A Numerical Analysis of Heat Transfer Performance of Gasifier Fins Using FEM

Vikash Kumar Choube[1], Dr. D S Rajput[2], Dr. Sachin Baraskar[3]
PG Scholar[1], Associate Professor[2], Associate Professor[3]
Department of Mechanical Engineering[1,2,3]
Sri Satya Sai University of Technology and Medical Sciences Sehore, Bhopal (MP)[1,2,3]

Abstract: *Gasifier is used for production of gas and pipes are employed to carry gases from gasifer to IC engines. Fins are the extended surface protruding from a surface or body and they are meant for increasing the heat transfer rate between the surface and the surrounding fluid by increasing heat transfer area. In gasifier power plant, condensing element used for decrease the temperature of hot gases which is found out from gasifier. In this condensing element fins are used for decreasing the temperature difference. The steady-state convection heat transfer from rectangular, triangular and proposed fins extending perpendicularly from vertical rectangular base was investigated numerically. The effects of geometric parameters and base-to-ambient temperature difference on the heat transfer performance of fin arrays were observed and the optimum fin separation values were determined. The aim of this investigation is to design a model and thermal analysis of gas carrying pipe with fins using ANSYS APDL finite element analysis software and also select a proper material composition for pipes. And in this present wok, thermal flux, temperature distribution, thermal gradient found via simulation software. The comparisons also made between the calculation results and software results.*

Keywords: *Gassifier, ANSYS, Steady-state, Temperature.*

1. Introduction

A fin is a surface that extends from an object to increase the rate of heat transfer to or from the environment by increasing convection. The amount of conduction, convection or radiation of an object determines the amount of heat it transfers.
Conventional sources of energy are being depleted at an alarming rate, which seems to put hurdle for sustainable development in future. On the other hand, in recent years, the issue of providing more efficient and reliable thermal systems in terms of reducing the size, weight, cost, and saving of energy has received substantial attention of Researchers. In order to fulfill these demands, many engineering techniques have been investigated over the years.

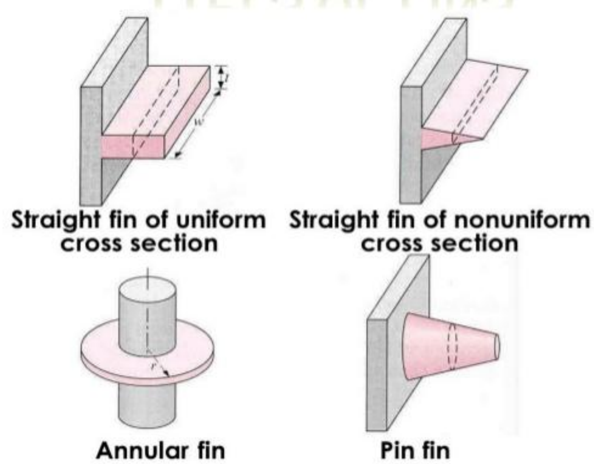


Figure.1 Types of fins

Techniques to Break the Laminar Sub-layer

Employing ribs or grooves on the inner surface of channels has been one of the frequent passive approaches to break the laminar sub-layer and create local wall turbulence due to flow separation and reattachment between successive corrugations, which reduces the thermal resistance and significantly enhances rate of heat transfer. The ribbed channel, because of its effectiveness in heat transfer, is a good candidate for engineering applications, such as cross-flow heat exchanger, gas turbine airfoil cooling design, solar air heater, blade-cooling system, and gas cooled nuclear reactor.

There is a lot of temperature (About 1500 degree Celsius) in the gasifier. But, when the area of fins not proper than the temperature difference decreases. For the solution of this above mention problem, new technique fins introduced in this present work.

2. Nomenclature

Ac	surface area of absorber plate (m2)
Cp	specific heat of air(J/kg/K)
D	equivalent or hydraulic diameter of duct(m)
E	rib height(m)
H	heat transfer coefficient(W/m2/K)
H	depth of duct(m)
I	turbulence intensity/intensity of solar radiation (W/m2)
K	thermal conductivity of air(W/m/K)
L	length of duct(m)
L_1	inlet length of duct(m)
L_2	test length of duct(m)
L_3	outlet length of duct(m) m mass flow rate(kg/s)
D_P	pressure drop(Pa)
P	pitch (m) q_u useful heat flux(W/m2)
Q_u	useful heat gain(W)
Q_L	heat loss from collector(W)
Q_t	heat loss from top of collector(W)
T_o	fluid outlet temperature(K)
T_i	fluid inlet temperature(K)
Ta	ambient temperature(K)
T_{pm}	mean plate temperature(K)
T_{am}	mean air temperature(K)
T_w	wall temperature(K)
Tm	bulk mean temperature(K)
U_L	overall heat loss coefficient(W/m2/K)
V	velocity of air in the duct(m/s)
W	width of duct(m)

3. Channel Geometry and Boundary Condition

Objectives

The main objectives of present investigation shows below:
i. Find the thermal effect (Conduction & Convection) on Aluminum Alloy by steady - state analysis.
ii. Compare the results of temperature distribution on different geometries of fins.
iii. Find the thermal gradient and Heat flux on different aluminum alloys and validate with analytical analysis.

Methodology

Collecting information and data related to fins of IC gasifier. A fully parametric model of the fin is created in ANSYS APDL software. Model obtained in Step 2 is analyzed using ANSYS APDL, to obtain the heat rate, thermal gradient, heat flux and nodal temperatures. Manual calculations are done. Finally, we compare the results obtained from ANSYS and manual calculations for different material, shapes and thickness.

Figure.2 Fins in Gasifier

4. Governing Equation

The steady 3-dimensional form of the continuity, the time-independent incompressible Naviere Stokes equations and the energy equation governs turbulent airflow through

artificially roughened solar air heater. These equations can be written as follows:
The assumptions made on the operating conditions of the ribbed channel are as follows:-
(1) The ribbed channel operates under steady-state conditions
(2) The fluid is incompressible and remains in single-phase along the channel
(3) The properties of the fluid and channel material are independent of temperature
(4) Uniform heat flux is incident on upper wall

The single-phase governing equations for flow and heat transfer in the ribbed channel can be written in the Cartesian tensor system as:-

Continuity equation:
Law of Conservation of Mass: Fluid mass is always conserved. (Equation 1)

$$\frac{\partial(\rho u_i)}{\partial x_i}=0 \tag{1}$$

Where ρ is the density of fluid and u_i is axial velocity

Momentum equation:
Newton's 2nd Law: The sum of the forces on a fluid particle is equal to the rate of change of momentum. (Equation 2)

$$\frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j} + \frac{\partial u_i}{\partial x_j}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_\iota u_J}\right) \tag{2}$$

Here μ is the viscosity of fluid,
Where μ, u_i and u_j are the fluid viscosity, fluctuated velocity, and the axial velocity, respectively, and the term the turbulent shear stress. The Reynolds-averaged approach to turbulence modeling requires that the Reynolds stresses $-\rho\overline{u_\iota u_J}$, in Eq. (2) needs to be modelled. For closure of the equations, the k-ε turbulence model is chosen. A common method employs the Boussinesq hypothesis to relate the Reynolds stresses to the mean velocity gradients:

$$-\rho\overline{u_\iota u_J} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_j}\right) \tag{3}$$

Energy equation:
First Law of Thermodynamics: The rate of head added to a system plus the rate of work done on a fluid particle equals the total rate of change in energy. (Equation 3)

$$\frac{\partial}{\partial x_i}(\rho u_i T) = \frac{\partial}{\partial x_j}\left[(\Gamma + \Gamma_t)\frac{\partial T}{\partial x_j}\right] \tag{4}$$

Where Γ and Γ_t, are molecular thermal diffusivity and turbulent thermal diffusivity, respectively and are given by

$$\Gamma = \frac{\mu}{pr}\ ,\ \text{and}\quad \Gamma_t = \frac{\mu}{pr_t} \tag{5}$$

The turbulent viscosity term μ_t is to be computed from an appropriate turbulence model. The expression for the turbulent viscosity is given as

$$\mu_t = \rho c_\mu \frac{k^2}{\varepsilon} \tag{6}$$

There are two additional equations for the k-ε turbulent model:
(i) Turbulent kinetic energy (k):

$$\frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\delta}{\delta x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_{k-}\rho\varepsilon \tag{7}$$

(ii) Rate of dissipation (ε):

$$\frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\delta}{\delta x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}G_k - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}\right] \tag{8}$$

In the above equation, G_k represents the rate of generation of the turbulent kinetic energy due to mean velocity gradients while $\rho\varepsilon$ is its destruction rate. The σ_k and σ_ε are effective Prandtl numbers for turbulent kinetic energy and rate of dissipation, respectively; $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are constants. G_k is written as:

$$G_k = -\rho u_i u_j \frac{\overline{\delta u_J}}{\delta x_j} \tag{9}$$

The boundary values for the turbulent quantities near the wall are specified with the enhanced wall treatment method. $C\mu=0.09$, $C_{1\varepsilon}=1.44$, $C_{2\varepsilon}=1.92$, $\sigma_k=1.0$, $\sigma_\varepsilon=1.3$ and $Pr_t=0.9$ are chosen to be empirical constants in the turbulence transport equations.
The governing equations are solved using a finite volume approach and the SIMPLE algorithm. The solutions are considered to be converged when the normalized residual values reach 10^{-5} for all variables.

PARAMETERS INVOLVED
To analyze and compare the flow characteristics and heat transfer of different configurations of ribbed channels, following as:-

(1) Hydraulic diameter (D_h):

$$D_h = \frac{4A}{P_h} \quad (10)$$

Where A is cross-sectional area and P_h is wetted perimeter of the cross-section

(2) Reynolds number (Re):

$$Re = \frac{\rho u_m}{\mu} D_h \quad (11)$$

Where u_m mean velocity of fluid in cross-section here is Reynolds numbers are taken a range of 3800-15000 in order to have turbulent regime.

(3) Nusselt number (Nu):

$$Nu = \frac{hD_h}{k} \quad (12)$$

Where h is convective heat transfer co-efficient and k is thermal conductivity of air

(4) Friction factor for fully developed turbulent flow (f)

The friction factor is computed by pressure drop, P across the length of test section, and can be obtained by

$$f = \frac{2\Delta pD}{\rho L u^2} \quad (13)$$

Where Δp is the pressure difference between inlet and outlet:

$$\Delta p = p_{av,\ inlet} - p_{av,\ outlet} \quad (14)$$

Here, $p_{av,inlet}$ and $p_{av,\ outlet}$ are the inlet and outlet average pressure, respectively.

(5) Thermo-hydraulic performance parameter (THPP):

Under a constant pumping power, the thermo hydraulic performance parameter has been used to estimate how effectively an artificially roughened surface enhances the heat transfer under constant pumping power constraints. In order to analyze overall performance of a solar air heater, thermo-hydraulic performance should be evaluated by considering thermal and hydraulic characteristics of the solar air heater simultaneously. Webb and Eckert suggested a Thermo hydraulic performance parameter which is used to compare the heat transfer of artificially roughened duct to that of a smooth duct. A value of thermo hydraulic performance parameter greater than one ensures the effectiveness of using an enhancement device and can be used to compare the performance of number of arrangements to decide the best among these.

$$\text{Thermo hydraulic performance parameter} = \frac{Nu_r/Nu_s}{(f_r/f_s)^{1/3}} \quad (15)$$

Here Nu_s and f_s are the Nusselt number and friction factor for smooth channel, respectively and Nu_r and f_r are the Nusselt number and friction factor for rough surface with different ribs shape.

Selection of Appropriate Turbulence Model

Results were validated by comparing the obtained numerical data with the available correlations developed for a smooth channel and also with the numerically conducted studies on transversely roughened channels.

First, the Nusselt number and friction factor obtained from the present smooth channel for turbulent flow are compared with correlations of Dittus-Boelter and Blasius, respectively.

Correlations of Dittus-Boelter:

$$Nu = 0.023 Re^{0.8} pr^{0.4} \quad \text{(for heating)} \quad (16)$$

Correlations of Blasius:

$$f = 0.316 Re^{-0.25} \text{ for } 3000 < Re < 20000 \quad (17)$$

The Nusselt number (Nu) of channel has been calculated by using three different turbulence models, including, the standard k-ε turbulence model, the renormalized group (RNG) k-ε turbulence model, and the standard k-ω turbulence model, and found that the RNG k-ε turbulence model gives the better result of Nusselt number as compared to other turbulence model when compared with the value of Nusselt number obtained from Dittus-Boelter equation.

5. Finite Element Analysis

The numerical analysis was been carried out for computing the temperature and thermal stress distributions in various types of fins using commercially available finite element software ANSYS with APDL. The preferences system option for thermal analysis was selected. The numerical analysis was based on the following assumptions:

i) Steady-state heat flow,
ii) The materials are homogeneous and isotropic,
iii) The convection heat transfer coefficient is the same all over the surface,
v) The temperature of the surrounding fluid is uniform,
vi) The thermal conductivity of the material is constant.

FEM Implementations

1. Preference (Thermal-steady state) –
    - Select Steady State Thermal Analysis from Analysis System tool bar.
    - Select Engineering Data and Insert the required Input Data, Material: Aluminum, Thermal Conductivity.
2. Preprocessing (Create FE model, Meshing and BCs) –
    - Input Analysis Parameter and boundary parameter.

3. FEM solver (assembles and solve the system of equations) –
    - Generate solution Click ‘temperature’; click ‘generate solution’; Click ‘temperature distribution’.
4. Post processing (sort and display in the form of graph the results)

Element Types

ANSYS APDL used to analyze the flux, gradient, stresses and deflections in the pipe/tube walls due to the internal temperature. Since the tube is subjected to internal temperature using three dimensional, element (SOLID 278).

Material Properties:

Table 1 shows material properties of three metal alloys.

Material Properties (unit)	Value
AA2024	
Density (g/cc)	3.26
Young’s modulus (GPa)	65
Poisson’s ratio	0.29
Thermal expansion coefficient	4.6
Thermal conductivity (W/m-k)	175
Tensile yield strength (MPa)	290
AA6063	
Density (g/cc)	3.5
Young’s modulus (GPa)	67
Poisson’s ratio	0.29
Thermal expansion coefficient	4.8
Thermal conductivity (W/m-k)	200
Tensile yield strength (MPa)	350

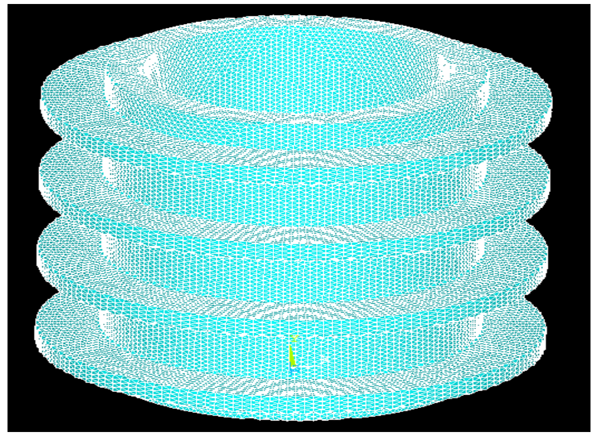
Meshing of tube and fins:

The meshing was performed by the ANSYS APDL auto-meshing tool after taking into the account the material type, the geometry of the model and the type of analysis being performed. Though, an increase in the number of elements generally means more accurate results, however for any given problem, the finite element solution converges to certain number of nodes. The number of nodes considered in this analysis was in the range 8000-10000. Below figure is showing the meshing of various types of fins.

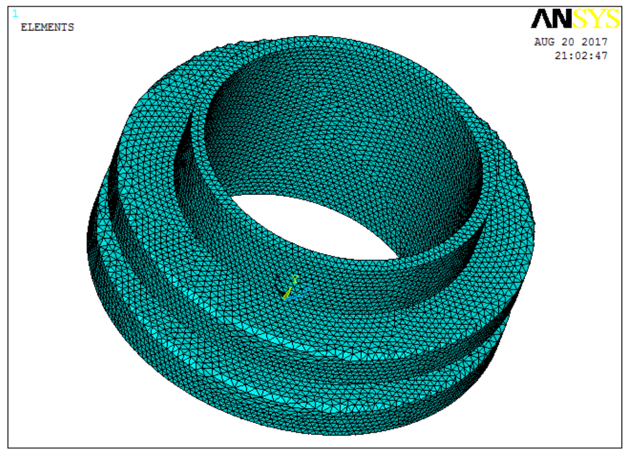
The temperature distribution of fins can be estimated by using ANSYS APDL for different nodes. The prediction of temperature distribution is given below:

Table:.2 Element type and meshing

Element type	Nodes	ELEMENT
SOLID	6500	8850



Meshing of tube and fins



Meshing of tube and fins

Boundary Conditions:

The modeled fins were subjected to convective boundary conditions on the inner and outer surface and the displacement boundary condition on the ends. Since only small vertical section of the tube was considered for analysis, the heat flux through the tube material can be safely ignored. Thus the top and bottom surface of the horizontal section of the tube has been kept as insulated. The fluid bulk temperature inside the tube was kept fixed at 960°C and the outside temperature for the fin was kept at 45°C.

The inner (gas and the tube) and outer (fin surface) convective heat transfer coefficient, h; are 1000 and 100 W/m^2 K, respectively. In order to provide resistance to bending, each element of the tube had length (L) to radius (R) ratio greater than or equal to 10. The tube edges at the top and bottom ends were constrained to have zero displacement in the y-direction and free displacements in the x-direction and z-direction. The heat flux was kept at zero at both tube edges.

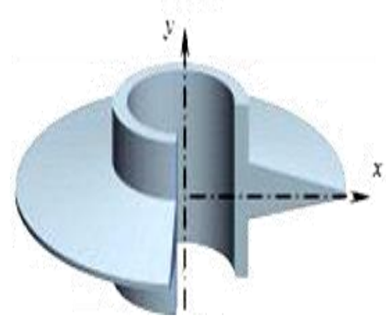
Solution:

In order to validate the thermal analysis results, the calculation were done and compare with ANSYS results by taking minimum thickness value 1 to 3 mm. The report gives the maximum deflection, thermal flux and thermal gradient distribution under the operating temperature. The purpose of this analysis is to ensure that the results of the finite element analyses were compared to the analytical findings values and the error of acceptance.

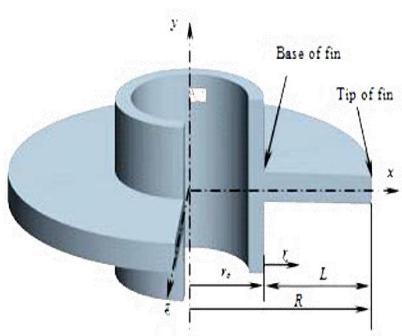
Table 3 Input Parameters

Parameters	Value
Diameter of pipe	22 mm
Length of pipe	17 mm
Thickness of fins	1 to 3 mm
Thickness of pipe	1 mm

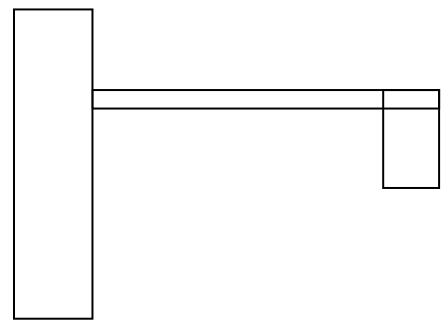
Finite element method is a mathematical technique used to design a fuel carrying pipes and performing the thermal analysis. In this the geometrical model is created and the model is sub divided into smaller elements. It is subjected to internal temperature and these Boundary conditions are applied at specified points.



(i) Triangular Fins



(ii) Rectangular Fins



(iii) Proposed Fins

Fig. 3 Circular fins with various profiles. (*i*) Triangular section,(*ii*) Rectangular section and (*iii*) proposed section.

Fin Profiles

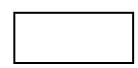
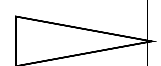
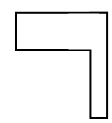
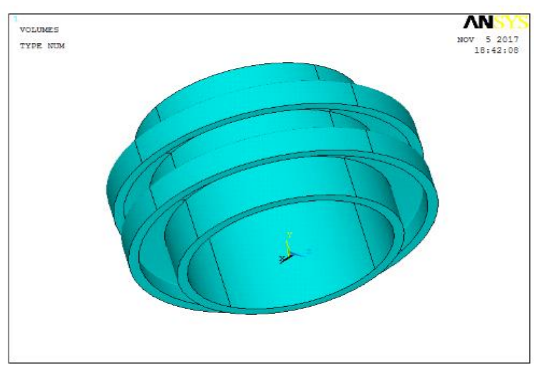
Types of radial fin profiles	Rectangle	Triangular	Proposed
Profile			

Fig. 4 depicts the circular fins with cross sections of the three different geometries considered for the thermal analysis. The triangular fins being axis-symmetric, rotation of the cross-sectional plane by 360° gives the solid model of the triangular fin.

VI. Result Analysis

In this present work, new type of fins is introduced. Figure 4 is showing that the FE model of new type of fins which used in gasifier condensing plant.



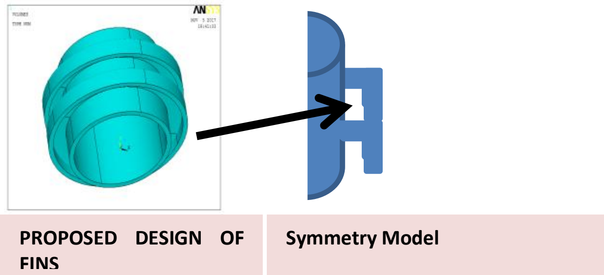


Figure 5 FE Model of Proposed Fins

Temperature distribution along the thickness of the fin with different profiles

A thermal-steady state analysis was carried out on AA 6067 tube with different profiles (Triangular, Rectangular and proposed) to determine the temperature distribution along the thickness of the fin. Temperature distribution contours in case of radius ratio 1 mm for the three different profiles are shown in Figure, and the effect of different fin profiles on the temperature distribution for various radius ratios are represented in the graph.

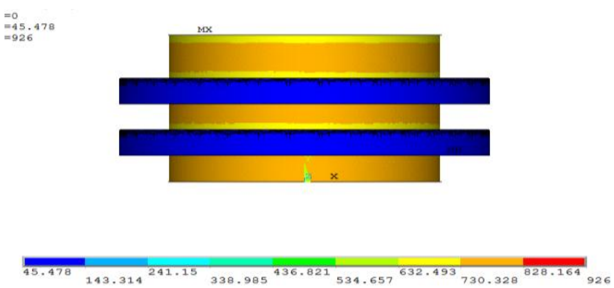


Figure 6 Temperature distribution of proposed fin (Thickness of fins = 1 mm

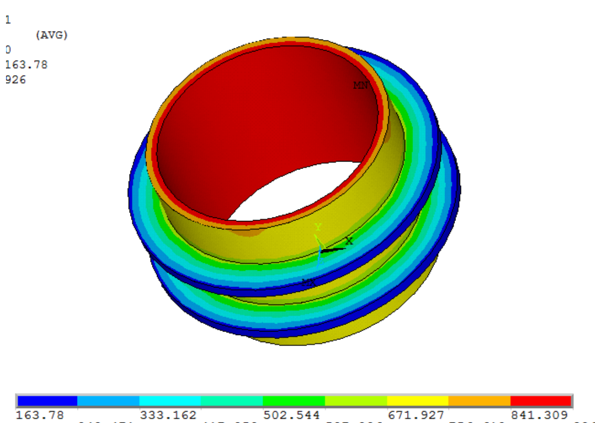


Figure.6.6 Temperature distribution of rectangular fin (Thickness of fins = 1 mm)

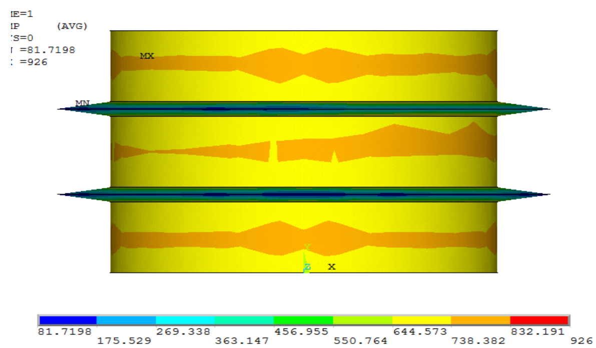


Figure 7 Temperature distribution of triangular fin (Thickness of fins = 1 mm)

From the below figure, it is evident that there is decrease in the temperature along the thickness of the fin for proposed profiles. It is found that the base temperature is maximum in the case of rectangular profile and minimum for proposed profile, while that of the triangular profile lies in between the proposed and rectangular profile. This is because area of fins is large in proposed fins will lead to more heat being transferred to the surrounding and less heat stored in the fin material, hence resulting in low base temperature.

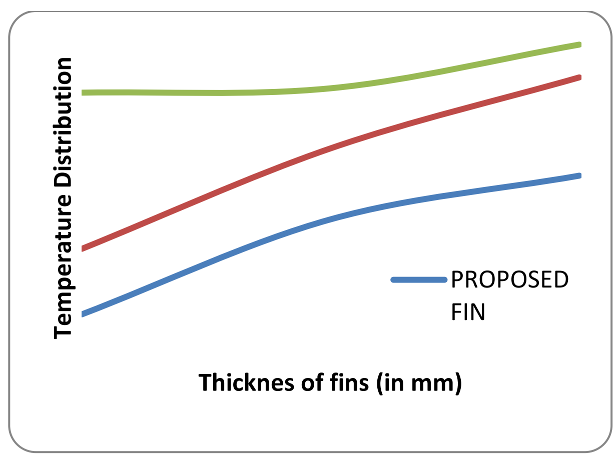


Figure 8 Temperature distribution of three different fins (Thickness of fins = 1 to 3 mm)

Temperature Difference

The FE analysis described in this paper incorporates several possible performance factors. These simplified terms represent a combination of several factors, such as material conductivity, lateral fin conduction, boundary layer formation, and effective cross section area. In comparing the circular fin of rectangular cross section, the triangular fin and proposed fin it was found that, the proposed profile fins yield a lower tip temperature distribution than other as shown in the table comparing with the length of the fin and space optimization rectangular fin can give better output than the real sample.

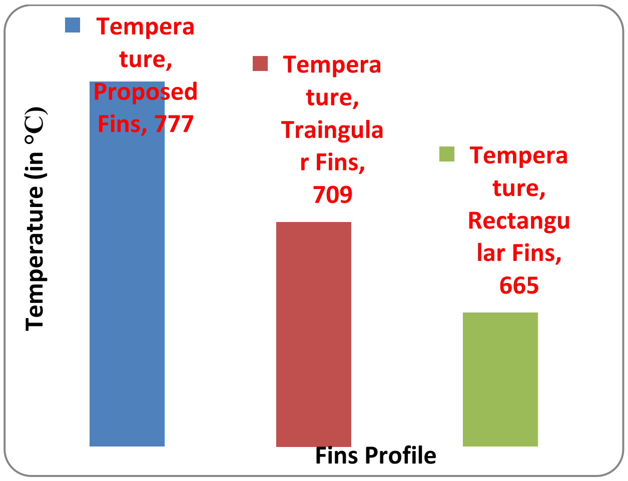


Figure 9 Temperature Difference

Below figure is showing the temperature difference on various points of proposed fin.

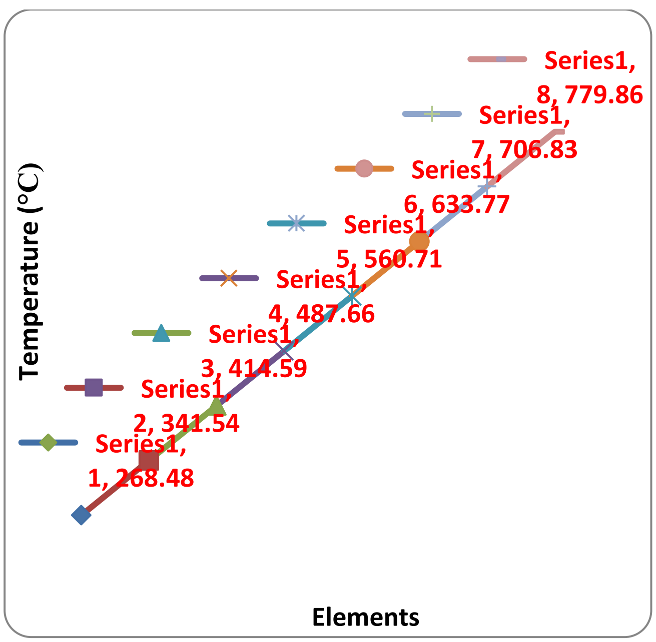


Figure 10 Temperature Difference on various points on Proposed fins

Key results of FEM

Below table is showing key results of FEM –

Rectangular Fins						
Thickness of Fins	Thermal Flux		Thermal Gradient		Nodal Temperature	
	Max	Min	Max	Min	Max	Min
1	56200.1	15271.7	312.223	84.8429	926	163.78
2	49503.4	21999.6	275.019	122.22	926	246.998
3	46587.6	21649.6	258.82	120.276	926	268.483
Triangular Fins						
Thickness of Fins	**Thermal Flux**		**Thermal Gradient**		**Nodal Temperature**	
	Max	**Min**	**Max**	**Min**	**Max**	**Min**
1	47813.8	20212.8	265.632	112.293	926	81.7198
2	42601.2	21487.6	236.673	119.375	926	191.781
3	40042.5	21003.6	222.458	116.687	926	259.509
Proposed Fins Design						
Thickness of Fins	**Thermal Flux**		**Thermal Gradient**		**Nodal Temperature**	
	Max	**Min**	**Max**	**Min**	**Max**	**Min**
1	44896.5	4446.04	249.425	24.7002	926	142.97
2	39607.6	4678.04	220.5	31.08	926	190.5
3	34508.2	4696.04	210.6	32.08	926	210.5

7. Conclusions & Future Scope

7.1 Conclusions

In this work, a new type of fins provided and this design modeled through ANSYS APDL FE Software. The aim of the investigation consists into find out the effect of various shapes of conventional fins over new type of fins.
According to the results –

I. FE simulation results showed that, Temperature distribution of proposed fins is maximum compare to different profiles (Rectangular and triangular fins). Temperature distribution is about 50% more in case of proposed fins.

II. From ANSYS APDL simulation results, it is found that the flux is maximum in the case of triangular profile and minimum for proposed profile, while that of the rectangular profile lies in between the proposed and triangular profile. For AA6061 grade, thermal flux is maximum compare to AA2024.

III. It is examine that there is a decrease in the heat flux along the types of fins. It is found that the flux is maximum in the case of rectangular profile and minimum for proposed profile, while that of the rectangular profile lies in between the proposed and rectangular profile.

IV. It was found that, the proposed profile fins yield a lower tip temperature distribution than other as shown in the table comparing with the length of the fin and space optimization rectangular fin can give better output than the real sample.

7.2 Future Scope

The shape of the fin can be modified to improve the heat transfer rate and can be analyzed. The use of Aluminum alloy 6061 and AA2024 as per the manufacturing aspect is to be considered. By changing the thickness of the fin, the total manufacturing cost is extra to prepare the new component.

References

[1]. P. Sai Chaitanya, B. Suneela Rani, K. Vijaya Kumar, “Thermal Analysis of Engine Cylinder Fin by Varying Its Geometry and Material”,IOSR Journal of Mechanical and Civil Engineering, Page No. 315 – 323, Vol. No. 24, Issue No. 1, 2007.
[2]. Bharti Sharma, Satyendra Singh, Vinod Kumar, Ravi Kumar, “Geometrical Study of Fins Using ANSYS as a Tool”, IJRMET, Page No. 1111-1121, 2007.
[3]. Deepak Gupta, Wankhade S.R., “Design and Analysis of Cooling Fins”, IRD India, Page No. 881-884, 2009.
[4]. Shekhar Shrivastav, Abhisek Badhan, Harekrushna Acharya, K.C. Rath, “THERMAL ANALYSIS OF AISI 1030 CARBON STEEL THROUGH FEA TECHNIQUE”, Archives of Computational Materials Science and Surface Engineering, Vol. No. 1, Issue No. 2, Page No. 106-111, 2009.
[5]. Komal Singh Rajput, Rohit Rajvaidya, Prabhash Jain, “Design and optimization of heat sink used in IBM PC to increase Heat transfer rate” Journal of Materials Processing Technology, Vol. no. 140, Page No. 555–561, 2003.
[6]. Shekhar Shrivastav, Abhisek Badhan, Harekrushna Acharya, K.C. Rath, “Thermal Analysis of Aisi 1030 Carbon Steel Through FEA Technique”, Global Journal of Advanced Engineering Technologies, pp 129-136, olume 5, Issue 2-2016.
[7]. Mohamed H.A. Elnaggar, M.Z. Abdullah, M. Abdul Mujeebu, “Experimental analysis and FEM simulation of finned U-shape multi heat pipe for desktop PC cooling”, Energy Conversion and Management 52 (2017) 2937–2944.
[8]. Jong Bum Lee, Hyun Jung Kim and Dong-Kwon Kim, “Thermal Optimization of Horizontal Tubes with Tilted Rectangular Fins under Free Convection for the Cooling of Electronic Devices”, Appl. Sci., 2017, 7, 352.
[9]. Joel Hemanth, “Experimental, Mathematical and Finite Element Analysis (FEA) of Temperature Distribution through Rectangular Fin with Circular Perforations”, Modeling and Numerical Simulation of Material Science, 2017, 7, 19-32.
[10]. R. Arularasan, P.Hemanandhan, T.Thamizhselvan, B.Arunkumar, S.Senthilnathan, S.Prathap, “Modeling and simulation of engine cylinder fins by using FEA”, International Journal of Advanced Research (2016), Volume 4, Issue 1, 1391- 1396.
[11]. Bhoopendra Dewangan, S. A. K. Jilani, “Behaviour and Effect of Temperature over Pressure Tube of Nuclear Reactor”, International Journal of Computer Applications, Volume 53, No.11, 11-17, 2012.
[12]. P Vijaya sagar, K Kiran chand, “Thermal Analysis of an Automobile Radiator with and without Louvered Fins”, Proceedings of International Conference on Recent Trends in Mechanical Engineering, 2015.
[13]. Hossein Shokouhmand , Koohyar Vahidkhah, Mohammad A. Esmaeili, Numerical Analysis of Air Flow and Conjugated Heat Transfer in Internally Grooved Parallel-Plate Channels, World Academy of Science, Engineering and Technology International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol:5, No:1, 2011.
[14]. Fahanieh, B., Herman, C., Sunden B., Numerical and experimental analysis of laminar fluid flow and forced

convection heat transfer in a grooved duct, Int. J. Heat Mass Transfer, Vol. 36, 1993, pp. 1609-1617.

[15]. Bilen, K., Yapici, S., Heat transfer from a surface fitted with rectangular blocks at different orientation angle, Int. J. Heat Mass Transfer, Vol. 38, 2002, pp. 649-655.

[16]. Herman, C., Kang, E., Heat transfer enhancement in a grooved channel with curved vanes, Int. J. Heat Mass Transfer, Vol. 45, 2002, pp. 3741-3757.

[17]. O. Manca, S. Nardini, D. Ricci, A two-dimensional numerical investigation on forced convection in channels with transversal ribs, ASME Conference Proceedings,(43826), pp. 1099-1107,2009.

[18]. H. Shokouhmand, K. Valiidkhah, M.A. Esmaeili, Numerical analysis of air flow and conjugated heat transfer in internally grooved parallel-plate channels, World Acad. Sci. Eng. Technol. 73 (2011).

[19]. R. Kamali, A.R. Binesh, The importance of rib shape effects on the local heat transfer and flow friction characteristics of square ducts with ribbed internal surfaces, Int. Commun. Heat Mass Transf. 35 (8) 1032-1040, (2008).

[20]. P. Promvonge, C. Thianpong, Thermal performance assessment of turbulent channel flows over different shaped ribs, Int. Commun. Heat Mass Transf. 35 (10) 1327-1334, (2008).

[21]. C. Thianpong, T. Chompookham, S. Skullong, P. Promvonge, Thermal characterization of turbulent flow in a channel with isosceles triangular ribs, Int. Commun. Heat Mass Transf. 36 (7) 712-717, (2009).