Use of "Thermal Boundary Layer Disruption" As Heat Transfer Enhancement Method

Mr. Dhaval Solanki[1] Prof. Viveck Joshi[2]
[1]P.G Student [2]Assistant Professor
[1,2]Department of Mechanical Engineering
[1,2]Parul Institute of Engineering and Technology, Vadodara

Abstract— Present research paper suggests geometrical modification methods to improve heat transfer rate of automobile air cooled engine fins by disruption of thermal boundary layer. Thermal boundary layer plays vital role in heat transfer from fins and act as barrier in convective heat transfer. So many methods and designs of geometry available to break thermal boundary layer, use of vortex generator, perforation on fin surface, and use of wavy fins are some of examples. Numerical simulation work has been carried out on combination of louvered shape and vortex generator geometries, combination of both geometries here are used on engine fin in order to enhance heat transfer by phenomenon of thermal boundary layer disruption.

Key words: Thermal Boundary Layer, Disruption, Vortex Generator, Louvered Shape, Pitch, Angle Of Attack, Heat Transfer Co-Efficient

I. INTRODUCTION

Automobile sector grows rapidly and subjected with an expected problem of fossil fuel crises, as a result of such problems researches rush towards alternative power trains and alternative fuels and its development in the existing technology is also that much important. To reduce fuel consumption rate and to increase performance rather than diverted towards the new power trains and fuels we are still able to develop conventional Internal combustion engines. Due to its compactness and light weight, air cooling system is still used in automobile engines. Using air as a cooling media and convection heat transfer phenomenon, it is cheaper to use and less maintenance requirement then liquid cooling system which is comparable complex and bulky. Still there are limitation regarding lower cooling rate of air cooled engine in high speed bikes and vehicles. Fins are not that much effective as liquid cooled system, to improve heat transfer rate of air cooled engine and to overcome problems regarding heat transfer rate researchers have done so many researches with fin materials and geometry.

Thermal boundary layer is one of the barrier which affect heat transfer enhancement and as thermal boundary layer growth and its thickness increases heat transfer rate decreases. Besides of fin material this is most affecting parameters, it is necessary to break thermal boundary layer formed on the surface of fin in order to enhance heat transfer rate. Present research deals with the combination of two different geometries named louvered fins and vortex generator on the engine instead of conventional straight fins. Numerical simulation work has been shown here at different geometrical parameter values, and behaviour of heat transfer with variation of these parameters also shown here.

II. EFFECT OF THERMAL BOUNDARY LAYER ON CONVECTIVE HEAT TRANSFER

The region of the fluid next to the surface in which energy exchange is occurring is known as thermal boundary layer. Thermal boundary layer develops if the free stream and surface temperatures differ. At the leading edge, the temperature profile is uniform, with $T(0,y) = T\infty$. Fluid particles that come into contact with the plate achieve the plate's surface temperature, Ts. In turn, these particles exchange energy with those in the adjoining layer, and the temperature gradients develop in the fluid.

The region of the fluid in which these temperature gradients exist is the thermal boundary layer, and its thickness δt is typically defined as the value of y for which the ratio $[(Ts-T)/(Ts-T\infty)] = 0.99$. With increasing distance from the leading edge, the effects of heat transfer penetrate further into the free stream, and the thermal boundary layer grows in a similar manner as does the hydrodynamic boundary. At any distance x from the leading edge, the local heat flux may be obtained by applying Fourier's law to the fluid at y =0 in terms of the thermal conductivity of the fluid, k, and the temperature gradient at the surface.

$$q''_s = -k\frac{\partial T}{\partial y}\bigg|_{y=0}$$

This expression is appropriate because at the surface, the fluid velocity is zero (no-slip condition) and energy transfer occurs by conduction. Recognize that the surface heat flux is equal to the convective flux, which is expressed by Newton's law of cooling. By combining the foregoing equations, we obtain an expression for the local convection coefficient

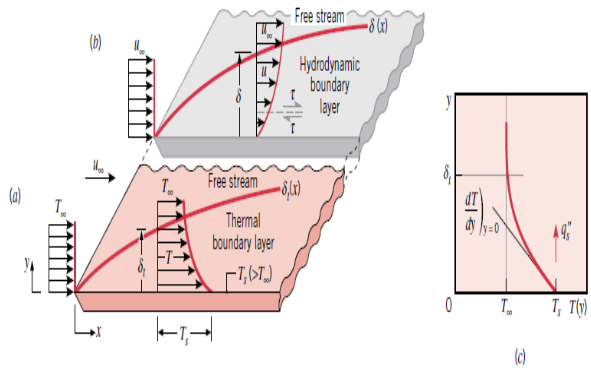


Fig. 1: Fluid with the uniform free stream velocity u_∞ temperature T_∞ in laminar flow over a flat plate with uniform temperature T_s ($T_s > T_\infty$).

$$q''_s = q_{con} = h_x(T_s - T_\infty)$$

$$h_x = \frac{-k\partial T/\partial y|_{y=0}}{T_s - T_\infty}$$

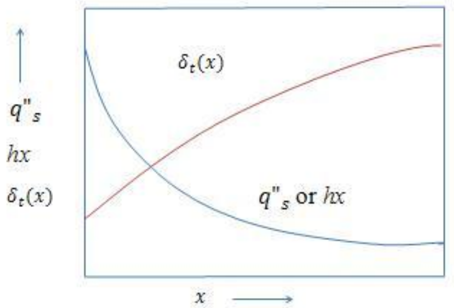


Fig. 2: Heat transfer co-efficient, heat flux, and boundary layer thickness varies with longitudinal distance x.

At some distance from the leading edge, small disturbances in the flow are amplified, and transition to turbulent flow begins to occur. Fluid motion in the turbulent region is highly irregular and characterized by velocity fluctuations that enhance the transfer of energy. Due to fluid mixing resulting from the fluctuations, turbulent boundary layers are thicker. Accordingly, the temperature profiles are flatter, but the temperature gradients at the surface are steeper than for laminar flow. Consequently, the local convection coefficient is larger than for laminar flow, but to decrease with *x* as shown in figure.

Thus from above discussion it is clear that thermal boundary layer is as its thickness increases reduces convective heat transfer co efficient which overall decrease heat transfer rate and act as an obstacle in convective heat transfer. To increase heat transfer in convective viscous flow it is necessary to break thermal boundary layer for the given surface flow.

III. PAST RELEVANT STUDIES

J.Cui and D.K.Tafti[1] found that with three dimensional multilouvered fin geometry with three different region angle louvered, transition region and flat region. Heat transfer enhancement shows increase 50% at the junction due to strong acceleration, when compared to the geometry in which angled louver-extends all the way to the tube surface.

Xiaoze Du, LiliFeng et al [2] studied longitudinal vortex generator punched on wavy finned flat tube Heat exchanger. They have used four types of longitudinal vortex generator and studied numerically with experimental verification. The delta winglet pair is the best longitudinal vortex generator for air side heat transfer enhancement among the four configurations. The average PEC (Performance Evaluation Criteria) of the delta winglet pair with angle of attack 250 can reach to 1.23 while the inlet air velocity varies from 1 m/s to 5 m/s.

Paul A. Sanders, Karen A. Thole[6]The experimental study presented in this paper concentrates on augmenting the heat transfer along the tube wall of the compact heat exchanger through the use of winglets placed on the louvers. The experiments were completed on a 20 times scaled model of an idealized louvered fin exchanger with a fin pitch to louver pitch ratio of 0.76 and a louver angle of 270. The Reynolds numbers tested, based on louver pitch, were between 230 and 1016. Heat transfer augmentation increased with increasing angle of attack, increasing winglet size (decreasing aspect ratio in this case), and with decreasing winglet distance from the wall.

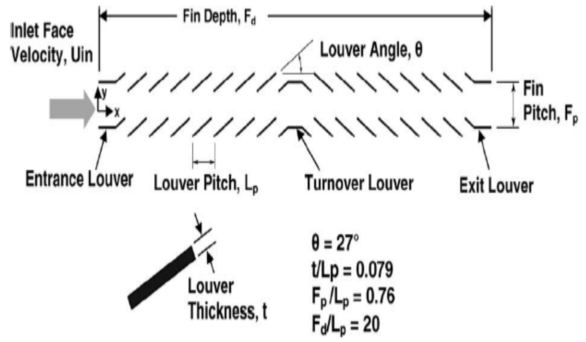


Fig. 3: Louvered array geometry and definitions used in this study

Geometrical parameters and configuration of louvered fins are showed in figure 3 and placement of winglet on surface of fin and its configuration are showed in figure 4. For an angle of attack of 200, the results showed little augmentation; however, angles of attack greater than or equal to 300 performed very well. An aspect ratio of 3 did not provide much benefit while an aspect ratio of 1.5 produced high augmentations. The best heat transfer augmentation was found with rectangular winglets giving 38%, 36%, and 3% at Reynolds numbers of 1016, 615, and 230, respectively

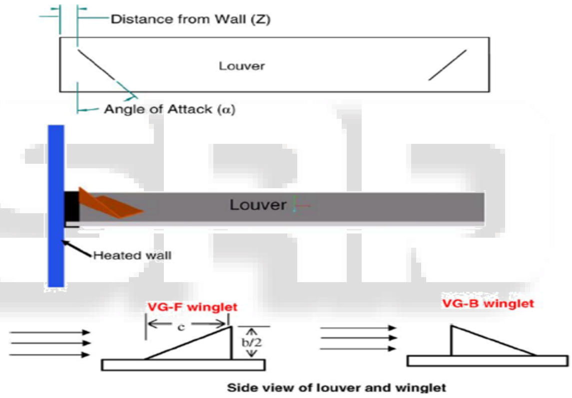


Fig. 4: Definition various winglet parameters

IV. NUMERICAL INVESTIGATION

Numerical simulation has been carried out on four louvered fin geometry models made in Creo parametric 2.0. Four models include straight rectangular fin, straight circular fin, Louvered fin array, and louvered fin array with vortex generator. Numerical simulation has been performed on these models with the help of ANSYS 15.0 Fluent tool. Instead of full engine assembly, half cylindrical shape with two fin array was used for simulation in order to decrease computational burden on hardware. As shown in figure louvered fin array consists one flat fin at upstream flow and one at downstream flow in order to stabilize flow and domain selected for air has been extended up to 200 mm for rear and front to avoid air recirculation. Louvered array consists one turn over louver in middle region of array which allows the fluid flow to change its direction. Total four flat fins, eight louvered fins, and two turnover fins are used in simulation work in two fin array.

A. *Governing Equations*

To examine the mathematical consequences of the boundary layer concept we begin with the governing equations for a simplified case. We make the following assumptions: (1) steady state, (2) two-dimensional, (3) laminar, (4) uniform properties, (5) no dissipation, and (6) no gravity. Based on these assumptions the continuity, momentum, and energy equations are

1) *Continuity equation:*

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

2) *Momentum equation for x direction:*

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial u}{\partial x}+\vartheta\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right)$$

3) *Momentum equation for y direction:*

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial v}{\partial x}+\vartheta\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)$$

4) *Energy Equation:*

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

These equations will be simplified using the boundary layer concept.

B. Geometrical Models

Geometry of louvered shape fin array on semi-circular cylindrical shape is created in ANSYS geometry module using design parameters list down in table, all parameters are in mm and geometry contains basic louvered shape fin array on semi cylindrical shape with vortex generator on its surface. Vortex generators are of delta winglet type.

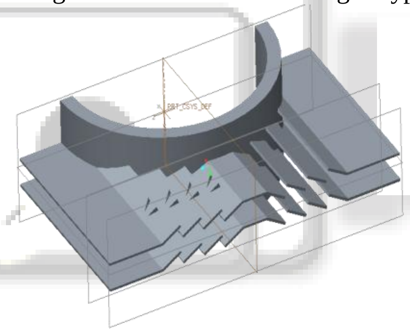


Fig. 5: Louvered fin Model (With vortex generator)

C. Boundary Conditions and Meshing

In this case tetrahedron mesh is selected which contains unstructured mesh elements for open air domain which encloses the geometry as well as fin geometry. Care should be taken to use the unstructured tetrahedron cells to an extent as high as possible. Meshing of all four models have been done with tetrahedron patch conforming method with proximity and curvature which generates 29,35,394 elements and 44,86,294 nodes. And with maximum skewness 0.85, which is within the limit of 0.98. Statistic of elements and skewness shows a good quality of mesh.

For the given problem inlet and outlet conditions are given as per problem requirement of forced convection. Boundary conditions given to encloser and geometry wall are as listed in bellow table.

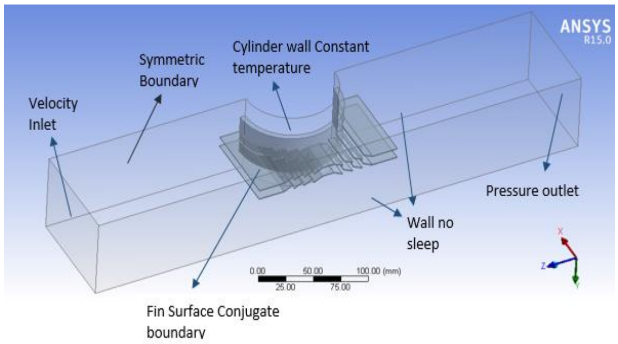


Fig 6: Boundary Condition Domain and fin

Boundary condition	**Parameter**	**Surface/ Direction**	**Value**
Inlet	Velocity,	Flow stream direction. Air inlet	10,15,20,25 m/s
	Temperature		27^0C (300K)
	Turbulence	Air Domain	5% Medium
Outlet	Gauge Pressure	Outer face	0 Pa
Wall	No sleep	Top, side, bottom and geometry surface	-
Base temperature	Temperature	cylinder wall	727^0C (1000K)

Table 1: Boundary Conditions (Thermal and Hydrodynamic)

D. Problem Setup

The type analysis is forced convective heat transfer type. The velocity and pressure are changed to absolute and heat is transfer from constant temperature semi cylinder wall to fins by conduction and from fins to it transfer to surrounding air by forced convection. Energy is set - ON position in Fluent Setup. As we prefer to analyze turbulent flow, so the viscous model is selected: "k-ε" model (2 equations).

Material properties have been changed to the properties of Aluminum alloy 6061 which is used as a fin material in engines in most of two wheelers. Second order upwind scheme used to calculate convective flow on boundary surface of control volume. This second order scheme is least sensitive to mesh structure.

V. RESULTS AND DISCUSSION

Simulation work has been carried out for two fin arrays which are mounted on semi-circular base which here act as an engine base. Simulation work has been done on conventional straight rectangular and circular fin model at varying wind velocity from 10 m/s to 25 m/s which is equivalent to 70- 80 km/h speed of bike. Simulation work carried out on louvered fin model without vortex generator for varying louvered angles between18^0-33^0 for given velocity range as mentioned above. Model with vortex generator are tested for varying angle of attack of winglet pair between 25^0-45^0 for aspect ratio 1.5, 2, and 3 for winglet pair.

A. Results Comparison for Varying Louvered angles With flat Rectangular and Circular Fins

Graph shows that up to velocity of 20 m/s all louvered angle gives same heat transfer increment, but after 20 m/s velocity all angle gives change in heat transfer rate which is in lower range of deviation with each other. Among these entire angles 30^0 Louvered fin shows maximum deviation and gives heat transfer rate maximum from cylinder wall. This is 33% augmentation on rectangular fin, 43.53 % over circular fin and 16.17% over other louvered angle fins.

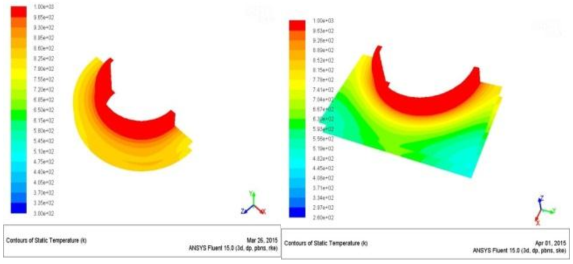


Fig. 7.Temperature contours for rectangular and circular fin array

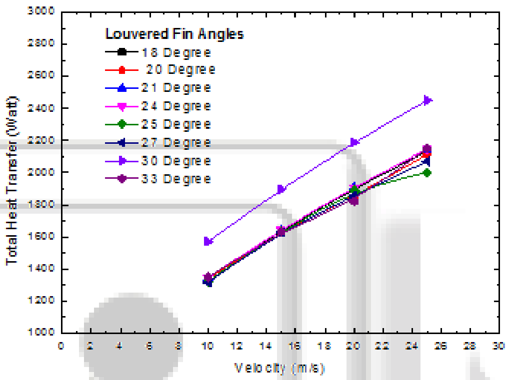
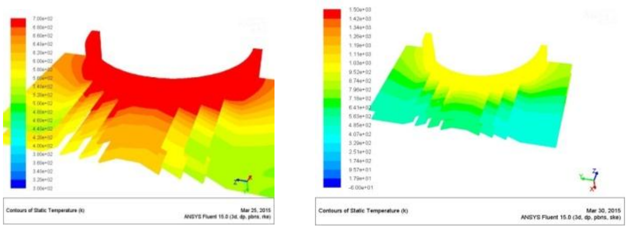


Fig. 8: Total Heat transfer V/S Velocity for given louvered angles.

B. Results Comparison for Vortex Generator and its Angle of Attack

Comparison for vortex generators varying angle of attack shows no major deviation in heat transfer rate for all three aspect ratio 1.5, 2, 3. At the same time if we compare the results of vortex generator louvered array and without vortex generator louvered array we found increase in heat transfer after placing vortex generator, this is due to disturbances generation in boundary layer flow by vortices generation which as a result cause thinning of thermal boundary layer, which in normal condition act as a barrier in convective heat transfer.



(a) (b)

Fig. 9: Temperature contours for louvered fin array a. without vortex generator and b. with vortex generator

Fins with winglet pair having aspect ratio 2 and 3 shows 7.21% higher heat transfer from base as compare to having aspect ratio1.5. But aspect ratio 2 and 3 shows same results for all given velocities and angle of attack. And louvered fin with vortex generator shows 7.45 % augmentation over louvered fins without vortex generator at 25 m/s air velocity.

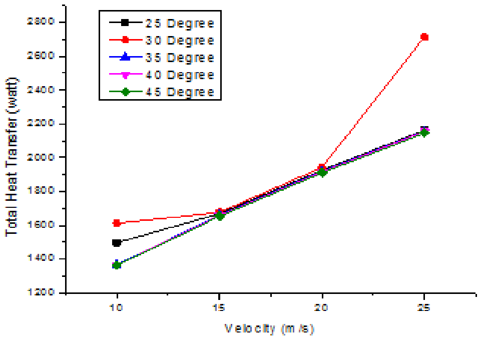


Fig. 10. Total Heat Transfer for 25,30,35,40 and 45 angle of attack of VG having Aspect Ratio 1.5

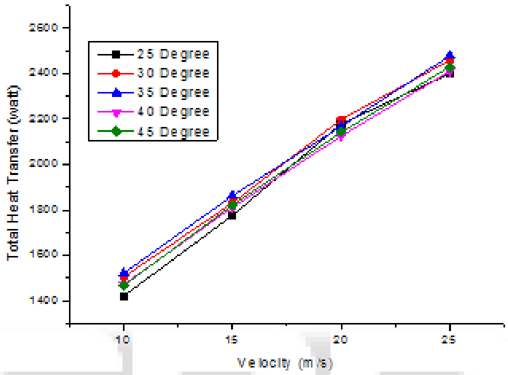


Fig 11. Total Heat Transfer for 25,30,35,40 and 45 angle of attack of VG having Aspect Ratio 2

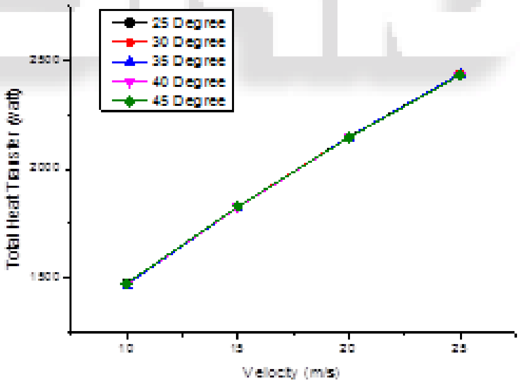


Fig 12: Total Heat Transfer for 25,30,35,40 and 45 angle of attack of VG having Aspect Ratio3

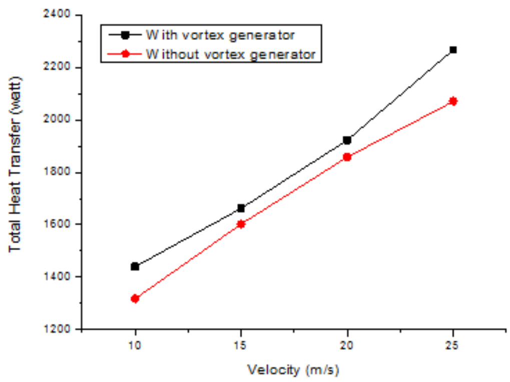
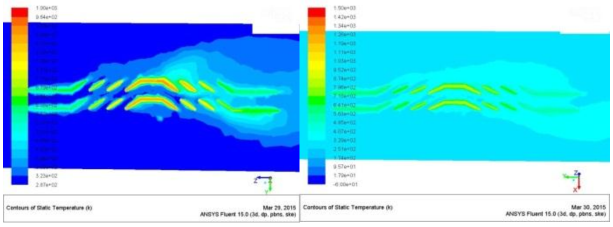


Fig. 13. Total heat transfer verses velocity for louvered fin with VG and without VG.



(a) (b)

Fig. 14. Heat dissipation temperature contours a. without vortex generator and b. with vortex generator

C. Results comparison for Aspect ratio 1.5, 2 and 3

Results for vortex generator as discussed above for different angle of attack it shows no much variation, but same results if we compare for aspect ratio 1.5, 2 and 3, aspect ratio 2 and 3 gives 7.21 % heat transfer augmentation over aspect ratio 1.5, Which is showed in bellow.

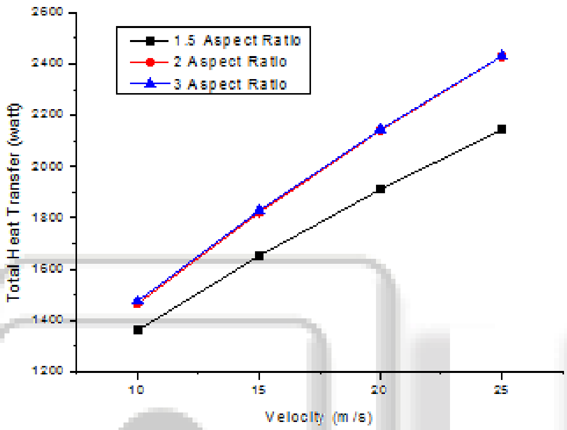


Fig. 15: Total heat transfer for aspect ratio 1.5, 2 and 3

D. Comparison of obtained simulation results with practical results

While comparing baseline results (without vortex generator) with the baseline results of Paul A.Sendars as shown in graph it gives very huge deviation at leading and trailing edge of fin and moderate deviation at middle part. Difference in fin thickness, fin material, number of fins, base design from flat to the circular as well as change in air velocity for inlet are the responsible factor for these deviation in Nusselt number.

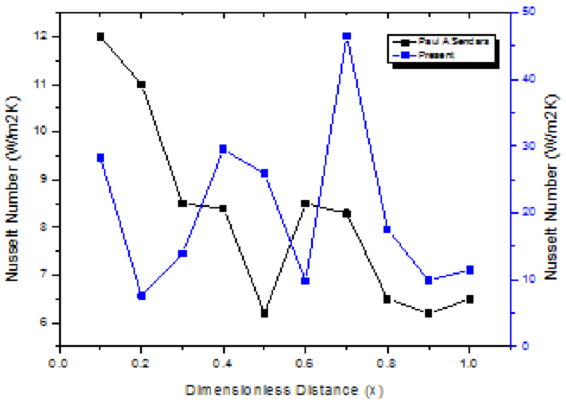


Fig. 16: Baseline Result Comparison with Results of Paul A. Sendars

VI. CONCLUSION

From above study we can say that heat transfer enhancement can be achieved at promising level by thermal boundary layer disruption method. There are so many other methods are available to disrupt boundary layer including wavy fins, perforation on fins, slit fins, and many other disturbances methods which can generates vortices and eddies as well as methods like perforation and slit fins can suck down the generated boundary layers, thus by creating more mixing, disturbances, and as a result thinning thermal boundary layer we can improve heat transfer rate.

REFERENCES

[1] J.Cui and D.K.Tafti, "Computations of Flow and Heat transfer in a three-dimensional multilouvered fin geometry", International Journal of Heat and Mass Transfer 45 (2002) 5007–5023.

[2] Xiaoze Du, LiliFeng, Li Li, Lijun Yang, Yongping Yang, "Heat transfer enhancement of wavy finned flat tube by punched longitudinal vortex generators", International Journal of Heat and Mass Transfer 75 (2014) 368–380

[3] B Ameel, J Degroote, H Huisseune, P De Jaeger, J Vierendeels, M De Paepe a paper on "Numerical Optimization of Louvered fin Heat Exchanger with Variable Louver Angles" Department of flow, heat and combustion mechanics, Ghent University, Belgium.

[4] Jiin-Yuh Jang and Chun-Chung Chen, "Optimization of the louver angle and louver pitch for a louver finned and tube heat exchanger" International Journal of Physical Science, Vol.8(43) pp.2011-2022,23 November 2013

[5] Manjunath S N, S.S.Mahesh Reddy, Saravana Kumar "Numerical Investigation of Automotive Radiator Louvered Fin Compact Heat Exchanger" International Journal of Mechanical Engineering and Technology (IJMET) Volume 5, Issue 7, July (2014), pp. 01-14

[6] Paul A. Sanders, Karen A. Thole. "Effects of Winglets to Augment Tube Wall Heat Transfer in Louvered Fin Heat Exchangers" International Journal of Heat and Mass Transfer 49 (2006) 4058–4069

[7] P. Ebeling, K.A. Thole, "Measurements and predictions of the heat transfer at the tube-fin junction for louvered fin heat exchangers", Int. J. Compact Heat Exchange. 5 (2004) 265–286.

