|| Volume 6 || Issue 5 || May 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH
AND ENGINEERING TRENDS

EXPERIMENTAL INVESTIGATION AND HEAT TRANSFER ANALYSIS OF DIFFERENT TYPES OF FIN WITH CFD

Dr. U. V. Kongre[1], Dhiraj Khobragade[2], Mohd. Usaid Patel[3], Akash Katekhaye[4]

HOD & Project Guide, Mechanical Engineering, JDIET, Yavatmal, India [1]

B. E. (Mechanical Engineering) Final Year Student[2 3 4], Mechanical Engineering, JDIET, Yavatmal, India [2 3 4]

ukongre@gmail.com[1], idkhobragade37k@gmail.com[2], usaidpatel7834@gmail.com[3], akashkatekhaye@gmail.com[4]

-- ***--

Abstract: - When any fluid flows through rectangular duct with heated object placed inside it, a respective pattern of flow carries heat through convection and radiation. Also the rate of heat transfer depends on the flow velocity, area of exposure and other parameters. Different profiled objects may give different rate of heat transfer. Hence the objects like pin fin with enhanced surface areas (Perforated) placed inside the duct, the heat transfer rate will be maximum. In this experimental paper, the study of different author's research and their papers is carried out to understand the flow through the rectangular duct. Rate of heat transfer, mass flow rate, inside velocity, pressure and other associated parameters are studied and explained.

Keywords: *Optimum heat transfer, thermal analysis, geometry, Fluid Flow.*

---***--

I INTRODUCTION

Component heating is a major cause under a various working condition in today's engineering applications so that the to increase the heat transfer rate from the heated component under the load or various thermal conduction. Therefore, for reducing the cost and increase the effectiveness is turned into the major challenge for design of heat exchanger component. Pin fin is majorly used for heat transfer which results in the heat exchange between the surface of the body and the fluid flow through it. There are various types of fins for a heat transfer. In this experiment we manufactured pin fin of various geometry and materials like copper fin of circular slotted shape, aluminum fin of rectangular bar shape, composite fin of aluminum and copper.

In this experiment we obtained the characteristics experimentally and compared with the CFD analysis. Where there is fluid, there is CFD. Having mentioned before, the initial stage to conduct a CFD simulation is specifying an appropriate mathematical model of reality. Rapprochements and assumptions give direction through solution processes to examine the case in the computational domain. For instance, fluid flow over a sphere / cylinder is a repetitive issue that has been taught by the lecturer as an example in fluid courses. The same phenomenon is virtually available in the movement of clouds in the atmosphere which is indeed tremendous.

We have used ANSYS 14.5 software to perform CFD Analysis on AC duct. ANSYS Fluent is the module which is used for the performance of CFD analysis. The basic simulating conditions are applied to the model. ANSYS is the one of the leading CAE software which gives the solution of CAE problems by means of various types of analysis. There are several modules are available to perform analysis in ANSYS workbench window like static structural, steady state thermal, Modal, Fluid Flow (Fluent) etc. We can use our required type of tool for the analysis

II.METHODOLOGY

Square Fins: We made models of the fin by using CATIA software by using sketch command, pad, pocket commands etc. The fin having a dimension of side 20 mm length and height 125 mm.

Circular Fin:

We make models of these fin with the CATIA software by using the various command in software like pad, pocket, mirror etc. these commands are used for making the models with software. The pin fin having diameter of 20 mm and height of 125 mm with square threaded on its circumference with height of 2.5 mm.

Composite Fin:

In this type of pin fin, we consider the two types of materials are copper and aluminum. This type of pin fin is manufactured by placing a copper rounded bar inside the aluminum rounded bar. On the outer periphery of rounded bar there is five holes are for the thermocouple which gives us temperature at various of point of fin. The dimension of the fin is having outer material is of aluminum having diameter 20mm and hollow diameter which is outer diameter of copper rounded bar 13mm. On the outer surface of the aluminum rounded bar there is five

|| Volume 6 || Issue 5 || May 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH
AND ENGINEERING TRENDS

screws are situated for thermocouple to know the temperature at various point.

IGES Conversion:

The models designed using CATIA software converted into IGES (Initial Graphics Exchange Specifications) format to perform CFD analysis in ANSYS software.

III. EXPERIMENTAL SETUP

A fin of different cross sections is fitted across a long rectangular duct the other end of duct is connected to the suction side of blower and the air flow pass the fin perpendicular to the axis. One end of the fin projects outside the duct and is heated by the heater. Temperature at five points along the length of fins. The air flow rate is measured by an orifice meter fitted on the delivery side of blower. Schematic diagram of setup is shown in figure

Apparatus Specifications:

•Fins: 20 mm diameter with 150 mm length and 6 thermocouples along the length

•Rectangular Duct: 150 mm*100 mm cross-section area with 1000 mm length one side of which is connected to blower.

•Centrifugal blower is used to supply forced air.

•Multichannel Digital Indicator: 0-300 C

•Dimmer-stat: 2 Amp, 240 Volt, for heater input control

•Voltmeter: 0 - 250 Volts.

• Ammeter: 0 - 2 Amp.

• Orifice-meter with water manometer.

Dimensional Specifications of Fins:

- Fins: 20 mm diameter with 150 mm length and 6 thermocouples along the length
- Rectangular Duct: 150 mm*100 mm cross-section area with 1000 mm length one side of which is connected to blower.
- Centrifugal blower is used to supply forced air.
- Multichannel Digital Indicator: 0-300 C
- Dimmer-stat: 2 Amp, 240 Volt, for heater input control
- Voltmeter: 0 - 250 Volts.
- Ammeter: 0 - 2 Amp.

Orifice-meter with water manometer

Experimental Investigation:

Sr. No.	CFD Result	Circular Fin		Square Fin		Composite Fin	
		Free Convection	Forced Convection	Free Convection	Forced Convection	Free Convection	Forced Convection
1	Pressure (Pa)	20.3	74.5	14.2	80.5	22.6	85.6
2	Temperature (k)	398	398	398	398	398	398
3	Turbulence ($k\text{-}m^2/s^2$)	4.16	10.9	1.48	6.25	2.99	9.37
4	Velocity (m/s)	7.5	15.2	5.36	13.3	**7.85**	**15.7**

Table 3.1: Tabulated Results genera ted through CFD Analysis

As we know that "more the velocity, higher the heat transfer rate". Hence, we found the maximum velocity in case of circular and composite fins. Composite fins are having slightly better results than the circular fins. Hence the Composite fin will have the maximum heat transfer rate as per the CFD Analysis.

IV CFD ANALYSIS

Steps to Perform CFD Analysis:

Meshing

This process is also called as discretization. In this process entire model is divided into small number of pieces called elements and they are connected together by means of points called nodes. Following figure 5.6 shows the meshed view of duct with circular fins along with number of nodes and elements.

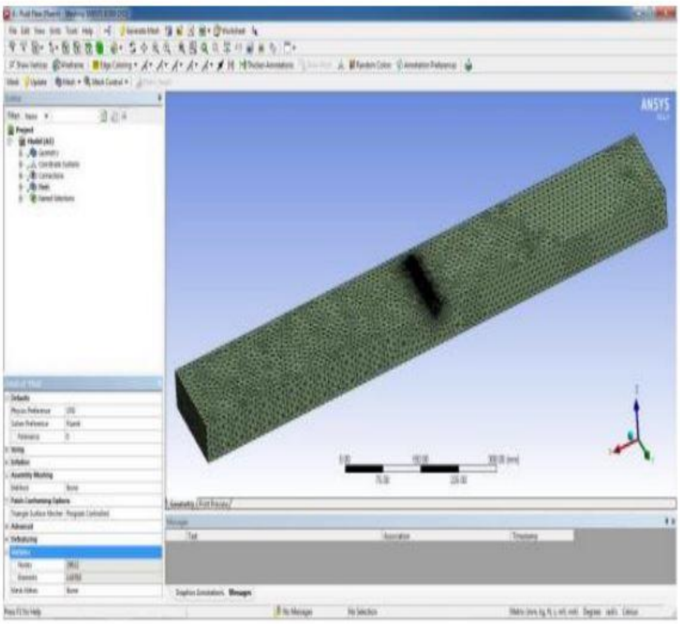

Fig.4.1: Meshed view duct with composite fins

|| Volume 6 || Issue 5 || May 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH
AND ENGINEERING TRENDS

Sr. No.	Parameter	Value
1	Number of Nodes	29822
2	Number of Elements	118783

Table 4.1: Nodes and Elements

Sr. No.	Parameter	Value
1	Number of Nodes	20258
2	Number of Elements	93911

Table 4.2: Nodes and Elements

Sr. No.	Parameter	Value
1	Number of Nodes	21531
2	Number of Elements	106701

Table 4.3: Nodes and Elements

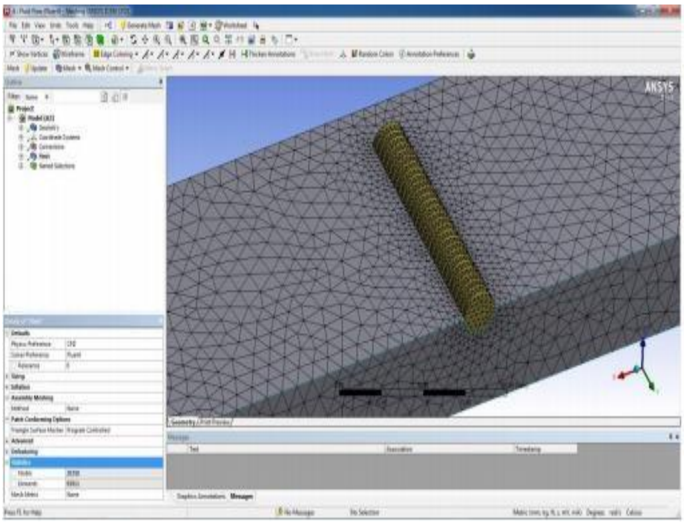

Fig.4.2: Meshed view duct with Circular fin

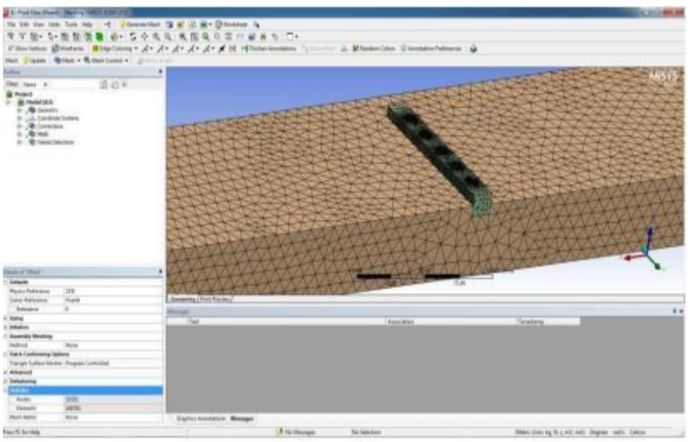

Fig.4.3: Meshed view duct with Square fin

V RESULTS GENERATED BY CFD ANALYSIS

Results for Circular Fins (Case–1: Natural Convection)

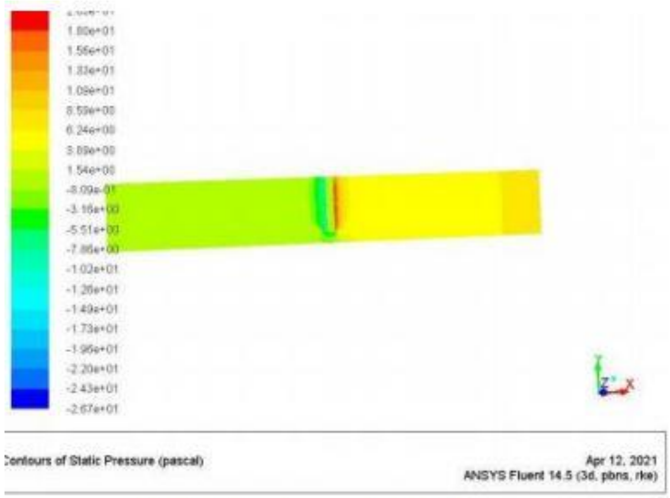

Fig.5.1: Contours of Static Pressure

The above figure shows the pressure variation inside the duct in case of free convection with circular fins. The flow of air is taken from right to left (i. e. in –x direction). By observing contours, it is found that the pressure in front of the fin is maximum. Whereas it is decreased just front of the fins. It is due to the obstacle created by the fin for air flow. Pressure before the fin is higher that pressure after the fin.

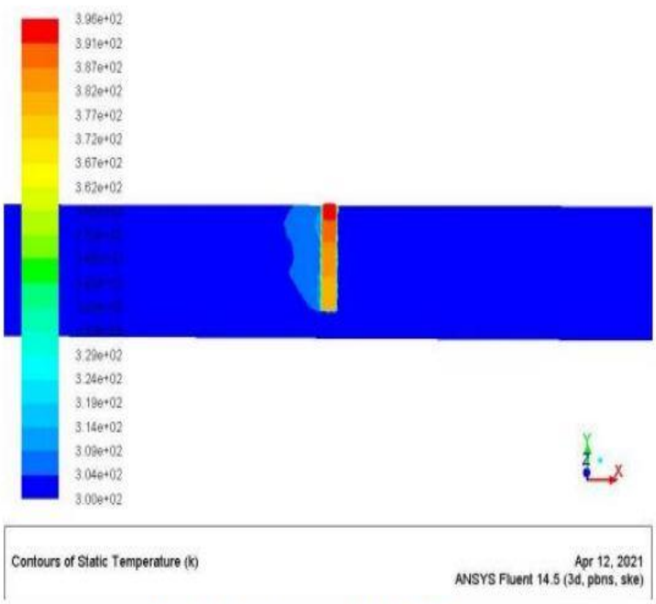

Fig.5.2: Contours of Static Pressure

The above figure shows the contours of temperature in a duct. Total 1200C temperature is applied on the circular fin. By observing temperature contours, it is found that the convection of heat takes place which reduces the fin temperature.

|| Volume 6 || Issue 5 || May 2021 || ISSN (Online) 2456-0774
INTERNATIONAL JOURNAL OF ADVANCE SCIENTIFIC RESEARCH AND ENGINEERING TRENDS

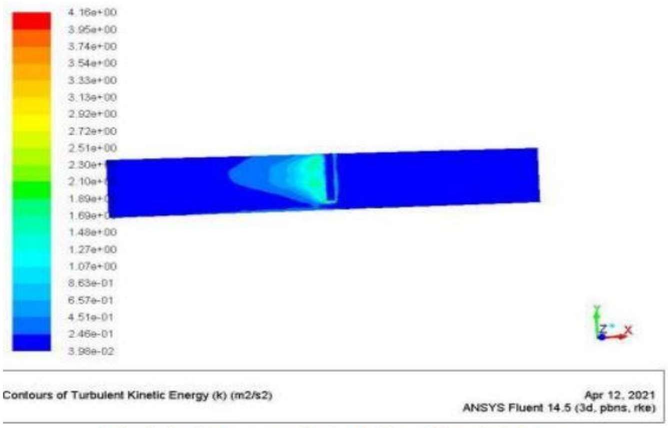

Fig.5.3: Contours of Turbulent Kinetic Energy

By observing above figure which shows the turbulent kinetic energy in a duct, it is found that the energy is increased beyond the fins due to swirling and obstacle of fin.

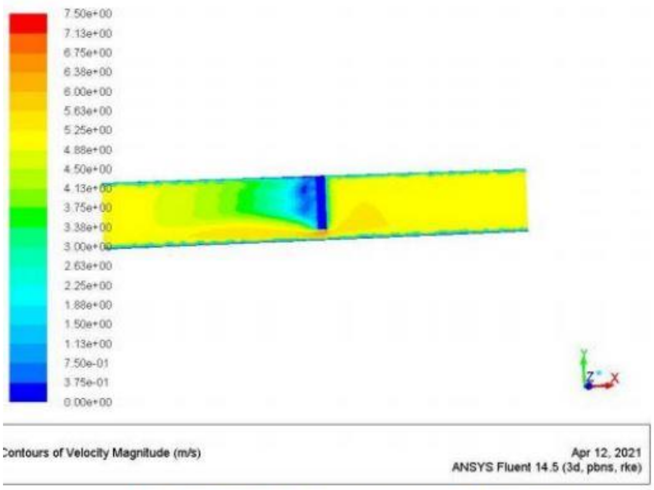

Fig.5.4: Velocity variation inside the duct

Velocity remains constant throughout the duct. Only the variation is observed near the fin. At the top edge of the fin velocity found maximum i. e. 7.5 m/s. Whereas the applied velocity is 5 m/s only. Pressure drop near the fin is the main reason of this variation.

Results for Circular Fins (Case–2: Forced Convection)

This is the case in which the air is forced to flow inside the duct with a 10 m/s velocity. Same fin i.e. circular fin is taken to find the temperature, pressure, and velocity and turbulence variation inside the duct. Following results are showing the same. These results are studied separately and compared with free convection results.

Figure is showing pressure variation inside the duct which indicates 74.5 Pascal as a maximum pressure. If we compare this result with previous pressure variation with free convection then it is more than free convection (20.3 Pascal). It is obvious as the is of forced convection.

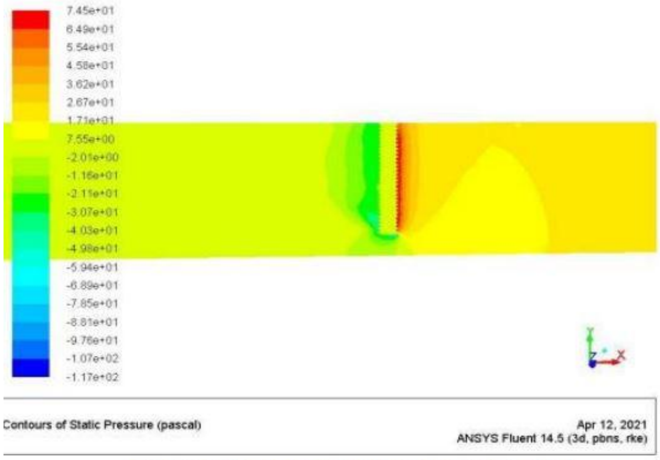

Fig.5.5: Pressure variation inside the duct

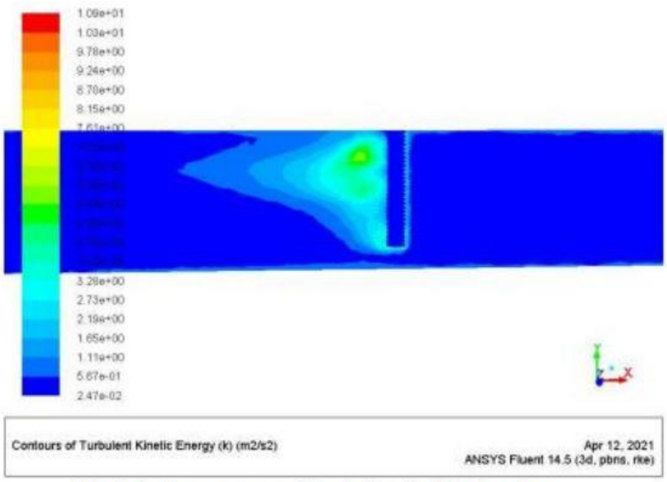

Fig.5.6: Temperature Variation inside the duct

By observing the temperature contours in above figure, convection is improved due to the forced air flow. It is maximum than the free convection. The temperature bands obtained in this case are broader than the previous case.

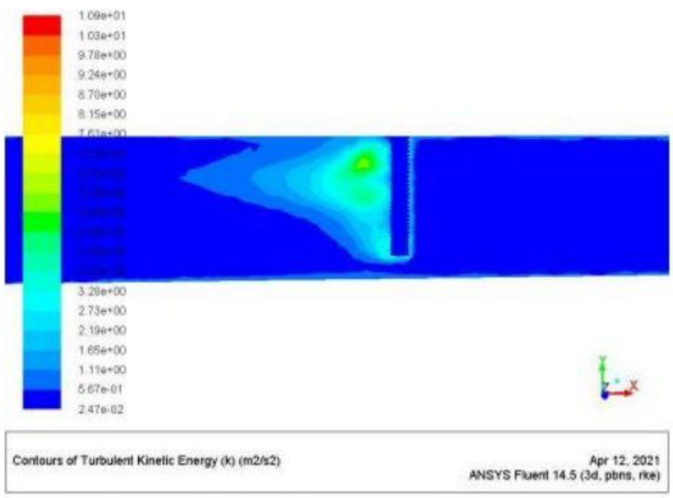

Fig.5.7: Contours of Turbulent Kinetic Energy

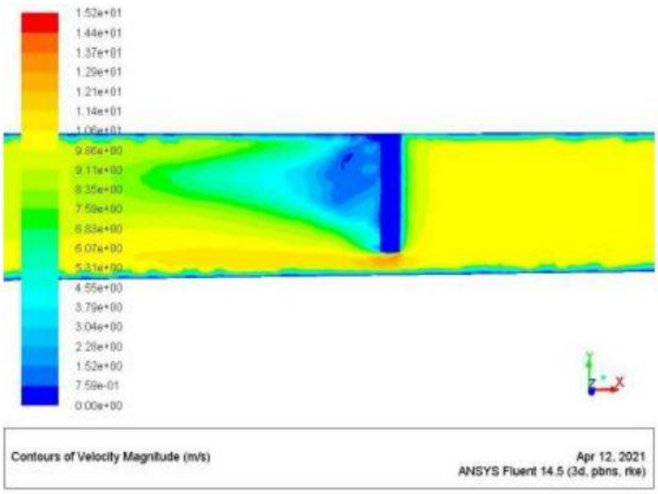

Fig.5.8: Velocity variation inside the duct

Velocity variation is found maximum near the fin and at the top of the edge. As compare with free convection velocity is more here i. e. 15.2 m/s. Similarly Following results are obtained.

VI CONCLUSION

It was designed an important graphical user interface to analyse and design fins for exchange of heat. Three case studies were performed, the temperature profile of different fins as a function of length for different materials, and the effect of thickness on the proper length of the fin and the heat transfer rate for copper and aluminium. The best working material was copper with the highest thermal conductivity(395 W/m·°C). Low thickness involves low temperature in the extreme of the fin resulting in a low heat transfer in that zone. The higher the thickness, the higher the proper length. Between copper and aluminium, the best material for the same thickness is aluminium because it has the shortest proper length. Though, it is necessary to see the amount of heat removed. Aluminium removes less heat than copper. The composite fin gives result in between two.

ACKNOWLEDGEMENT

The authors are thankful to all Professors who help for the preparation of this paper and would like to thank the anonymous reviewers and main authors for their comments which were very helpful in improving the quality and presentation of this paper.

REFERENCES

1. Balendra Singh, Satish Singh, "A Research Paper on Heat Transfer in Notch Fin and UN Notch Fin" International Journal for Research in Applied Science & Engineering Technology (IJRASET). ISSN: 2321-9653.Volume 4, Issue IX, September 2016.

2. Basavaraj Olekar, Ganesha T, "Experimental and Numerical Analysis of Cylindrical Pin Fins having Square Thread with and without Perforations by Natural and Forced Convection.", IJISET - International Journal of Innovative Science, Engineering & Technology, ISSN (Online) 2348 – 7968, Vol.

3 Issue 7, July 2016.

3. H. Azarkish, S.M.H. Sarvari, A behzadmehr, "optimization design of a longitudinal fin array with convection and radiation heat transfer using a genetic algorithm".

4. International Journal of thermal Sciences Vol-49. ELSEVIER, 2010. pp 589-602.

5. Ganesha Murali J. Subrahmanya S Katte, "Experimental Investigation of heat transfer enhancement in radiating pin fin", Jordan Journal of Mechanical and Industrial Engineering, ISSN: 1995-6665, Vol- 2, No- 3, Sep 2008. pp 163-167.

6. Javid Siddiqui, A. V. Gadekar, "A review paper on CFD Simulation of square threaded & helical twisted pin fin array in a rectangular channel to enhance heat transfer", International Journal of Science and Research, ISSN 2319-7064. 2013. pp 365-372.

7. S. M. Wange, R. M. Metkar, "Computational Analysis of Inverted Notched Fin Arrays Dissipating Heat by Natural Convection", International Journal of Engineering and Innovative Technology (IJEIT) Volume 2, Issue 11, May 2013.

8. Shivdas S. Kharche, Hemant S. Farkade, "Heat Transfer Analysis through Fin Array by Using Natural Convection", International Journal of Emerging Technology and Advanced

Engineering, Volume 2, Issue 4, April 2012.

9. Ganesha B, G V Naveen Prakash, "Forced Convection Heat Transfer through the Rectangular Fins of Different Geometry of Perforations". International Journal of Recent Technology and Engineering (IJRTE) ISSN: 2277-3878, Volume-8 Issue-1S2, May 2019

10. C. Mageswaran, R. Karthikeyan, R.Muthukumaran, "Heat Transfer Enhancement of Perforated Square Pin-Fin Array in a Rectangular Duct". International Journal for Research in Applied Science & Engineering Technology (IJRASET), ISSN:

2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887, Volume 5 Issue IX, September 2017-Available at www.ijraset.com