Numerical Investigation of Heat Transfer Enhancement by Providing Vents on Fins of Air Cooled SI Engine

Surada Jaideep[1], Srinivasa Prasad Katrenipadu[2]

[1]*P.G.Scholar, Thermal Engineering*
[2]*Assistant Professor, Department of Mechanical Engineering*
[1,2]*JNTUK-UCE Vizianagaram, A.P, India.*

Abstract - *In this present work, CFD analysis was done on the fins of SI engine by making vents to boost the overall performance. To enhance the heat transfer rate from existing fins there are numerous studies are taking place. Perforation is a new approach to increase the heat transfer rate from the fin with a merit of reduction in weight. Weight reduction plays a key role in automobiles. This work considered 100 cc engine head and change the existing fin by making two different vents (i.e. circular, taper shapes) on it and considered different flow patterns, different materials and different running speeds (i.e. 45 KMPH, 60 KMPH, 75 KMPH). The entire analysis was done on ANSYS fluent 16.0. Results show that vent fin has a higher heat transfer coefficient than existing solid fin and the temperature at the tip can be effectively reduced by 6 ^{0}C by making vents on fins. Compare with inline flow pattern, staggered pattern exhibits good performance in heat transfer rate due to more turbulence generation. If the existing material is replaced with Mg E21 alloy, the weight may get reduced approximately 29% and it can further cut by 2% by making vents on it.*

Key Words: **CFD, Flow patterns, Heat transfer rate, Inline, Staggered, Vents, Weight reduction.**

1. INTRODUCTION

Fins are generally surface extended components to increase heat flux from the surface. In most air cooling methods fins are employed because the specific heat capacity of air is low. Providing fins are not always advisable and it can be judged by effectiveness. In low capable engines, air cooling method would prefer than water cooling method due to some advantages. Perforation is a new approach to enhance heat flux from the existing fin surface. The perforated fins increase the effective heat transfer area however the rise in the surface area of the fin depends on the perforation dimensions [1]. T.K. Ibrahim [2] numerically and experimentally studied about the impact of perforation on the fin by considering different perforation shapes, heat flux, and velocities. They concluded that using perforation can increase heat flux from fin surface and square perforation shows 4.6% lower nusselt number than circular perforation. Using perforation also reduce friction factor by 5.9%. M.E. Nakhchi [3] numerically investigated the heat transfer enhancement by using perforated cylinder fitted in heat exchanger tube. For this work, they considered different perforated index cylinders, diameter ration and Reynolds number. They validated their numerical results with earlier experimental results. They concluded that turbulent kinetic energy increase with increase diameter ratio. Peak thermal performance value of 1.456 could be reached for the case of d/D=0.74 and PI=24% at Re=6000. In another study, thermal enhancement of heat sink by perforation method was analyzed by M.R. Shaeri [4] using a numerical method. In this number of perforation are varies and change the Reynolds number. They suggested that an increasing number of perforation enhance heat transfer rate and pressure drop and friction factor reduces by using increasing perforation number. Subodh Kathane [5] the authors considered the square and circular shape of perforation and different materials, the CFD analysis was conducted on ANSYS fluent and finally they concluded that using circular gives high heat transfer than square. By replacing existing material with aluminium 7075 shows best results. J.S. Chavan [6] computationally studied that impact of fin spacing, flow arrangements, perforation. For this work, they designed five models and conducted CFD analysis by using fluent. They concluded that increasing interruption length cause disturbed thermal boundary layer leads to better heat transfer rate. Perforated fin with staggered arrangement shows a bit higher value. Susmitha Sundar [7] experimentally studied the radial heat sink with perforated fins in a staggered manner under free convection and radiation. They validated experimental results with their numerical results. They suggested that by using perforation technique, the thermal resistance of fin get decreased by 7-12%. Using perforation overall weight of components get reduced by 9%.

The thermo hydraulic performance of perforated plate fin heat sink was studied by Waleed Al-Sallami [8]. For this work they considered different type of notches, slots, and multiple circular perforations on fin. The numerical analysis was done on Ansys fluent. Finally they concluded that all type of perforation can enhance heat transfer rate but notch fin offers slightly better performance in terms of heat transfer and pressure drop.

After interpreting the all of journals as mentioned above they only considered a constant area of perforation but in this present study varying area of vent or perforation was used and observe how this will affect the thermo hydraulic performance of fin.

2. METHODOLOGY

For conducting CFD analysis on the different type of fins, initially all the models were designed in CATIA V5 software. Table 1 shows the design details. In this, number of vents are constraint and only change the shape and size, flow pattern of the vent. After completion of modelling those were exported into ". stp" format to conduct the CFD analysis on it.

Table -1: Design details of Models

Fin type	Solid type	Vent type
Fin pitch	8 mm	
Fin thickness	3 mm	
Fin length	50 mm	
Fin width	15-10 taper 1:10	
No. of fins	27	
CC of engine	100	
Vent type	Taper	circular
Vent size	1.5 mm	2.5 mm
Flow pattern	Inline	Staggered

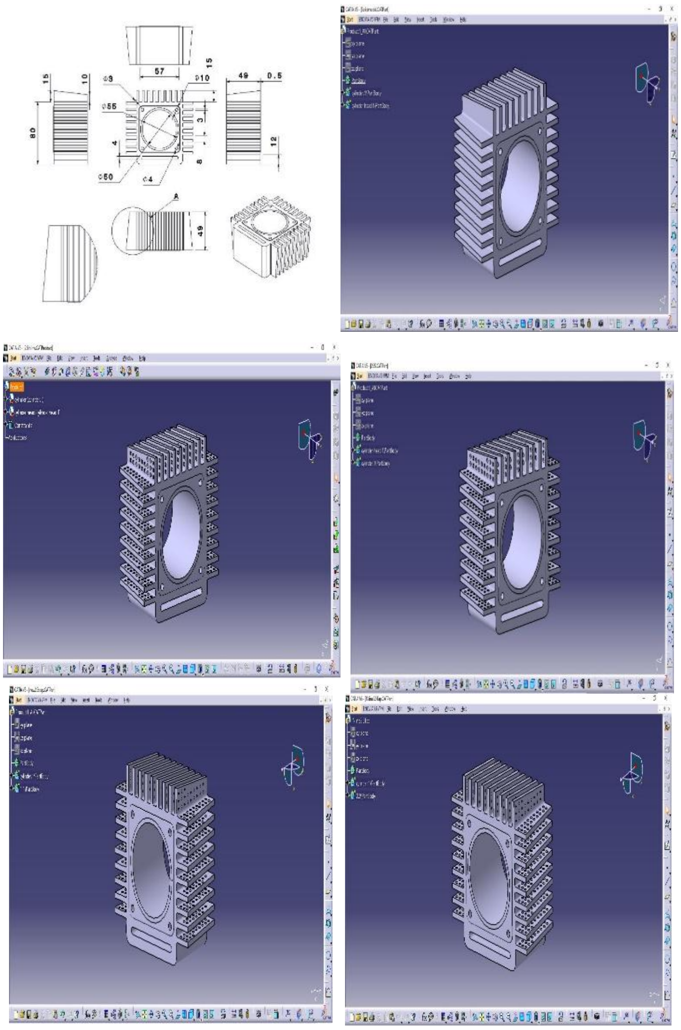


Fig -1: Various Vent fin models with Inline and staggered pattern

A. NUMERICAL ANALYSIS:

For conducting numerical analysis, in this work used ANSYS fluent workbench. After modeling those were exported into .stp format and imported into ANSYS fluent workbench. In modeling section fluid domain around the engine head was created. Then mesh was generated by using tetrahedral elements with an element size 4 mm. After conducted the grid independence test, the result shows 4 mm was optimum size with less computation time without deviating the accuracy in the outcome. Graph 1 shows grid independence plot.

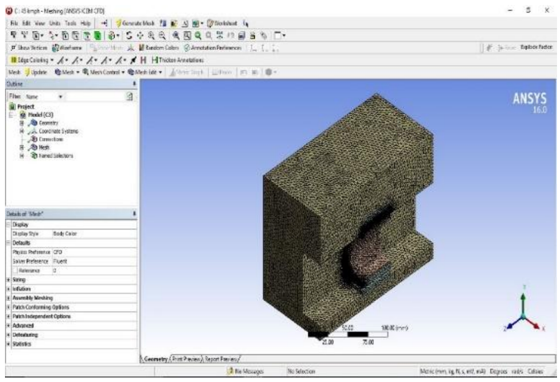
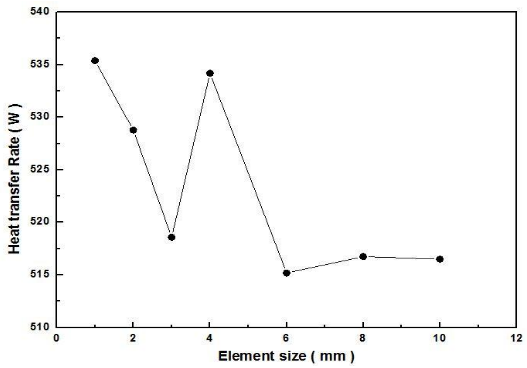


Fig-2: Meshing of Entire domain



Graph 1: Grid Independence Plot

From the grid independence plot, it is clear that element number 4 gives as accurate as element number 1. Element number 4 takes much lesser time to complete the calculation. Figure 2 depicts the mesh generation of the entire domain. In this work only half of the section was used to save simulation process time. Meshing is nothing but discretization of the domain into finite volumes and solves governing equations. After completion of meshing, analysis was conducted by using required boundary conditions. Boundary conditions applied on the model as given:

At Inlet:

U= 45,60,75 kmph

Temperature of inlet air=303 K

Temperature of Cylinder=493 K

Temperature of cylinder outer part=via system coupled

Turbulent Intensity=1%

Turbulent Viscosity ratio=10

At outlet:

Outlet pressure outlet

Turbulent Intensity=5%

Turbulent Viscosity ratio=10

In this present work 3 different materials were taken based on thermal conductivity, specific gravity and density because these are given a rough idea about heat flux from the surface and weight reduction. The material properties were given in table 2.

Table-2: Thermo physical properties of materials

Material Type	Density (Kg/m^3)	Specific Heat (J/KgK)	Thermal conductivity (W/mK)
Al356	2670	963	151
Mg-E21	1800	1086	116
Al Sic	3010	741	180

In this two materials were considered with addition to existing one. One is metal alloy and the another one is composite. Alloys are mix of different metals in order to give some advantages and composites are mix of two materials, one of the main benefits of composite is these best suited where weight is a troubling factor. Due to these reasons in this work considered one alloy and composite and check whether they make a benefit or not.

B. GOVERNING EQUATIONS:

Continuity:

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \boldsymbol{u}) = 0$$

Momentum equation:

X, Y, Z momentum equation:

$$\frac{\partial(\rho u)}{\partial t} + \nabla(\rho u \boldsymbol{u}) = -\frac{\partial p}{\partial x} + \nabla(\mu \ grad \ u) + S_{Mx}$$

$$\frac{\partial(\rho v)}{\partial t} + \nabla(\rho v \boldsymbol{u}) = -\frac{\partial p}{\partial y} + \nabla(\mu \ grad \ v) + S_{My}$$

$$\frac{\partial(\rho w)}{\partial t} + \nabla(\rho w \boldsymbol{u}) = -\frac{\partial p}{\partial z} + \nabla(\mu \ grad \ w) + S_{Mz}$$

Energy Equation:

$$\frac{\partial(\rho i)}{\partial t} + \nabla(\rho i \boldsymbol{u}) = -p \nabla \boldsymbol{u} + \nabla(k \ grad \ T) + \Phi + S_i$$

The pressure velocity linked by SIMPLE algorithm and second order upwind scheme was used to discretise the energy and momentum equations. Finally conducted the transient analysis for 1min by standard initiation.

C. ASSUMPTIONS WERE CONSIDERED IN THIS WORK:

1. Transient heat transfer and effect of radiation is negligible
2. Thermo physical properties are constant.
3. Flow is turbulent in nature
4. No heat generation.

D. DATA COLLECTION:

Nusselt Number calculation:

$$Nu = \frac{Q_{conv.}}{Q_{cond.}} = \frac{h}{k/l} = \frac{hL}{k}$$

Vent Fin effectiveness:

$$\varepsilon_{vfe} = \frac{q_{vf} - q_{sf}}{q_{sf}}$$

Mass reduction parameter:(η)

$$\eta = \frac{M_{sf} - M_{vf}}{M_{sf}}$$

Vent factor:(ψ)

$$\psi = \frac{A_{vf}}{A_{sf}}$$

3. RESULTS & DISCUSSIONS:

3.1 Temperature distribution

Figure 3 shows temperature contours from CFD post processing. The fin temperature at the tip can effectively reduce up to 6 ^{0}C compared with a solid fin and this may proportional to vent size. This happens due to the weaken of the TBL around the fins due to vents. Thus decreasing of TBL rises more heat flux from the surface of fin. Graph 2 shows temperature distribution over different type of fins with flow patterns. From the graph 2, it is evident that increasing vent cause more temperature drop occurs and compare with circular vent, taper vent gives a bit high temperature drop. The reason might be due to the taper shape is induce more

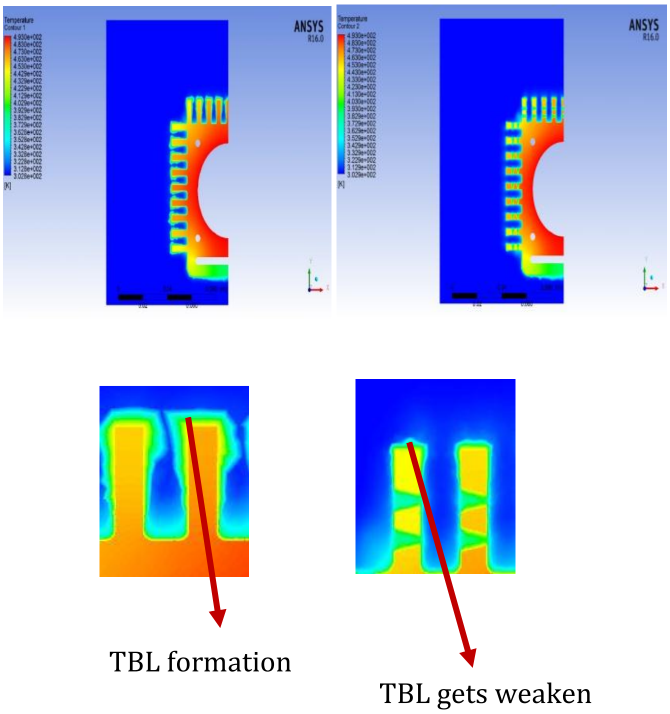
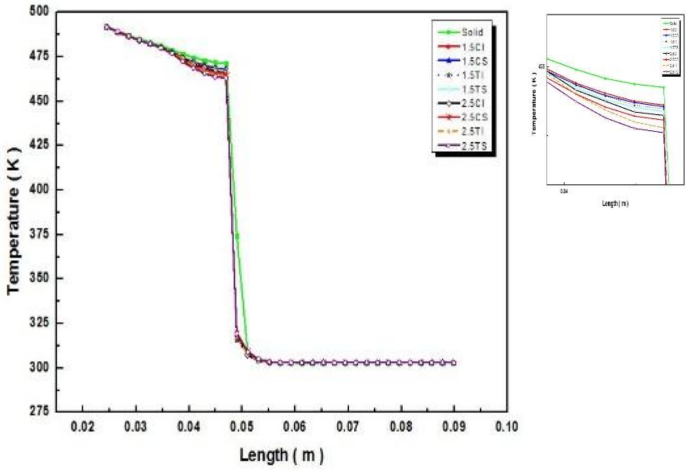


Fig-3: Temperature contours



Graph 2: (a) Temperature distribution over fins
(b) Close up view of temperature distribution

KE to fluid particles leads to better heat transfer occurs. In contrast with inline, staggered flow pattern offers bit higher value because of more turbulences occurs among the fins hence TBL gets weaken. When increase the vent size from 1.5 to 2.5 mm increase more interaction occurs between cold fluid to the hot surface. Thus increase the size of vent could increase temperature drop.

3.2 Velocity plots:

It is essential to comprehend the flow behaviour to improve the heat transfer from the surface. Figure 4 shows velocity vector plots. It is very clear that making vents causes the disturbance created in fluid flows. From figure 4 solid fin arrangement shows weak flow vortices and by making vent cause increase flow vortices among fins.

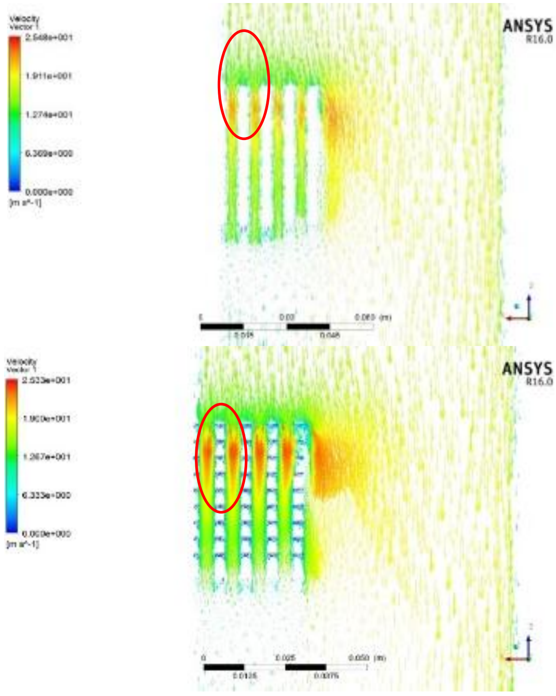
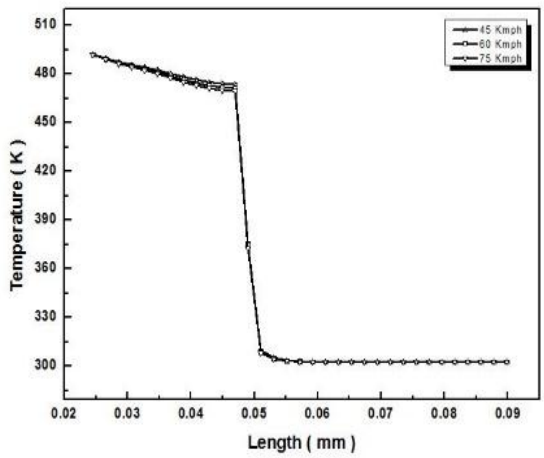


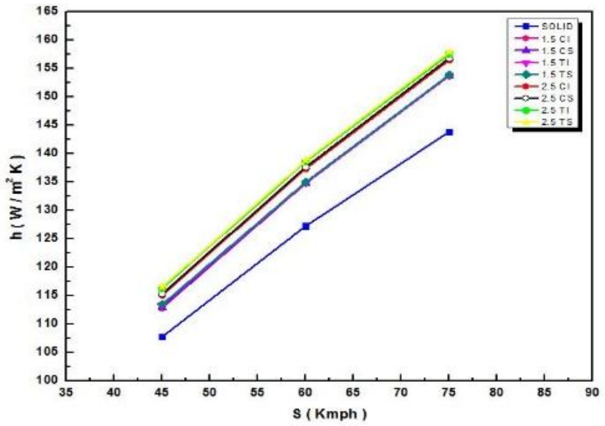
Fig-4: Velocity Vector Plots

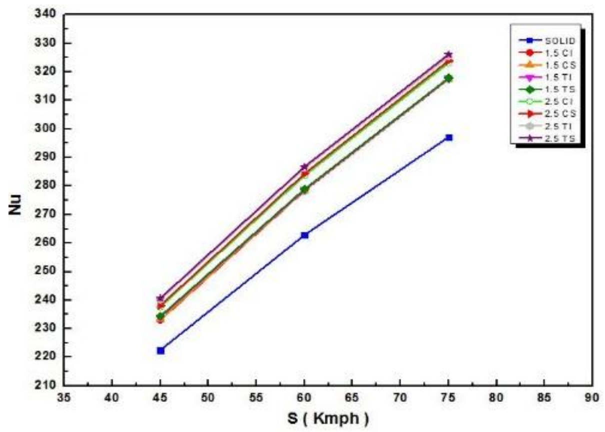
This turbulences collapse the velocity boundary layer. So that in compare with solid fin model, vent fin model exhibit bit higher velocity. This is the reason why vent fins show higher temperature drop. Graph 3 depicts that when running speed increases the temperature drop is more because the heat gets rapidly transfer to the cold fluid. But running engine at higher speeds cost more fuel consumption. Hence making vents cause more heat gets transferred at low speeds also.



Graph 3: Temperature distribution for all speeds

3.3 Effect of vents on h and Nu:



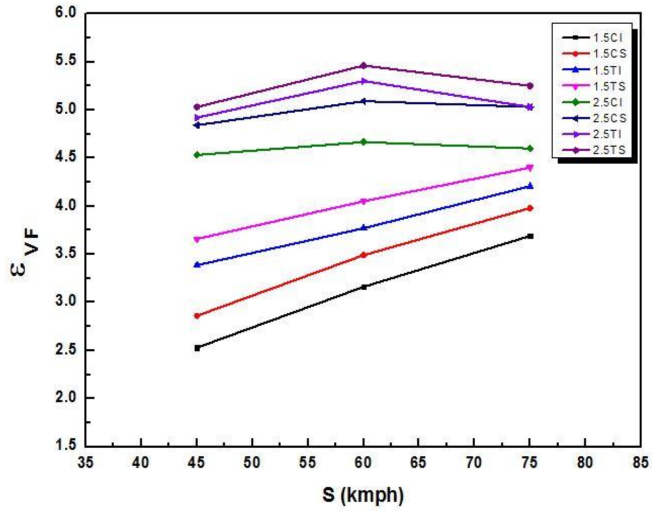


Graph 4: (a) h Vs. Speed (b) Nu Vs. Speed

In thermal application, Heat transfer coefficient is one of the main parameter to judge a fluid whether it has a good heat transfer capability or not. In this analysis heat transfer coefficient and nusselt number graphs were drawn with respect to speeds. From the above graph 4 can say that vent fin models shows high heat transfer coefficient and nusselt number. The nusselt number and heat transfer coefficient are interlinked parameters if one is known other can calculate by formula that was mentioned previously. By observing graphs compare with 1.5 mm vent 2.5 mm vent indicates higher values and taper with staggered flow pattern exhibits better h and Nu values. By increasing size of vent cause rise in the convective area so that it shows bit higher values. And staggered flow pattern leads to more flow separations cause more h and Nu. For 1.5 mm vent gives around 5.98% improved h and for 2.5 mm gives 8.53% improved h value. The nusselt number graph also follows a similar fashion.

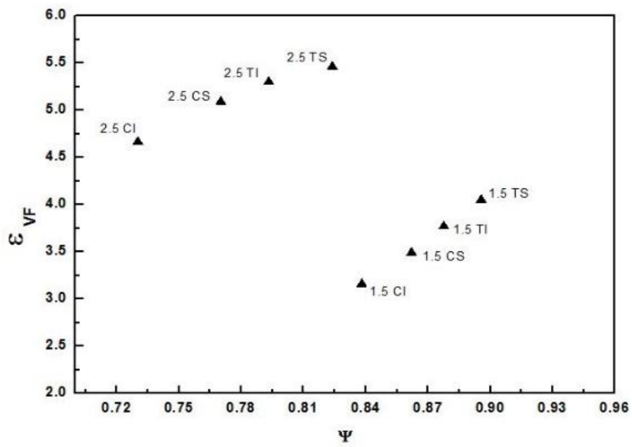
3.4 Effect of vents on Effectiveness:

Effectiveness of fin means how much amount of improvement in heat transfer by providing fin. In this vent fin effectiveness was calculated by using formula for different speeds and this is a slight modification to the original formula. The graph 5 shows the vent fin effectiveness. Its clear that rising vent size shows improvement in effectiveness. Due to more exposure area 2.5 mm vent gives higher VFE than 1.5 mm vent. In fact the effectiveness is based upon heat flux so that taper with staggered arrangement have high heat flux thus it gives higher effectiveness.



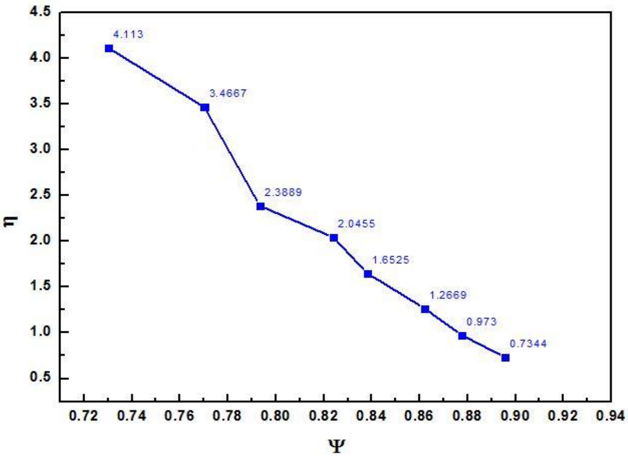
Graph 5: Vent fin Effectiveness Vs. Speed

3.5 Effect of Vent factor on VFE and Mass reduction Parameter:



Graph 6: Vent factor vs. Vent fin Effectiveness

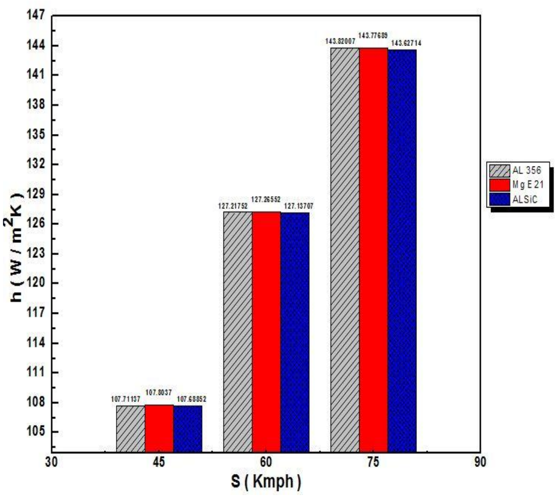
Graph 6 shows the relation between vent factor and VFE. Its depicts that when decreasing the vent factor it increases the VFE also from the graph it comes to know that vent factor not solely judge the VFE, other factors such as vent shape and flow pattern can also affect the VFE. In contrast with 1.5 mm, 2.5 mm vent factors give higher VFE thus it can say that the vent factor is inversely proportional to VFE.



Graph 7: Vent factor Vs. Mass reduction parameter

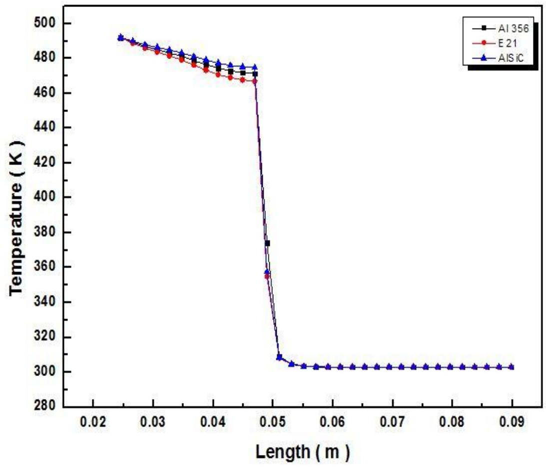
Graph 7 shows the relationship between the mass reduction parameter to vent factor. This mass reduction plays a vital role in the automobile sector. From the graph its evident that mass reduction parameter gradually declines when increasing the vent factor. At lowest factor value gives highest mass reduction. When the size of vent increase, mass reduction also increases. This graph only shows how the weight reduction varies with vent factor. When number of vents are more the weight reduction is more. so that it can say that mass reduction is inversely proportional to vent factor. On average 2.3% overall mass get reduced by making vents on fins. Thus providing vents on fins can reduced weight with addition of increase heat transfer coefficient.

3.6 Effect of replacing existing material with proposed materials on various parameters:



Graph 8: h Vs. S for different materials

This study used two materials along with existing one i.e. AL356. One is Mg electron alloy which is a recent trending alloy used in car engines. Another one is AlSiC composite this is usually used in heat sink applications. Graph 8 exhibits heat transfer coefficient changes with respect to speed for different materials. There is no visible difference between these materials but if keenly observe Mg alloy offers a bit higher h value than others.



Graph 9: Temperature distribution for all materials

In graph 9 shows temperature distribution of fin by using these materials. It shows Mg alloy has lesser temperatures. But AlSiC composite shows disappointed values in heat transfer coefficient and temperature distribution. Even though Mg alloy does not shows any higher difference in heat transfer coefficient but it offers great weight reduction.

Table 4: weight reduction data for all materials

Material	Mass (Kg)	Mass reduction Percentage %
Al 356	0.5446	0
Mg electron 21	0.3847	29.36%
AlSiC	0.6067	-11.40%

Table 4 shows weight reduction for different materials. Among all, Mg alloy offers great weight reduction. But alsic materials shows negative sign that means if alsic used overall weight would be increased. If the existing material replaces with Mg alloy on average 29% weight gets reduced and it can further be increased by 2% by making vents. Thus finally replace existing material i.e. al356 with Mg alloy and adopt vent fins can significantly reduce overall weight without compromise on the heat transfer parameters.

4. CONCLUSIONS

In this present study comparisons were made between existing IC engine fins with vent fin models. The following conclusions are obtained:

- Providing vents causes thermal boundary layer gets disturbed. Hence, compare with solid fins all vent fin models show high temperature drop. On average 6^0C temperature drop occurs at the tip due to these vents. If vent size increase temperature drops also increase because of more fluid and hot surface interaction.

- In comparing with circular, taper shape exhibits bit greater temperature drop. The reason might be taper shape increase KE of fluid particles passes through it. Among all 2.5 mm, taper shape with staggered flow pattern shows higher values.
- From the velocity vector plots it can be concluded that compare with existing solid fin, vent fin offers more turbulence leads to more heat transfer occurs. Heat transfer coefficient gets improved about 5.98% when 1.5 mm vent was used if further increased to 2.5 mm h values improved 8.59%.
- Vent factor is inversely proportional to mass reduction parameter and weight gets reduced about 2% by making vents. If the existing material replaces with Mg E21 alloy, weight will reduce about 29%. Using Mg alloy instead of existing al356 alloy can increasing bit higher heat transfer rate along with weight reduction.
- Finally if the existing material replaces with Mg E21 alloy and also providing vents increase heat transfer rate from the engine with addition of reduction of overall weight.

REFERENCE:

1. R.S. Matos, J.V.C. Vargas, T.A. Laursen, A. Bejan, Optimally staggered finned circular and elliptic tubes in forced convection, Int. J. Heat Mass Transf. 47 (2004) 1347–1359.
2. Thamir K. Ibrahim, Ahmed T. Al-Sammarraie, Manar S.M. Al-Jethelah, Wadhah H. Al-Doori, Mohammad Reza Salimpour, Hai Tao, The impact of square shape perforations on the enhanced heat transfer from fins: Experimental and numerical study, International Journal of Thermal Sciences 149 (2020) 106144.
3. M.E. Nakhchi, J.A. Esfahani, Numerical investigation of heat transfer enhancement inside heat exchanger tubes fitted with perforated hollow cylinders, International Journal of Thermal Sciences 147 (2020) 106153.
4. M.R. Shaeri, M. Yaghoubi, Thermal enhancement from heat sinks by using perforated fins, Energy Conversion and Management 50 (2009) 1264–1270.
5. Subodh Kathane, Mr. Ravindra Randa, CFD Analysis of Engine Cylinder FIN, International Journal for Research in Applied Science & Engineering Technology June (2017), issue VI
6. J.S. Chavan, R.D. Shelke, H.N. Deshpande, numerical analysis of interrupted perforated fins under natural convection, Journal of Emerging Technologies and Innovative Research Aug (2017),issue 08
7. Susmitha Sundar, Gihyun Song, Muhammad Zeeshan Zahir, J.S. Jayakumar, Se-Jin Yook, Performance investigation of radial heat sink with circular base and perforated staggered fins, International Journal of Heat and Mass Transfer 143 (2019) 118526
8. Waleed Al-sallami,Amer al-damook, HM Thompson, A numerical investigation of the thermal-hydraulic characteristics of perforated plate fin heat sinks, International Journal of thermal sciences, 121(2017) 266-277.