Optimization of Heat Transfer Process Parameters for Heat Sink using CFD

Yogesh Kumar¹, N. V. Saxena²

¹Research Scholar, ²Assistant Professor,
1-2Department of Mechanical Engineering, Millennium Institute of Technology, Bhopal, Madhya Pradesh, India

ABSTRACT
As secondary source of heat transfer, heat sinks are primarily used. Heat generated by various equipments is first transferred to heat sink via conduction, after that heat is transferred to the environment. Heat sinks are thus used to maintain temperature of equipments under desirable range. CFD has recently emerged as an effective tool to judge performance of any equipment in design phase. CFD can also be used for design optimisation of heat sink also. In the present work, CFD analysis of heat sink has been carried out to find out the effect of change in air velocity. Effect of different materials used for the construction of heat sink on heat transfer (aluminium alloys that is AA-6063, AA-6061, AA-6060 and AA-1050) has been carried out. It has been found that material AA-1050 shows the maximum heat transfer as compared to other materials. The work has also been carried out for various thicknesses of base metals. It has been found that heat sink having 10 mm base thickness has maximum heat transfer.

KEYWORDS: Heat sink, Velocity, Heat transfer, Computational fluid dynamics, Base thickness

INTRODUCTION
In electrical and electronic gadgets, heat is additionally produced which needs to be rejected to keep the gadget safe and in proper working condition. This work can easily be done by the use of heat sinks. Heat is absorbed by the heat sinks and then rejected to the atmosphere leading to proper functioning of the equipments and increasing their life.

With time Computational Fluid Dynamics (CFD) has emerged as a powerful tool to predict the performance of any equipment in design phase. CFD can also be used for performance prediction of heat sinks. CFD uses the basic principles of fluid flow and solves the equations to get the approximate solution to a problem.

A lot of work has been done in recent past. And there is further scope of research.

LITERATURE REVIEW

Heat sink is one of the important device for rejection of heat and CFD can be very effective for its performance prediction in design phase only.

GEOMETRIC MODELING
The geometry of heat sink was modelled in ANSYS Design Modeler with dimensions given in Table 1.


Creative Commons Attribution License (CC BY 4.0) (http://creativecommons.org/licenses/by/4.0/)

Table 1: Geometric parameters of heat sink

<table>
<thead>
<tr>
<th>Parameters</th>
<th>values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length of complete channel</td>
<td>450 mm</td>
</tr>
<tr>
<td>Base thickness (t_b)</td>
<td>8 mm</td>
</tr>
<tr>
<td>Heat sink fin thickness</td>
<td>3 mm</td>
</tr>
<tr>
<td>Total height of heat sink</td>
<td>50 mm</td>
</tr>
<tr>
<td>Length and width of heat sink</td>
<td>150 mm</td>
</tr>
<tr>
<td>Width of heat source</td>
<td>60 mm</td>
</tr>
</tbody>
</table>
MESHING
For getting good results, refined mesh needs to be used. In this work the solid model of heat sink is discretized into 162699 number of element.

INPUT PARAMETERS AND BOUNDARY CONDITIONS
In order to validate the CFD model of heat sink, aluminium alloy AA-6063 has been considered. Other boundary conditions such as velocity of air was verified to check its effect, temperature of air at the inlet of domain is kept as 25°C, heat flux of 100 W was considered. For analysing the effect of different velocity of air, three different velocity of air that is 2, 4 and 6 m/s have been considered.

RESULTS AND DISCUSSIONS
ANSYS Fluent was used for analysis. Following temperature contours were obtained.

Through CFD analysis, it is found that the value of thermal resistance and nusselt number measured were near to the values obtained from the base paper. After validating the CFD model of heat sink here it analysed the effect of different materials used to manufacture heat sink and find the optimum material for the manufacturing of heat sink. It also calculates the effect of different base thickness of material, base thickness of heat sink also responsible for heat transfer.

So in order to optimize the base thickness of heat sink here in this work it considered six different base thickness of heat sink.
Nusselt number of AA-1050 material heat sink is higher, the value of thermal resistance for this material lower than the other materials. Through analysis it is found that change in thermal resistance of heat sink with respect to velocity of air flowing inside the channel follow same nature of variation.

It is found that the value of Nusselt number is higher in case of heat sink having 10 mm base thickness. Heat transfer is highest for 10 mm base thickness heat sink. So heat sink having 10 mm base thickness is most efficient for heat transfer.

CONCLUSIONS
Through CFD analysis, temperature distribution inside the heat sink has been found out. Through analysis it is found that as the Nusselt number of AA-1050 material heat sink is higher as compared to other aluminium alloys. The value of thermal resistance for AA-1050 material is lower than the other materials used for heat sink. Thermal conductivity of AA-1050 material is higher than the other materials so heat transfer is highest for it. It has been found that change in thermal resistance of heat sink with variation in velocity of air follows same nature of variation for all materials. As the velocity of air increases, the value of thermal resistance of heat sink decreases. Heat sink having 10 mm base thickness shows the maximum heat transfer as compared to other base thickness.

REFERENCES


