iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 6 **Issue:** VI **Month of publication:** June 2018

DOI: http://doi.org/10.22214/ijraset.2018.6138

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VI, June 2018- Available at www.ijraset.com

Performance Calculation of Vehicle Radiator Group using CFD

Mr.Sonu Thomas[1], Mr.V. Karthikeyan[2], Dr.G. Nallakumarasamy[3]
[1]PG Scholar, Department of Mechanical Engg, Excel Engineering College, Tamilnadu
[2]Assistant Professor, Department of Mechanical Engg, Excel Engineering College, Tamilnadu
[3]Head of the Department of Mechanical Engg, Excel Engineering College, Tamilnadu

Abstract: The efficiency of the engine consists of the efficiency of its radiator. As we know that the 30% engine power will be transferred to the surroundings using the radiator. So the heat transfer through radiator must be efficient and fast in order to reduce the overheating of the engine cabin. And in the most of the modern vehicles, the radiators are arranged in the same air flow path. The engine cabin contains all other accessories, which will get over heated as the engine cabin gets overheated. And this becomes the serious issue. In this project the author try to analyses the efficiency of group radiators with different types of fins like square and ribbon [louvered] fins using CFD in a practical system. However detailed evaluation through the testing alone is difficult and not possible during the vehicle development. The focus of the study is to find the performance of the square tube radiators with various fins and their arrangement using CFD. Also the outlet air coming from the last radiator is to be analyzed to find the engine cabin cooling efficiency. Numerical simulation is carried out using a commercially available CFD code, Fluent. Realizable k-e RANS turbulence model was used to model turbulence. Result of this study gave a detailed vision of the arrangement of various types of radiators for the better performance of the vehicle group radiator heat transmission. This paper studied about the efficiency of different radiator type according to the fins used and their efficient arrangement for the better cooling of the engine compartment in a practical system.

Keyword: Radiator, CFD, Engine Component, Turbulence, Square Tube

I. INTRODUCTION

A radiator is a cross flow heat exchanger, which transfers heat from hot coolant to air by fins placed on the tube throughout its length via conduction and convection. The coolant circulates over the engine block and absorbs heat from the engine during combustion process. Hot coolant coming from engine is passed to radiator for cooling the coolant. It regulates the engine temperature at optimum value. Radiator is the primary component of the cooling system in automobiles. Failure of engine takes place mainly due to excessive heat produced in the engine components. This can be avoided by employing the proper cooling system. Radiators have been classified depending upon the flow and type of the material used. Radiators are preferred based on heat dissipation rate of the engine. Mostly cross flow heat exchangers are used as the radiators in vehicles. In these radiators, fins, which are extended surface, are using to increase the conduction rate and the surface area.

II. OBJECTIVES

A. To analyse the Square tube radiator.
B. To analyse the Square fin, Ribbon fin in the efficient tube.
C. To analyse the arrangement of radiators for the best cooling of engine compartment.

III. PROBLEM IDENTIFICATION & METHODOLOGY

A. Identification Of The Problem

1) Overheating of engine compartment
2) Arrangement of different radiators in group.

B. Problem Analysis And Rectification

The outline of the simulation process is summarized as follows:

1) Pre-Processing
a) Modeling the geometry and the flow domain
b) Establishing the boundary and initial conditions

c) Mesh generation

2) *Solving*

a) Reading the mesh file and grid check

b) Establishing the simulation strategy

c) Establishing the input parameters and files

d) Performing the simulation

e) Monitoring the simulation for convergence

3) *Post-processing*

a) Post-processing the simulation to get results graphs, plots, contour plots etc.

b) Arranging the radiators back to back.

c) The number of radiators is assumed to be (1+2), this is done to reduce the complexity of the model and applying the obtained results according to the requirement.

d) The pipes, tubes and other components also neglected.

e) The flow pattern of the air and fluid is carried under no load condition.

f) Air domain is used to get the engine cabin.

4) *Pre-processing:* For modelling the geometry, it is a general purpose pre-processor for CFD analysis which provides meshing capabilities wherein the model can be meshed and subsequently imported into FLUENT and solved. Predefined grid topology templates are used to minimize grid setup time and optimize the mesh for the given application.

5) *Modeling procedure*: In numerical simulations, approximations of the geometry and simplifications may be required in an analysis to ease the computational effort. Especially, for the case of radiator simulations, it is very difficult to model the pipes, tubes, pumps,etc. of the engine compartment unit because these are much smaller compared to the compartment dimensions and also because of their complicated geometry. It increases the computational effort because of the increased number of nodes and meshed elements making the problem set-up complicated. The geometry was created using SOLIDWORKS 2011. The basic dimensions of Volvo TWD 1240 series was taken for modeling. Modeling was done with both the square and louvered fins. The dimensions of the radiators are:

a) CROSS FLOW RADIATOR: 27.62mm*10.56mm*10mm
Tube height: 3.55mm
Fin height: 3.55mm

b) DOWN FLOW RADIATOR 1(Right): 13.52mm*10.56mm*10mm
Tube height: 3.24mm
Fin height: 6.56mm

c) DOWN FLOW RADIATOR 2(Left): 13.59mm*10.56mm*10mm
Tube height: 4.53mm
Fin height: 4.53mm

After that the geometry will imported to the Ansys FLUENT 2015.

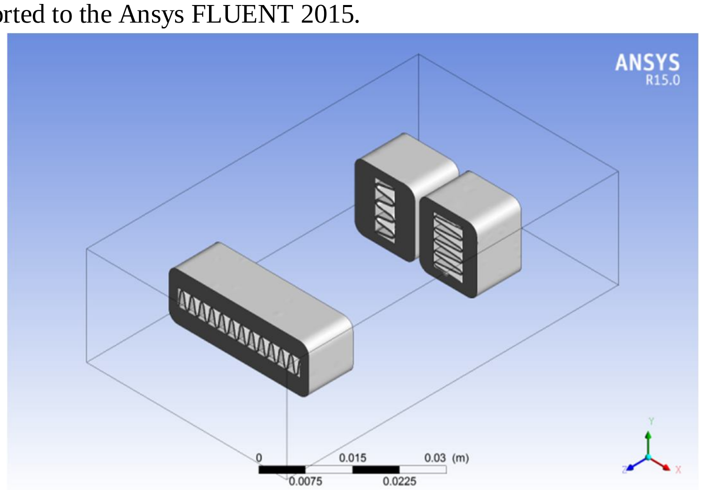


Figure3.1 Imported Geometry Of The Louvered Fin Radiator

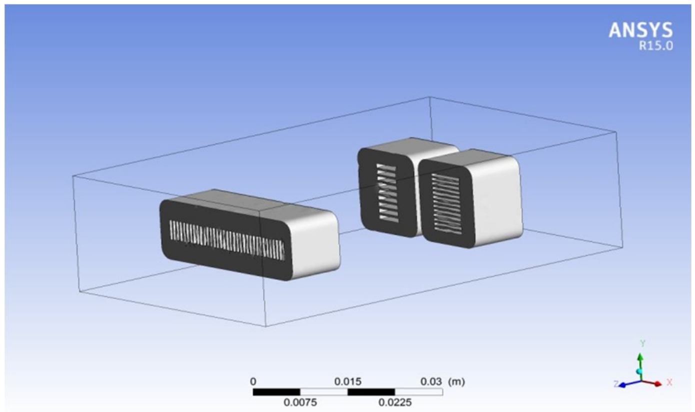


Figure3.2 Imported Geometry Of The Louvered Fin Radiator

6) *Mesh generation*: The entire face is divided into innumerable small finite number of elements. This process is called meshing and the grid generated is called a mesh. Meshing gives us a scope to study the behavior of various parameters (such as pressure, velocity etc.) at each of these elements. The finer the mesh (more elements) the better is the scope for analysis since it gives us more number of points to study the behavior of parameters. The model is then divided into 3 radiators and 1 air domain which transports and circulates air throughout the system. The outlet is placed at the back of air domain. All these are of standard dimensions.
7) *Establishing the Boundary Conditions*: Once the mesh is generated, various edges of the grid are given names for easy understanding and for setting the appropriate boundary conditions while solving. The continuum type (fluid/solid) is also specified. Finally, this meshed model is exported as mesh file in a format that can be directly into FLUENT.
8) *Solving*: Solving is an important phase in CFD analysis. The mesh file is read into FLUENT and a routine grid check is performed to detect the presence of any skewed cells. Skewness is the difference between the shape of the cell and the shape of an equilateral cell of an equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. These skewed cells can be weeded out either by use of smooth/swap grid option in FLUENT. Then, we may proceed to setting up the problem. Establishing the simulation strategy, to perform the simulation, we must lay out some rules that affect the physics of the problem. The problem is then set-up based on these under-lying assumptions. The pressure based solver is used for low-speed incompressible flows while the density based solver is used for high speed compressible flows, where the velocity and pressure are strongly coupled (high pressures and high velocities). The indoor airflow simulation falls under the category of low-speed incompressible flow, which can be deduced from extensive literature survey .The air velocities at the inlets, also indicate the same. Thereby, in the present study a pressure based solver has been employed. In this method, governing equations are solved sequentially (i.e. segregated from one another). Because the governing equations are non-linear, several iterations of the solution loop must be performed before a converged solution is obtained. Once the grid is checked, the pressure based implicit solver is applied.

C. Results

The solution after the iteration process will be available once all the iterations converge together.

The results of the air circulation inside the domain can be shown using different plots and graphs as well with values.

The results obtained will be in a wireframe stage , we will have to make some changes in order to obtain results using different method

1) Streamline plots: The velocity of the inlets and the characteristics and direction of air flow is shown,

Streamline flow shows the complete method and direction and the amount of air flowing throughout the system.

2) Contour plot (using plane): A plane is created in the system, and it is placed in or near the bus, then the values or results obtained are displayed on the plane we created.

3) Vector plots: Vector flow is similar to streamline and shows the air flow through a particular region.

The input value for the system is taken by the error and trial method. When the velocity is at 3m/s, the result gives the least difference for both outlet temperature and outlet velocity. The inlet temperature for air is taken as constant of ambient temperature (300k).

The inlet velocity of the air for the radiator analysis is necessary for the improvement of the heat transfer in the radiators. So we have to take utmost care for the selection for the inlet air velocity as the inlet air velocity will vary according to the fan speed. So an ideal velocity has to be selected.

RADIATOR	TEMPERATURE(K)
CROSS FLOW	333
DOWN FLOW 1(Right)	306
DOWN FLOW 2(Left)	356

Table3.1 Fluid Temperature For Radiators

SQUARE FIN RADIATOR		LOUVERED FIN RADIATOR	
VELOCITY(m/s)	OUTLET TEMPERATURE(K)	VELOCITY(m/s)	OUTLET TEMPERATURE(K)
2	304.27	2	304.931
3	304.081	3	304.651
4	303.746	4	304.358
5	303.451	5	304.142

table3.2 outlet temperature for 3m/s

SQUARE FIN RADIATOR		LOUVERED FIN RADIATOR	
VELOCITY(m/s)	OUTLET VELOCITY(m/s)	VELOCITY(m/s)	OUTLET VELOCITY(m/s)
2	2.14225	2	2.09802
3	3.182	3	3.14533
4	4.2083	4	4.19241
5	5.22789	5	5.2012

table3.3 outlet velocity for 3m/s

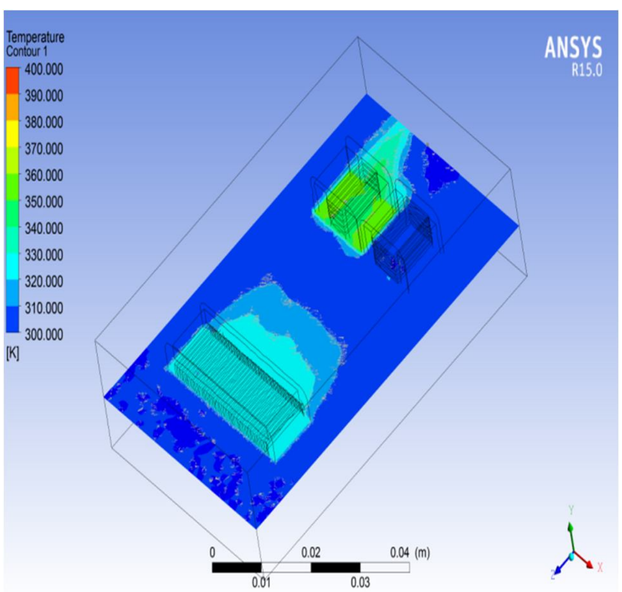


Figure 3.3 Contour Plot On The (X-Z) Plane, Temperature (Square Fin)

911

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VI, June 2018- Available at www.ijraset.com

The temperature of the system is varies according to the varying fluid temperature in different radiators. The main parameter affecting the efficiency of radiator is temperature and so the temperature distribution in radiator has to be studied in practical conditions. And the result is plotted .

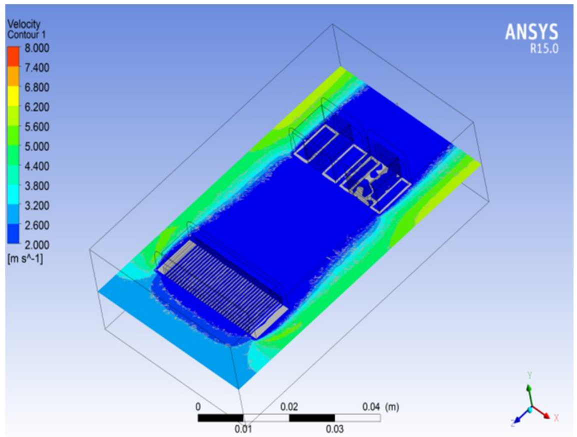


Figure 3.4 Contour Plot On The (X-Z) Plane, Velocity (Square Fin)

The velocity distribution throughout the system is plotted as the velocity will be varied according to varying system temperature. The velocity distribution in system has to be studied as there are chances of the swirling of air in the system. If the air will be holded in peculiar region for more time, then the heat dissipation will get affected.

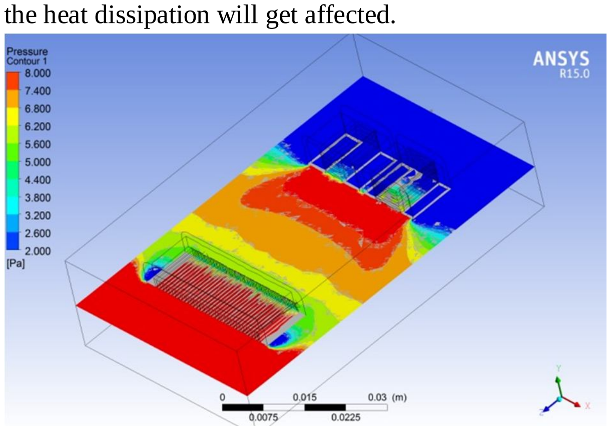


Figure 3.5 Contour Plot On The (X-Z) Plane, Pressure (Square Fin)

Pressure distribution through the system is plotted. If the pressure exceeds, the heat dissipation at the region will be low. So the pressure distribution has to studied properly.

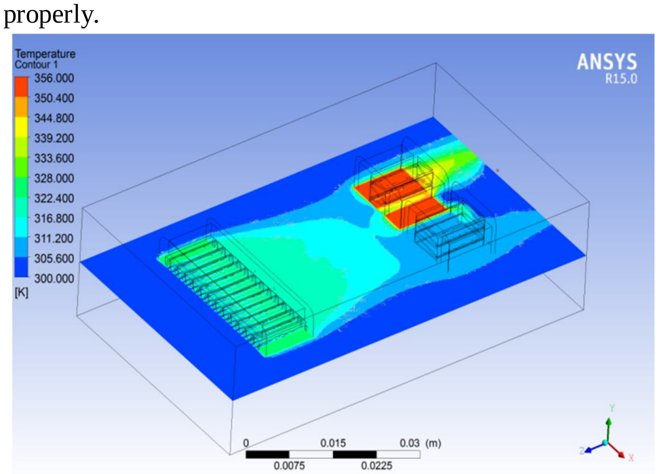


Figure 3.6 Contour Plot On The (X-Z) Plane, Temperature (Louvered Fin)

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VI, June 2018- Available at www.ijraset.com

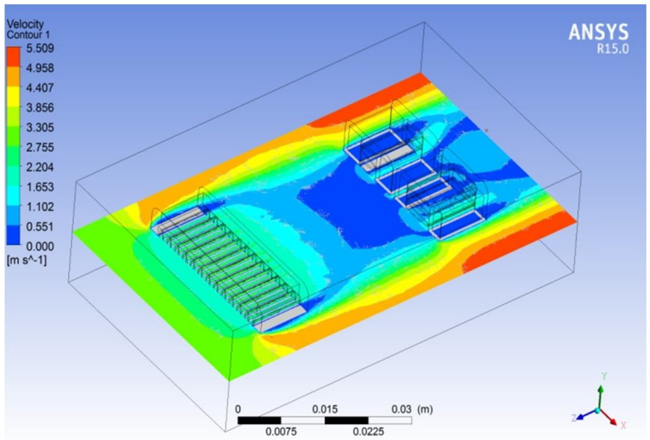


Figure 3.7 Contour Plot On The (X-Z) Plane, Velocity(Louvered Fin)

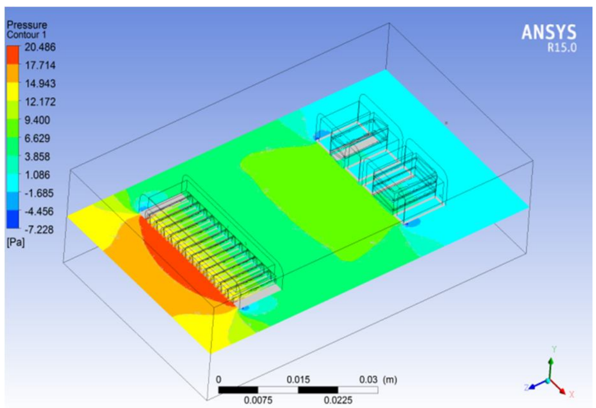


Figure 3.8 Contour Plot On The (X-Z) Plane, Pressure (Louvered Fin)

D. Case Study: 1

The temperature of the inlet air will be varied as the weather condition changes. So I made a study of the heat dissipation in both the weather condition in both type of radiators.

1) Summer Conditon: The temperature of the inlet air will more than ambient temperature in summer conditions, and this will create changes in other parameters in the system. And I took 313k as the inlet air temperature in summer condition.

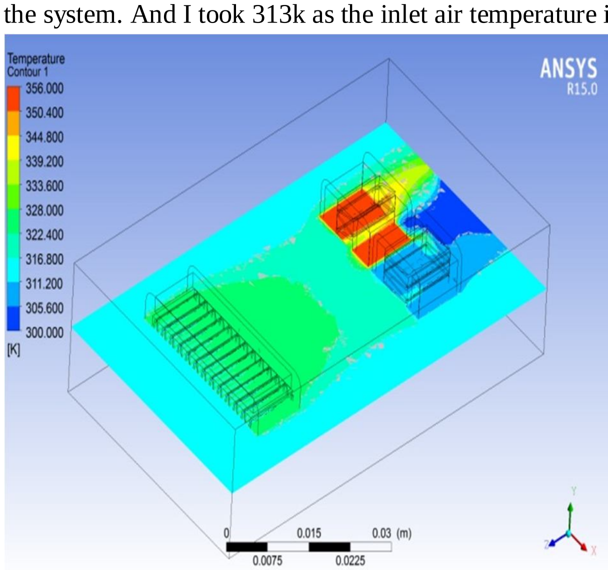


Figure 3.9 Contour Plot On The (X-Z) Plane, Temperature (Louvered Fin)

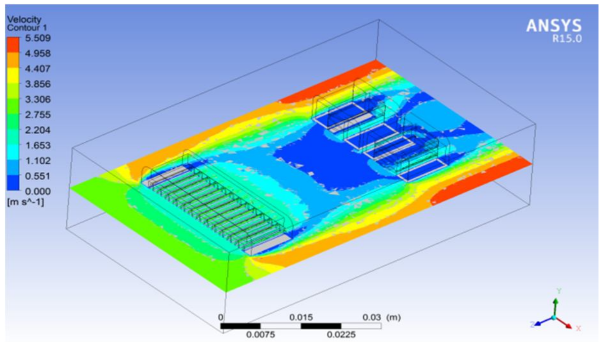


Figure 3.10 Contour Plot On The (X-Z) Plane, Velocity (Louvered Fin)

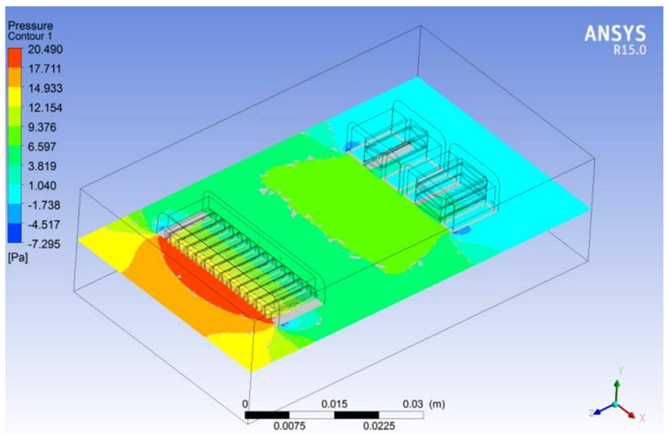


Figure 3.11 Contour Plot On The (X-Z) Plane, Pressure (Louvered Fin)

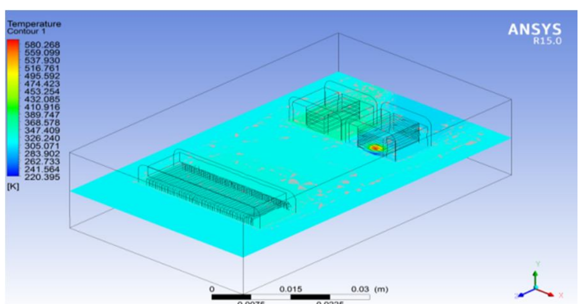


Figure 3.12 Contour Plot On The (X-Z) Plane, Temperature (Square Fin)

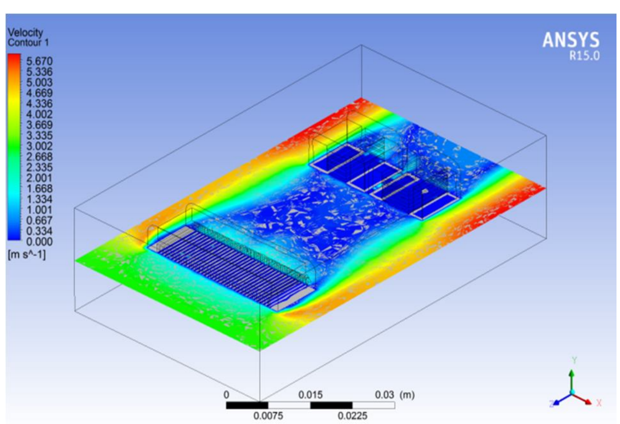


Figure 3.13 Contour Plot On The (X-Z) Plane, Velocity (Square Fin)

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VI, June 2018- Available at www.ijraset.com

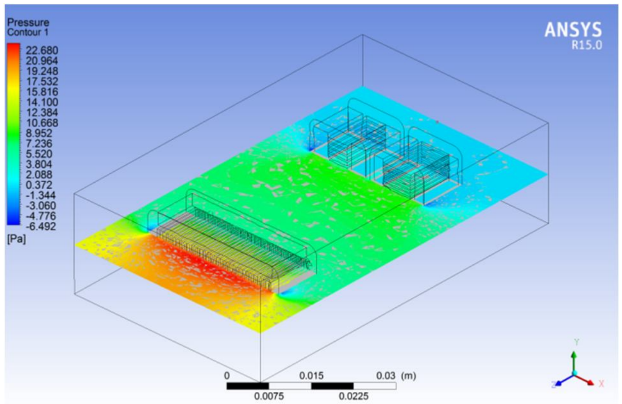


Figure 3.14 Contour Plot On The (X-Z) Plane, Pressure (Square Fin)

The outlet parameters for the given inlet parameters at summer condition.

PARAMETERS	SQUARE FIN	LOUVERED
TEMPERATURE (K)	314.057	314.85
VELOCITY (m/s)	3.182	3.14533
PRESSURRE (Pa)	-0.0287401	-0.0252043

Table3.4 outlet parameters on summer.

2) *Winter Condition:* The temperature of the inlet air in winter condition is assumed as 295k in practical aspect. And the outlet temperature, outlet pressure and outlet velocity is plotted on contour plot.

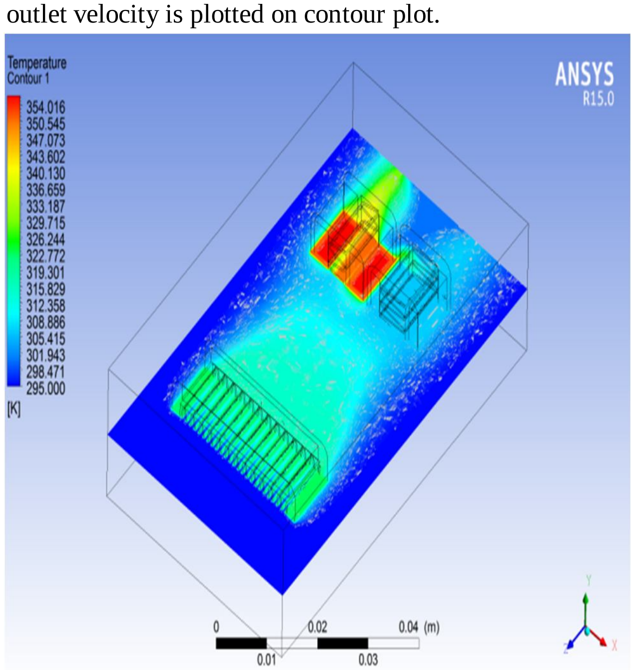


Figure 3.15 Contour Plot On The (X-Z) Plane, Temperature (Louvered Fin)

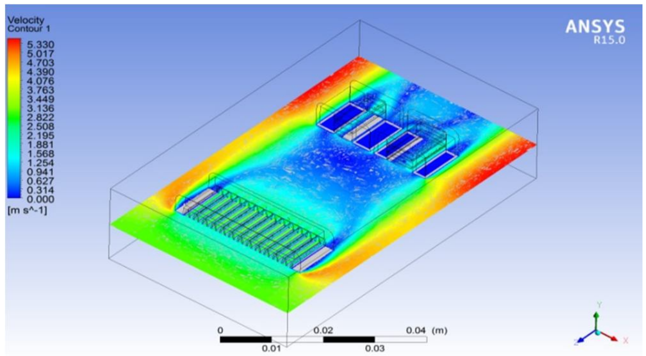


Figure 3.16 Contour Plot On The (X-Z) Plane, Velocity (Louvered Fin)

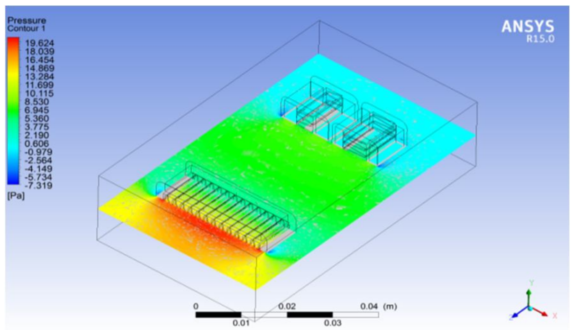


Figure 3.17 Contour Plot On The (X-Z) Plane, Pressure (Louvered Fin)

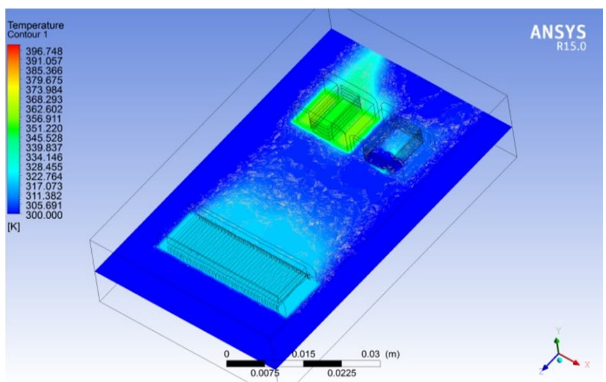


Figure 3.18 Contour Plot On The (X-Z) Plane, Temperature (Square Fin)

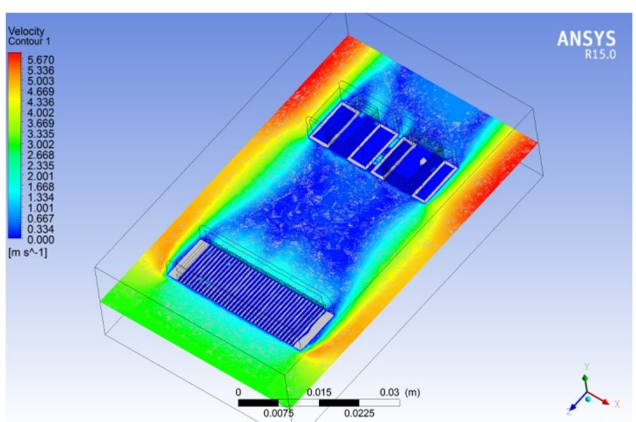


Figure 3.19 Contour Plot On The (X-Z) Plane, Velocity (Square Fin)

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VI, June 2018- Available at www.ijraset.com

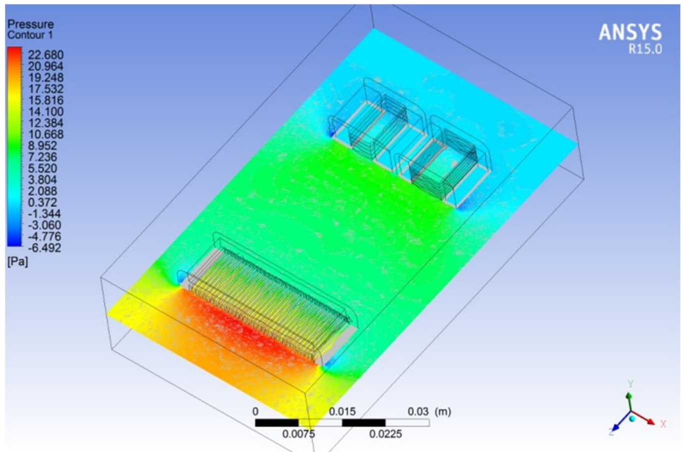


Figure 3.20 Contour Plot On The (X-Z) Plane Pressure (Square Fin)

The outlet parameters for winter condition at given inlet parameters.

PARAMETERS	SQUARE FIN	LOUVERED
TEMPERATURE (K)	301.05	301.498
VELOCITY (m/s)	3.182	3.14533
PRESSURRE (Pa)	-0.0287401	-0.0252043

Table3.5 outlet parameters on winter.

E. Case Study: 2distance Between The Radiators

The second case study is the distance between the radiators. As the distance between the radiators is changed the heat dissipation in system has been studied. The two distances studied are 20mm and 30mm between cross flow and the two down flow radiators. And the contour plot of the outlet temperature, outlet velocity and outlet pressure are plotted.

1) *DISTANCE 20mm:*The contour plot of the outlet temperature, velocity and pressure for both square fin and louvered fin are potted.

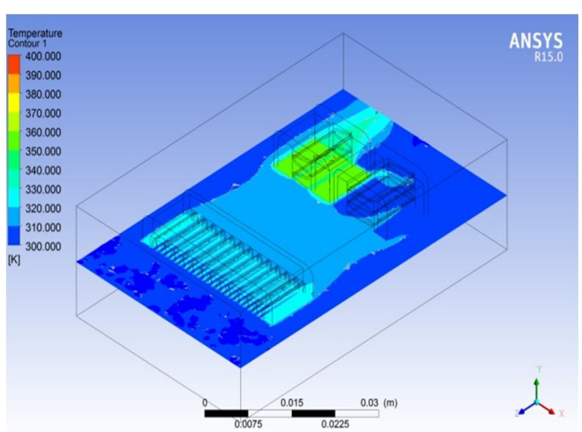


Figure 3.21 Contour Plot On The (X-Z) Plane, Temperature (Louvered Fin)

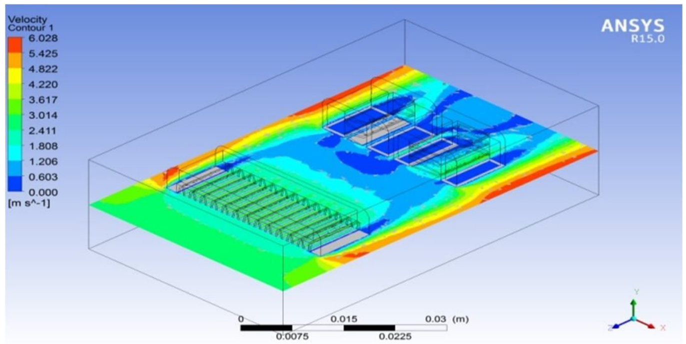


Figure 3.22 Contour Plot On The (X-Z) Plane, Velocity (Louvered Fin)

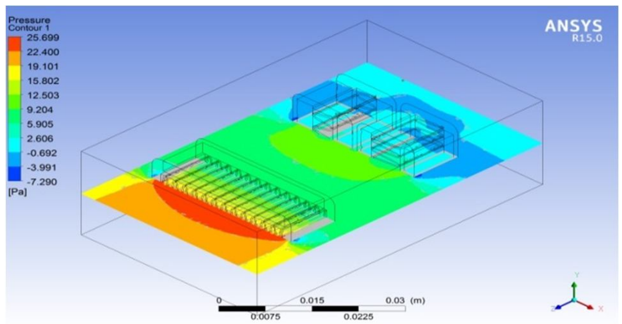


figure 3.23 contour plot on the (x-z) plane, pressure (louvered fin)

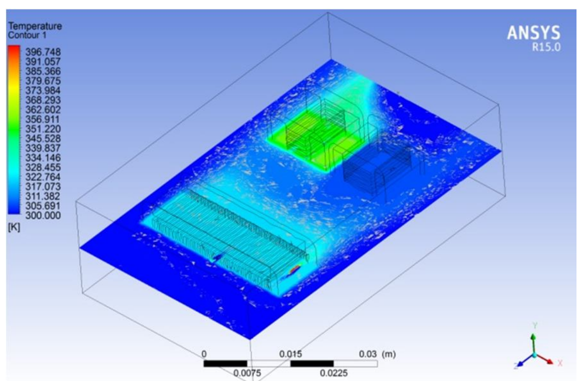


figure 3.24 contour plot on the (x-z) plane, temperature (square fin)

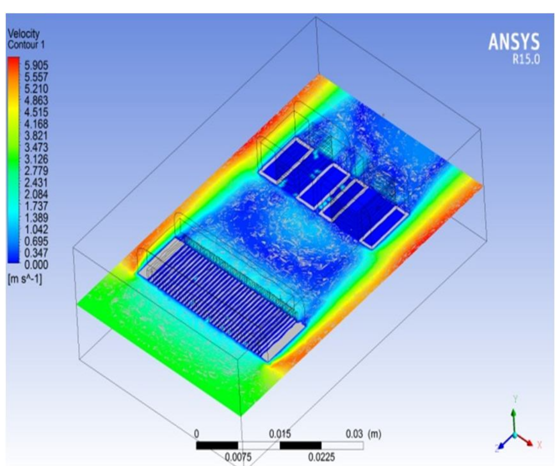


Figure 3.25 Contour Plot On The (X-Z) Plane, Velocity (Square Fin)

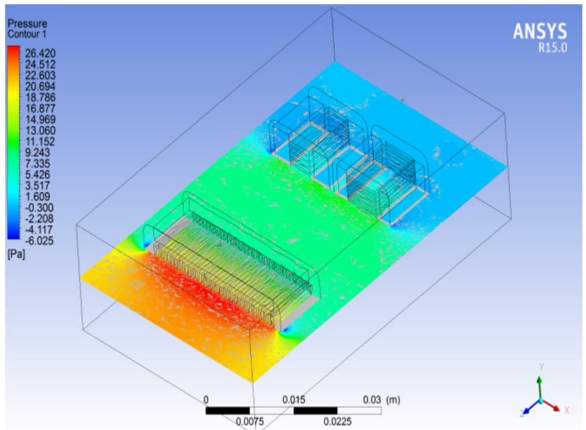


Figure 3.26 Contour Plot On The (X-Z) Plane, Pressure (Square Fin)

The outlet parameters in distance of 20mm for given input parameters.

PARAMETERS	SQUARE FIN	LOUVERED
TEMPERATURE (K)	304.248	305.603
VELOCITY (m/s)	3.20547	3.18571
PRESSURRE (Pa)	-0.0322095	-0.0381106

. TABLE3.6 OUTLET PARAMETERS AT 20mm

2) *DISTANCE 30mm:* The contour plot of the outlet temperature, velocity and pressure for both square fin and louvered fin are potted

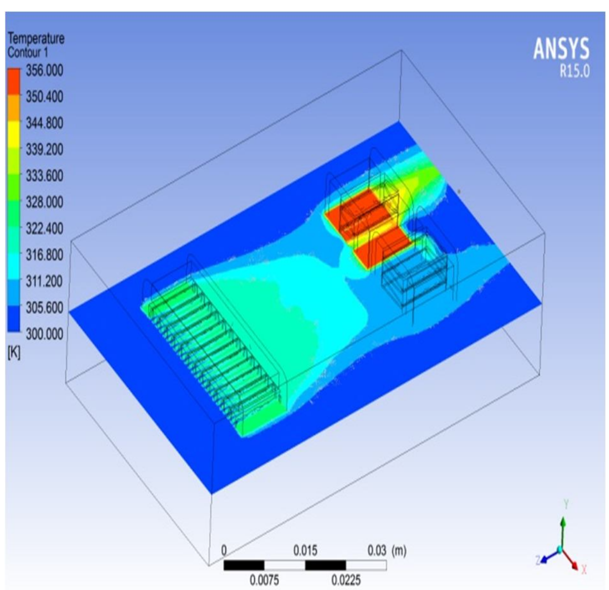


Figure 3.27 Contour Plot On The (X-Z) Plane, Temperature (Louvered Fin)

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 6.887
Volume 6 Issue VI, June 2018- Available at www.ijraset.com

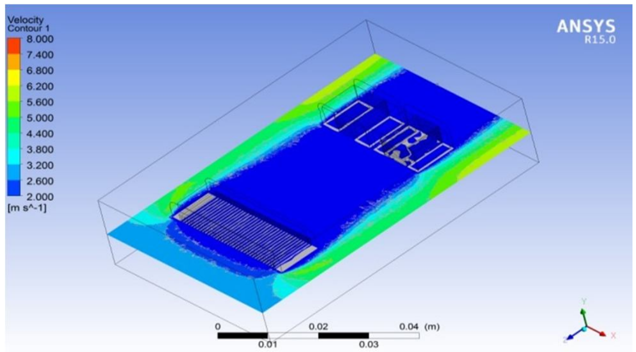


Figure 3.28 Contour Plot On The (X-Z) Plane, Velocity (Louvered Fin)

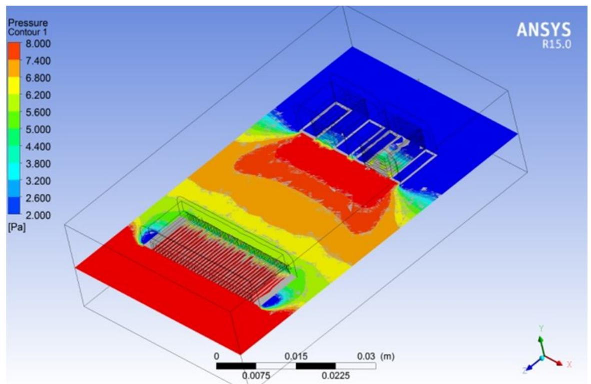


Figure 3.29 Contour Plot On The (X-Z) Plane, Pressure (Louvered Fin)

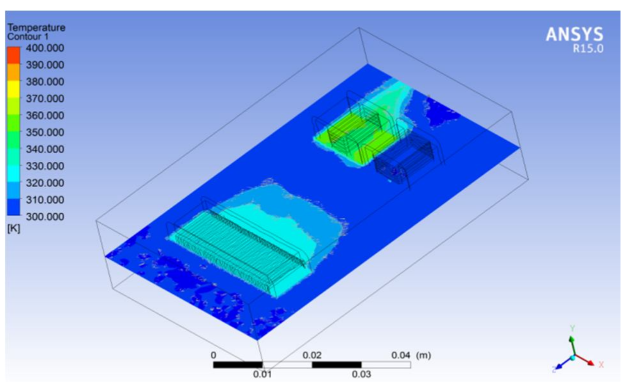


Figure 3.30 Contour Plot On The (X-Z) Plane, Temperature (Square Fin)

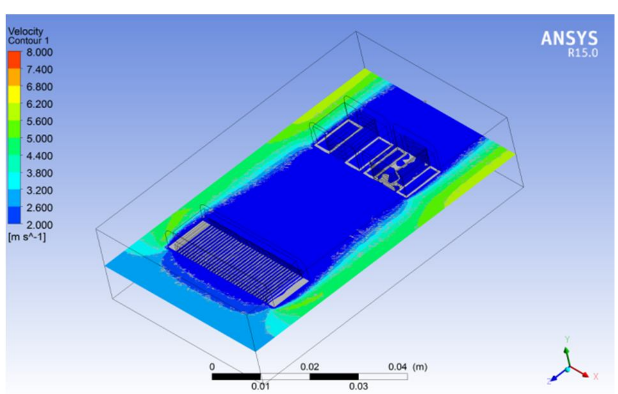


Figure 3.31 Contour Plot On The (X-Z) Plane, Velocity (Square Fin)

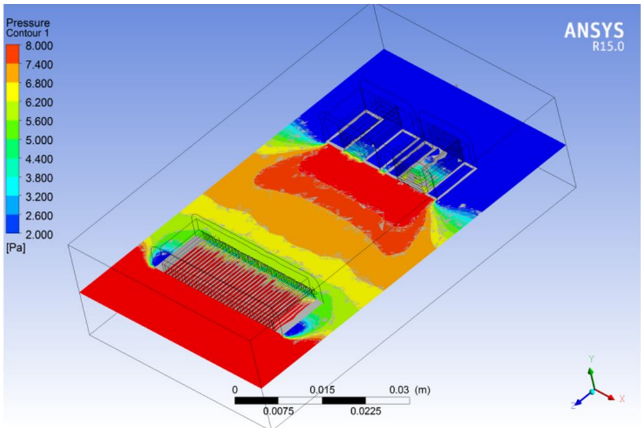


Figure 3.32 Contour Plot On The (X-Z) Plane, Pressure (Square Fin)

The outlet parameters in 30mm distance with given inlet parameters.

PARAMETERS	SQUARE FIN	LOUVERED
TEMPERATURE (K)	304.081	304.651
VELOCITY (m/s)	3.182	3.14533
PRESSURRE (Pa)	-0.0287401	-0.0252043

TABLE3.7 OUTLET PARAMETERS AT 30mm

III. CONCLUSION

A. The efficiency of the louvered fin radiator is more than the square fin radiator. In the case of group radiator also the louvered fin radiator gives more heat dissipation than square fin radiator.

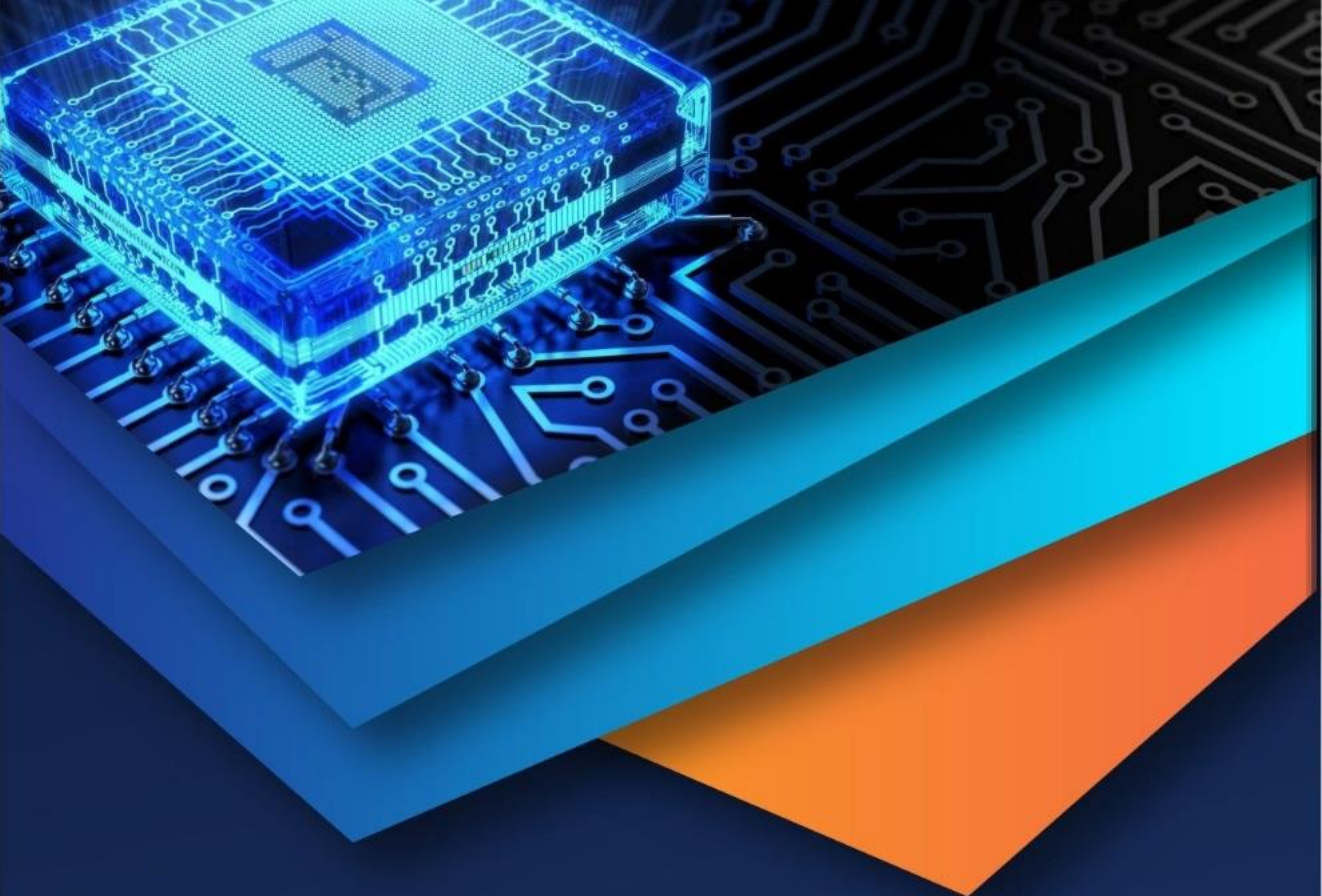
B. The square fin radiator is better for engine cabin cooling than the louvered fin radiator. If the square fin radiator efficiency is sufficient for the singular radiators, then square fin radiator will be good for the other components.

C. The distance between the radiators in group radiator is an important factor in heat dissipation. The heat dissipation decreases as the distance between the radiators increases for both square fin and louvered fin radiators.

D. The change in inlet temperature plays an important role in heat dissipation in the group radiators.

REFERENCES

[1] Alok Vyas, Dr. Alka Bani Agarwal(February 3013) Offset-Strip Fin Heat Exchanger A Conceptual Review Study, IJERA, Vol.3, Issue1
[2] Chetan Kulkarni, Deshpande M.D, Umesh S, Chetan Rawal (April 2012) Underhood Flow Management Of Heavy Commercial Vehicle To Improve Thermal Performance, Vol 11, Issue 1, Apr 2012
[3] J R Patel, A M Mavani(January 2014) Review Paper On CFD Analysis Of Automobile Radiator To Improve Its Thermal Efficiency, IJSRD, Vol.2, Issue01.
[4] Vinod M Angadi (August 2014) CFD Analysis Of Heat Transfer Enhancement Of A Car Radiator Using Nano Fluid As A Coolant, IJERT, Vol.3, Issue08.
[5] Salvio Chacko(2005) Numerical Simulation For Improving Radiator Efficiency By Air Flow Optimisation, Scribd, Tata Technologies Limited, Pune, India.
[6] Babu Male(December 2015) Performance Improvement Of An Automobile Radiator Using CFD Analysis, Vol 2, ISSNO: 23484845.
[7] Karthikeyan V(May 2015) Heat Transfer Enhancement Analysis Of Micro Channel, IJESRT, ISSN NO: 2277-9655.
[8] Shui Chang Liu, Zheng-Qi Gu, Yong Zhang, Wang Dong Zhao(October2014) Performance Calculation Of A Vehicle Radiator Group Based On CFD Simulation With Modified Standard Functions, SAE Technical Paper 2014-01-2586, 2014.



10.22214/IJRASET

INDEX COPERNICUS
45.98

IMPACT FACTOR:
7.129

TOGETHER WE REACH THE GOAL
IMPACT FACTOR:
7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY