Three-Dimensional Numerical Simulation of Heat Transfer for Different Shapes of Heat Sinks

Sinan A. Ali†, Reyadh Ch. Al-Zuhairy‡, Ammar I. Kubba††

†Control and Systems Engineering Department, University of Technology, Baghdad, Iraq.
‡Ministry of Higher Education and Scientific Research, Baghdad, Iraq.
††Prosthetics and Orthotics Engineering Department, Al-Nahrain University, Baghdad, Iraq.

*Corresponding Author E-mail: reyadhc@yahoo.com

ABSTRACT

In this research, a three-dimensional simulation of fluid flow and heat transfer using Computational Fluid Dynamics (CFD) for various types of heat sinks are presented. A new pins design has been proposed, which consists of a pyramid pin fins form was simulated by using (ANSYS Fluent software 17.2), the software was used to simulate the heat sink three-dimensional model with standard (k-ε) turbulence mathematical model with different factors. There were six unique fins pins models: square, circular, triangular, pyramid circular, pyramid square, and pyramid triangular pin fin heat sink various designs. The thermal performance of pin fin heat sinks designs was founded by several attempts of simulations with a heat flux of (23088 W/m2) and air travel velocities range that have a Reynolds number range between 8515 to 22312. These conditions were applied to the heat sink model to calculate the heat dissipation from the pin fin heat sinks designs. Numerical predictions of temperature and velocity vector in the heat sinks are found from the model solution results. The founded results also indicate that the performance of the prism pin fin heat sink (PPF) is to be the most effective design that dissipating heat from the other fins designs; this performance of the prism pins can be due to the largest surface area compared to the other designs.

KEYWORDS

Computational Fluid Dynamic, Pin fin heat sink, thermal resistance, pyramid pin fin.

INTRODUCTION

Since the progress of computers designs during the 1940s, the need for effective heat removal system was a key factor in developing reliable new generations of computer designs [1]. Air-cooled heat sink system is the common type of cooling system because this system has many advantages, such its fabrication is much easier and costs lower than other systems, it also needs less maintenance. Many researchers have considered utilizing heat sinks in computer systems to dissipate thermal energy [2]. A heat sink is an object designed to absorb heat and dissipate heat from another system with thermal contact (either direct or radiant). The conventional heat sink fluid medium is air, but other fluid mediums can be also used as water, oil, or refrigerants. Heat sink applications are very wide, such as microprocessors cooling in electronics devices, refrigeration, and heat engine systems. In common, its metal object attached with a hot electronic component’s surface. It also has thin material between the two surfaces that improve heat transfer between them. Semiconductors and microprocessors are examples of electronics systems which need a heat sink system attached to them to reduce their temperature by increasing their thermal mass and heat dissipations (by conduction, convection and radiation). Wide range of heat sink design geometries are in today. The fins Pin’s (spines) are uniform straight fins, uniform tapered straight fins, and uniform annular or splines fins are possible. The common designs are pin fins, which has a cross-section that can be square, round, hexagonal, elliptical, or any suitable shape. These fins designs’ by engineers who were trying to obtain its design with the minimum thermal resistance in addition to the lowest pressure drop [2].

There are extensive studies that investigate heat sinks fluid flow and heat transfer characteristics. Yoav Peles, et al. 2005 [3] investigated micro pin fins heat transfer besides the pressure drops over the micro pin system. The simplified expression of the total thermal resistance expression has been introduced and experimentally
confirmed. Many studies were utilized on the designing shapes and the thermo-hydraulic parameters of the pin fin heat sink, which observed the major effect on the total thermal resistance. The findings of these studies shows that the pin fin design has very low thermal resistances in heat sink design. For an optimum design for the heat sink. An early investigation on the fluid flow and heat transfer properties of a typical pin fin was done. Then more investigation was utilized to hybrid design pin fin and splayed design pin fin for a heat sink system have been studied throughout CFD modeling software and simulations by Md. Abdul Raheem Junaidi, et al. 2014 [4]. It was found from the collected data that the optimal heat dissipation was carried off by hybrid and splayed pin fin design for heat sinks. The hybrid designs of the heat sink can promise to reduce the electronic circuit systems temperature by 20 to 40% compared to an ordinary pin-fin designs.

Mohamed H.A. Elnaggar 2015[5] analytical investigation found that the number of the fins and its thickness would affect the heat sink performance. The collected data shows that increasing both the number and the thickness of the fins will increase the rate of heat transfer from the heat sink, but any significant increase in the numbers of the fins, will has larger effect on the rate of the heat transfer compared to the growth in the fins thickness does. Some growth in the fins thickness will increase the heat transfer rate, but any extra increase in the fins thickness will result in a reduction in the space among the fins. Also, various arrangements of fins geometry have been studied by R. Rosli, et al. 2015 [6]. Using COMSOL Multiphysics software to mimic the range of pin fins arrangement and design of several models of heat sinks. The collected data have found that new pin-fin arrangement geometry is able to offer better thermal performances in which 4.1% and 0.5% in circular type fins and 0.2% and 0.4% in square fins type, theses data were compared to inline as well as staggered arrangement. The advantage of using heat sinks with multiple perforations pin fin designs were studied experimentally as well as computationally on the pressure drops and heat transfer in pin fin heat sinks by Amer. Al-Damook, et al. 2015 [7]. The perforated pin fin design effects were studied on the heat transfer in addition to the pressure drops through the heat sinks.

The experimental and numerical results found that the Nusselt number increases significantly with the perforated pin number, while the drop in the pressure in addition to the fan required power to overcoming the pressure drop all decrease significantly. Mohammad Saraireh 2016 [8] presented three-dimensional (CFD) simulation of heat transfer and fluid flow in two types of heat sinks. Thermal and hydraulic performances of plate fin and circular pin fin heat sinks are obtained. Numerical predictions of pressure drop and thermal resistance in the heat sinks were compared with experimental data from the literature and an excellent agreement is found. The results also show that the circular pin fin heat sink has enhanced performance compared to the plate fin heat sink. Chao, et al. 2017 [9] investigated by numerical solution the hydraulic in addition to the thermal features of forced convection in turbulent airflow conditions through a perforated circular PIFHSs with constant heat flux. A circular perforated pin fin was numerical modeled in a heat sink system with several variable parameters including perforation diameter, perforation space, circular pin fin diameter in addition to a range of Reynolds number values. The perforated pin fin model has larger averaged Nusselt number compared to the solid pin fin, besides that when increasing the diameter of the pin fin, both averaged Nusselt number and the friction factor increased. The increasing the averaged Nusselt number trend is more distinguished than the increase of the friction factor, which leads to a raise in the amount of heat transfer with the raise of the pin fin diameter.

Alhassan Salami Tijani and Nursyameera Binti Jaffri 2018[10] investigated the heat sink thermal performance effect on perforated pin fins with forced convection conditions. The results of this study gave an important chance to improve the understanding of the effect of a wide range of parameters such as temperature distribution, pressure drop, and the level of perforation on the heat sink thermal performance. A three-dimensional model was simulated in rectangle shape channels. The collected result demonstrations that heat sink design with perforated pin fins and flat plate improves the thermal efficiency 1-4% in contrast with both solid flat plate and solid pin fins heat sink design. S. Bhattacharyya, et al. 2020 [11] investigate the effect of air cooling strategies on the heat transfer in a range of heat sink designs attached to a circular cylinder CPU chip, this investigation also searched the design of a micro pin fin and the thermal fluid behavior of the heat sink. This study shows that the Nusselt number (Nu) increases with the increasing of the airflow speed, which improves the heat dissipation from the CPU. The heat sinks have been studied with nanofluids and several other modifications.
The results showed that the relationship between the rate of free heat transfer and the vibration amplitude is proportional with the tilt angles. The effect of the concentration of CuO nanofluid has been discussed with Reynolds number value and with the velocity of fluid [12-13]. The heat transfer was studied both experimentally and numerically in different applications. It was found that the Darcy number, Rayleigh number, aspect ratio, and porosity considerably influenced characteristics of flow and heat transfer mechanisms [14-19]. In this research, a numerical model investigation on the characteristics of the thermal performance in pin fin heat sink (PFHS) using a range of pin shapes. The effect of the fins configurations on both the fluid flow and the heat transfer characteristics is studied. Three-dimensional numerical simulation with ANSYS FLUENT (17.2) software. The numerical simulation investigated a range of parameters for six different types of pin shapes using the same hydraulic diameter: square, circular, triangular, pyramid square, pyramid circular and pyramid triangular, these pins shapes were for the fins in the heat sink system. From the results of this research, the heat sinks thermal performance can be improved for various configurations.

NUMERICAL METHODOLOGY

The purpose of this research is to present a pin–fin model for a heat sink (PFHS) by using forced convection with different shapes of the pin fin heat sink system. The 3D models of the heat sinks have been generated in Solid works software and imported into ANSYS FLUENT 17.2 workbench. In the CFD simulation, there were three main principle phases: Pre-Processing, then Solver Execution, and in the end Post-Processing. Pre-Processing is the phase where the settled objective is set and the computational element is made by the meshing process. Fins were considered as aluminum that has a specific value of heat-flux input value at the base side surface for the array system, using fluid flow properties and the geometry of the pin, surrounding boundary conditions in addition to the numerical analysis models are settled to start the solving process at the second step in the solution. The solver model continues to run until finding the meeting point for the solution. The data will be sent to the post-processing phase for viewing.

Physical model

In this present study, the material used for the (PFHS) was aluminum for all the different geometries. The different heat sink configurations present the pin fins were placed on the base plate having dimensions W × L (60*60 mm). These geometries are located inside a channel, in manner that the zones before and after the studied arrangement (PFHS) allow to develop flows and avoid the reversed flow phenomenon, respectively. Several arrangements have been studied in this paper: circular, square, triangle, and pyramid shapes. The pyramid form is characterized by lower surfaces of the pyramid as circular, square, triangle. The section of circular, square, triangle pin fins is characterized by the diameter (D), and the height of pin fins (H) as shown in Figures (1) and (2). The pitch length ratio of pin fin (SL/H) which can be defined as the ratio of pitch length of pin fins (distance between center to center the pin fins, or SL) to the height of channel (H) and it is fixed to 0.5. Where the pitch transverse ratio of pin-fin (SW/H) which can be defined as the ratio of pitch transverse of pin fins (SW) to the height of channel (H) and it is set to 0.5 to the height of channel (H) is equal to the height of pin fins (HP). The pin fins for all configurations are distributed over the base plate in staggered arrangements.

Table 1. Geometrical parameter of heat sink

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Heat sink Dimensions (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fins Height H</td>
<td>25</td>
</tr>
<tr>
<td>Fins Dimeter D</td>
<td>5</td>
</tr>
<tr>
<td>Base Height Hb</td>
<td>7</td>
</tr>
<tr>
<td>Heat Sink Length L</td>
<td>80</td>
</tr>
<tr>
<td>Heat Sink Width W</td>
<td>65</td>
</tr>
<tr>
<td>Pin space distance in streamwise direction S_L</td>
<td>7</td>
</tr>
<tr>
<td>Pin space distance in spanwise direction S_T</td>
<td>7</td>
</tr>
<tr>
<td>Fin Number N</td>
<td>54</td>
</tr>
</tbody>
</table>
Three Dimensional Numerical Simulation of Heat Transfer for Different Shapes of Heat Sinks

Figure 1. The pin-fín model heat sinks investigated in this research.
Figure 2. The pyramid pin-fin model heat sinks investigated in this research.

Mesh generation

The distribution of mesh and the duct computational domain is shown in figure (3) in addition to the PPFHS is shown in figure (4). For the duct computational domain, the inlet and the outlet (pressure and velocity) boundary conditions as well the wall and symmetry boundary conditions were set. Figures (3) and (4) demonstrate the computational grid in x, y, z-direction for the geometrical pin-fin model for the heat sink researched in this article with a straight channel. The discretized 3-D model of pin-fin heat sinks were created in ANSYS FLUENT 17.2 in the first phase of the solution (pre-processor). The selected element to model and simulate the pin-fin heat sink geometry is a hexahedral element that creates an accurate thermal model for the heat sink system. The model mesh for a selected heat sink design is shown in figures (3) and (4). The amount of the model element can affect the resulted data of the simulation. For that reason, a mesh independence check was completed for a range of grid models as listed in table (2). From the collected results in that table, the data accuracy of the simulated model does not rise sharply with the growth of the grid elements number. Accordingly, the selected mesh for all the case studies in this research was the lower number of the grid element which can cause a significant reduction in the computation time.

Table 2. The effect of the amount of grid elements on the collected data.

<table>
<thead>
<tr>
<th>Pin fin type</th>
<th>Number of grid elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circular Pin – Fin</td>
<td>2824620</td>
</tr>
<tr>
<td>Square Pin – Fin</td>
<td>2851983</td>
</tr>
<tr>
<td>Triangle Pin – Fin</td>
<td>2807461</td>
</tr>
<tr>
<td>Pyramid Circular Pin - Fin</td>
<td>3087223</td>
</tr>
<tr>
<td>Pyramid Square Pin – Fin</td>
<td>3201978</td>
</tr>
<tr>
<td>Pyramid Triangle Pin – Fin</td>
<td>2985046</td>
</tr>
</tbody>
</table>

(a) Circular Pin – Fin

(b) Square Pin – Fin
Boundary conditions

A uniform temperature and velocity have been used at the domain inlet area with a symmetrical boundary condition was employed to minimize the solution time of convergence. In the outlet, the atmospheric pressure is considered; the no-slip boundary condition is imposed on the solid surfaces. Air at standard conditions is considered as the working fluid. The aluminum type and 202 W/m.K of thermal conductivity are selected as a material for the heat sink. The thermo physical air properties and aluminum were assumed constant. The Boundary conditions are applied to the geometric entities for forced convection cases. A heat flux of 23088 W/m² i.e. 50 W was applied for the base area of the present heat sink. The pressure outlet was set as the right face as presented in figure (5), the remaining faces will act as walls for the fluent model simulation. The left face of the fluid domain as figure (5) shows is set to be velocity inlet with as 2 m/s.

At the inlet, boundary conditions are:
At the outlet, boundary conditions are:
\[ P = P_{\text{out}}, \quad u = v = 0, \quad \frac{\partial T}{\partial x} = 0 \]

At the walls, boundary conditions are:
\[ u = v = w = 0, \quad \frac{\partial T}{\partial x} = \frac{\partial T}{\partial y} = 0 \]

Heat flux surface, boundary conditions are:
\[ u = v = w = 0, \quad \dot{q} = -k_s \frac{\partial T}{\partial y} \]

Governing equations

Continuity, momentum and energy equations are used to govern turbulent regime, incompressible flow and heat transfer through PFHS. The continuity, energy, and momentum equations are run the fluid flow forced convective. In addition, the fluid flow conditions were considered as steady-state, turbulent, and incompressible air properties. As well, the air thermo-physical properties are set to remain constant. The bouncy impacts and radiation heat transfer are irrelevant. A stated by the previous assumptions and using the standard \( k - \varepsilon \) model the 3D Navier-Stokes governing equations will be [2]:

Continuity equation is:
\[ \nabla \cdot (\rho u) = 0 \]  
(1)

For the fluid region the momentum equation is shown as:
\[ (\nabla \cdot u) \rho u = -\nabla P + \mu (\nabla^2 u) \]  
(2)

The energy equation is expressed as:
\[ \rho c_p u (\nabla T) = \nabla \cdot (k \nabla T) \]  
(3)

The Governing equation about heat sink system is shown as:
\[ \nabla^2 T = 0 \]  
(4)

The \( k - \varepsilon \) standard is turbulence model used in this paper. This model is largely used in the literature to predict the turbulent flows in channel fitted obstacles as the geometries analyzed in the present study.

Figure 5. Proposed geometry and flow direction
The continuity equation, momentum equation, and energy equation are solved using the finite volume method with the continuum approach to solving the integral form of the governing equations. As follows, the precise equations that define pressure, velocities, and temperature distribution are accomplished. The convergence factors for mass, energy imbalance, and momentum were fixed to be below than $10^{-6}$.

THERMAL METHOD

The thermal resistance ($R_{th}$) is calculated as follow:

$$ R_{th} = \frac{T_w - T_{ai}}{\dot{q}} $$

Where $T_w$ is the average wall temperature, $T_{ai}$ is the air inlet temperature, $\dot{q}$ is the heat flux applied to the base plate. The Reynolds number ($Re$) is expressed as follow:

$$ Re = \frac{\rho_a u_a D_h}{\mu_a} $$

Where ($\mu_a$) is the air dynamic viscosity.

The hydraulic diameter ($D_h$) of the rectangular duct is determined as:

$$ D_h = \frac{4A_c}{P} $$

Where: $A_c$ is cross section of duct and $P$ is perimeter of rectangular duct.

The total heat energy supplied to the heat sink base ($Q_h$) was calculated from the value of voltage and current supplied by:

$$ Q_{heater} = I \times V $$

It assumed that the amount of heat transfer rate from the heating element to the cooling medium corresponds to the supplied electrical power. Hence the convection heat transfer from the heat sink projected area ($Q$) which can be demonstrated as:

$$ Q_{convection} = Q_{heater} = h A_p (T_s - T_f) $$

In which ($A_p$) is the projected area for the heat sink and its value can be stated as:

$$ A_p = LW $$

In which

($L$) and ($W$) are the length and the width of the heat sink base side.

($T_s$) is the average temperature of the plate base surface as measured by a thermocouple.

The average value of Nusselt number was founded by using the coefficient of the convective heat transfer ($h$) as following:

Nusselt number is determined from:

$$ Nu = \frac{h D_h}{k_a} $$

where ($k_a$) is thermal conductivity of air.

RESULTS AND DISCUSSION

Using the commercial FLUENT 17.2 software the Standard $k-\varepsilon$ model with turbulence fluid flow abilities, in addition to heat transfer analysis for the pin fin heat sink (PFHS) have studied a range of configuration designs. Heat transfer simulation analysis involves convection and conduction heat transfer forms. To find the PFHS thermal performance, several simulations were achieved with a heat flux of (23088W/m2) and range of air
velocities with (range of Reynolds number between 8515- 22312) to calculate the heat dissipation for different pin fin heat sinks designs. Figure (6) shows the contour of the temperature of the cylindrical, square and triangles pin fin heat sinks at the heat flux of 23088 W/m² with air traveling velocity of 2 m/s in a wind tunnel. Unusually, it is shown that increasing heat transfer surface increases heat transfer execution, but not always if we considered others physical phenomenon such as recirculation zones and velocity fluctuations near the walls. The highest temperature of the various configurations is observed at the base of the hotplates, as well as in the corners formed by the base and the pins due to the formation of the recirculation zones at low speeds. The cylindrical square and triangles pin fin heat sinks are largely used in literature, but its real exploitation results several problems such as the formation of lower heat transfer areas, which is lead to reduce the heat transfer performances of PFHS. In this paper, we focused on the pyramid pin fin heat sinks at different configurations of pyramid which is presents circular, rectangular form (i.e. the lower and upper surfaces of the pyramid are identical) and triangle.

Indeed, the temperature is high on the base of the pins and it begins to decrease upwards as shown in figure (7). For this reason, the pyramid shape has been proposed to minimize the heat exchange surface in the upper parts of the pins and we are increasing this area near the source of heat flux. Also, it is very clear that the outlet air temperatures are higher than the inlet. Because, through the heat sink, the air captures the heat through the base and the pins walls and comes out hot. For the purpose of explain the outcome of the new design of this research, figures (8,9, and 10) shows the temperature contour in pin fin heat sinks with several configurations with the heat flux condition of 23088 W/m² with air traveling velocity of 2m/s in a wind tunnel. Figures (11, 12 and 13) show the temperature contour and vector of the circular, square and triangle pin fin heat sink with staggered arrangements. As a new design, the pyramid pin fins present a better configuration which merits to be considered in the design of PFHSs shown in figures (14, 15, and 16). Compared with cylindrical, square, and triangle pin fins heat sinks, the configurations of pyramid pin fins create larger reattachment zones which are present a suitable advantage for better execution of heat transfer. Thus, the pyramid pin fin ensures the smallest stagnation area, which merits to be considered in the PFHSs. The pyramid shape has been proposed to minimize the heat exchange surface in the upper parts of the pins and we are increasing this area near the source of heat flux. Also, it is very clear that the outlet air temperatures are higher than the inlet. Because, through the heat sink, the air captures the heat through the base and the pins walls and comes out hot. The obtained results demonstrate that the insertion of the pyramid pin fin shape on the baseplate of heat sinks will enhance both thermal and hydrodynamic aspects inside the heat sink by decreasing the pressure drop around the pins and decreasing the thermal resistance of the heat sink. Also it found that the pyramid pin fin heat sink (PPF) have more dissipating heat efficiency against the other fins configurations; this can be detected mainly due to its larger surface area.

![Figure 6. The contour of temperature for a-circular shape, square shape, and triangular shape PFHS.](image-url)
Three Dimensional Numerical Simulation of Heat Transfer for Different Shapes of Heat Sinks

Figure 7. The contour of temperature for pyramid circular shape, pyramid square shape, and pyramid triangular shape PFHS.

Figure 8. A temperature contour comparison for circular PFHS at different planes.
Figure 9. Temperature contour comparison for square PFHS at different planes.

Figure 10. Temperature contour comparison for triangle PFHS at different planes.
Figure 11. The temperature contour and vector for the staggered arrangements of circular pin fin heat sink.
Figure 12. The temperature contour and vector for the staggered arrangements of square pin fin heat sink.
Figure 13. The temperature contour and vector for the staggered arrangements of triangle pin fin heat sink.
Figure 14. Temperature contour comparison for pyramid circular PFHS at different planes.
Figure 15. Temperature contour comparison for pyramid square PFHS at different planes.
CONCLUSION

In this study, the PFHS computational fluid dynamics analysis was utilized by Ansys workbench Fluent 17.2 software. Different types of designs for the fin geometries using aluminum materials as PFHS were simulated with forced convection boundary conditions. Range of fins arrangements was simulated in this study for example; staggered arrangements with various pin-fins shapes. Fin geometries such as cylindrical squares and triangles pin fin heat sink at a heat flux of 23088 W/m² and an air traveling velocity of 2 m/s in the wind tunnel. Also, pyramid pin fin heat sinks at different configurations of the pyramid which is presents circular, square form (i.e. the lower surface of the pyramid are square) and triangle at the heat flux of 23088 W/m² and an air velocity of 2 m/s in a wind tunnel. The heat sink that has prism pin fin (PPF) was found to have more dissipating heat in compared to other fins designs; this was detected due to the larger heat sink total surface area.

REFERENCES


