MAXIMISING THE HEAT TRANSFER THROUGH FINS USING CFD AS A TOOL

Sanjay Kumar Sharma[1] and Vikas Sharma[2]

[1,2] Assistant Professor, Department of Mechanical Engineering, Gyan Vihar University, Jaipur, Rajasthan, India

sanjaybagra84@gmail.com, vikasmeche@gmail.com

ABSTRACT

This study presents the results of computational numerical analysis of air flow and heat transfer in a light weight automobile engine, considering three different morphology pin fins. A numerical study using Ansys fluent® (Version 6.3.26) was conducted to find the optimum pin shape based on minimum pressure drop and maximizing the heat transfer across the Automobile engine body. The results indicate that the drop shaped pin fins show improved results on the basis of heat transfer and pressure drop by comparing other fins. The reason behind the improvement in heat transfer by drop shape pin fin was increased wetted surface area and delay in thermal flow separation from drop shape pin fin.

KEYWORDS

CFD, Continuum Type, FLUENT, Optimization, Simulation, Turbulence

1. INTRODUCTION

Performance of various devices are based on heat transfer and widely used in the many industries, especially in power distribution sector (transformers), Automobile sector (engine cooling), Power Plant Sector, electric components, space industry etc.

One of the useful methods to take away heat transfer from surface area of thermal device was extended surface or fins. Pin fin is suitable for numerous applications including heat transfer removal from air cooled I C engines, Electrical Small Transfers etc.

"Pin fin geometry highly affects the different heat exchangers efficiency although these devices are used in various industries. Drop shaped pin fins can show more heat transfer with lower pressure drop from system and it was used for heat exchange purpose from past decades."
In past this type of research work was based on experimental study, but having large technical and financial issues which was overcome by use of CFD techniques. A computational study was performed by various researchers using commercial software's to find out optimal shaped fins. Various researchers considered heat transfer and pressure drop across the thermal devices surface area. CFD analysis follow top to bottom procedure to perform simulation for any type of research problems. The first step is known as pre-processing, in which geometry making, mesh generation and boundary conditions of particular problem were defined by user. The heat transfer and associated pressure drop behaviour are characterized by second step known as solution of problem statement made in first step. To find optimum shape or performance of any thermal device third step was very useful because in this step post processing of results was performed and conclusion was made by researches.

The objective of this study was to find out optimum type of fins used for heat removal application for automobile engine. This task was performed by using CFD as a tool. Three basic shapes of pin fins will be used in this study to find best shape. Maximising the heat transfer and minimising the pressure drop will be main criteria for selection of optimum pin fin.

2. GEOMETRICAL MODELLING AND MESH GENERATION

Methodology:-

For CFD simulation, first of all geometry of the wind duct was created using GAMBIT (a software). After geometry creation next step is to mesh the geometrical model, which was also done using GAMBIT. Next step in GAMBIT is to declare continuum type and boundary type for the surfaces generated. Finally a mesh file is created, which is imported in FLUENT.
After importing mesh file in FLUENT, dimensional units for CFD domain are specified. In FLUENT desired turbulence model was selected for viscous modelling on the basis of literature review. After selection of turbulence model boundary conditions are specified. Fluent has capability to store value of physical parameters for any point in the domain for analysis. Seven points were created to store the value physical parameters such as temperature, velocity and pressure.

FLUENT is now ready to simulate flow problem. Simulation was done for unsteady mode. Finally, post processing was done for result analysis.

3. DESCRIPTION OF DUCT AND TEST POINTS

3.1 Size of Duct

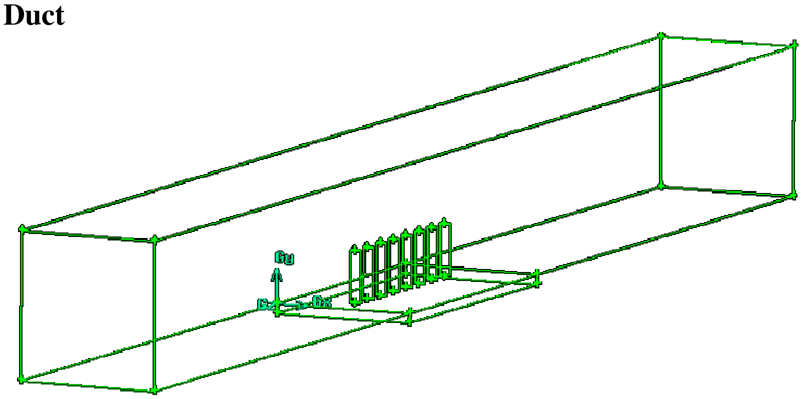

Figure 1 Diagram of Test Ring in CFD

3.2 Geometric modelling

Geometry: geometry generation is first step for making CFD domain. In gambit we can create both 2-D and 3-D shapes. In this case 3-D geometry of the duct was created.

In most of the problems shape of the domain is very complex. Some special operations are given in geometry mode to model complex geometries.

The most significant operations are: unite, subtract, split, move, copy, align, rotate, translate etc. Figure 1 shows 3-D geometry used in our case. Furniture has been subtracted from main body of the room.

3.3 Meshing of CFD domain

After making geometry of the CFD domain, next step is to mesh the domain. To perform better results using CFD tool it was mandatory to use better quality of mesh. In fluent this task was done in GAMBIT software, where various tools are available to complete this task like mesh healing, dynamic refinement, aspect ratio etc. In this study various parameters, such as aspect ratio, internal angle, face war-page, right handedness, negative volumes, cracks, and tetrahedral quality were used by authors, but in this paper only limited results were shown.

Mesh sizes were kept different for zones. In Gambit, for meshing hexahedral element with sub-map scheme was selected. No boundary layer was created in this case. Figure 2 represent meshed domain with hexahedral type meshing.

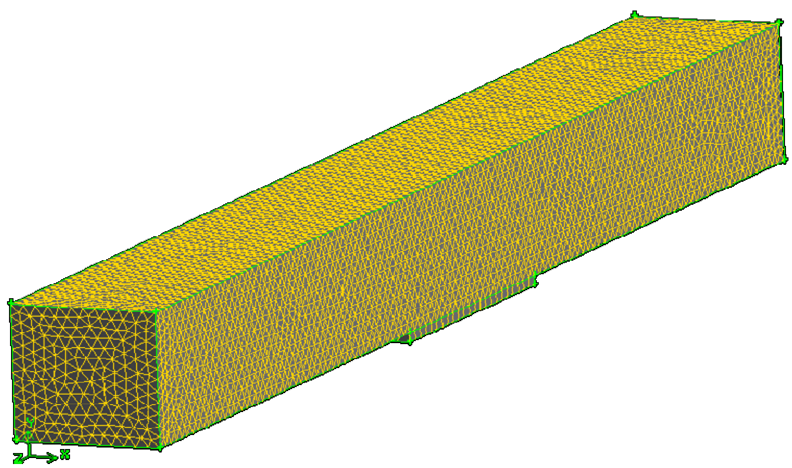

Figure 2 Meshed Domain

After mesh generation quality of mesh was checked in Gambit. Table 1 shows quality parameter like equisize skew. Here "from value" to "to value" represents quality parameter. Zero represent best and one represent worst element of grid. Figure 3 shows quality parameter aspect ratio, which shows that quality of the grid generated is good.

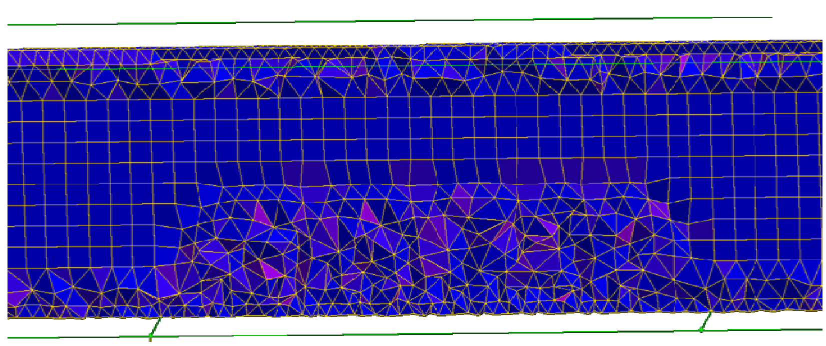

Figure 3 Aspect Ratio of Domain

Another Figure 4 shows that of the view of grid in Y-Z plane.

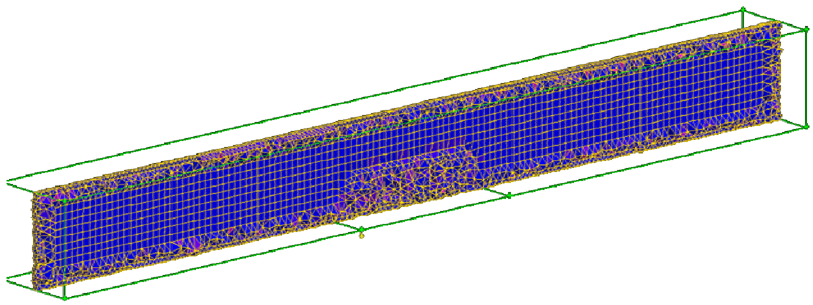

Figure 4 View of Grid in Y-Z Plane

Table 1 Equi-size skew in percentage

From value	To value	Count in range	Total count (in %)
0	0.1	28964	16.19
0.1	0.2	13512	7.55
0.2	0.3	30378	16.98
0.3	0.4	61324	34.27
0.4	0.5	22278	12.45
0.5	0.6	14135	7.90
0.6	0.7	6205	3.47
0.7	0.8	2145	1.20
0.8	0.9	0	0.0
0.9	1.0	0	0.0

The worst element for equi-size skew has a quality value of 0.6 to 1.0. This is ok, because only some elements are in the worst elements range.

3.4 Boundary Conditions for Domain

After mesh generation boundary conditions are defined for CFD domain. This process is done in Gambit. “Specify boundary type” icon is used to create boundaries. Gambit can be used to make mesh files for many different CFD Softwares. In this case FLUENT 6 was selected. After boundary creation next and last step is to define continuum type. In Gambit both fluid and solid continuum type can be defined. As we are studying air flow, fluid continuum type was selected.

4. CFD SIMULATION

In this work FLUENT software is used for simulation. Main focus of this work is on heat transfer analysis of a duct for different types of Pin Fins. In CFD simulation selection of turbulence model is an important issue. Although in most of the research papers STD k-ε turbulence model is used for building simulation but k-ω SST show better results.

Governing Equations and turbulence modelling

The governing equations for fluid dynamics are conservation equations for mass, momentum, and energy. The Governing Equations have actually been known for over 150 years. In the 19th century two scientists, Navier and Stokes described the equations for a viscous, compressible fluid, which are now known as the Navier-Stokes Equations. These equations form a set of differential equations. The generic form of these relationships follows the advection diffusion equation:

$$\partial/\partial t\ (\rho\phi)+\mathrm{div}(\rho\nabla\phi-\Gamma\_\phi\ \mathrm{grad}\phi)=S$$

4. RESULT AND DISCUSSION

By completion of all the test runs in Fluent, several key performance indicators were studied to understand the heat transfer characteristics and trends for each pin-fin configuration. To understand results we study Temperature based results in graphical mode, Velocity results and Pressure based results.

4.1 Temperature Contour Results

Figure 5 to figure 19 provides a temperature contour like the previous figures, but in a vertical surface (from x-plane). Here it is clearer that heat transfer coefficient (total temperature) is higher between drop-shaped fin arrays. Heat transfer performance is based on turbulence effect created by fins shapes. The other important point is that because of high conductivity of material for both base of device and extended surface which an enhancing parameter is for heat transfer.

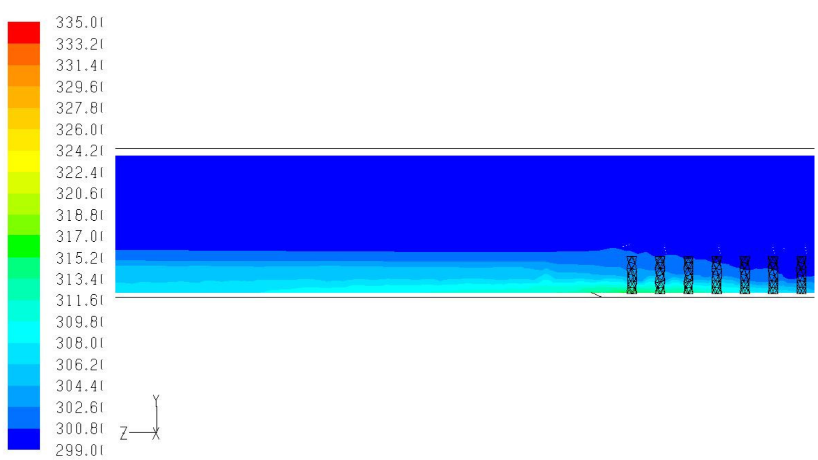

Figure 5 Temperature Contours of Cylindrical fin at plan x=25mm

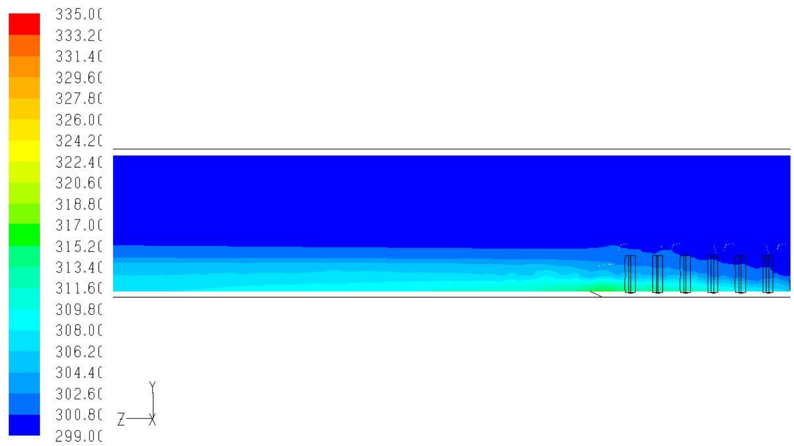

Figure 6 Temperature Contours of Drop fin at plan x=25mm

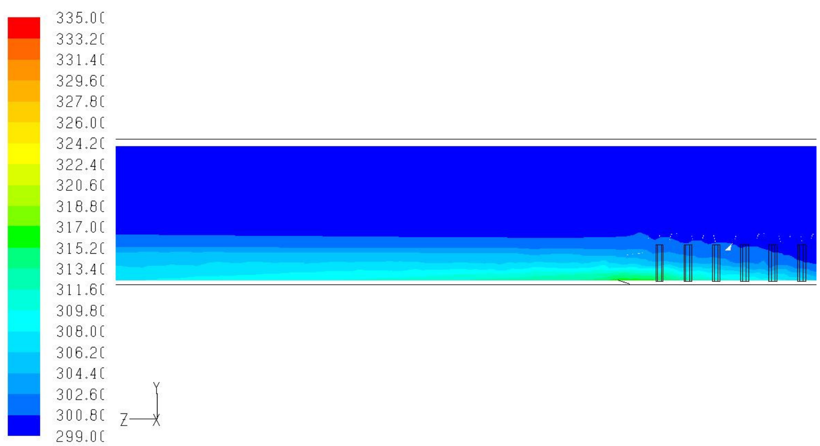

Figure 7 Temperature Contours of Rectangular fin at plan x=25mm

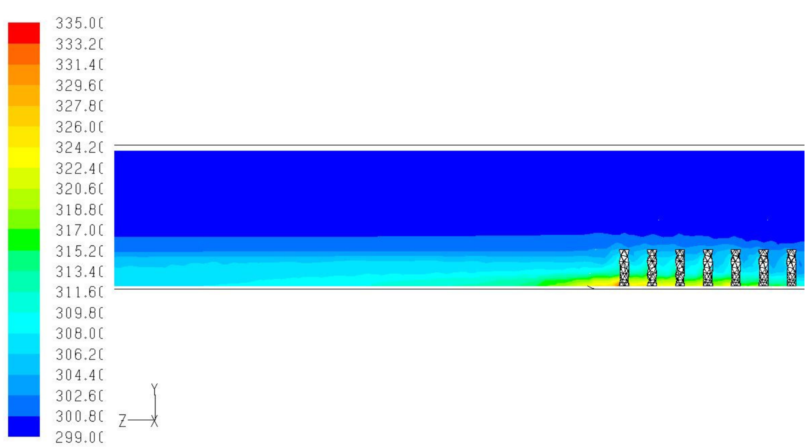

Figure 8 Temperature Contours of Cylindrical fin at plan x=37.5 mm

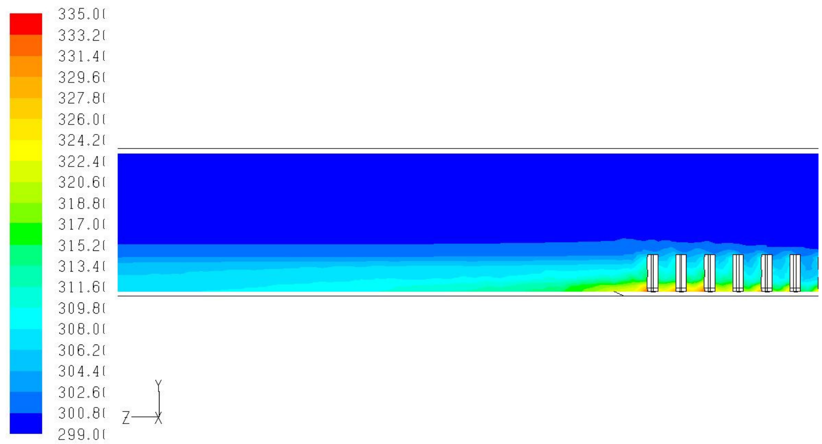

Figure 9 Temperature Contours of drop fin at plan x=37.5 mm

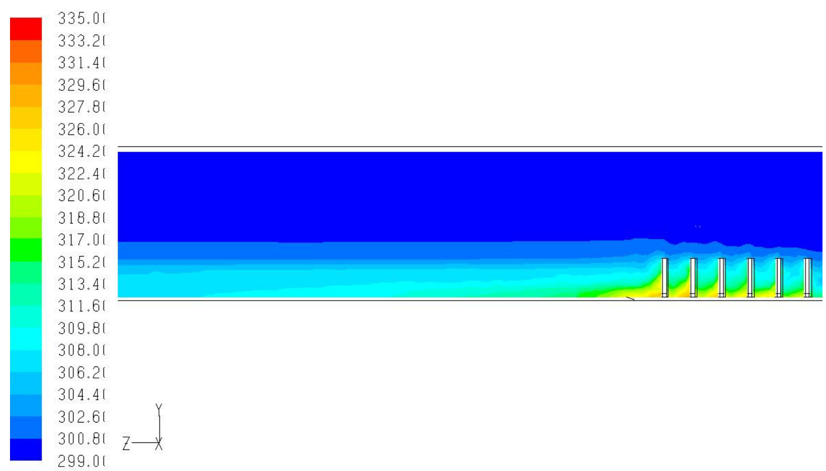

Figure 10 Temperature Contours of rectangular fin at plan x=37.5 mm

Figure 11 to figure 16 provides a contour plot for two virtual plane in y direction to show change in temperature profile for various fins used in this study. It is evident that the fins are affecting this temperature distribution. In the drop-shaped fins, thermal flow has reached the complete developed mode, more quickly than the other fins. The heat transfer coefficient has the highest value for the drop-shaped fins and the lowest value in the rectangular fins, because of geometry shapes and surface area of fins. The reason is that there is strong recirculation flow between fins in the rectangular fins. This recirculation reduced in the cylindrical and drop-shaped fins. Recirculation flow acts as a wall preventing the fresh air contributes in heat transfer theory.

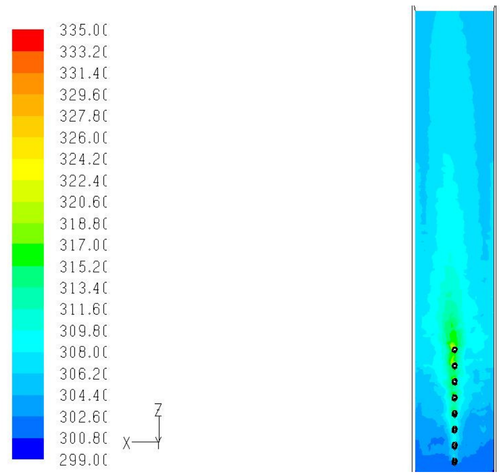

Figure 11 Cylindrical fin count temp at Y=5mm

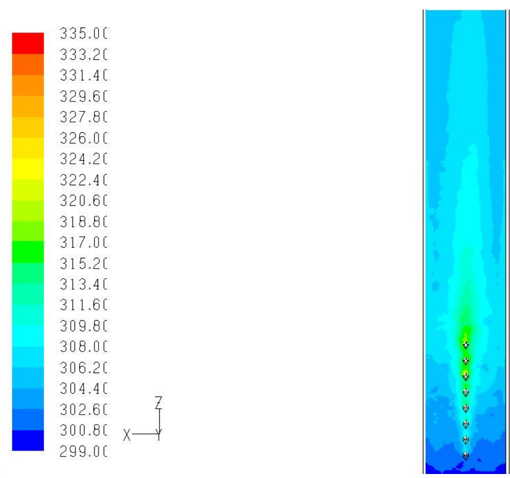

Figure 12 Drop fin count temp at Y=5mm

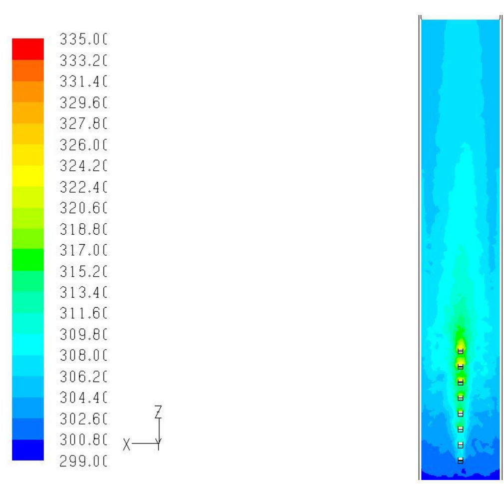

Figure 13 Rectangular fin count temp at Y=5mm

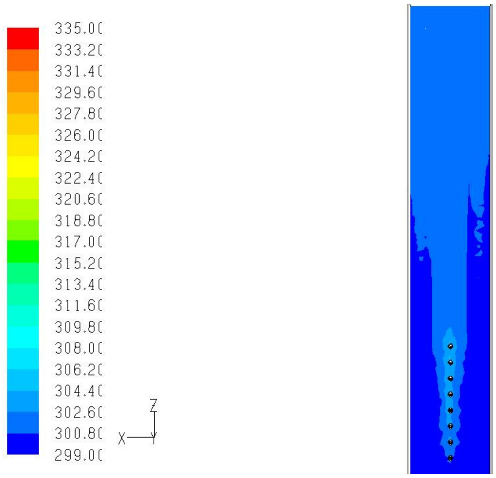

Figure 14 cylindrical fin count temp at plan Y=20mm

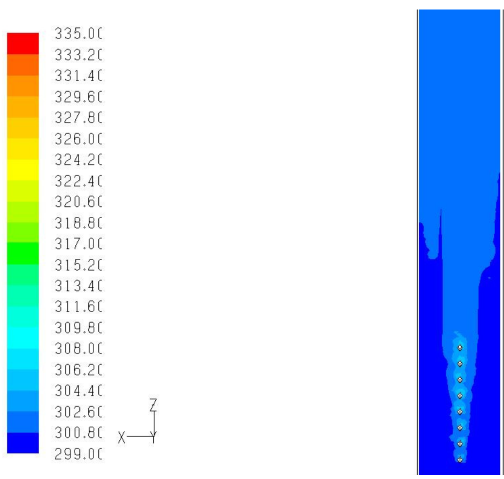

Figure 15 Drop fin count temp at plan Y=20mm

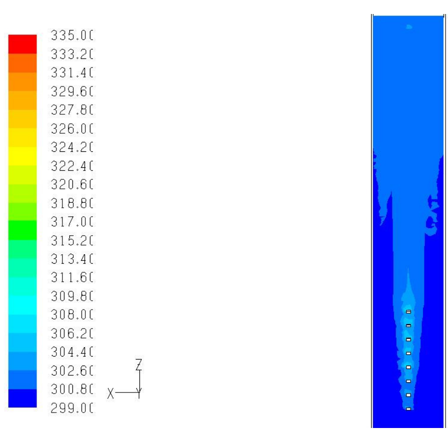

Figure 16 Rectangular fin count temp at plan Y=20mm

Figures 17 to 19 consist of different plots showing the temperature distribution in different parts of solution. From the figure 17, the temperature distribution seems close for all three cases used in this study; where there is a small difference between outlet temperatures occurs. The difference is less than 1°C.

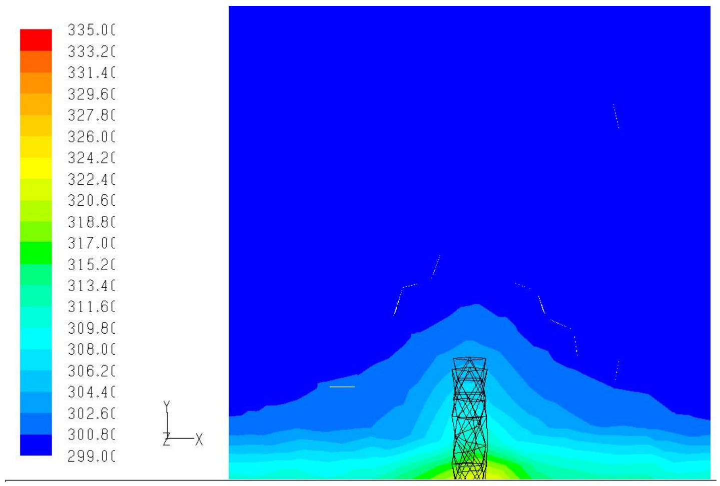

Figure 17 Temperature Contours of cylindrical fin at plan Z=38 mm

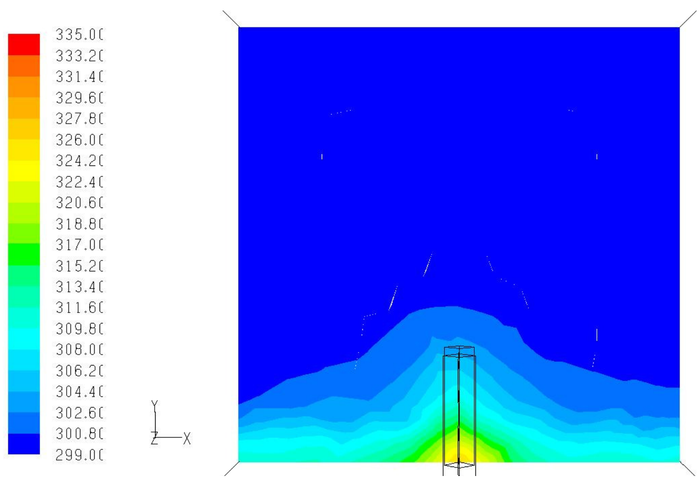

Figure 18 Temperature Contours of drop fin at plan Z=38 mm

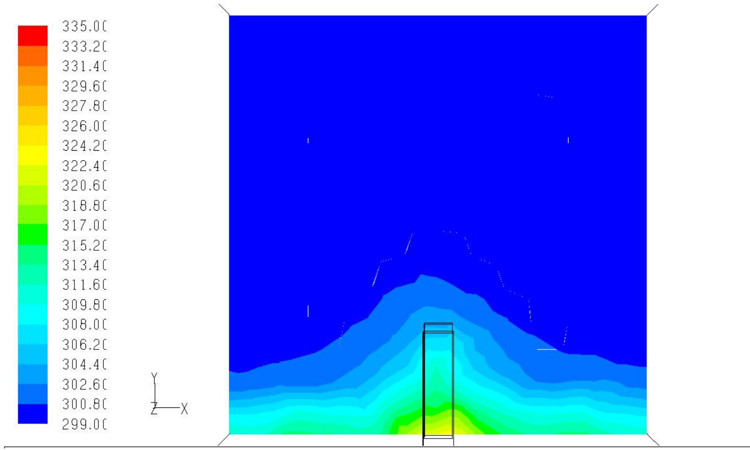

Figure 19 Temperature Contours of rectangular fin at plan Z=38 mm

4.2 Pressure plots (inlet outlet)

It is seen that the pressure loss is higher in the rectangular pins which was shown in figure 20 to figure 25 when comparing with all case used in this study. The drop-shaped fins have the least pressure loss. The reason is that in rectangular fins case, air flow particles follow a smoother path line structure. Figure 20 to Figure 25 shows Total pressure plots on inlet and outlet conditions.

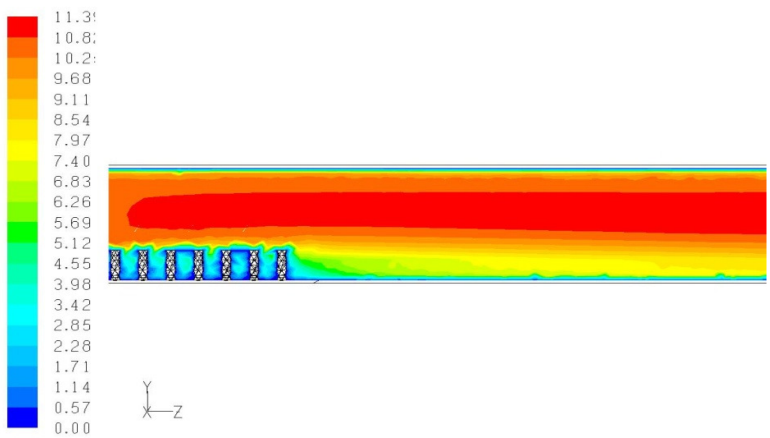

Figure 20 Velocity Plot for Cyl Pin Fin

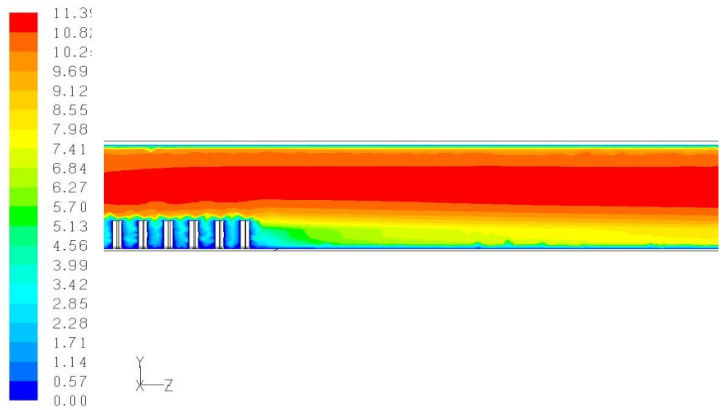

Figure 21 Velocity Plot for Drop Pin Fin

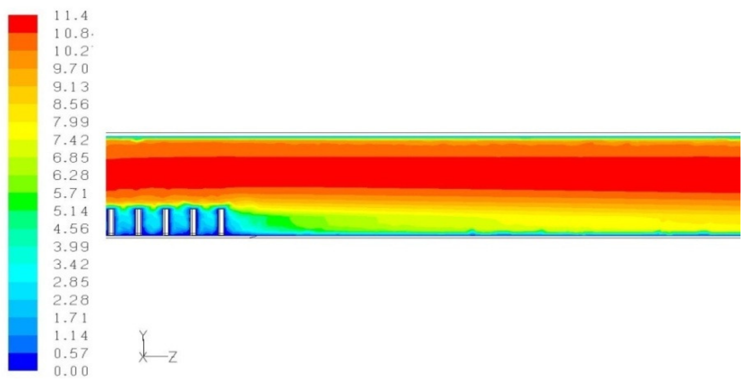

Figure 22 Velocity Plot for Rect Pin Fin

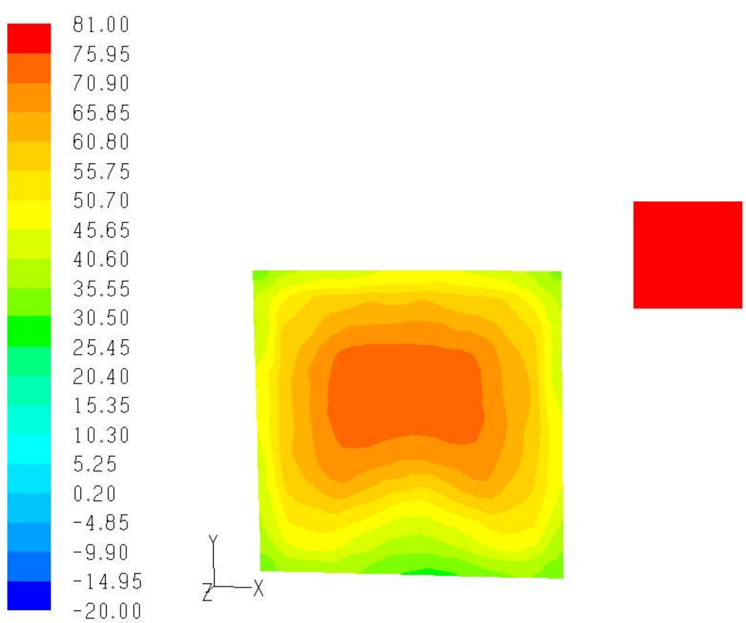

Figure 23 Total Pressure Plot for Cyl Pin Fin

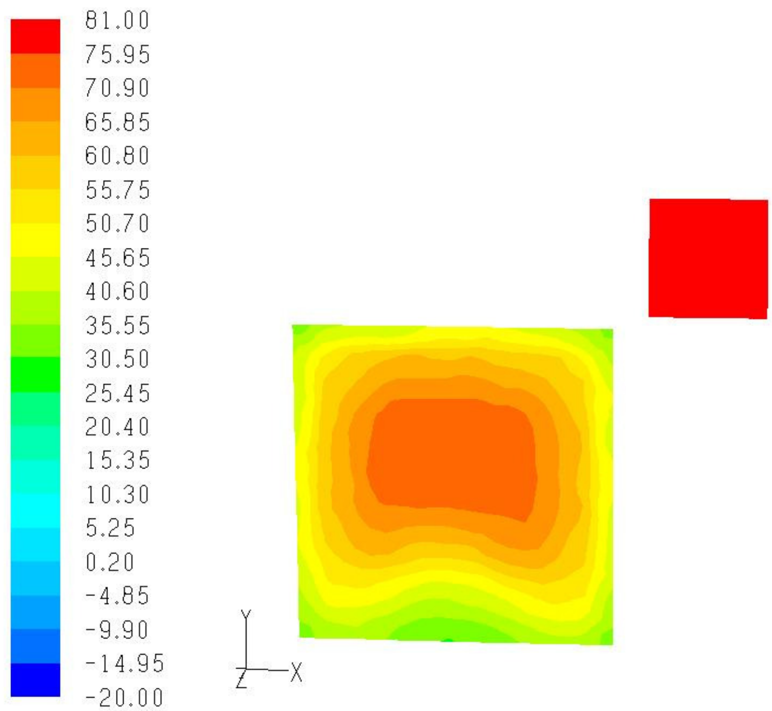

Figure 24 Total Pressure Plot for Drop Shaped Pin Fin

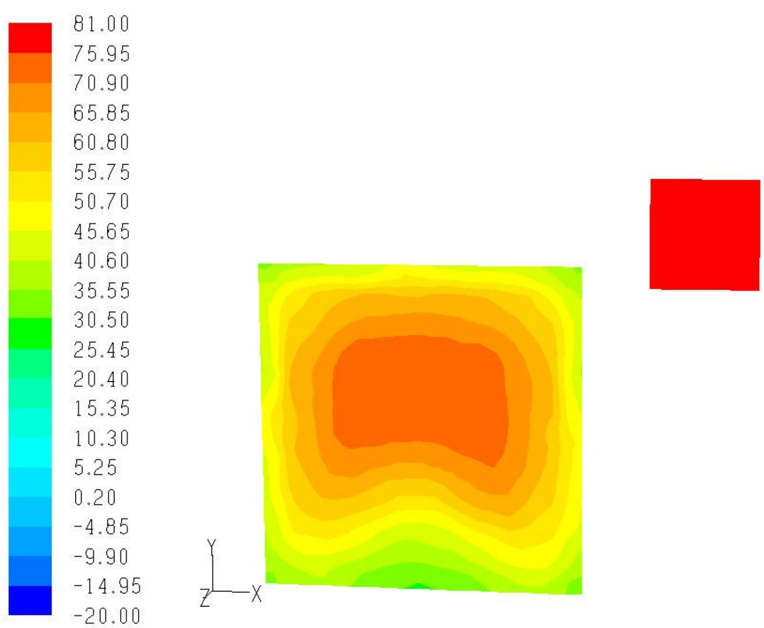

Figure 25 Total Pressure Plot for Rect Pin Fin

4.3 Velocity plots

Velocity plots are very helpful to understand air flow analysis in fins. It should be noted here that the H/D ratio for drop-shaped pin-fins is smaller than the other ones (but wetted surface area is equal for all). This behaviour can be reasoned by noticing the figure 26 to 28. It can be seen that for Drop-shaped pin-fins, flow is accelerated in pin-fin section which could cause more friction and pressure drop in fins top section

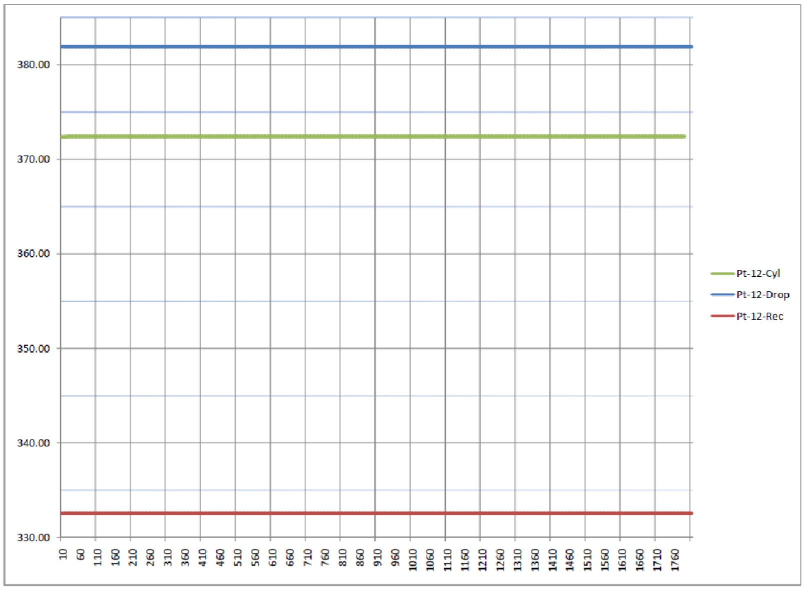

Figure 26 Temperature plot on VP-12 for all type Fins

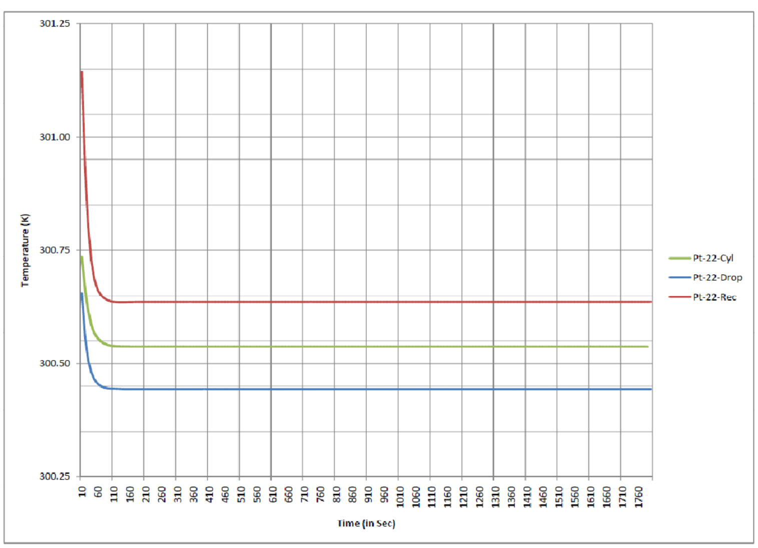

Figure 27 Temperature plot on VP-22 for all type Fins

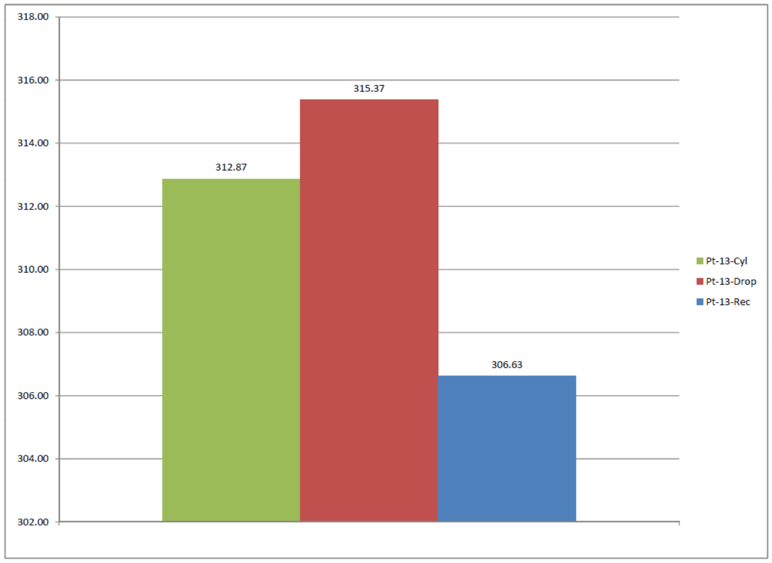

Figure 28 Temperature plot on VP-13 for All type Fins

3. CONCLUSIONS

A reasonable comparison of various pin-fin geometries has been attempted. A three-dimensional conjugate problem has been studied with a three –dimensional CFD model .These were greatly simplified by assuming 1-column in-line pin-fins with axes perpendicular to the flow and isothermal heat transfer surfaces. At lower values of pressure drop and pumping power, drop shaped fins work best. At higher values, drop-shaped fins and Circular fins offer highest performance. For high Reynolds numbers, the fins thermal efficiency and effectiveness show same behaviour, but drop-shaped fins configuration always stand a little bit upper. Also its variation in different flow regimes is smoother, which means that the engine performance varies according the working load conditions.

In last it was important to show that CFD tool is good approach to analysis thermal behaviour of any thermal device. For future work refinement of mesh was good approach to refine results and validation of CFD results will give more benefit to users.

REFERENCES

[1] S.R Mcilwain, "Improved prediction methods for finned tube bundle heat exchangers in cross flow", PhD Thesis, University of Strathclyde, Glasgow, 2003

[2] S.R Mcilwain, "A comparison of heat transfer around a single serrated finned tube and a plane finned tube", IJJRS, pp 88-94, 2010

[3] C. Weierman, "Pressure drop data for heavy duty finned tubes", Chemical engineering progress, 73, pp 69-72, 1977

[4] C. Weierman, "Correlations ease the selection of finned tubes", The Oil and Gas Journal, Vol.74, pp 94-100, 1976

[5] V. Ganapathy, "Design and evaluate finned tube bundles", Hydrocarbon processing, Vol.75, No.9, pp103-111, 1996

[6] Poulikakos, A. and Bejan, A., "Fin Geometry for Minimum Entropy Generation in Forced Convection" ASME Journal of Heat Transfer, Vol 104, pp. 616-623.

[7] Incropera F. P., DeWitt D. P., 1996, “Fundamentals of heat and mass transfer”, 4th Edition, John Wiley & Sons, Pg. No. : 147-172.
[8] P. K. Nag, 2006, “Heat & Mass Transfer”, 2nd Edition, Tata McGraw Hill Co. Pg. No. : 86- 108 & 425-449
[9] J. P. Holman, 2004, “Heat Transfer”, 9th Edition, Tata McGraw Hill Co,” Pg. No. 43-53 & 315-350
[10] Yunus A. Çengel, 2004, “Heat Transfer- A Practical Approach”, SI units 2nd Edition, Tata McGraw Hill Co., Pg. No. : 156-168, 333-352 & 459-500
[11] H.Y. Pak, K. Park, and M.S. Choi, “Numerical Analysis of the Flow and Heat Transfer Characteristics for Forced Convection-Radiation in Entrance Region of an Internally Finned Tubes,” KSME Int. J., Vol. 12 no. 2, 1998, pp. 310~319.
[12] X.Liu, and M.K. Jensen, “Geometry Effects on Turbulent Flow and Heat Transfer in Internally Finned Tubes,” ASME J. of Heat Transfer, Vol.123, 2001, pp. 1035~1044.
[13] G. Fabbri, “Heat Transfer Optimization in Internally Finned Tubes Under Laminar Flow Conditions, Int. J. of Heat and Mass Transfer, Vol.41, No.10, 1998, pp.1243-1253.
[14] J. Lee, J, S. Lee, and K. Park, , Flow/Heat Transfer Analysis and Shape Optimization of a Heat Exchanger with Internally Finned Tube, Trans, of the KSME (B), Vol.29, No.4, 2005, pp.1620-1629.
[15] Sanghwan Lee ,Juhee Lee, Kyoungwoo Park, An Application of Multi-Objective Global Optimization Technique For Internally Finned Tube, Korean Journal of Air- Conditioning and Refrigeration Engineering, Vol 17, No. 10, 2005, pp. 938-946.
[16] A.C.Poloni, A. Giurgevich, L Onesti, and V.Pediroda, Hybridisation of a Multi-Objective Genetic Algorithm, a Neural Network and a Classical Optimizer for a Complex Design Problem in Fluid Dynamics, Dipartimento diEnergetica Universita di Trieste, Italy, 1999.

Authors

Sanjay Kumar Sharma completed M.tech. (Thermal Engineering) RTU, Kota and working as Assistant Professor, Gyan Vihar University, Jaipur.His area of interest is thermal efficiency improvement of heat exchangers, CFD use in Solar applications, Building energy simulation
E-mail address: sanjaybagra84@gmail.com

Vikas Sharma completed M.tech. (Energy Engineering) MNIT, Jaipur and working as Assistant Professor, Gyan Vihar University, Jaipur.His area of interest is use of CFD in buildings, solar thermal devices and building energy simulation.
E-mail ID: vikasmeche@gmail.com