Spectrum Journal of Innovation, Reforms and Development

Volume 36, February- 2025 | **ISSN (E): 2751-1731**

WEBSITE: WWW.SJIRD.JOURNALSPARK.ORG

ENHANCEMENT HEAT TRANSFER RATE BY USING NEW CONE SHAPE FINS WITH ELLIPSE BASE

Nagham Yass Khudair
University of Babylon / College of Materials Engineering/
Metallurgy Department. Babylon, Iraq.
naghamalbakry@gmail.com

Abstract

The application of using fins to lose excess heat is affected by many factors such as fin surface area, weight of fins, heat transfer coefficient, fins' material, orientation of fin surface of fins, Reynolds value due to the mutually effect between the inertia and viscous force for fluid flow. In this manuscript, Numerical simulation and experimental work was done to investigate the improvement of heat convection by using cone shape fin with ellipse base at low Reynolds number and arrangement as staggered. Where, the fluent Ansys 2020 R1 program is presented using modified K-□ (turbulent model). Moreover, the effect of curvature of the fin surface on the amount of heat driven was studied. The thermal dissipation of each type of fins are verification with the values of (h) heat transfer coefficient that obtained from relations which derived from conservation of energy. The various results of h at different value of fin distribution at air velocity (1 m/s and 2.5 m/s) are obtained. The results show that, the heat transfer coefficient increase as the fine surface increase. Also, there are a good relation between the surface shape and the amount of heat dissipation. The result compared with experimental values and gives a good agreement.

Keywords: New shape fin, Ansys 2020R1, heat transfer rate, K-□ turbulent model, forced convection.

Introduction

Nomenclatures

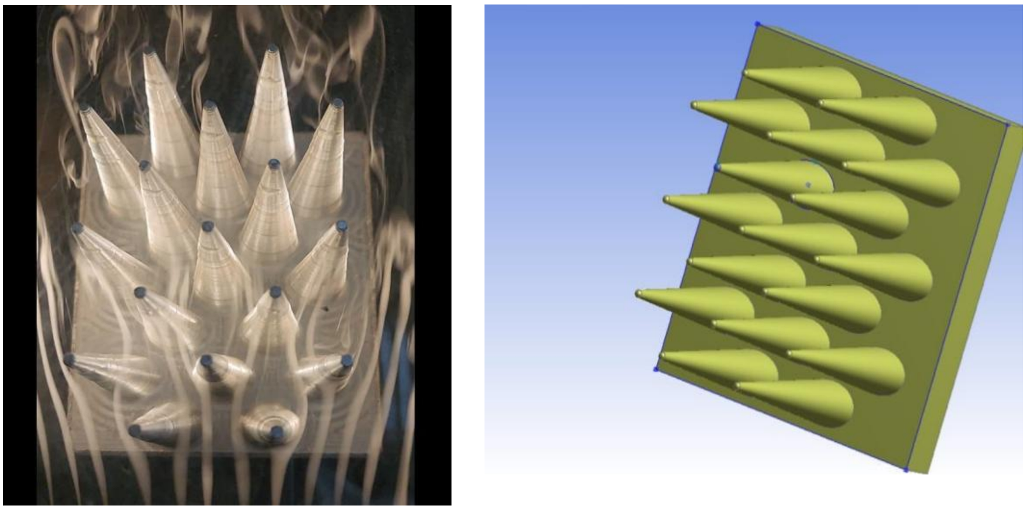
A area, [m^2]	*k* thermal conductivity, [W/(m·K)]
Cp coefficient of friction, [-]	*l*, L length, [mm] or [m]
cp specific heat, [J/(kg·K)]	Nu average Nusselt number, [-]
A area, [m2]	*p* pressure, [Pa] or [N/m2]
cp specific heat, [J/(kg·K)]	*Po* atmospheric pressure, [Pa]
Cf coefficient of pressure, [-]	Pr Prandtl number, [-]
C1 arbitrary constant, [-]	*Qs* heat supplied, [W]
D diameter, [mm] or [m]	Re Reynolds number, [-]
H height, [mm] or [m]	
h heat transfer coefficient, [W/(m^2·K)]	
T: temperature, [K]	

In many heat transfer applications, the forced convection from extended surfaces which called fins has not been studied well; despite its great practical importance. In most engineering applications like cooling of electronic components the fluid flow velocity is very low with high viscosity low space between fins in which the thermal control is very important [1]; this situation the viscosity is dominated. Therefore, this flow type will produce low Reynolds number. On the other hand, the locations of fins in flow region may produce turbulent flow by the drag force due to flow obstruction by fins. Many parameters that may increases of contribution to increases the intensity of turbulent in this case like the effect of angle of flow incidence, obstruction body shape and its distribution behind the flow source. The references [2-4] presented experimentally and numerically investigation of flow around circular and rectangular cylinder in several flow media (air and water) that produce turbulent flow at low Reynolds number. There are many works reveals that this flow formation in vicinity of fin will enhance the heat rate. Also, there are other parameters depends on the applications of fin can be used for improved the performance of fins like geometry of fins, weight, size. Where, Saroj Yadav et al [5] conducted the numerically and experimentally investigation for triangular with aluminum fins as heat sink. For numerical solution they used the finite element method (FEM) to simulate velocity streamline, direction as well as the fins' surface temperature [K]. However, they studied the effect of many parameters, number of fins, its arrangement's and features of fins length with different orientation angles. The shape of fin has a wide effect on the fin performance as presented by numerous previous researchers. Where, Nabil et al [6] presented innovate shape fin induced from egg shape to increase the heat dissipation. Also, they used the FEM to improve that the fin shape alone may be improved the fin performance. Moreover, Khan et al. [7] presented Y-shaped fin for enhancing the heat transfer rate along the bottom surface of square cavity. While, Yatendra and Sahu [8] conducted a vertical pin fins array that used as heat sinks to increase its thermal performance. Hamid and Mohammed [9] presented the numerically analysis of the elliptical pin fin heat sinks. The region of flow were laminar that around and above the elliptical foam fin. They found that the fin matel gives pressure drop but increases with heat transfer dissipation. Also, Navid et al [10] presented a numerically solution using FEM to investigated the multiple shapes effect of fin on enhancement of its thermal performance. However, they investigated numerically the thermal features of rectangular, convex, parabolic, exponential, sinusoidal, and cosine shape fin. Many researchers presented new models for modification to enhance heat rate like added perforated fins, design new shapes…etc. Consequently, Vidyadhar Karlapalem Sukanta Kumar Dash [11] designed new perforated fin laminar flow and found the maiximum heat dissipate happened in narrowest space. While, Rosnadiah et al [12] designed Hybrid pin fin with connector Heat Sink and discovered that the arrangement of pin fins has a huge effect of heat flow rate. Ambarish et al [13] increased the heat dissipation rate by conducted circular perforated fins. On the other hand, pressure drop will be lower with perforated fin compare with flat one depends on the number and size of holes.

In the present work, new cone fin shape is experimentally and numerically thermally investigated. The vertical wind tunnel is used for experimental work as that shall elaborate in subsequent articles. The experimental is very important for distinguish the flow behavior in front and among the new shape fin that arrangement with staggered way. On the other hand, the Ansys 2020R1 program is used for numerical solution at low Reynolds number for simulation and validation of the fin arrangement effect on its thermal performance.

2. The Numerical and Experimental Model

The base of the fin is ellipse, while the top of fin is half of sphere as shown in figure (1). This fin used as heat sinks also used to induce turbulent flow by new shape that will enhancement transfer rate. All the thermal fins are made of aluminum placed on a base with dimensions (110) mm Length, (100) mm width and the arrangement of fin is staggered. The staggered arrangement is the best one for enhancement the heat dissipation. The height of the hot cone fins are (68) mm.



a) Experimental model of fins b) Numerical model of fins

Figure (1): Fins under investigation a) experimental model of fins and b) Numerical model of fins.

3. Governing Equations and Boundary Conditions

The Governing Equations of Continuity, Momentum, And Heat Transfer Rate That used for numerical simulation of the heat transfer rate and turbulent induced by new fin shape have been presented in this part. Also, the temperature distribution which induced by air flow over or in vinicity region of each hot fin and the its base for thermal featuers will be presented as follow [15],[16] and [14]:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \qquad \ldots (1)$$

$$\rho(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \qquad \ldots (2)$$

$$\rho(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) + \rho g\beta(T - T_c) \qquad \ldots (3)$$

$$\rho c_p(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}) = \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \qquad \ldots (4)$$

The velocity analysis parts in the x is u while in y-direction have expressed by v. However, T refers to the temperature of the fluid very near the hot cone fin surface. The main numbers and flow properties will be presented as following:

$$Re = \frac{\rho V D_h}{\mu} \qquad \ldots (5)$$

Where D_h is haydrolic diameter and μ is air viscosity,.

The Nusselt Number can be given by

$$Nu = \frac{h\, D_h}{K} \qquad \ldots (6)$$

And the heat trasfer coefficient can be given by

$$h = \frac{1}{A_s}\int_{A_s}\frac{Q}{T_w - T_a}\,dA \qquad \ldots(7)$$

The symbol T_a represent the air flow temperature in vinicity region to the fin that have cone shape and A_s is new shape fin outer surfces area and T_w is the fin surface wall temperature. Moreover, the k – ε model was choosn with in ANSYS 2020 R1 software and to apply in this case with the following assumptions:

1. The volume of new size of fins in numerical and experimental is identical
2. No slip of boundary layer around curved triangle.
3. Steady flow rate and effect dominate between viscouse force and heat dissipation that comes from temperature distribution within fluid and around the hot cone fin.
4. The fins made from aliminum and have ellipse base shape with a small hemisphere at the top for each one.

the *k*–ε turbulence model can be given by [17]

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial xi}(\rho k ui) = \frac{\partial}{\partial xj}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial xj}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_K \qquad \ldots(8)$$

And

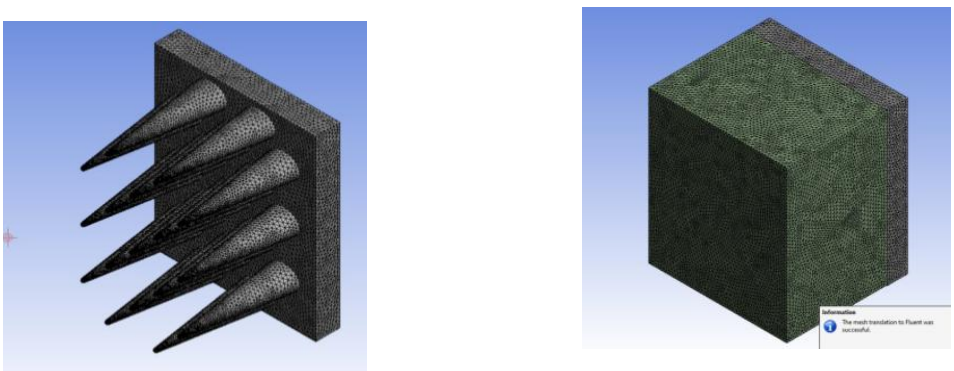
$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial xi}(\rho\varepsilon ui) = \frac{\partial}{\partial xj}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial xj}\right] + G_{1\varepsilon\frac{\varepsilon}{k}}(G_k + C_{3\varepsilon}G_b) - C_2\rho\frac{\varepsilon^2}{k} + S_\varepsilon \qquad (9)$$

4. Mesh Generation and Validation

Numerically analyses using the finite volume method that base in Fluent ANSYS 2020 R1. Figure (2) conducted the mesh generation in the confined fluid among the cone fin shape in two fluid flow velocities cases in this work. The element that used is tetrahedron with no. of element 997236. The power supply is constant 50w/m^2.

Table 1: The number of element with h (heat transfer coefficient) for cone fin with.

Grid types	coarse	fine	finer
Number of elements	243132	423365	997236
h (heat transfer coefficient)	57.72367	58.896768	58.98652



a) Fins b) air flow over fins(control volume)

Figure 2. (a, b) Mesh generation for all cases.

5. Experimental Device

Experiments are presented to show the flow performance of a set of aluminum fin that arranged in a staggered way.
Figure (1a). The figure 3, shows the experimental set-up used in the current work smoke wind tunnel.
The set-up consists of a square test section as shown with air flow by located at apparatus with outlet temperature Sensor.
Air enters the tunnel from the bottom and leaves it from up way.

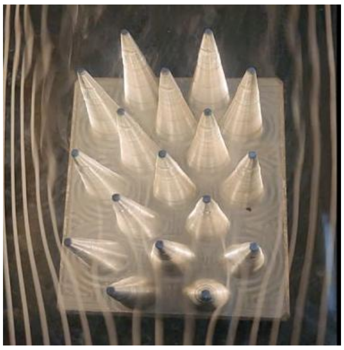
Figure (3) smoke wind tunnel.

The base plate is joined with the heater unit to apply constant heat power behind the base plate (50 W/m^2). The main purpose of using this apparatus is to investigate the flow visualization over and around the cone fin that may induced turbulent flow by its shape and this will increase the thermal performance of fin arrangement.

Results and Discussions

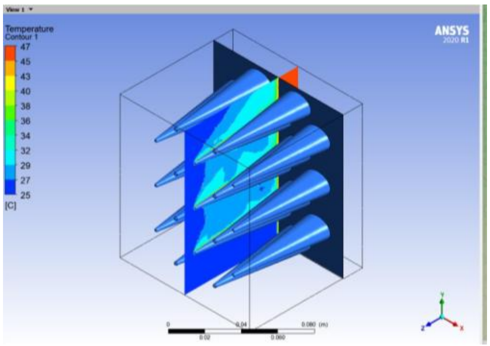
The fin design is very important parameter can be used to increase its thermal performance. The novelty of this work is presented new shape fin with ellipse base and cone shape. The main purpose of this shape is to induce turbulent flow between the fin system and behind the each one fin. Where, this flow type will increase heat dissipation from cone fins. The figure (4) represents the experimental flow visualization over cone fin with ellipse base at different air flow velocity. Where, part a) at air flow velocity 1m/s while part b) at air flow velocity 2.5 m/s. As shown this figure the turbulent concentrated behind the fins and grow up with vertical distance due to the narrow distance between adjacent fins. The ellipse base of each fin will increase the intensity of turbulence addition to the effect of staggered arrangement. The figure (5) represents the contour of temperature distribution on the vertical distance. These figure show the thermal boundary rapidly growth by the large and sudden spread of heat drawn from the fins. The ellipse base effect can be shown significantly in figure (6). The figure (6) shows that air velocity near to the base region around the ellipse base that attach base plate have the maximum velocity that will take the red color. The increase in velocity at base will produce the difference in pressure distribution that induced disturbance in flow and calling more heat energy from the outer surface of new shape fin with plat at bottom. This effect is very clear in figure (5). The overall effect of increasing of air flow velocity can be show in figure (7) which represents the numerical temperature distribution of air flow due to hot cone fin with ellipse bas at different air flow velocity. The comparison of the results of heat transfer coefficient of new fin with other researchers with identical boundary conditions of air flow and heat supply can be seen in figure(8). Also, the weight of the new fin with cylindrical type that used in reference [18] is the same by equating their volumes and materials type. This figure show the modification that happened by new fin design. So, this new shape of fin can be used for more thermal performance compare with cylindrical classical one

a) air velocity 1m/s

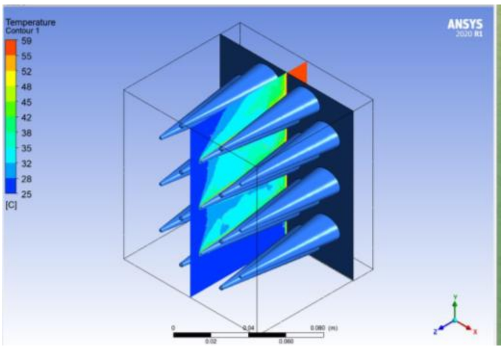


b) air velocity 2.5 m/s

Figure (4): Experimental flow visualization over cone fin with ellipse base at different air flow velocity.

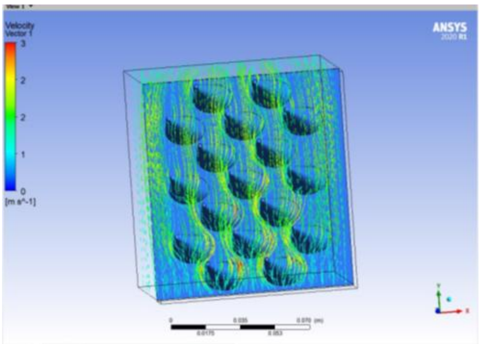


a) air velocity 1m/s

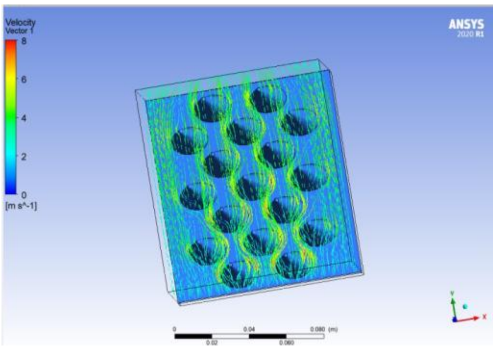


b) air velocity 2.5 m/s

Figure (5): numerical results for temperature contour in the middle of fins staggered system at different air flow velocity.

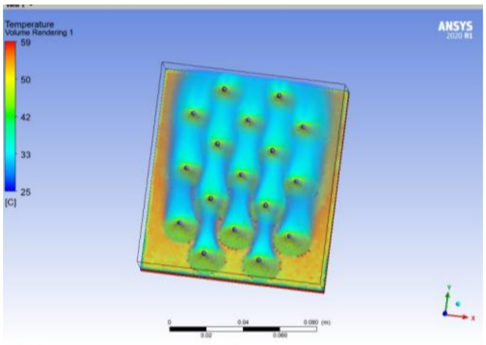


a) air velocity 1m/s

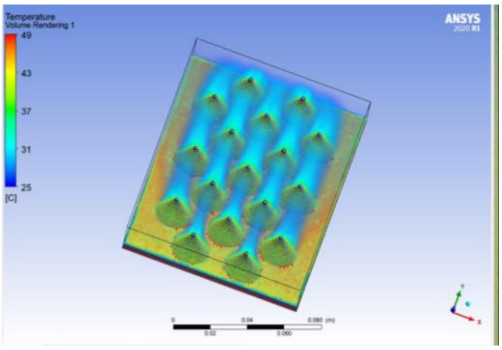


b) air velocity 2.5 m/s

Figure (6): Numerical flow velocity over cone fin with ellipse bas at different air flow velocity.



a) air velocity 1m/s



b) air velocity 2.5 m/s

Figure (7): Numerical temperature distribution of air flow due to hot cone fin with ellipse bas at different air flow velocity.

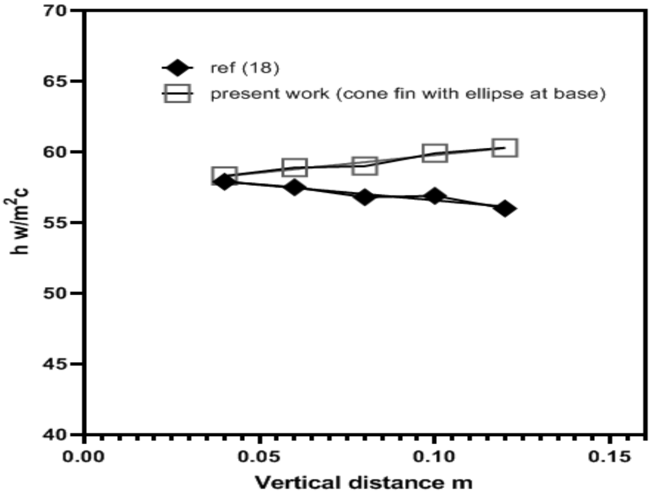


Figure (8): comparison the present work results with references [18] circular cylinder that have the identical situations.

Conclusions

The present work conducted the experimental and numerical thermal analysis of new shaped fins (cone fin with ellipse base) at different air flow velocity. A comparison is done at staggered arrangement with reference [18] and it's observed that the new shape fin gives a heat transfer rate greater that of classical cylindrical cylinder. Moreover, it very clear that the new shape will help strongly to produce more intensity turbulent flow that will enhance heat dissipation from the outer surface of fin systems. The numerical solution is very useful to understand the flow behavior in several levels of flow behavior. The new fin shape gives high flexibility in realizing the real effects, air movement and heat transfer under different flow conditions.

References

1. H. A.K Shahad, T. A. Husain , and F. H. Ali, "Experimental and Numerical Study of Fin Heat Sinks Systems", The Iraqi Journal for Mechanical and Materials Engineering · November 2009.

2. A. SLAOUTI and J. H. GERRARD, " An experimental investigation of the end effects on the wake of a circular cylinder towed through water at low Reynolds numbers ", J. Fluid Meeh. (1981), vol. 112, pp. 297-314. Printed in Great Britain.
3. S. H. LEE and LD. G. LEAL, "Low-Reynolds-number flow past cylindrical bodies of arbitrary cross-sectional shape" J. Fluid Mech. (1986), vol. 164, pp. 401427. Printed in Great Britain.
4. A. SOHANKAR, C. NORBERG AND L. DAVIDSON," Low-Reynolds-Number Flow around A Square Cylinder at Incidence: Study of Blockage, Onset of Vortex Shedding and Outlet Boundary Condition", INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS, VOL. 26, 39±56 (1998).
5. Saroj Yadav – Krishna Murari Pandey," A Parametric Thermal Analysis of Triangular Fins for Improved Heat Transfer in Forced Convection", Strojniški vestnik - Journal of Mechanical Engineering 64(2018)6, 401-411, Strojniški vestnik - Journal of Mechanical Engineering 64(2018)6, 401-411, DOI:10.5545/sv-jme.2017.5085.
6. Nabil L. K. Al-Saffar, Hayder K. Rashid, Mohammed K. Hamza and Mohammed Y. Jabbar, " Enhancement of Fin Efficiency: A New Proposal", international Review of aerospace engineering (IREASE), Vol.14, No. 1, (2021). DOI: 10.15866/irease.v14i1.18829
7. Khan, Zafar Hayat, Waqar Ahmad Khan, and Muhammad Hamid. "Non-Newtonian fluid flow around a Y-shaped fin embedded in a square cavity." Journal of Thermal Analysis and Calorimetry (2020): 1-13.
8. Yatendra Singh Tomar and M. M. Sahu “Comparative Study of Performance of Pin Fin under Forced Convection Heat Transfer” International Journal of Engineering Research and Technology, 2013, Vol. 2, PP. 364-374.
9. Hamid Reza Seyf and Mohammad Layeghi, " Numerical Analysis of Convective Heat Transfer From an Elliptic Pin Fin Heat Sink With and Without Metal Foam Insert", Journal of Heat Transfer Copyright © 2010 by ASME JULY 2010, Vol. 132.
10. Navid Namdari, Morteza Abdi, Hamidreza Chaghomi, Farzad Rahmani " Numerical Solution for Transient Heat Transfer in Longitudinal Fins" International Research Journal of Advanced Engineering and Science, Volume 3, Issue 2, pp. 131-136, 2018.
11. Vidyadhar Karlapalem *, Sukanta Kumar Dash," Vidyadhar Karlapalem *, Sukanta Kumar Dash", International Communications in Heat and Mass Transfer 120 (2021) 105071.
12. Rosnadiah Bahsan1*, Muhammad Aniq Asyraf Mohd Zamri1, Alhassan Salami Tijani1, Jeeventh Kubenthiran1, Sajith Thottathil Abdulrahman2 and Ibrahim Kolawole Muritala," Thermal Performances of Hybrid Pin Fin with Connector Heat Sink Under Natural Convection" , Pertanika J. Sci. & Technol. 30 (3): 2077 - 2096 (2022).
13. Ambarish Maji, Dipankar Bhanja, and Promod Kumar Patowari," Effect of Knurled Fin Surface on Thermal Performance of Perforated Fin Heat Sink",JOURNAL OF THERMOPHYSICS AND HEAT TRANSFER, 2018.
14. LI Xiuzhen1, WANG Lin, FENG Rong, WANG Zhanwei, ZHU Dongsheng," Thermal-Hydraulic Characteristics of Twisted Elliptical Tube Bundle in Staggered Arrangement", Journal of Thermal Science, Institute of Engineering Thermophysics, CAS and Springer-Verlag GmbH Germany, part of Springer Nature 2021.

15. M. Ishak, Tahseen Ahmad Tahseen And Md. Mustafizur Rahman, " Experimental Investigation on Heat Transfer and Pressure Drop Characteristics of Air Flow Over A Staggered Flat Tube Bank In Crossflow", International Journal of Automotive and Mechanical Engineering, Volume 7, pp. 900-911, January-June 2013.
16. Gary L. Hammock, " Cross-Flow, Staggered-Tube Heat Exchanger Analysis for High Enthalpy Flows, " Thesis, The University of Tennessee, Knoxville, 2011.
17. R. A. W. M. Henkes, F. F. Van Der Vlugt and C. J. Hoogendoorn, "Natural-Convection Flow in a Square Cavity Calculated with Low-Reynolds-Number Turbulence Models", Int. J. Hui Mass Transfer. Vol. 34, No. 2, pp. 377-388, 1991.
18. H. A.K Shahad , T. A. Husain , and F. H. Ali," Experimental and Numerical Study of Fin Heat Sinks Systems", The Iraqi Journal for Mechanical and Materials Engineering · November 2009.