Thermal Analysis of Pin Fin with Different Shape Forms using ANSYS

R. Sudheer kumar Reddy¹, Dr .k. Govinda Rajulu², Dr. S.M. Jameel Basha³, E. Vijay Gowd⁴, P. Veera Prathap⁵, C.N. Vishnu Vandhan.A⁶
Assistant Professor, Mechanical Engineering Department, SRIT, Anantapuramu, A.P India¹
Professor and Principal of J.N.T.U.A, Pulivendula, Y.S.R Dist., A.P, India²
Professor and Head of Mechanical Engineering Department, SRIT, Anantapuramu, A.P India³
U.G Students, Mechanical Engineering Department, SRIT, Anantapuramu, A.P India³, ⁴, ⁵, ⁶

Abstract:
Pin fins are most commonly used in heat exchanging microelectronic devices such as computer CPU heat sinks, and semiconductor devices power transistors and optoelectronic devices such as lasers and light emitting diodes (LEDs). The objective of this study is to examine the effects of the fin shape of the heat sink on the thermal performance. The various types of Perforation on pin fin are used for effective heat transfer rate under constant heat flux condition. Cooling is done by forced convection utilizing fan. Air-based cooling technologies have been widely employed for thermal management of electronics. For low power CPUs, aluminium heat sinks are often capable of dissipating the heat. If better performance is required, copper heat sink may be used for higher heat sink performance, but aluminium is used because of its lower weight and lower cost than copper. This numerical simulation is accomplished by 3D modeling and analysis using CATIA and ANSYS, 12.0. This will help in finding out the new fin topologies with heat transfer characteristics that will do better than conventional plane fin. The main goal is to increase the heat transfer rate through the fin surface and to decrease material cost. The effect of different shape forms of pin fins on the overall performance of the heat sink studied in this paper.

Keywords: pin fin apparatus, perforated pin fin, fin efficiency, fin effectiveness.

1. INTRODUCTION
The design of heat sink device is predicated upon optimizing the opposite demands of thermal dissipation rate. Rapid development in manufacturing technology and consumer demands has driven the electronic technology towards increasing the functionality and compactness of the components. The miniaturization of electronics results in higher rate of power dissipation per unit volume. Thus effective thermal management becomes ever more critical to maintain the operation temperature which would ensure the efficiency and reliability of the electronic components [1]. Pin fins are widely used to enhance forced convection heat transfer across various industries, finding application in turbine blade trailing edges, electronics cooling, and broadly for compact heat exchange. Increasing miniaturization of high speed multifunctional electronics demands ever more stringent thermal management [2]. The present work investigates analyses the use of perforated pin fins to enhance the rate of heat transfer in these devices. In particular, the effects of the number of perforations and the diameter of perforation on each pin are studied.

2. LITERATURE SURVEY

Most studies that have demonstrated the beneficial effects of perforations on the rate of heat transfer in pin fin arrays have used only a single perforation per pin. The logical extremis of this parameter is the metallic foam-like porous pins. Indeed, numerical simulations, e.g. by Pradeep singh[3], stated that the rectangular shaped extended surfaces shows the high rate of heat transfer when compared to other extensions at same length. Kang Hiechan[4], has made many experiments to find the fin efficiency and concluded that the efficiency of fin is useful when the value of NTU is zero otherwise the fin efficiency is high when the NTU is high and is used in air conditioning systems. Shivdas S Kharche[5], explained that when a notch is provided on the surface of fin with a rectangular shape the fin supports for much heat transfer and compared the heat transfer rate of fins by changing the material from Aluminum to copper and found that copper shows much heat transfer value than aluminum. Sandhya Mirapalli[6], had made a conclusion that for a triangular fin when the length is increased the heatflow percentage also increases at constant base temperature compared to rectangular fin. I.Lakshmi Anusha[7], concluded that total weight of the system made of splayed pin fins can be reduced to the minimum level by using the advanced composite materials like polyphenylene sulphide (PPS), carbon foam, graphite epoxy at the same thermal inputs. Zhang, H.S[8], resulted that a fabric heat sink temperature distribution is so nearer to common pin fin heat sink but the temperature decreases in axial direction by increasing the pin fin length. Sampath S S[9], compared the temperature distribution of a cylindrical element at various points is carried out by providing the thermal conductivity and heat transfer coefficient and with prescribed boundary conditions and analyzed with the help of simulation software and DOT NET software[12] and the results are almost equal expect at the middle of the specimen it is just deviated. Amol B. Dhumne[10], resulted that to achieve high thermal
performance the cylindrical perforated pin fins are used they leads to high heat transfer than the cylindrical pin fins. M. P. Shah[11], compared the study of cylindrical pin fin was done by changing the material conductivity and heat transfer coefficient for copper ,AA1100,AA2011 materials[13] and concluded that the heat transfer rate increases when thermal conductivity of material increases.

3. ANSYS

ANSYS stands for Analysis System. ANSYS Mechanical is a finite element analysis tool for structural analysis, including linear, nonlinear and dynamic studies. This computer simulation product provides finite elements to model behaviour, and supports material models and equation solvers for a wide range of mechanical design problems.

3.1 USE OF ANSYS WORKBENCH

Anssys provide a cost effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping. With virtual prototyping techniques, users can iterate various scenarios to optimize the product long before the manufacturing is started. This enables reduction in the level of risk, and in the cost of ineffective designs.

4. THE GRAPHICAL USER INTERFACE COMPONENTS OF ANSYS WORKBENCH

4.1 TOOLBOX
The ansys workbench toolbox presents the types of data that you can add to your project. The toolbox in content sensitive, as you select different items in the project schematic or the other workspaces, the contents of the toolbox may change to reflect the components and actions available to you. When working in other workspaces, such as engineering data or parameters, you can return to the project workspace by clicking the return to project button on the toolbar.

4.2 PROJECT SCHEMATIC
The project schematic captures the project and workflow of your project, providing a visual representation of the objects in the project and their relationship to each other. Analyses are shown as systems, which are comprised of different individual cells.

4.3 TYPES OF CELLS
1. Engineering data
2. Geometry
3. Model/Mesh
4. Setup
5. Solution

6. Results

4.3.1 ENGINEERING DATA
Use the Engineering data cell with Mechanical systems or the Engineering data component system to define or access material models for use in an analysis. Double click the engineering data cell, or right mouse click and choose EDIT from the context menu to display the engineering data workspace to define material data. For more information see engineering data.

The required material for the analysis is selected from the options provided in the engineering data sources. There are different categories of materials available in the engineering data sources for this steady state thermal analysis we are going to consider the material as Aluminium. This material is available in the general materials.

4.3.2 REPLACE GEOMETRY:
Select browse to open a dialog box that allows you to navigate to an existing geometry file, or select a file from the list of recently viewed files to replace the currently specified file.

UPDATE FROM CAD:
Generates an existing CAD Geometry using the parameter values as defined in the cad system.

PROPERTIES:
Displays a properties pane where you can select basic and advanced geometry properties. For a detailed description of the options available from the properties pane, see geometry preferences in the Mechanical application help.

The design modeller will be generated in a new window where the geometric model can be imported according to our requirement. Click on the import option so that the design modeller imports the geometric created CAD model. The geometric model can be imported by right mouse click on the import and then click on generate. This will generate the created CAD model into the design modeller window.
The imported geometric model has to be converted or formed into a single solid component. Hence all the small parts of the generated CAD model are selected and converted into a solid part. To form a single solid part, right mouse click on the selected solids and click on form new part. The required CAD model can be generated as a single solid part. The created model can be used for further modelling or meshing based on the requirement for the steady state thermal analysis.

4.3.3 MODEL/MESH:
The Model cell in the Mechanical application analysis systems or the Mechanical model component system is associated with the Model branch in the Mechanical application and affects the definition of the geometry, coordinate systems, connections and mesh branches of the model definition. The Mesh cell in the Steady State Thermal analysis systems or the Mesh component system is used to create a mesh using the Meshing application. It can also be used to import an existing mesh file.

EDIT:
Launches the appropriate Model or Mesh application (the mechanical application, meshing etc).

Now the generated model has to be meshed in order to get appropriate results. The left side box outline is having options mainly geometry, mesh, steady state thermal etc. The material can be again selected at the geometry option. The aluminium material will be available at the geometry. The boundary conditions can be given according to the required analysis. Now right click on the geometry to get an option called part. Now click on the part so that the required material can be selected as shown in the bottom left side box. The properties of the material and the solid can be changed based on the results that are to be attained. The meshing can be generated by right mouse click on the mesh option in the left side toolbox. Click on the generate mesh option. The system takes a few seconds to generate the required mesh.

The meshing of the created CAD model can be made by giving the element size. There is a sizing option in the left side bottom toolbox. Click on the sizing option, the element size is to be defined. The smaller the element size the greater is the accuracy in the results.

4.3.4 SETUP:
Use the Setup cell to launch the appropriate application for that system. You will define your loads, boundary conditions, and otherwise configure your analysis in the application. The data from the application will then be incorporated in the project in ANSYS workbench, including connections between systems. For the mechanical application systems, you will see the following setup options, in addition to the common options.

EDIT:
Launches the mechanical application with the geometry loaded and with cells mapped to their respective tree locations in the mechanical applications. In this module the boundary conditions are to be defined. We are considering the steady state thermal analysis in which the temperature values are to be specified for the thermal analysis. There are two types of boundary conditions in this thermal analysis. First one is the temperature boundary condition and the second one is the convection boundary condition.
condition. Now right mouse click on the steady state thermal option which is available in the left side toolbox. Click on insert and then select the temperature.

Now the initial boundary condition is given as 105°C. In this thermal analysis, the initial condition is given at the heater i.e., the maximum temperature at which the conduction of the fin starts. The required temperature value can be given beside the magnitude option. After giving the temperature value, select all the faces of the solid body where the temperature is to be applied. Here the temperature is given to the source as an input i.e., the initial boundary condition. A graph is shown in the below graphic window which indicates the temperature variation along the fin.

The second boundary condition is the convection along the fin. Now in the graphic window, right mouse click on the steady state thermal and then click on insert, then click convection. Here the value of heat transfer coefficient is given according to the calculations.

The convection is applied only to the fin surface. Hence the value of heat transfer is to be calculated earlier based on the attained temperature values. Now give the value of heat transfer coefficient in the toolbox beside the film coefficient in the left side bottom box. Select all the surfaces on the fin and then click apply so that the convection is applied on the fin. The ambient temperature can be defined based on the requirement. Generally the ambient temperature refers to the temperature inside the duct where the air flow takes place. In this case the ambient temperature can be considered as the atmospheric temperature as 25°C.

The convection is applied on the surface of the fin. A tick mark is displayed on the left side of the convection option. Convection heat transfer coefficient depends on the surface area that is considered for the calculation. A graph is shown below the solid part which is straight along the length of the fin i.e., the convection takes place linearly along the length of the fin. The value of heat transfer coefficient changes based on the surface area considered and the initial boundary conditions of temperature.

4.3.5 SOLUTION:
From the solution cell, you can access the solution branch of your application, and you can share solution data with other downstream systems (for instance, you can specify the solution from one analysis as input conditions to another analysis). If you have an analysis running as a remote process, you will see the solution cell in a pending state until the remote process completes.

IMPORT SOLUTION:
Displays the open dialog, where you can specify the solver results file to load. When the results file is loaded, the system will display only the solution cell and the results cell.

The solution in the ansys workbench gives the temperature distribution along the considered surface or fin. To get the temperature distribution, right mouse click on the solution and click on insert and then select thermal. Then click on the temperature so that the temperature distribution can be measured along the length of the fin. The maximum temperature will be at the source and the temperature gradually decrease along the length of the fin.
After inserting temperature, right click on it and select evaluate all results. The temperature distribution is shown from the source to the tip of the fin. The minimum temperature at the end indicates the heat transfer rate that has been taken place. The temperature of the fin decreases if the heat transfer increases.

The temperature distribution can be determined by using ansys workbench. The maximum temperature will be at the source and the minimum temperature will be at the end of the fin. The temperature gradually decreases from one end to other end based on the given heat transfer coefficient.

4.3.6 RESULTS:
After the evaluation of temperature distribution along the length of the fin, a path can be defined to analyze the variation in the temperature from source to the free end of the fin. Hence to define the path, construction geometry is to be considered. Now right mouse click on the model and then click insert. The re select the construction geometry option.

Now in this graphic window, the required path can be defined based on the given boundary conditions. Click on construction geometry, then select the path option. Here we require a path from the source to the free end of the fin. After the selection of path, the graphic window shows options where the required path can be defined.

We need to define the path based on two points i.e., the source and the end of the fin. Now select the first point at the source and then click apply. Again select second point at the free end of the pin fin and then click apply. To generate the required path, we have to insert temperature in the solution option. After defining the path, insert temperature from the solution option available in the left side toolbox. Right mouse click on the temperature and then select evaluate all results. The required path will be generated as shown in the graphic window below.

Here the path can be generated based on the given boundary conditions. The maximum and minimum temperatures are defined based on the steady state thermal analysis. The variation of temperature can be determined from point 1 to point 2.

5. TEMPERATURE DISTRIBUTION FOR DIFFERENT MATERIALS

Figure.1. Temperature Distribution for Copper
Figure.2. Temperature Distribution for Aluminium
Figure.3. Temperature Distribution for Brass
From the above temperature distribution analysis of different materials copper have highest heat sink performance than the other materials, but aluminium heat sinks are often capable of dissipating the heat, because of its lower weight and lower cost than copper.

**ALUMINIUM:**
Aluminium is a chemical element in the group with symbol Al and its atomic number is 13. It is a silvery white, soft, nonmagnetic, ductile metal. Aluminium is the third most abundant element (after oxygen and silicon). The chief ore of Aluminium is Bauxite. Its low density, excellent corrosion resistance, good thermal and electrical conductivity etc are the other good properties. In the present work an Aluminium pin fin of diameter 12 mm and the length of 135 mm is used.

### 6. RESULTS AND DISCUSSIONS

#### 6.1 PURE SOLID (S1)

**Fig 6:** Cad model for pure solid

**Description of Solid:**
- Length = 135mm
- Diameter = 12mm

**Figure 4. Temperature Distribution for Stainless Steel**

**Figure 5. Thermal conductivity of different materials**

**Figure 6. Temperature Distribution for plane solid**

**Figure 7. Temperature Distribution for plane solid**

**Figure 8. Temperature varying along length of plane solid**

**Figure 9. Cad model for solid with perpendicular holes (P.H)**

**Figure 10. Temperature Distribution for solid with P.H**

**6.2 SOLID WITH PERPENDICULAR HOLES (S2)**

**Description of Solid:**
- Length = 135mm
- Diameter = 12mm
- Diameter for perpendicular holes = 3mm
- No. of Perpendicular Holes = 4
Figure 11. Temperature linear graph along the length of solid with P.H.

6.3 SOLID WITH PERPENDICULAR HOLES AND THROUGH WHOLE OF ONE BY THIRD LENGTH (S3)

Figure 12. Cad model for solid with P.H and axial hole of one by third length.

Description of Solid:
Length = 135mm
Diameter = 12mm
Diameter of the Perpendicular holes = 3mm
No. of Perpendicular holes = 4
Length of the Through Hole = 45mm
Diameter of the Through Hole = 6mm

Figure 13. Temperature Distribution for solid with P.H and axial hole of one by third length.

Figure 14. Temperature linear graph along the length of solid with P.H and axial hole of one by third length.

6.4 SOLID WITH PERPENDICULAR HOLES AND THROUGH WHOLE OF ONE BY TWO LENGTH (S4)

Figure 15. Cad model for solid with P.H and axial hole of one by two length.

Description of Solid:
Length = 135mm
Diameter = 12mm
Diameter of the Perpendicular Holes = 3mm
No. of perpendicular holes = 4
Length of the Through Hole = 67.5mm
Diameter of the Through Hole = 6mm

Figure 16. Temperature Distribution for solid with P.H and axial hole of one by two length.

Figure 17. Temperature linear graph along the length of solid with P.H and axial hole of one by two length.
Table 1: Heat transfer coefficient and Heat transfer rate for various pin fin configurations

<table>
<thead>
<tr>
<th>Profile</th>
<th>Heat Input (watts)</th>
<th>Velocity of air (m/s)</th>
<th>Heat transfer coefficient, $h$ (w/m²k)</th>
<th>Heat transfer rate (watts)</th>
<th>Maximum Temp. (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plane solid</td>
<td>80</td>
<td>0.914</td>
<td>28.32</td>
<td>8.32</td>
<td>105</td>
</tr>
<tr>
<td>Solid with P.H</td>
<td>80</td>
<td>0.914</td>
<td>28.328</td>
<td>8.40</td>
<td>103.8</td>
</tr>
<tr>
<td>Solid with P.H and $L_A = 80$</td>
<td>80</td>
<td>0.914</td>
<td>28.334</td>
<td>9.12</td>
<td>102.6</td>
</tr>
<tr>
<td>Solid with P.H and $L_A = 80$</td>
<td>80</td>
<td>0.914</td>
<td>28.428</td>
<td>9.89</td>
<td>101.2</td>
</tr>
</tbody>
</table>

7. CONCLUSION

From the investigation the following conclusion were made:
- It is found that the temperature drop along the perforated fins length is consistently higher than that for the non perforated fins.
- It is found that the heat transfer rate is more for different perforated fin compare to plane fin.
- It is also concluded that from various perforated fins Plane solid have minimum heat transfer rate whereas solid with perpendicular hole and axial hole of one by two length have highest heat transfer rate.
- It is found that the Nusselt number, heat transfer and heat transfer coefficient is maximum in case of solid with perpendicular hols and axial hole one by two length fin.
- Perforated fin leads to decreases the expenditure of the fin material.

7. REFERENCES


