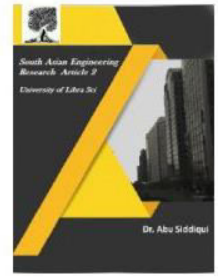
International Journal For Recent Developments in Science & Technology

2581-4575

Crossref

A Peer Reviewed Research Journal



DESIGN OPTIMISATION AND ANALYSIS OF EXISTING IC ENGINE RADIATOR

MAHIJA SANDEEP[1], RAJESH[2]

1. M.tech Student, Thermal Engineering , Narasimha Reddy Engineering college, Hyderabad.
2. Assistant Professor, Mechanical Engineering , Narasimha Reddy Engineering college, Hyderabad.

Sandeep.mahija@gmail.com ,rajesh.sc32@gmail.com

ABSTRACT

In continuous development and research, an automotive industry is trying to increase the performance of efficiency engines. A high efficiency engines meant not only bases on its performance but also for good fuel economy, good looking and attractive design and to maintain less emission rate.Radiator is important part of the internal combustion engine automotive cooling system. The manufacturing and maintenance cost of the radiator is 25 percent of whole cost of the engine components. So improving overall performance and reducing cost of radiators are necessary needs to be research and development on specific region. For higher cooling capacity of Radiator, by adding fins is one of the approaches to increase the cooling rate of the radiator and it leads to weight increase in the automobileHere, in this documentary project we dealt with the designing the radiator by doing geometrical modification. We did not involve with variations with mass flow rate, heat transfer fluids. We applied one of the case studies shows how radiator can be modified and we did as we required the design. An automotive radiator (Tube Cube Concept) model is modeled on modeling software CATIA V5 & SOLIDWORKS 2017 and performance characteristics of model and evaluation is done on processing software ANSYS WORKBENCH. The temperature contour and velocity contour distribution of coolant and air are analyzed by using Computational fluid dynamics environment software CFD. Results have shown that the modified tube cube concept is having good results with existing radiator.

Keywords: Radiator, Turbo Tube cube concept, SOLIDWORKS, ANSYS WORKBENCH.

1. INTRODUCTION

A radiator could be a sort of warm exchanger. It is outlined to exchanging warm from the hot coolant and that streams through it to air blown by the fan. Most modern cars utilize the material aluminum. Those radiators are manufactured by brazing lean aluminum material balances to smooth the radiator aluminum circular tubes. The coolant fluid or air streams from the channel where it enters to the outlet through numerous tubes mounted in a parallel arrangement course of action. The circular tubes in some cases have a sort of blade embedded into them called a turbulator, which increments the turbulence of the liquid streaming through the tubes. On the off chance that the liquid streamed exceptionally

International Journal For Recent Developments in Science & Technology
2581-4575
Crossref
A Peer Reviewed Research Journal

easily through the circular tubes, touching the tubes would be cooled specifically. The sum of warm exchanged to the tubes from the liquid running e contrast in temperature between the tube and the fluid touching it. So the liquid that's in contact with the circular tube cools down rapidly, less warm will be exchanged. By making turbulence interior the tube, all of the liquid blends together, keeping the temperature of the liquid touching the tubes up so that more warm can be extricated, and all of the liquid interior the tube is utilized effectively. Radiators ordinarily within the picture over, you'll be able see the channel and outlet where the oil from the transmission enters the cooler.

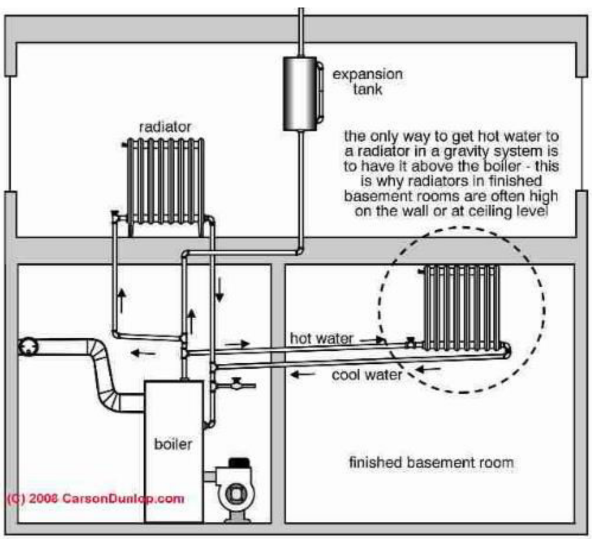


Figure 1.1: Working of the radiator

2. LITERATURE REVIEW

Oliet et al [1]. (2007) considered diverse components which impact radiator execution. It incorporates discuss, blade thickness, coolant stream and air inlet temperature. It is capture that warm exchange and execution of radiator unequivocally influenced by discuss & coolant mass stream rate. As air and coolant stream increments cooling capacity moreover increments. When air channel temperature increments, the warm exchange and thus cooling amount diminishes. Littler blade dispersing and more noteworthy louver blade point have higher warm exchange. Blade thickness may be increased till it pieces air stream and warm exchange rate diminished.

Sulaiman et al [2].. (2009) uses the computational Fluid Dynamics (CFD) modeling reenactment of discuss stream distribution from the automobile radiator fan to the another radiator. The task attempted the demonstrate the geometries of the fan and its environment is the primary step. It’s about appear that the outlet discuss speed is 10 m/s. The blunder of average outlet discuss speed is 12.5 % due to distinction within the tip of the fan hub.

Chacko [3]... (2005) utilized the concept that the effectiveness of the vehicle cooling framework emphatically depend on air stream towards the radiator center. A clear understanding of the stream design interior the radiator cover is required for optimizing the radiator cover shape to increment the stream toward the radiator center, in this manner progressing the warm efficiency of the radiator. CFD examination of the baseline design that was approved against test information appeared that irreplaceable region of re-circulating stream to be interior the radiator cover. This distribution diminished the stream towards the radiator center, driving to a notoriety of hot discuss pinches near to the radiator surface and consequent disfavor of radiator warm productivity. The CFD make able optimization driven to radiator cover setup that eliminated these distribution range and

International Journal For Recent Developments in Science & Technology
2581-4575
Crossref
A Peer Reviewed Research Journal

expanded the stream towards the radiator center by 34%.

Jain et al. [4]. (2012) showed a computational fluid dynamics (CFD) modeling of discuss the ponder uncovered that a left oriented edge fan with counterclockwise revolution 5 performed the same as a right arranged edge fan with pivoting the left to right direction. The CFD comes about were in agreement with the test information measured amid physical testing.

Singh et al. [5]. (2011) examined almost the issues of geometric parameters of a centrifugal fan with in reverse- and forward-curved blades has been assessed. Centrifugal fans are utilized for moving forward the warm scattering from the inner combustion engine surfaces. The parameters examined in this ponder are number of edges, outlet point and breadth proportion. Within the run of parameters considered, forward bended edges have 4.5% lower productivity with 21% higher mass stream rates and 42% higher power consumption compared to in reverse bended fan. Test examinations recommend that motor temperature drop is significant with forward bended edge fan with inconsequential impact on mileage. Consequently, utilize of forward fan is suggested on the vehicles where cooling necessities are tall about propose that fan with distinctive edges would appear same an activity underneath high-pressure coefficient. Increment within the number of edges increments the stream coefficient take after by increment in control coefficient due to better stream direction and decreased losses.

Kumawat et al. [5]. (2014) defined approximately the essential stream fans, whereas unable of expanding tall weights, they are well relevant for dealing with expansive volumes of discuss at comparatively moo weights. In common, they are moo in taken a toll, have great proficiency and airfoil shape. Hub stream fans appear great efficiencies, and can to work at tall inactive weights in case such operation is necessary. The introduction of a hub fan was mimicked utilizing CFD results were displayed within the shape of speed vector and streamlines, which given real stream characteristics of discuss around the fan for distinctive number of fan edges. The different parameters comparative temperature, weight, fan clamor, turbulence and were too considered whereas performing CFD examination. The study uncovered that a fan with an ideal number of fan edges performed well as compared to the fan with less number of fan blades. In common, as a compared between the proficiency and taken a toll, five to 12 edges are great down to earth arrangements.

Jama et al. [6]. (2014) The wind stream dissemination and non-uniformity over the radiator of a full measure Comes about from these tests have shown the most excellent strategy for protecting the front conclusion of the vehicle in terms of wind current balance to be the even way taken after by the vertical strategy. These protecting strategies moreover created the tall normal wind current speed over the radiator which is analogous to way better cooling. It appeared that the strategy to shield the front-end of a traveler vehicle would be to utilize a flat

International Journal For Recent Developments in Science & Technology
2581-4575
Crossref
A Peer Reviewed Research Journal

method. This protecting strategy created the tall uniform cooling wind current dissemination coordinated to the other strategies. By extension it ought to moreover create the lesser decrease in cooling capacity for a given admissions zone.

Leong et al. [7]. (2010) portrayed utilize of Nano fluids based coolant within the motor cooling framework and its impact on cooling capacity. It is found that Nano-fluid having higher thermal conductivity than base coolant like 50% water and 50% ethylene glycol. It increments warm exchange. So for same warm transfer, radiator center region can be diminished coordinated to base one. It finds better arrangement to play down range. Warm execution of a radiator utilizing Nano liquids is expanded with increment in pumping control required compared to same radiator utilizing ethylene glycol as coolant.

Trivedi et al. [8]. (2012) outlined the impact of pitch tube for best arranged radiator for ideal introduction. Warm transfer increases as the surface range of the radiator center is expanded. This leads to alter the geometry by adjusting arrange of tubes in car radiator to extend the surface range for more prominent warm exchange. The alteration in arrange of tubes in radiator is carried out by examining the impact of tube pitch by CFD investigation. Comes about Appears that as the tube pitch this diminished or expanded than optimum pitch of tubes, the warm exchange rate increments. So it can propose that ideal productivity is coming at the pitch of 12 mm.

Yadav et al. [9]. (2011) displayed parametric think about on car radiator. Within the activity assessment, a radiator is introduced into a test setup. The different parameters counting channel coolant temperature, mass stream rate of coolant, and etc. are shifted. Taking after remarks are watched amid learning. Impact of coolant mass stream cooling capacity of the radiator has clear connection with the coolant stream rate. With an increment within the value of cooling stream rate, comparing increment within the esteem of the viability and cooling capacity. Impact of coolant gulf temperature is increment within the gulf temperature of the coolant the cooling capacity of the radiator increments.

Bozorgan et al. [10]. (2012) This paper displayed a numerical examination of the utilize of copper oxide water nanofluids as a coolant in a radiator of Chevrolet Rural IC motor with a given warm trade and pumping control for CuO water capacity. The nearby convective in general warm exchange coefficients Nano liquid at distinctive volume divisions (0.1% to 2%) was of the coolant Reynolds number and the considered beneath turbulent stream conditions. Too the impacts car speed on the radiator performance are consider within the work. The simulation results show that the full warm transfer coefficient of Nano liquid is better than that of water alone.

3. CONCEPT ASSESSMENT_ TURBO TUBE CUBE CONCEPT

The turbo circular tube cube shape concept permits for an expansive surface region to disseminate warm mutual with a turbo-charged to extend gust stream over the coolant tubes. These two plan thoughts. The turbocharger will have adjustable geometry in arrange to keep steady wind current amid sit still and moo engine speeds. In arrange to induce a get a handle on how huge this concept would truly ought to be, we utilized a basic scientific demonstrate to appraise the total surface zone required to dismiss 147kW of heat.

For the straightforward scientific show, we expected a straight tube with a consistent rate of cooling. We utilized Newton's Law of Cooling to appraise the surface zone required to dissipate the 147kW of warm. In this condition Q = 147.5 kW, h is the convection liquid (we assumed h=100 W/m2K), and T0 is the liquid temperature and Ta is the encompassing temperature (assumed to be 25°C). We accepted h since that's the esteem we are anticipating for our model (we are incapable to calculate it explicitly because we don't have the measurements of the plan).

$$Q = hA(T_0 - T_a)$$

After running through the calculations, we found a basic 3d shape with circular tubes to be as well large to meet our prerequisites. In this manner we changed our plan marginally to compensate for the required surface range. We utilized a finned tube plan which gave us two to three times the surface zone on the tubes. We moreover occupied from the cubic plan to a more rectangular design with bowed tubes that permitted us to create the tubes somewhat longer without expanding the length of radiator. The bowed tubes too permit us to center the turbocharger into the inner curved segment and accomplish decently reliable wind current over the tubes.

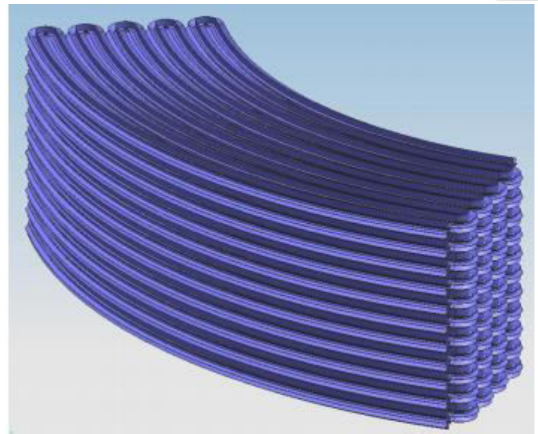


Figure 3.1: Isometric CAD 3D model of the circular Tube Cube Concept.

3.2.1 Design evaluation

Upon advance thought, both of our selected concepts were in feasible. The tube 3d shape idea required as well numerous coolant tubes, making the modern plan much better than current designs. The cooling concept would require a bigger condenser to attain the specified warm dismissal.

3.2.2 Reconsideration

Due to the not availability of feasibility of our concepts, we chosen to rethink the utilize of carbon foam in our plan. Carbon froth may be a absorptive froth, which is made from coal. When warmed in abundance of 2000°C, the carbon takes the shape of graphite material, which is the essential fabric in carbon (graphite) foam.

3.2.3 Advantages

Carbon froth gives an expansive surface range per unit volume due to huge and various pores. This expansive surface zone will increment the surface region un covered to the air and hence decrease air side resistance. Carbon froth is exceptionally lightweight when compared to ordinary materials used in current radiators (aluminum or copper alloy). This can be

International Journal For Recent Developments in Science & Technology
2581-4575
Crossref
A Peer Reviewed Research Journal

fabricated from a square to any desirable shape by implies of processing, cutting, boring, etc. Carbon froth too could be a sponge-like material, which is more strong compared to aluminum blades.

3.2.4 Disadvantages

The major drawback related with carbon head is that it is costly to deliver, with a commercial taken a toll around $5.00 per cubic inch. Be that as it may, modern generation strategies seem possible to lower the cost within the close future. Too, the numerous little pores in carbon froth can become clogged with street flotsam and jetsam or creepy crawlies, but a sifting shelter ought to keep the froth clean for our application. It too requires extra bracing for bolster.

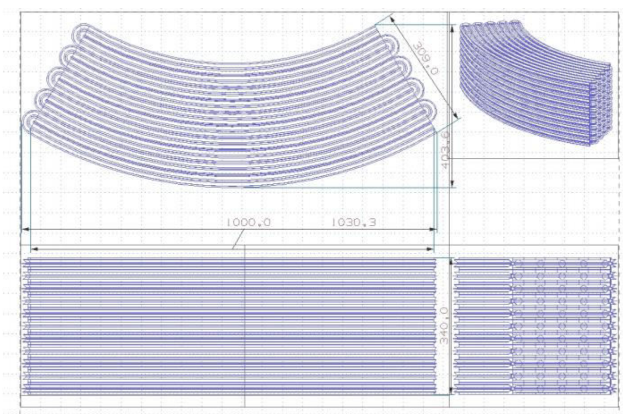


Figure 3.2: Drawing views of Concept design radiator

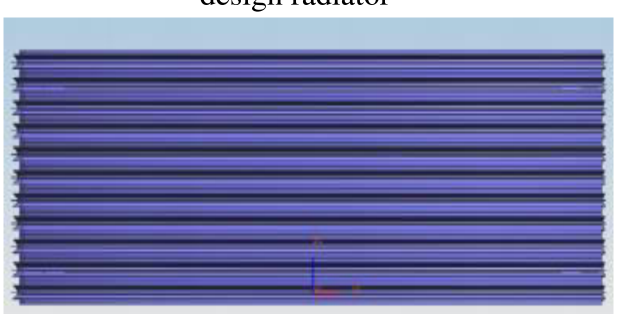


Figure 3.3: Front view of radiator

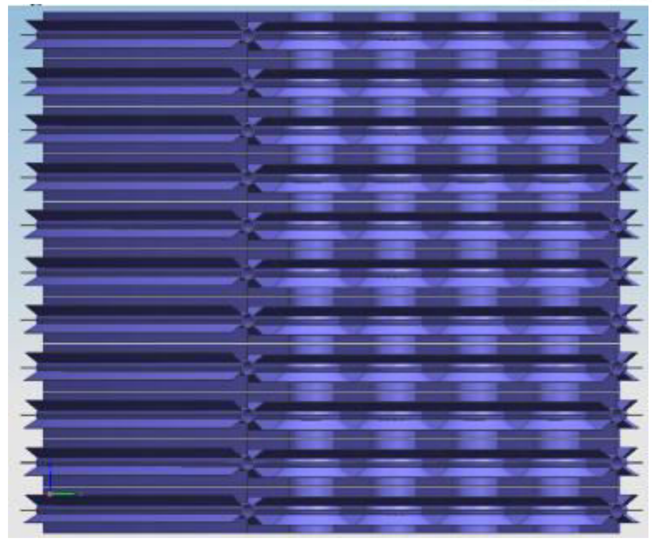


Figure 3.4: Right view of tube cube radiator

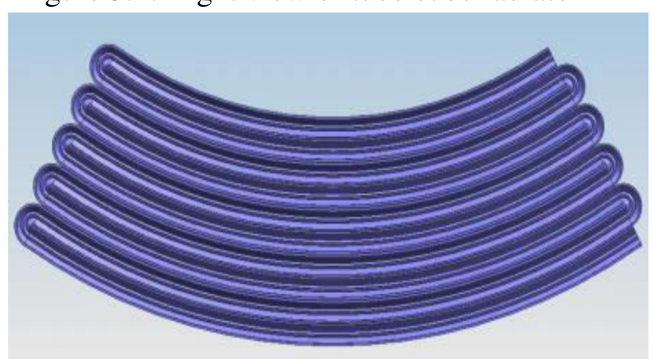


Figure 3.5: Plan view of radiator

4.1 DESIGN OF RADIATOR

4.1.1 Design of Existing Radiator:

Dimensions are measured for existing radiator and modelled as shown in below.

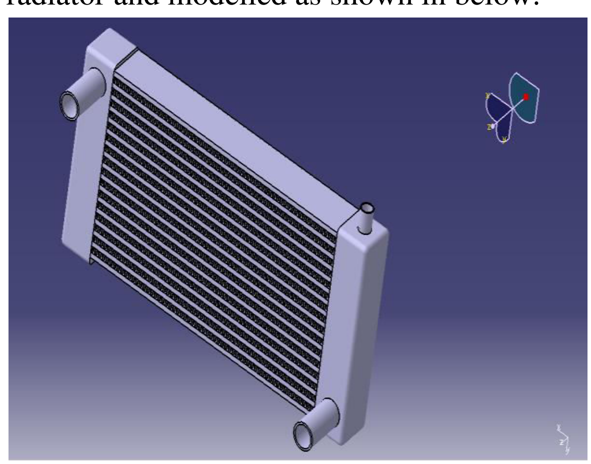


Figure 4.1: Isometric view of radiator

2581-4575
International Journal For Recent Developments in Science & Technology
Crossref
A Peer Reviewed Research Journal

4.1.2 NEW CONCEPT DESIGN OF RADIATOR:

Step1: Select a right plane and a draw a sketch for the path.

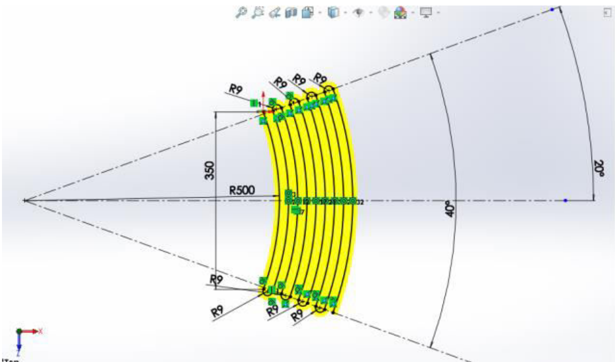


Figure 4.2: Path sketch for new concept radiator design

Step 2: Do sweep by selecting profile and path.

Figure 4.3: Isometric view of new concept radiator design

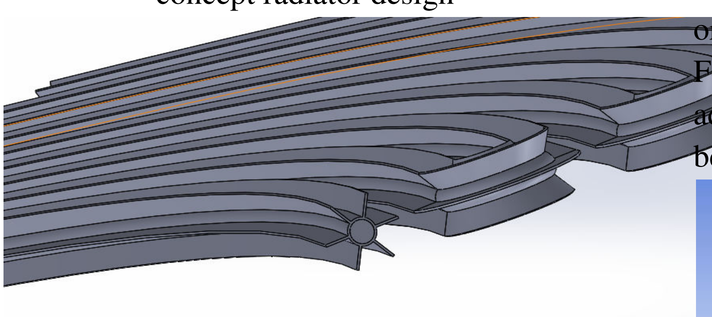


Figure 4.4: Section view of new concept radiator design

Step 3: Create domain for analysis purpose.

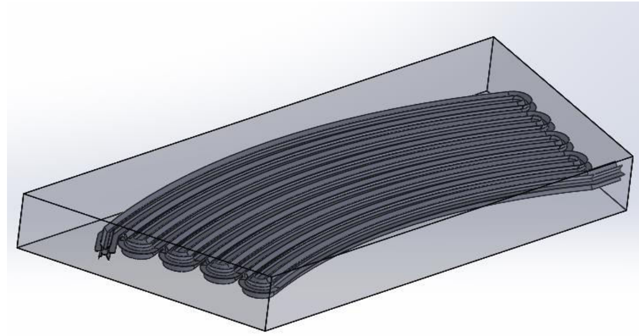


Figure 4.22: Isometric view of radiator with domain

5 COMPUTATIONAL ANALYSIS OF RADIATOR

5.1 PROCEDURE FOR ANALYSIS ON EXISITNG RADIATOR

In this ANSYS work bench we are going to do the CFD analysis on the convective plates which are designed in SOLIDWORKS 2017. To perform the CFD analysis we need to follow these steps:

Step1: Now go to main window -> right click on the geometry -> import the geometry from external file (Radiator) which is saved in stp form. Then click on generate.

Step 2: Right click on the mesh tab to generate mesh.

After this it generated the automatic or default mesh over the body will generated. For getting the more accurate of the meshing adding the sizing to the parts as shown in below figure.

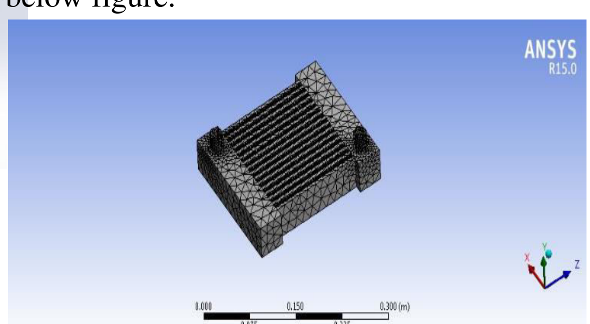


Figure 5.1: Radiator with meshing

International Journal For Recent Developments in Science & Technology
2581-4575
Crossref
A Peer Reviewed Research Journal

Step 3: Right click on face or edge -> click Create Named selection -> Give the names for faces or blocks as shown in below figures.

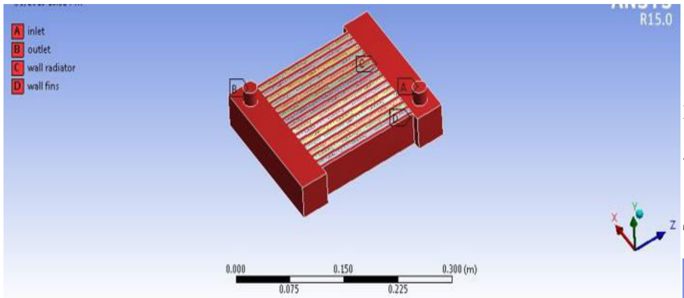


Figure 5.2: Named Selection for Radiator

Step 4: General -> type -> select Pressure based -> velocity formulation -> select absolute.

Models -> energy is on -> select viscous K-Epsilon (2 Eqn) -> click ok.

- ➔ Inlet -> Velocity inlet -> Value as 1 m/s.
- ➔ Solid wall should be wall ->Click on edit ->go to thermal tab -> Change the material -> give the Temperature.
- ➔ Outlet -> Keep it as Pressure outlet -> Use default parameters.
- ➔ Interior part -> Interior -> Use default parameters.
- ➔ Wall part -> Wall -> Use default parameters.
- ➔ Reference values -> compute from -> select inlet
- ➔ Solution -> solution methods -> formulation -> select implicit ->flux type as Roe-fds -

>gradient -> select least square cell based ->flow as second order upwind ->modified

Turbulent viscosity as first order upwind.

If you want more accurate values click -> monitors ->residuals, print, plot -> give the

Values as in order of 10^-6 -> click ok.

- ➔ Solution controls-> give courant number as less than 1
- ➔ Solution initialization ->standard initialization -> compute from inlet ->reference

Frame as relative to cell zone -> initialize.

5.2 ANALYSIS FOR NEW CONCEPTUAL DESIGNED RADIATOR

The process will repeat as explain as above.

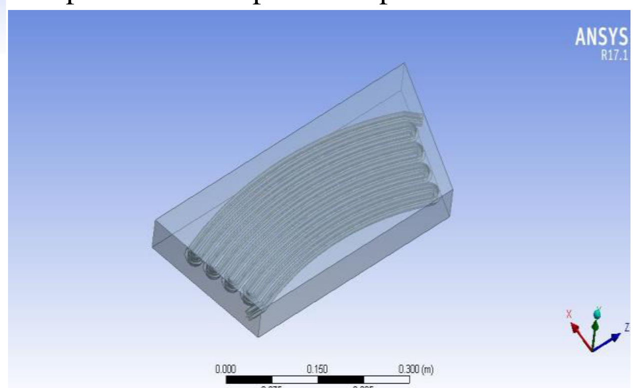


Figure 5.3: Imported Geometry in Ansys

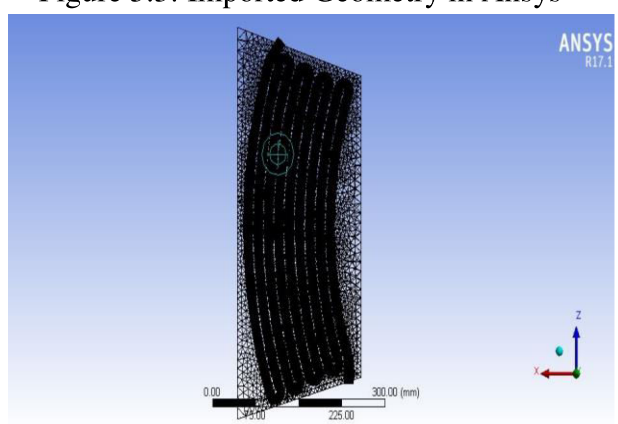
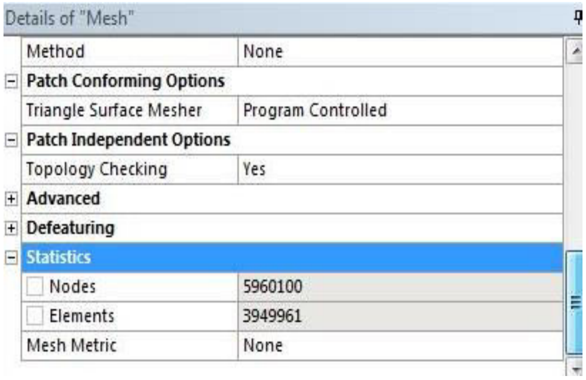


Figure 5.4: Radiator meshing

International Journal For Recent Developments in Science & Technology
2581-4575
Crossref
A Peer Reviewed Research Journal



Details of "Mesh"	
Method	None
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
Patch Independent Options	
Topology Checking	Yes
Advanced	
Defeaturing	
Statistics	
Nodes	5960100
Elements	3949961
Mesh Metric	None

Figure 5.5: Details of Mesh

6 RESULTS & DISCUSSION

For iterations, it takes more time as it depends on number of iterations.

After completion of iterations, see the results as follows.

- ➔ Setup ->Results ->graphics and animations -> contours -> select wall Fluxes -> click on thermal related results like surface heat transfer, heat transfer coefficient, Surface Stanton Number and Surface Nusselt number etc. > click display

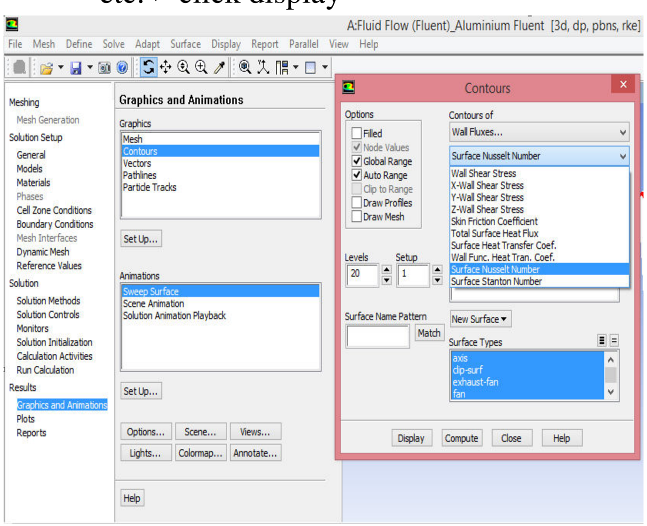


Figure 6.1: Setup for Results in Fluent

We can also get results from Results tab in Ansys main window. For this follow the method as follows.

- ➔ Go to Ansys main window -> double click on Results ->New tab will open.
- ➔ Click on part details (Named selections given in the mesh) -> Click the boxes.
- ➔ Now, go to Insert -> Location -> Iso surface -> give the variable as your own. (X, Y or Z Planes) -> Click apply.
- ➔ Then, go to Insert -> Contour -> give the location as iso surface -> give the variable related to temperature -> Click apply -> Now, see the results.

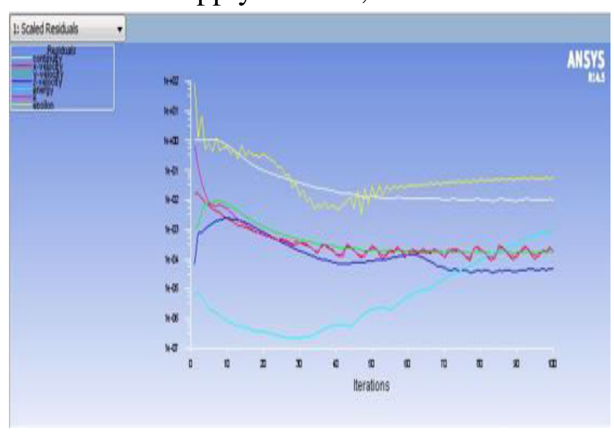


Figure 6.2: Scaled residuals

6.1 POST PROCESSING OF EXISTING RADIATOR

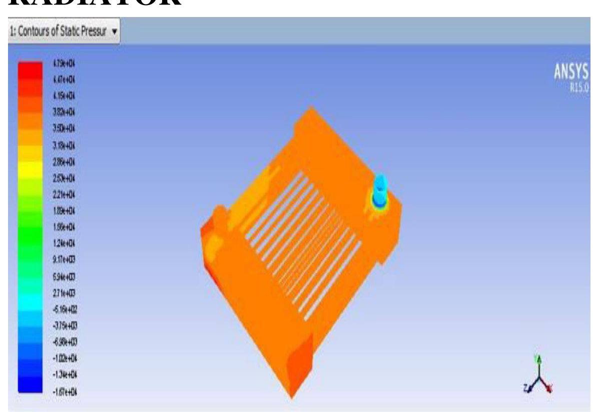


Figure 6.3: Contours of static pressure

2581-4575

International Journal For Recent Developments in Science & Technology

Crossref

A Peer Reviewed Research Journal

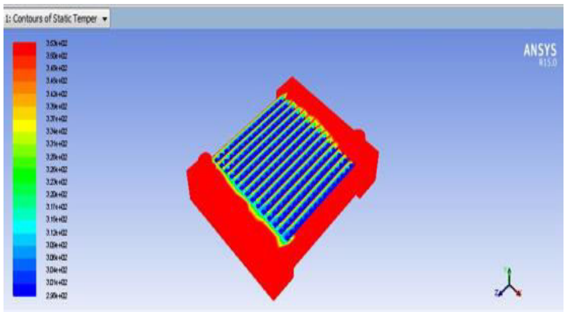


Figure 6.4: Contours of static temperature

6.2 POST PROCESSING OF CONCEPTUAL RADIATOR

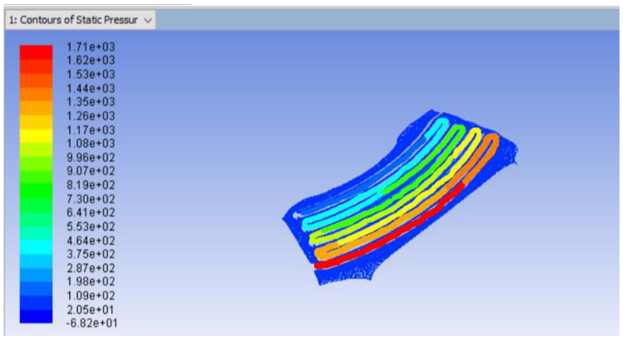


Figure 6.5: Contours of static pressure

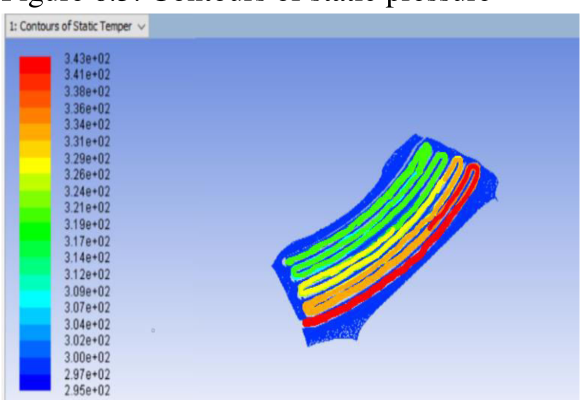


Figure 6.6: Contours of static temperature

S.No.	Parameter	Existing Radiator design	Conceptual radiator design
01	TEMPERATURE(C)	83.467	86.918
02	THERMAL ERROR	1.4563e6	1.6542e6
03	HEAT FLUX(W/mm2)	1.468	0.9854

CONCLUSION

In this paper, the aim is to design a new concept for existing an automotive radiator. It was studied with new concept design for the radiator and we compared with existing radiator. We found that the new concept design is good compare to existing radiator. If we see manufacturing capability between these two, found that existing radiator is good but in the performance wise tube cube concept is best for using like radiator.Heat transfer analysis is performed to analyze the heat transfer rate to determine the thermal flux. The material taken is Aluminum alloy 6061 for thermal analysis. By observing the thermal analysis results, thermal flux is increased by 14% for the modified model.So it can be concluded that modifying the radiator model yields better results.

REFERENCES

[1] Golakiya Satyamkumar, Sarvaiya Brijrajsinh, Makwana Sulay. (2006), 'Analysis of Radiator with Different types of Nano fluids by using ansys', Engineering Research and Studies, vol. 6, no. 1, pp. 1-3.

[2] Jain S and Deshpande Y. (2012), 'Design & CFD Analysis of a Radiator Axial Fan for Air Flow Distribution', World Academy of Science, Engineering and Technology, vol. 6, no. 11, pp. 1085–1090.

[3] Jama H, Watkins S, Dixon C. (2004) 'Airflow Distribution through the Radiator of a Typical Australian Passenger automobile Car', Australasian Fluid Mechanics Conference, vol. 15, pp. 1-4.

[4] Kim D, Kwon Y, Cho Y, Cheong S, Hwang Y. (2009) 'Convective heat transfer characteristics of nanofluids under laminar & turbulent flow conditions', Current Applied Physics, vol. 9, no 2, pp. 119-123.

[5] Kumawat H. (2014), 'Modeling and analysis of Axial Fan Using CFD', Mechatronic and Manufacturing Engineering, vol. 8, no. 11, pp. 1858–1862.

[6] Leong K.Y, Saidur R, Kazi S.N and A.H. (2010), 'Performance investigation of a car radiator operated with nanofluid- based coolants (nanofluids as a coolant in a radiator)', Applied Thermal Engineering , vol. 2, pp. 1-10.

[7] Mohd Yusoff Sulaiman, Shamsul Bahari Azraai and Wan Mokhtar Wan Abdullah. (2009), 'CFD Modelling of Air Flow Distribution from a heat exchanger Fan', Proceeding of International Conference on Applications and Design in Mechanical Engineering, pp. 11-13.

[8] Naraki M, Peyghambarzadeh S. M, Hashemabadi S, Vermahmoudi, Y. (2013) 'Theoretical study of overall heat transfer coefficient of CuO/water nanofluids in a car radiator', International Journal of Thermal Sciences, Vol. 66, pp. 82-90.

[9] Navid Bozorgan, Komalangan Krishnakumar, Nariman Bozorgan. (2012), 'Numerical Study on CuO-Water Nano fluid in Automotive Diesel Engine Radiator', Modern Mechanical Engineering, vol. 2, pp. 130-136. 57.

[10] Nguyen C.T, Roy G, Gauthier C, Galanis N. (2007) 'Heat transfer enhancement using Al2O3-water nano fluid for an electronic liquid cooling system in a heat exchanger', Applied Thermal Engineering, vol. 27, pp. 1501-1506.

[11] Oliet C, Oliva A, Castro J and Pérez-Segarra C.D. (2007), 'Numerical studies on automotive radiators' Applied Thermal Engineering, vol. 27, no. 11–12, pp. 2033–2043.

[12] Peyghambarzadeh S, Hashemabadi S, Hoseini S, Seifi Jamnani. (2011) 'Experimental study of heat transfer enhancement using water/ethylene glycol based nanofluids', International Communication in Heat and Mass Transfer, vol. 52, no 1, pp. 1283–1290.

[13] Sai Sasank P and Naik G. (2014), 'Experimental Studies on Automotive Radiator Performance using Water and Al2O3 Nano fluid with different concentartions', International Journal of Engineering Research & Technology, vol. 3, no. 9, pp. 1347–1350.

[14] Salvio Chacko, Biswadip Shome and Vinod Kumar. (2005), 'Numerical Simulation for Improving Radiator Efficiency by Air Flow Optimization using Ansys', Engineering Research Center, pp.1-10.

[15] Singh O. P, Khilwani R, Sreenivasulu T and Kannan M. (2011), 'Parametric & Numerical Study of Centrifugal Fan Performance : Experiments and Numerical Simulation', International Journal of Advances in Engineering & Technology, vol. 1, no. 2, pp. 33–50.

[16] Trivedi P. K and Vasava N. B. (2012), 'Effect of Variation in Pitch of Tube on Heat Transfer Rate in Automobile Radiator by Computational Fluid Dynamics', vol. 1, no. 6, pp. 180–183.