Thermal Analysis Report of a Heat Sink

This report presents a comparative thermal analysis of a heat sink under two different heat loading conditions. The primary objective is to understand how heat distribution affects the thermal performance and maximum temperature of the heat sink.

Objective

To compare the thermal performance of a heat sink when:

1. Heat flux is applied uniformly across the entire base.
2. Heat flux is applied only to discrete components (8 blocks) simulating real electronic components.

Heat Sink and Component Specifications

- **Heat Sink Dimensions**: 100 mm × 50 mm × 20 mm
- **Material**: Aluminium , Copper
- **Component Dimensions**: 10 mm × 10 mm each
- **Number of Components**: 8
- **Power per Component**: 18 W
- **Total Power Input**: 144 W
- **Cooling Condition**: Natural convection applied to all non-heated surfaces

Simulation Configurations

Case 1: Uniform Heat Flux on Base

Case 2: Localized Heat Flux on Components

Simulation Procedure in ANSYS

The thermal analysis of the heat sink was conducted using ANSYS Workbench, following these standard steps:

Material Selection

- First, materials were selected in the Engineering Data section.

CAD Model Import

- The CAD geometry of the heat sink was imported into ANSYS Workbench.

- Two geometries were used:
  Case 1: Heat sink only
  Case 2: Heat sink with 8 square blocks (10 mm × 10 mm) placed on the base

Geometry Setup in ANSYS Mechanical

- The model was opened in ANSYS Mechanical.

- The assigned material (Aluminum, Copper) was applied to all relevant parts of the model.

Meshing

- The geometry was discretized using the Meshing tool in ANSYS.
- Mesh quality was ensured with adequate element sizing and refinement.

Applying Boundary Conditions

- In the steady-state thermal module, boundary conditions were set:
  Case 1: A heat flux equivalent to 144 W(28800W/m2) was applied uniformly to the entire bottom surface of the heat sink.
  Case 2: Each of the 8 component blocks (10 mm × 10 mm) received a heat flux corresponding to 18W (1.8 e+005 W/m2)

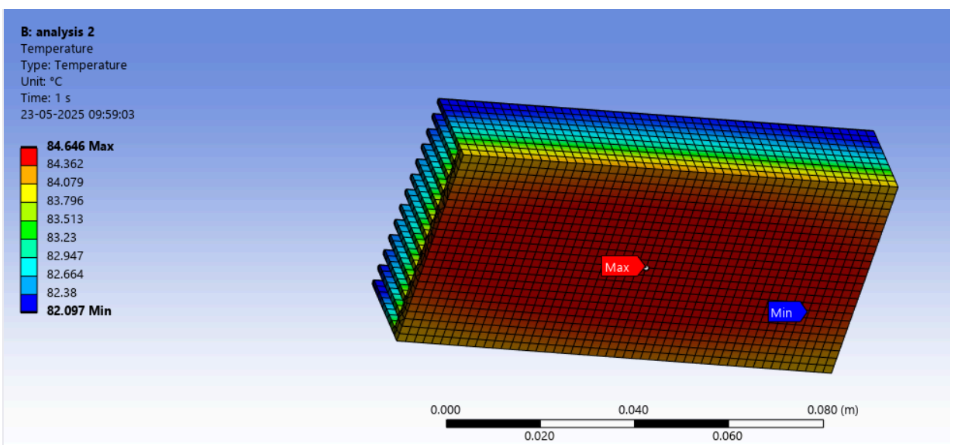
In both cases, all other exposed surfaces were assigned natural convection with a suitable heat transfer coefficient (20 $W/m^2 \cdot K$) and ambient temperature (25°C).

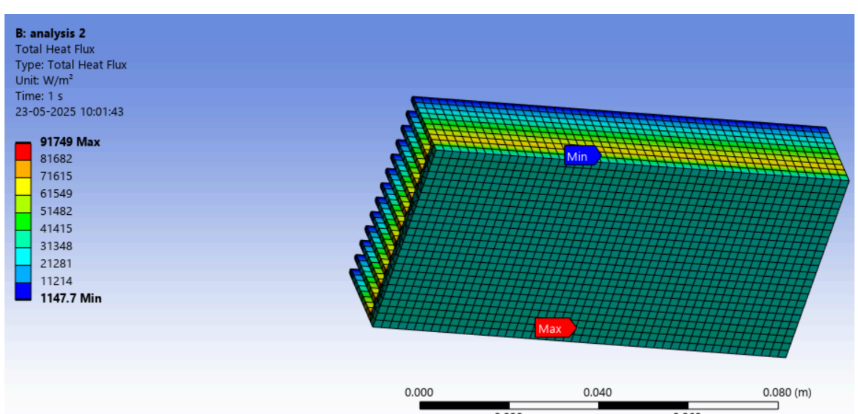
Simulation and Results

- The setup was solved using the ANSYS solver.
- Key outputs included:
  Maximum Temperature in the heat sink
  Temperature distribution plots
  Total heat flux values to verify power balance

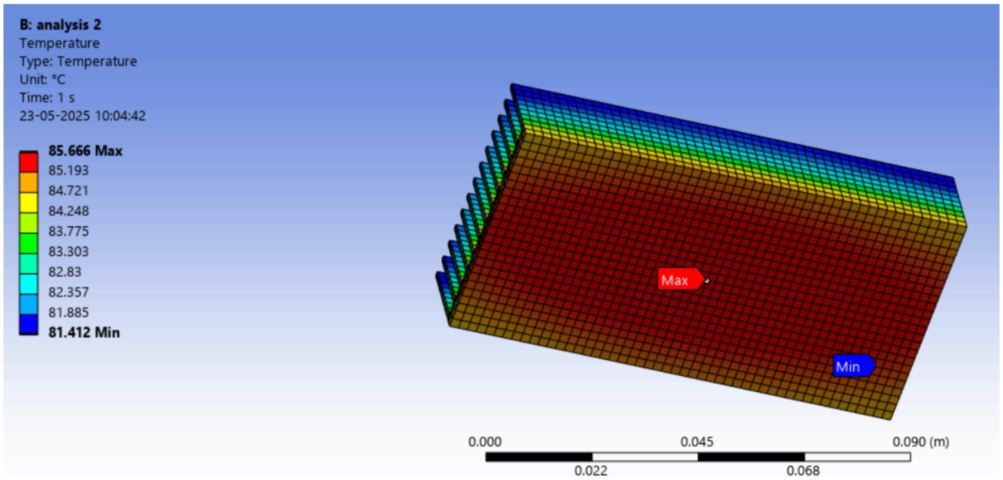
Case 1: Uniform Heat Flux on Base

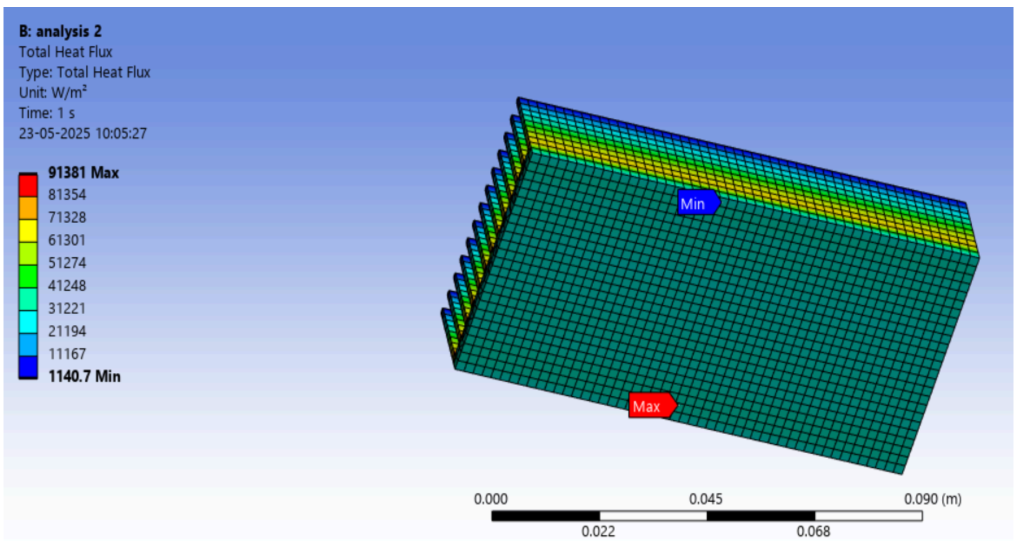
- **Description**: A total heat input of 144 W applied evenly over the entire bottom base of the heat sink (100 × 50 mm).
- **Boundary Conditions**: Convection applied to all other surfaces.
- **Result**:
  Maximum temperature = 84.646°C (Copper)





Maximum temperature = 85.666°C (Aluminium)



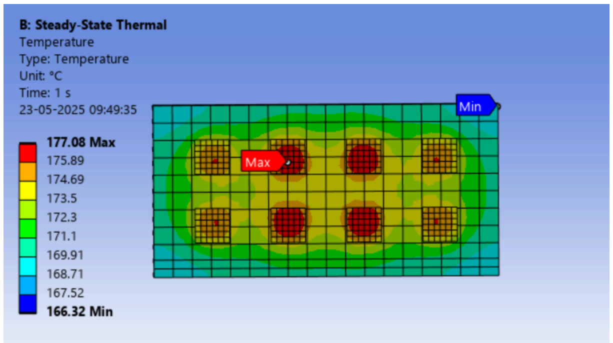


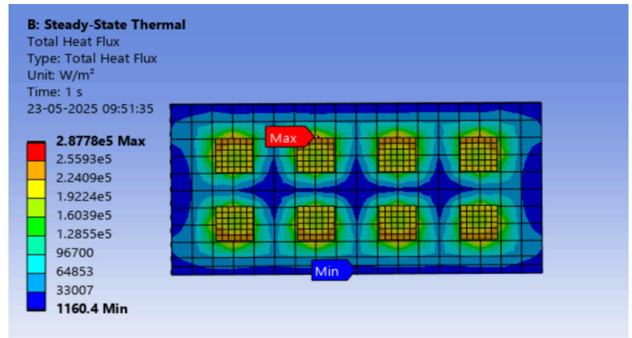
Case 2: Localized Heat Flux on Components

- **Description**: Eight 10 × 10 mm blocks on the heat sink base are defined as discrete heat sources. Each block receives 18 W, totaling 144 W.

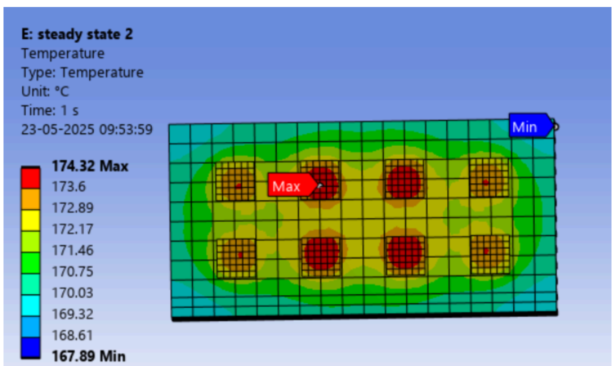
- **Boundary Conditions**: Convection applied to all other surfaces.

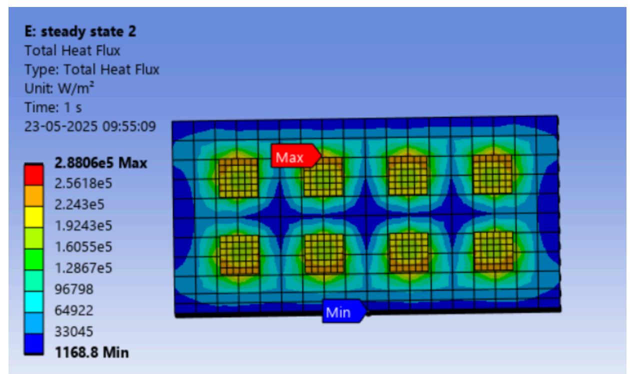
- **Result**:
Maximum temperature = 177.08°C (Aluminium)





Maximum temperature = 174.32°C (Copper)





Analysis and Discussion

- **Temperature Comparison**:
  $T_2 > T_1$ indicates that localized heating leads to poor thermal spreading and increased hot spots.

- **Thermal Distribution**:
  Uniform heating (Case 1) spreads thermal energy more evenly, reducing peak temperature.
  Localized sources (Case 2) create high local gradients and less effective spreading through the heat sink base.