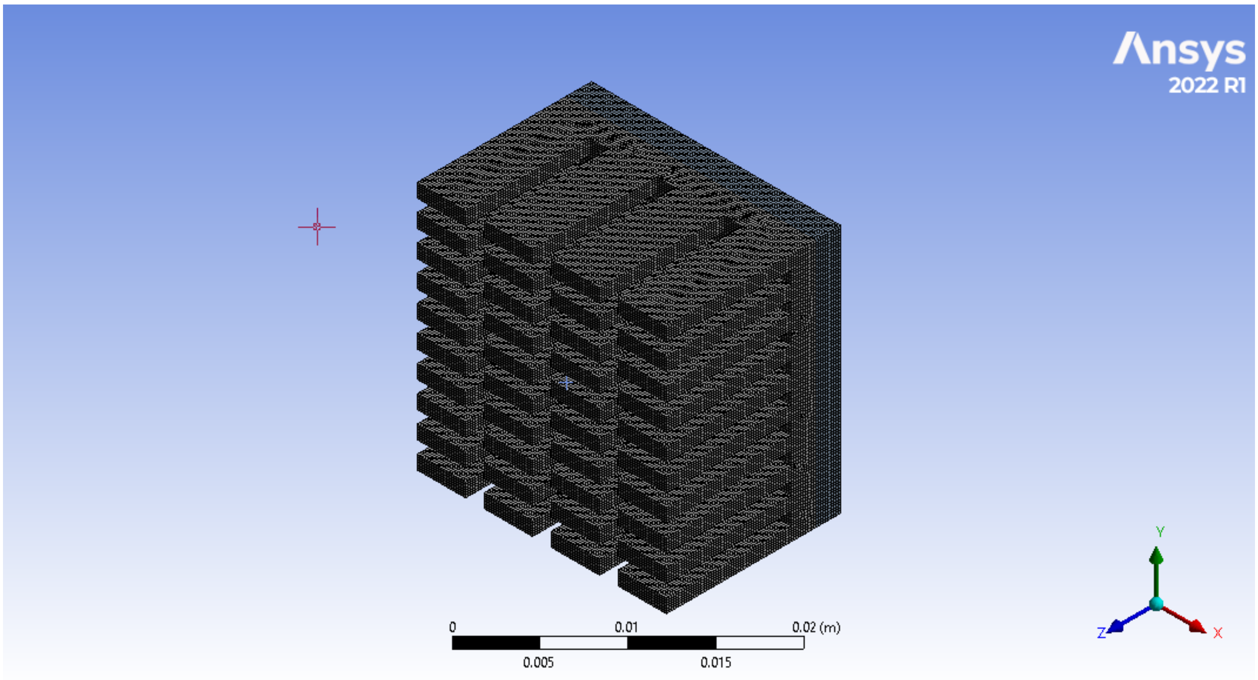

Figure 5: Meshed Geometry

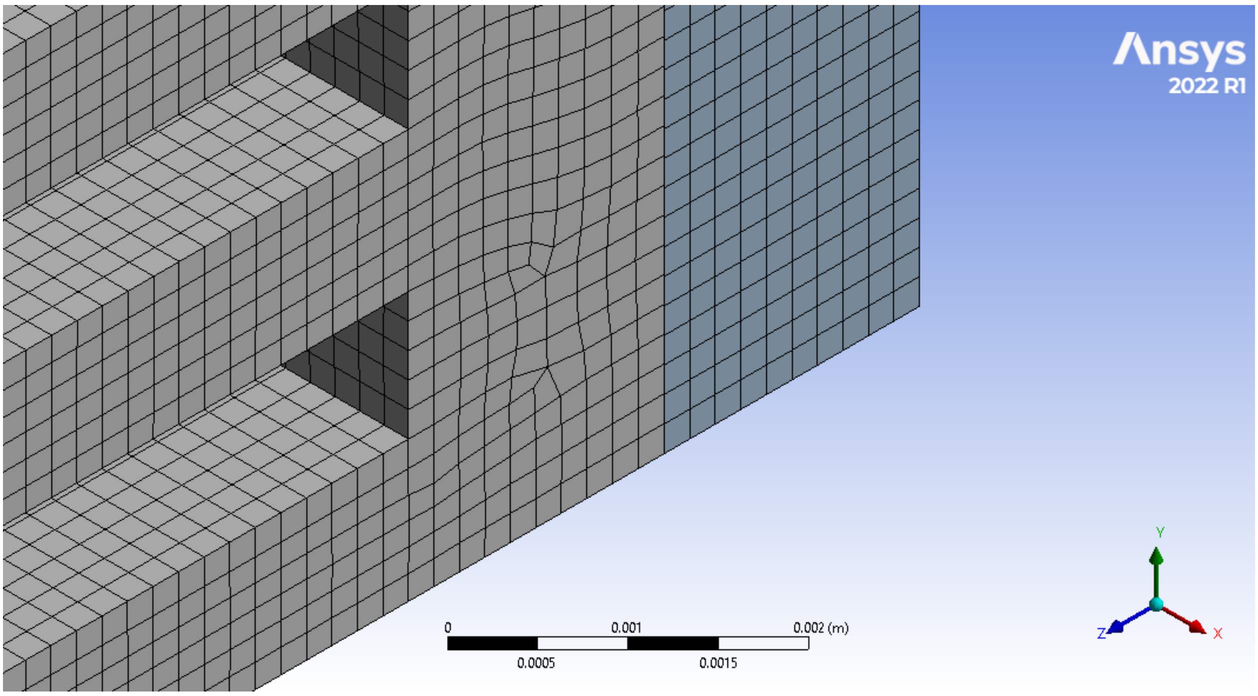

Figure 6: Meshed Details 1

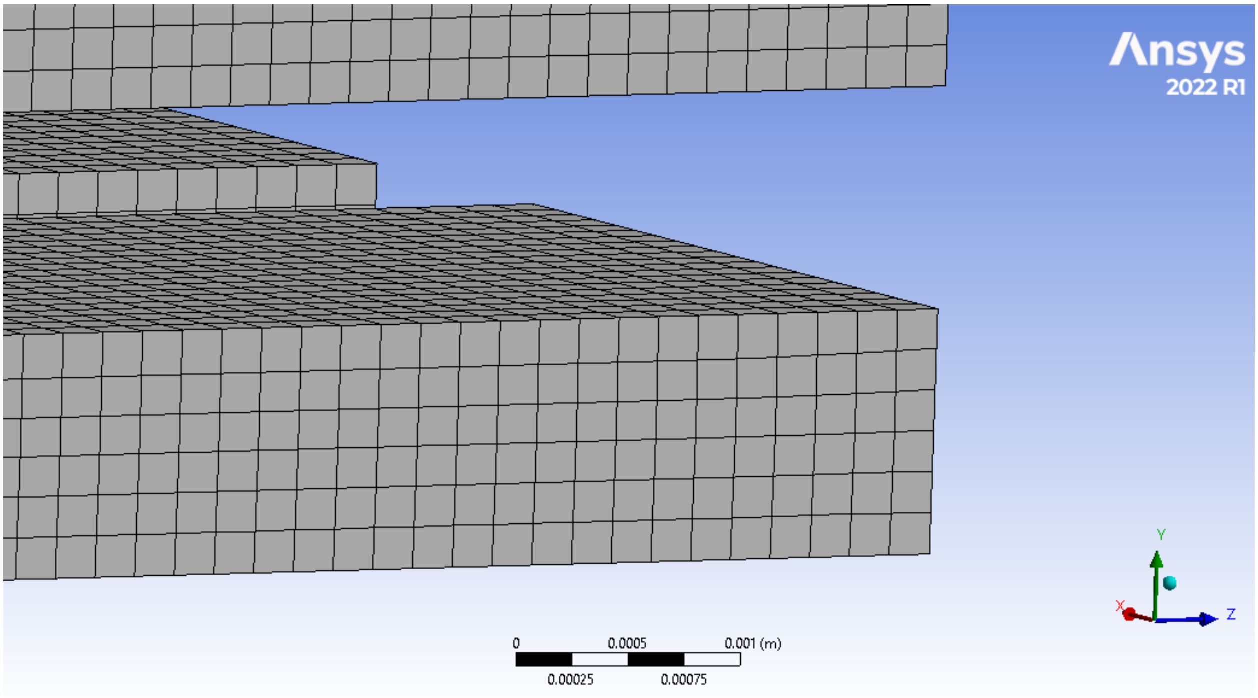

Figure 7: Meshed Details 2

Mesh Parameters:

These mesh parameters were used in the discretization of the geometry:

Parameter	**Setting**
Physics Preference	CFD
Solver Preference	Fluent
Element size	0.0002 m
Element order	Linear
Method	Multizone
Mesh type	Hexa/Prism

Table 1: Mesh Parameters

These were the no. of nodes and elements after discretization:

Parameter	**Count**
No. of Nodes	522804
Elements	455673

Table 2: Node and Element counts

7 Solution Parameters

These parameters were used in pre-processing of the CFD solution for the problem. The parameters not defined in the tables are left as defaults. The naming are used as defined in the Section 4.

A single core "CPU G3250 @ 3.20GHz" Intel® Pentium® processor with 8 Gb RAM was used to sun the solution in ANSYS Fluent environment. For initiation, the initial values for Turbulence Kinetic Energy, m^2/s^2 and Specific Dissipation Rate, s^{-1} was set to 1. And the initial Temperature, K was set to 283 K. 460 Iterations with Reporting interval of 1 was used to run calculation.

Turn on **Energy Equation** in Models and set the below parameters.

Body	Material
Wall 1	Aluminium
Wall 2	Copper
Fins	Copper
Ambient Fluid	Air

Table 3: Materials of the bodies

Location	Temperature, K	Heat Transfer Coefficient, $\frac{W}{m^2K}$	Thermal Conditions
Wall 1	283	25	Convection
Wall 2	283	25	Convection
Source	363	-	Temperature

Table 4: Boundary Conditions

Residual	Absolute criteria
Continuity	0.001
x-velocity	0.001
y-velocity	0.001
z-velocity	0.001
Energy	1e-07
k	0.001
omega	0.001

Table 5: Residual Criteria

8 Solution

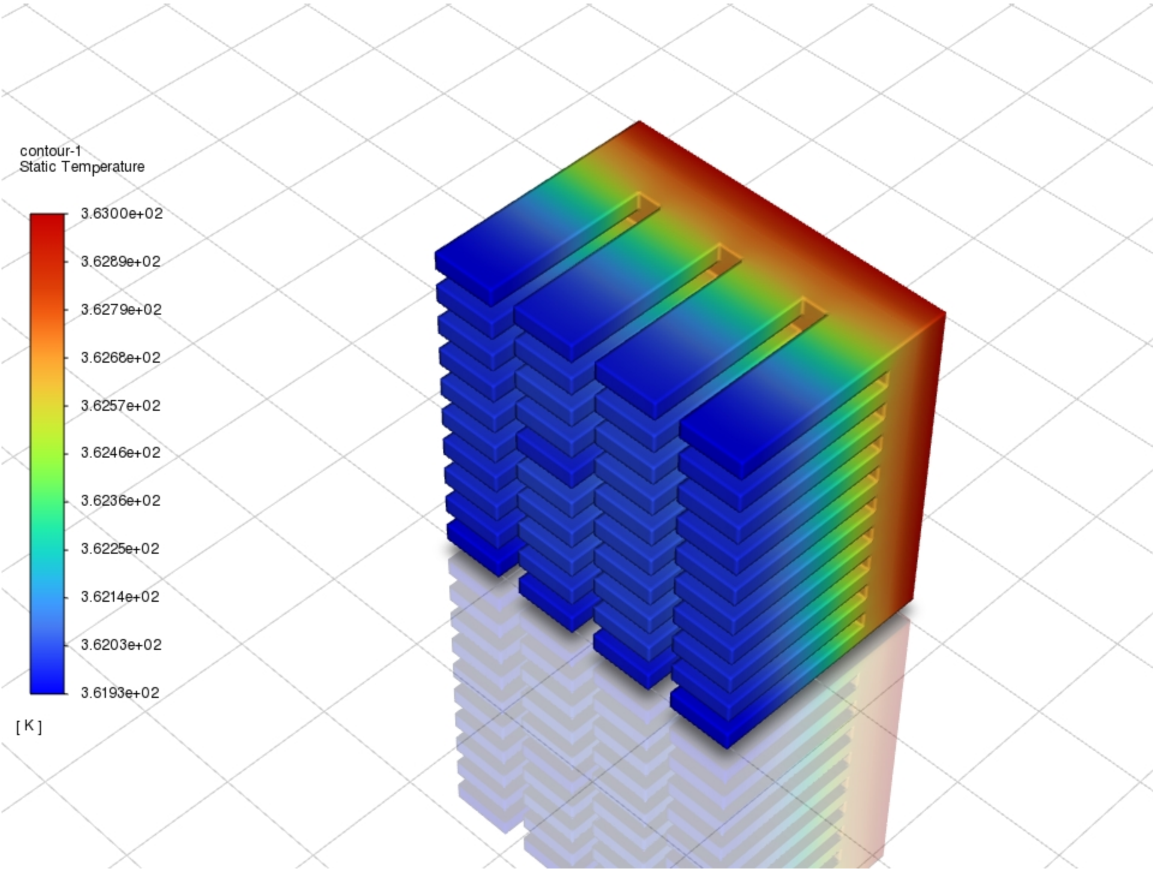

Figure 8: Solution: Isometric View

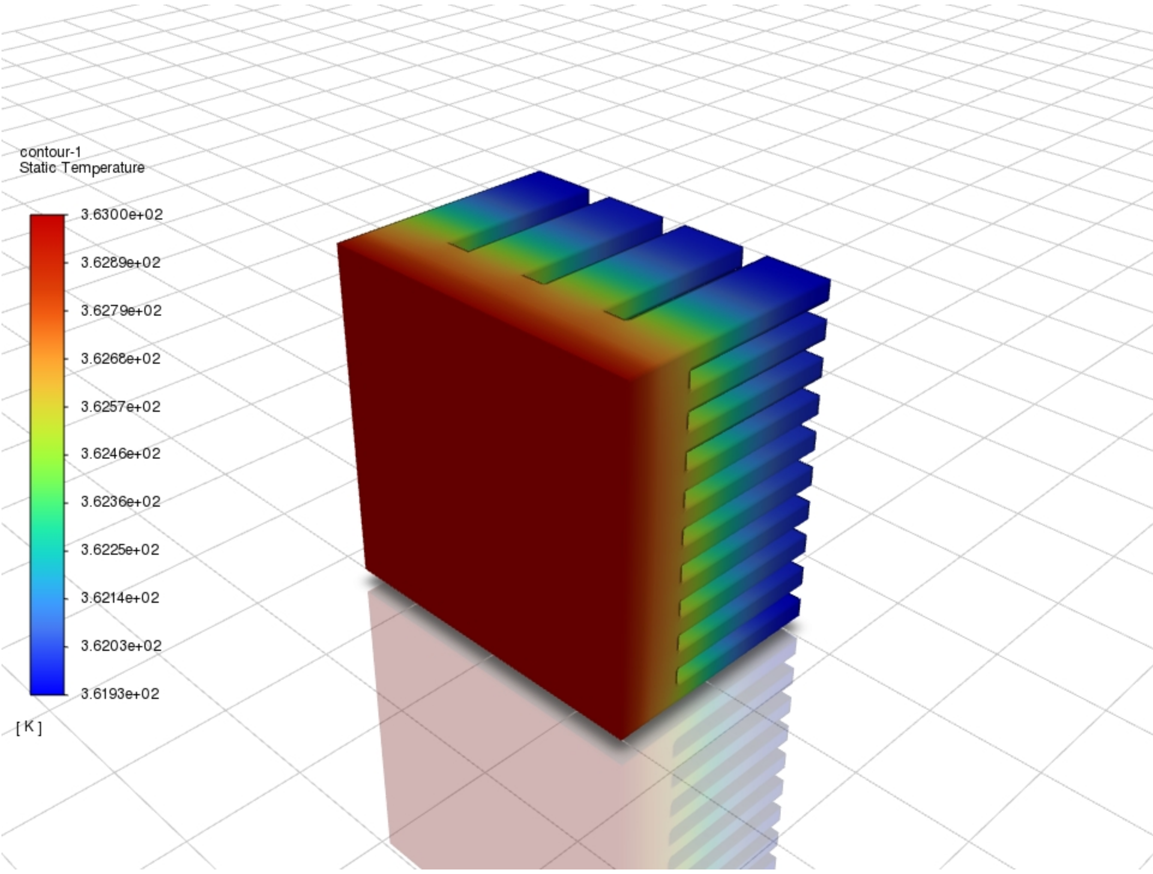

Figure 9: Solution: Isometric View (Back)

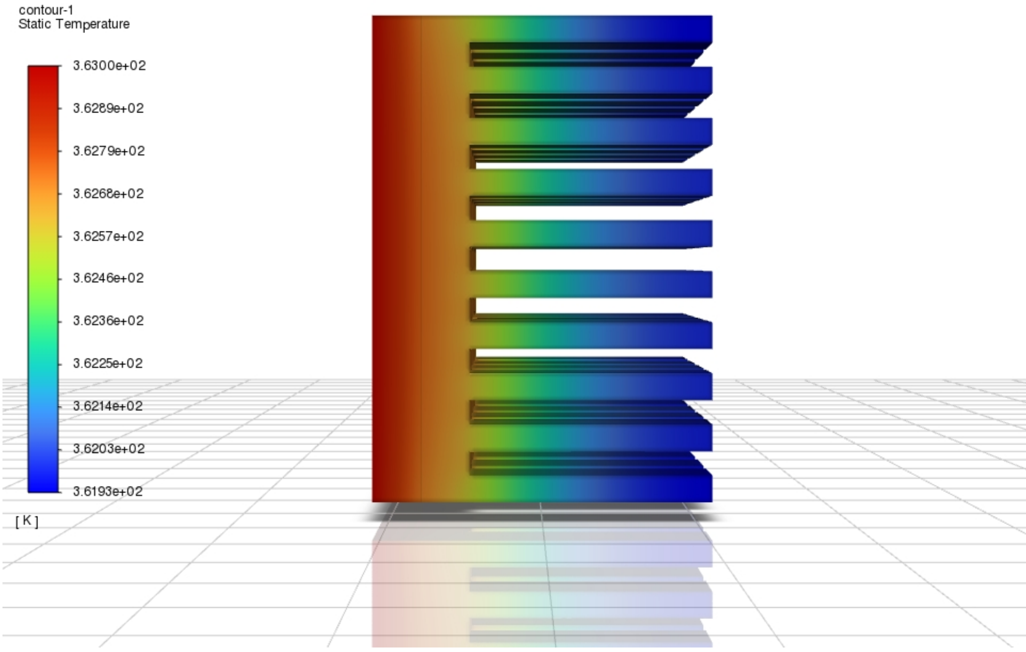

Figure 10: Solution: Right View

This is a free convection for a short fin. As it could be observed the heat (Red) at the source (Aluminium) gets transferred steadily to the interface-1. The as the heat gets transferred to interface-2 (Copper) the heat then suddenly dissipates to the ambience and by the tip of the fin, the temperature is almost drops by 3 K (blue). The heat from source - 363 K drops to ≈ 361.9 K while the ambient Temperature is 283 K.
Hence it could be inferred that for a longer fin and lower ambient temperature, with a forced convection, the heat transfer achieved could be more.

9 Graphs

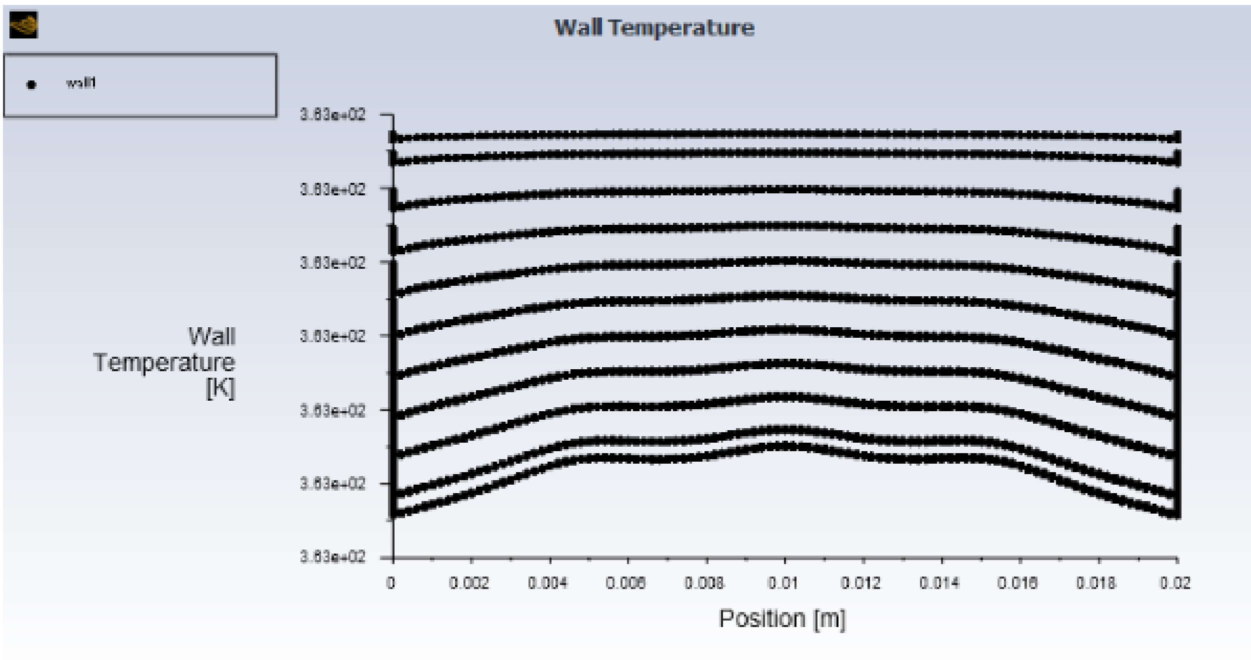

Figure 11: Static Temperature, K Vs. Position of Wall 1

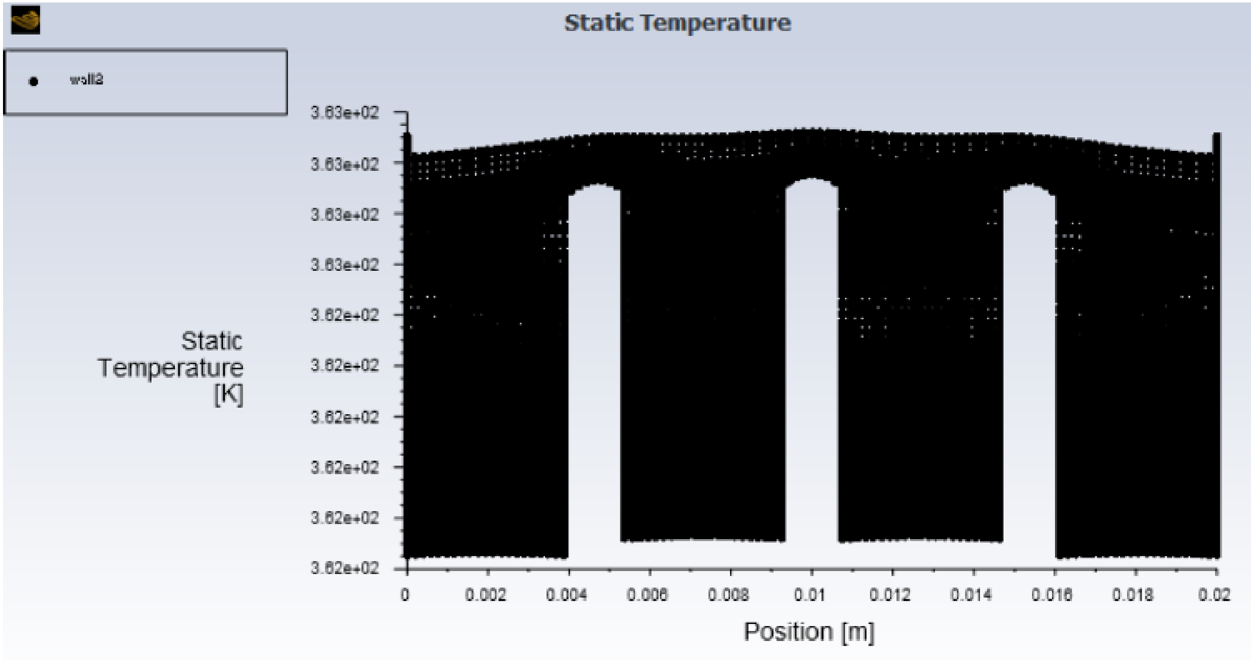

Figure 12: Static Temperature, K Vs. Position of Wall 2

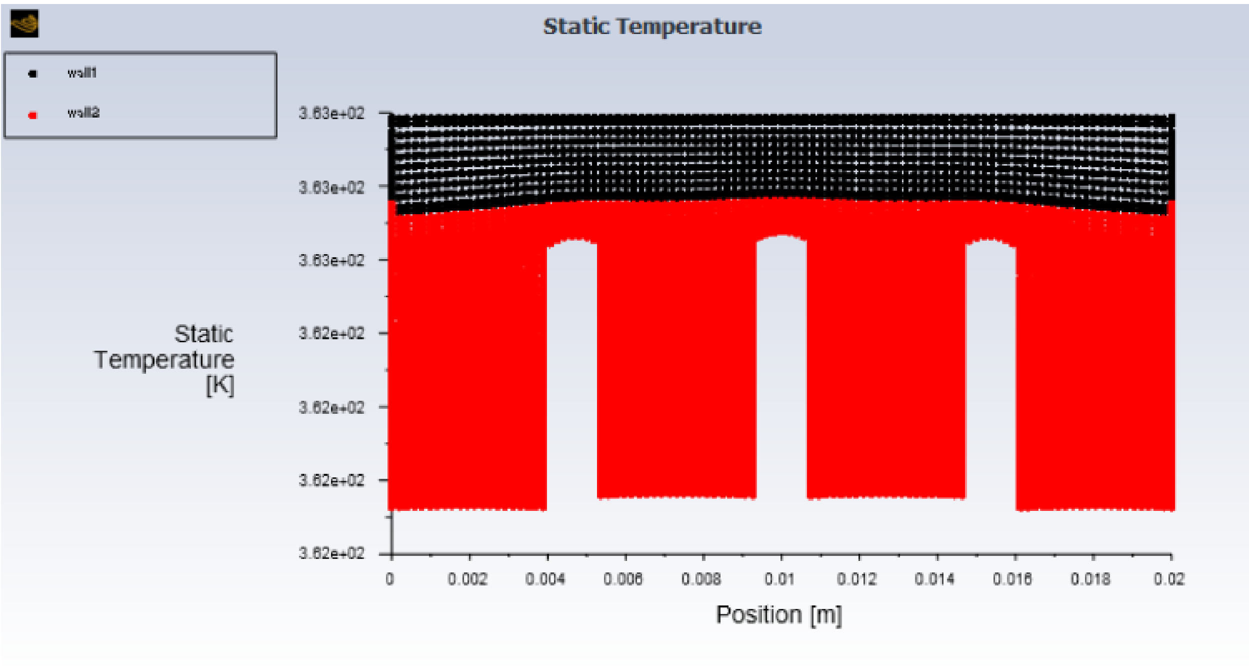

Figure 13: Static Temperature, K Vs. Position of Wall 1 + Wall 2

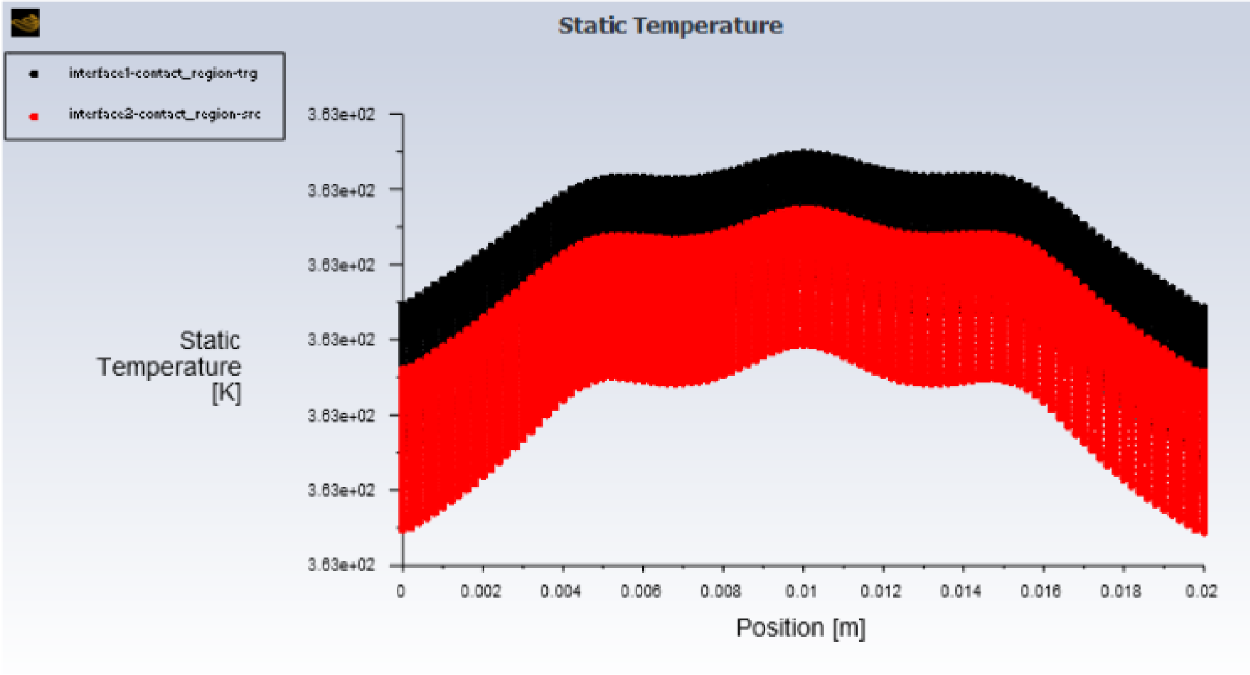

Figure 14: Static Temperature, K Vs. Position of Interface Regions

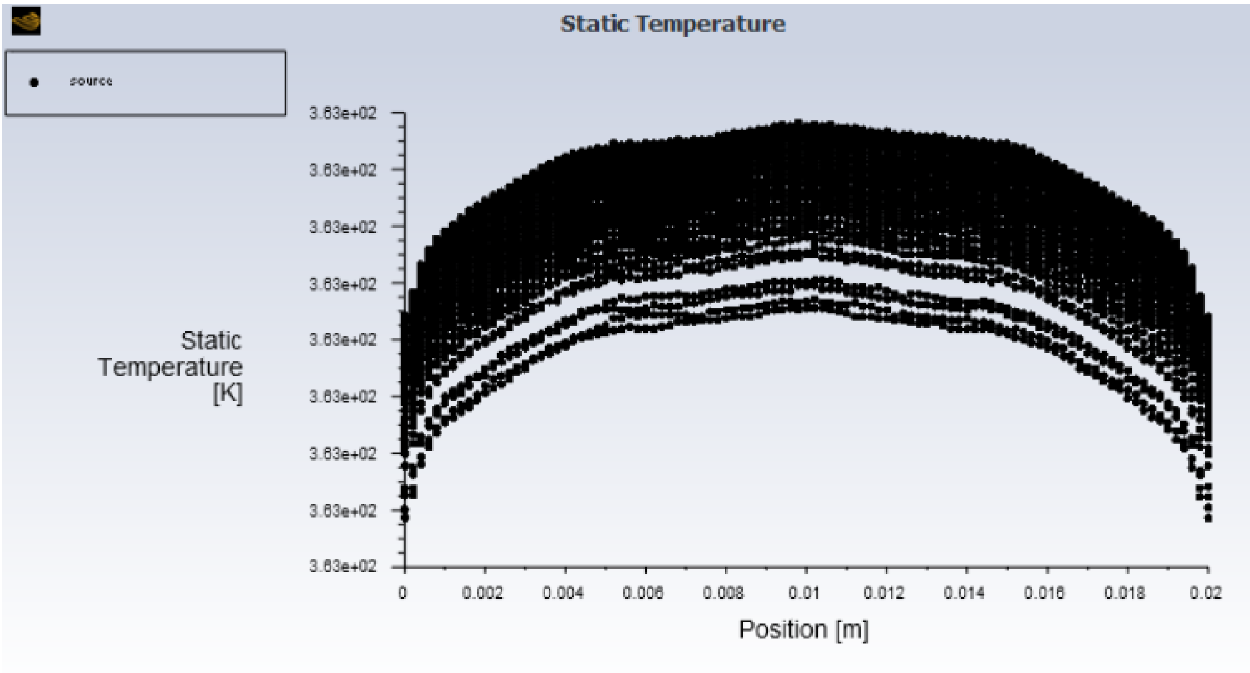

Figure 15: Static Temperature, K Vs. Position of Source

10 Application

Heat sinks are used everywhere where heat is to be absorbed and dissipated. Its main applications are in industrial facilities, power plants, solar thermal water systems, HVAC systems, gas water heaters, forced air heating and cooling systems, geothermal heating and cooling, and electronic systems. This designed heat sink block can be used in small furnaces for heat transfer and cooling. Multiple such blocks arranged in a wall with forced convection could be used.

11 Conclusion

Thus, the analysis of a heat sink block for a low temperature furnace wall with free convection was done in ANSYS Fluent and the essential contour plots and Temperature Vs. Position graphs were plotted. Based on the results obtained, it is inferred that, lower the ambient temperature, higher the heat transfer due to convection. Using this heat sink with a forced convection and lower ambient temperature would produce much higher heat transfer due to convection. On the other hand, Aluminum and Copper were used for heat transfer due to conduction. Since they are the only good conductors while being cost effective too, no further improvements can be more on the conduction heat transfer.

This report was compiled in LaTeX.
The codes and the PDF document is available at github.com/codeynamics/LaTeX/tree/main/heatSinkCFD