

# Performance Analysis of Automobile Radiator

Shailendra Singh Verma

Lecturer, Department of Automobile Engineering, Kalaniketan Polytechnic College Jabalpur, India

**ABSTRACT:** In a car, a radiator is a device that is used to transmit heat energy from one medium to another. They're mostly used in the built-in functions of cars and other devices. The radiator is a heat transmission source to the environment that can be accomplished by transferring heat from the automobile's engine to the environment. The radiator, as its name suggests, uses convection to transfer a large amount of heat to the surroundings. The primary function of an automotive radiator is to cool the engine by bypassing the coolant through cylinder water jackets. Solidworks is utilised in this article to do a CFD analysis of a radiator at various flow rates. The CFD analysis of a radiator is performed in this paper using regular water and coolant. The automobile radiator is constructed out of aluminium and brass.

**KEYWORDS:** Modeling; Analysis; Water; Coolant; Solidworks; Fins

## I. INTRODUCTION

As we all know, car speed and performance are growing in comparison to older vehicles, while vehicle size is shrinking and more modifications are being made for better performance and comfort. Because of the increase in vehicle speed, cooling systems are required to cool the engine. Radiators are utilised for cooling because they are the most significant component of the engine. We know that the movement of the piston, the crankshaft, and the connecting rod generates heat. With the help of the engine oil and exhaust gases, it will release some heat. The cooling system dissipates the remaining heat inside the engine, allowing the temperature to be maintained and preventing the engine from overheating.

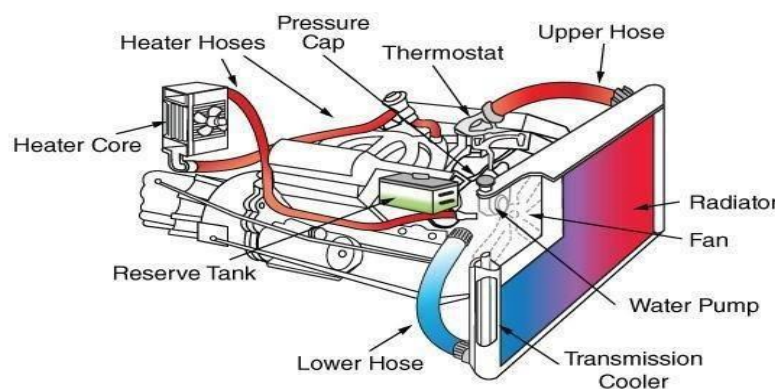


Fig1: - The working condition of the radiator (GOOGLE)

The thermostat is used for measuring the temperature. The working of the thermostat is to release the coolant at a time when the temperature of the engine increases and absorb the heat make it cool and release it to the surrounding and make the engine cool. Nowadays the Aluminum is used for making radiator fins. Inside the radiator, small fins tube the hot water

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

is supplied from the engine through inlet valve and then passes through tubes which led to the decreasing the temperature of the liquid and make the water cool and the supplies it through the outlet valve to the pump and the pump suppliers to the engine. A tabulator is present inside the tubes which are made of aluminum films. The liquid gets cooled faster as it gets more and more in contact with films if the speed of the liquid is slow. As we know that the more the liquid touches the films which led to the transfer of more heat to the environment. For the better transfer of heat, turbulence must be created in the liquid for the touching of more and more liquid to the films which led to the transfer of more heat to the environment. If the transfer of heat from the liquid to the environment is more the more heat will be absorbed from the engine and the performance of the engine will increase.

### Type of radiator

There are two types of radiator

- 1) Tabular type
- 2) Cellular type core

#### Tabular type

It is the series combination of upper and lower tanks through which water passes. Fins are attached to improve heat transfer around the tubes. Outside of the tubes, the air is passed between the fins that absorbing heat from the water. The water passes through all the tubes the cooling effect of the entire tube is lost if one of the tubes becomes clogged. On a cellular radiator, the clogging of any passage results in a loss but of a small part of the total cooling surface.

#### Cellular type core

Air passes through the tubes and the water flows in the spaces between them in cellular type core. The core contains a large number of air cells that are surrounded by the radiator. It is known as a honeycomb radiator because of its appearance as the cells in front are hexagonal. On a cellular radiator, passage clogging results in a loss but small areas will be affected by it. It consists of many small tubes equipped with a honeycomb-like structure of fins to dissipate heat rapidly and cools hot liquid from the engine.

#### Parts of radiator

- 1) Upper tank
- 2) Lower tank
- 3) Tubes
- 4) Filler cap
- 5) Fins

### Upper Tank:

Due to absorbing heat from the engine coolant get hot, the liquid expands and creates pressure in the radiator additionally.

The pressure causes the coolant to get higher than the pressure cap, to prevent leakage excess coolant needs to be captured somewhere. The excess fluid flows into the pipe and goes into the overflow tank. When the driver parks turn off the engines of the vehicle and the heat dissipates which causes the coolant. The coolant will then contract instead of expanding; resulting in the volume of the coolant. The vacuum effect will take place where the pressure decrease allows the excess coolant in the overflow tank to flow back so it can return to the radiator. Tanks could also be made of brass, plastic, a polyamide).

### Lower Tank:

Just after it has passed through the heat radiating tubes and fins in the body of the radiator the bottom tank receives the cooling water. The significant temperature takes place. From the block, the thermostat releases water at 180 degrees

Fahrenheit. That water can lose as 100 def. F. depending on the ambient air temperature and the efficiency of the radiator by the time it reaches the bottom tank. The water pump holds this cooled water. It will back into the block where it is again heated up.

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

### **Tubes:**

On its way to the opposite tank, as the coolant passes through the radiator tubes, it transfers heat to the tubes that transfer the heat to the fins that are attached between the rows. The fins head the heat flow to the ambient air. Radiator tubes are made up of brass. The use of aluminum increased, eventually taking over the vast majority of vehicular radiator applications.

### **Filler cap:**

Since the coolant expands the high coolant temperature leads to an increase in pressure in the cooling system. Coolant is press in the tank that will increase the pressure in the tank.

A pressure relief valve in the filler cap opens. It allows air to escape. Partial vacuum forms in the cooling system when the coolant temperature returns to normal. This causes a vacuum in the tank because the Coolant is extracted from the tank.

### **Fins:**

Fins are surfaces that are used to increase the rate of heat transfer to or from the environment and they extend from the surface by increasing convection. Fins increase the surface area and can be an economical solution to heat transfer problems.

### **COOLANT**

Its primary function is to keep the engine at a constant safe operating temperature but also, with the addition of other chemicals called inhibitors, to control the formation of scale and rust. Inhibitors also prevent cavitation, that is, oxidation from water passing through the cooling system. Anti-freeze / Anti-Boil coolants lower the freezing point of the coolant while raising the boiling point. Corrosion inhibitors have little or no Anti-Freeze / Anti-Boil properties and are used primarily to protect surface materials in the cooling system from corrosion.

### **TYPERS OF COOLANT**

There are mainly two types of coolant

- 1) Type A
- 2) Type B

Type A has an Anti-Freeze Anti-Boil package included present as Ethylene Glycol or similar, an additive package which raises the boiling point of the water and lowers the freezing point.

Type B does not have an Anti-Freeze Anti-Boil additive package added and are true corrosion inhibitors only with no or little AF/AB properties.

### **Property of coolant**

There are six property of coolant

1. Good heat transfer.
2. Protection against deposits.
3. High-temperature protection.
4. Safe to use with hard waters.
5. Won't damage and be neutral with internal components
6. Reduced foaming tendency.

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

### CALCULATION

In fluid regions SOLIDWORKS Flow Simulation solves the Navies-Stokes equations, which are formulations of mass, momentum, and energy conservation laws:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}(\rho u_i u_j) + \frac{\partial P}{\partial x_i} = \frac{\partial}{\partial x_j}(\tau_{ij} + \tau_{ij}^R) + S_i \quad (2)$$

$$\frac{\partial \rho H}{\partial t} + \frac{\partial \rho u_i H}{\partial x_i} = \frac{\partial}{\partial x_i} (u_j (\tau_{ij} + \tau_{ij}^R) + q_i) + \frac{\partial p}{\partial t} - \tau_{ij}^R \frac{\partial u_i}{\partial x_j} + \rho \epsilon + S_i u_i + Q_H \quad (3)$$

$$H = h + \frac{u^2}{2}$$

For calculation of high-speed compressible flows and flows with shock waves the following energy equation is used:

$$\frac{\partial \rho E}{\partial t} + \frac{\partial \rho u_i \left( E + \frac{p}{\rho} \right)}{\partial x_i} = \frac{\partial}{\partial x_i} (u_j (\tau_{ij} + \tau_{ij}^R) + q_i) - \tau_{ij}^R \frac{\partial u_i}{\partial x_j} + \rho \epsilon + S_i u_i + Q_H \quad (4)$$

$$E = e + \frac{u^2}{2}$$

These equations are supplemented by fluid state equations defining the nature of the fluid, and by empirical dependencies of fluid density, viscosity, and thermal conductivity on temperature. Inelastic non-Newtonian fluids are considered by introducing a dependency whereby their dynamic viscosity is dependent on flow shear rate and temperature.

Special models are used for the description of real gases, volume condensation, and vaporization, cavitations, as well as for porous media.

### II. SOLID WORKS

Flow Simulation can consider both laminar and turbulent flows. Laminar flows occur at low values of the Reynolds number, which is defined as the product of representative scales of velocity and length divided by the kinematic viscosity. When the Reynolds number exceeds a certain critical value the flow transitions smoothly to turbulent. To predict turbulent flows, the Favre-averaged Navies-Stokes equations are used, where time-averaged effects of the flow turbulence on the flow parameters are considered, whereas the large-scale, time-dependent phenomena are taken into account directly. Through this procedure, extra terms known as the Reynolds stresses appear in the equations for which additional information must be provided. To close this system of equations, SOLIDWORKS Flow Simulation employs transport equations for the turbulent kinetic energy and its dissipation rate, using the **k-ε** model.

The modified **k-ε** turbulence model with damping functions proposed by Lam and Bremhorst (1981) describes laminar, turbulent, and transitional flows of homogeneous fluids consisting of the following turbulence conservation

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

laws:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho k u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right) + \tau_{ij}^R \frac{\partial u_i}{\partial x_j} - \rho \epsilon + \mu_t P_B, \quad (5)$$

$$\frac{\partial \rho \epsilon}{\partial t} + \frac{\partial \rho \epsilon u_i}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \left( \mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_i} \right) + C_{\epsilon 1} \frac{\epsilon}{k} \left( f_1 \tau_{ij}^R \frac{\partial u_i}{\partial x_j} + C_B \mu_t P_B \right) - f_2 C_{\epsilon 2} \frac{\rho \epsilon^2}{k} \quad (6)$$

$$\tau_{ij} = \mu s_{ij}, \quad \tau_{ij}^R = \mu_t s_{ij} - \frac{2}{3} \rho k \delta_{ij}, \quad s_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k}, \quad (7)$$

$$P_B = - \frac{g_i}{\sigma_B} \frac{1}{\rho} \frac{\partial \rho}{\partial x_i}, \quad (8)$$

Where  $C_\mu = 0.09$ ,  $C_{\epsilon 1} = 1.44$ ,  $C_{\epsilon 2} = 1.92$ ,  $\sigma_k = 1$ ,  $\sigma_\epsilon = 1.3$ ,  $\sigma_B = 0.9$ ,  $C_B = 1$  if  $P_B > 0$ ,  $C_B = 0$  if  $P_B < 0$ , the turbulent viscosity is determined from:

$$\mu_t = f_\mu \cdot \frac{C_\mu \rho k^2}{\epsilon}, \quad (9)$$

Lam and Bremhorst's damping function  $f_\mu$  is determined from:

$$f_\mu = \left( 1 - e^{-0.025 R_y} \right)^2 \cdot \left( 1 + \frac{20.5}{R_t} \right), \quad (10)$$

Where:

$$R_y = \frac{\rho \sqrt{k} y}{\mu}, \quad (11)$$

$$R_t = \frac{\rho k^2}{\mu \epsilon}, \quad (12)$$

$y$  is the distance from a point to the wall and Lam and Bremhorst's damping functions  $f_1$  and  $f_2$  are determined from:

$$f_1 = 1 + \left( \frac{0.05}{f_\mu} \right)^3, \quad f_2 = 1 - e^{-R_t^2}. \quad (13)$$

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

Lam and Bremhost's damping functions  $f\mu$ ,  $f1$ ,  $f2$  decrease turbulent viscosity and turbulence energy and increase the turbulence dissipation rate when the Reynolds number  $Ry$  based on the average velocity of fluctuations and distance from the wall becomes too small. When  $f\mu =1$ ,  $f1 =1$ ,  $f2 =1$  the approach reverts back to the original  $k-\epsilon$  model.

The heat flux is defined by:

$$q_i = \left( \frac{\mu}{Pr} + \frac{\mu_t}{\sigma_c} \right) \frac{\partial h}{\partial x_i}, \quad i = 1, 2, 3 \quad (14)$$

Here the constant  $\sigma_c=0.9$ ,  $Pr$  the Prandtl Number, and  $H$  is the thermal enthalpy.

A particular computational task is finally specified by the definition of its geometry, boundary, and initial conditions. All data for such conditions are defined directly on the native CAD model.

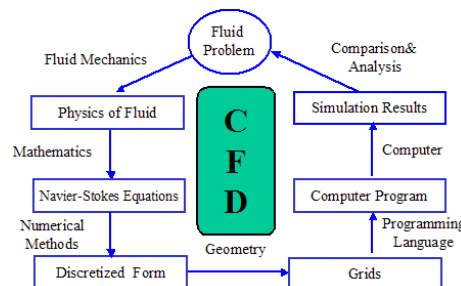


Fig2:-GOVERNING EQUATION

## III. RESULTS

### FINITE ELEMENT ANALYSIS CFD FLOW SIMULATION

Nowadays it is not possible to prepare the model and test it. Designing software is used for preparing models and analyzing

the models. Finite element analysis is one of the major parts of the design

#### Computational fluid dynamics (CFD):-

As the name suggested CFD (computational fluid dynamics) is the study of the fluid flow inside the design body software. Surface conditions, boundary conditions, and materials are also being selected for the design. A Solidworks CFD is also used for the flow simulation.

#### Solidworks Flow Simulation Introduction

As the name suggested Solidworks flow simulation is a part of Solidworks. It is used to perform the flow simulation. Flow

simulation is present in the add-in options in the Solidworks. There are two types of analysis

- 1) Internal flow analysis.
- 2) External flow analysis. Internal flow analysis-

Radiator, valves, pump are all examples of internal analysis studies. Because SolidWorks flow simulation function inside of

Solidworks setting up a computational domain, boundary condition, and heat source can be done in a minute

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

External flow analysis –

It is used when we need to understand the fluid behavior around our model based on the environmental condition, winds turbine. In external analysis the computational domain act as our virtual wind turbine.

### Dimension used for designing of radiator

The following data is 30% less than the original data

Parts	Size
Radiator length	0.45m
Radiator width	0.43m
Radiator height	0.024 m
Tube width	0.024m
Tube height	0.0015m
Fin width	0.024m
Fin height	0.011m
Fin thickness	0.0002540m
Distance between tubes	0.0015m
Number of tubes	33

### 3D-MODEL OF RADIATOR

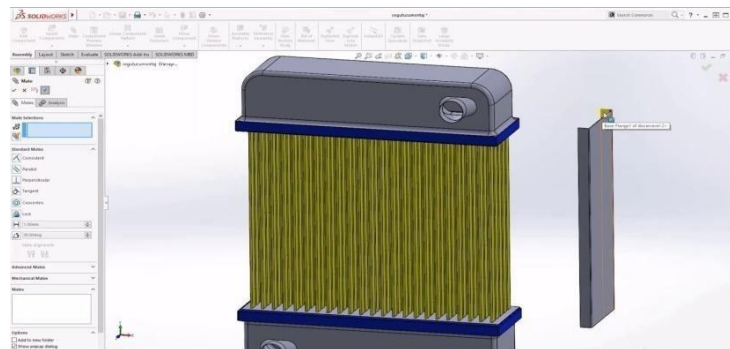


Fig3:- RADIATOR

After completing the designing part the flow analysis has to be performed and the result has to be calculated. During performing the analysis part the two medium has to take

- a) Water medium.
- b) WATER AND Coolant as a medium.

As we see that in the 1<sup>st</sup> case water as a medium has to take. Whereas in the 2<sup>nd</sup> case water and coolant as a medium (1:4)

ratio. After that data has to be collected and comparing both results.

### 1) USING WATER AS MEDIUM General Settings CFD analysis

In the Solidworks when we performed the CFD analysis some general setting has to be selected. In this software, two types of analysis have to be done internal and external analysis is done. It depends on the model what type of analysis has to be selected. After selecting the analysis type the physical features has to be selected.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016



Fig 1.1 – Internal analysis and heat conduction has to be selected

## Water as fluid

In the Solidworks when we performed the CFD analysis some general setting has to be selected. After selecting the analysis the medium has to be selected and flow characteristics have to be selected (Laminar and turbulent).

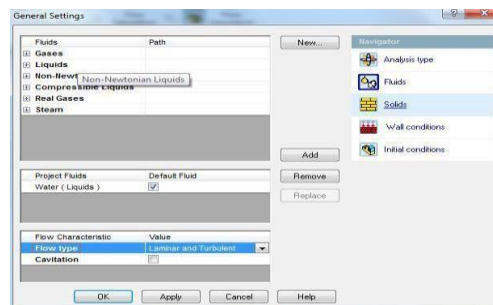


Fig 1.2- water as fluid and laminar and turbulent flow has to be selected

## Computational Domain

In the Solidworks when we perform the CFD analysis some general setting (material and flow characteristic) has to be selected and the computational domain has to be defined. The computational domain defines the area within which flow simulation has to be performed.

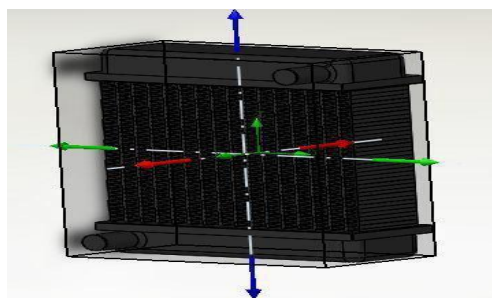


Fig 1.3- computational domain has to be selected along X-axis, Y-axis & Z- axis



# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

## BOUNDARY CONDITIONS

In the Solidworks when we performed the CFD analysis some general setting has to be selected. After selecting the analysis, the medium, and the computational domain the boundary condition has to define. Some parameters have to be defined according to the simulation.

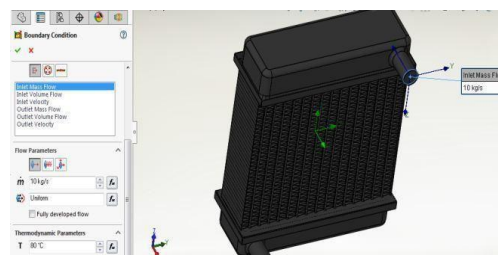


Fig1.4 – Inlet mass flow rate 10KG/S and temperature 80°C

## Material

In the Solidworks when we performed the CFD analysis some general setting has to be selected. One of the most important parts is the material has to be defined. If the material is not defined the simulation will show an error during the simulation process. After selecting the material LID has to be selected for creating a vacuum.

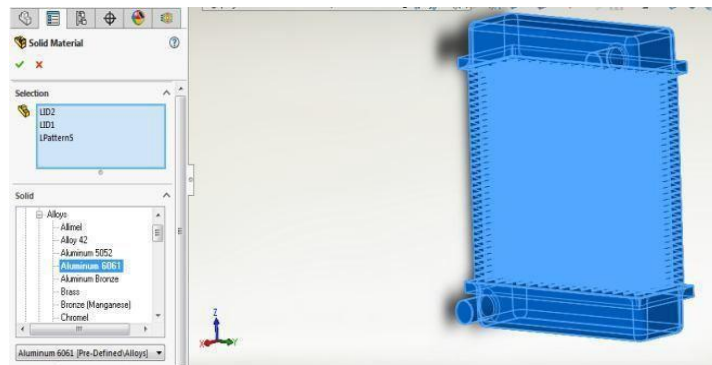


Fig1.5- LID and aluminum 6061

## Outlet

In the Solidworks, we perform the CFD analysis some general setting has to be selected. After selecting the analysis, the medium, the computational domain, the boundary condition, and the material the static pressure has to be defined. It states the force per unit area by the fluid.

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

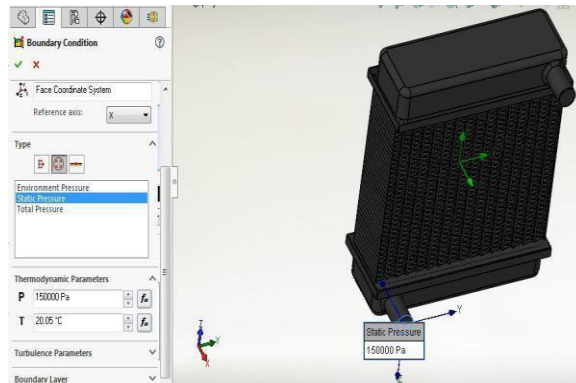


Fig1.6- static pressure has to be 150000Pa and temperature 20.05°c

### Wall

In the Solidworks when we performed the CFD analysis some general setting has to be selected. After selecting the material and the static pressure the outer wall temperature has to be defined. If we did not select the outer wall temperature transfer of heat will not take place.

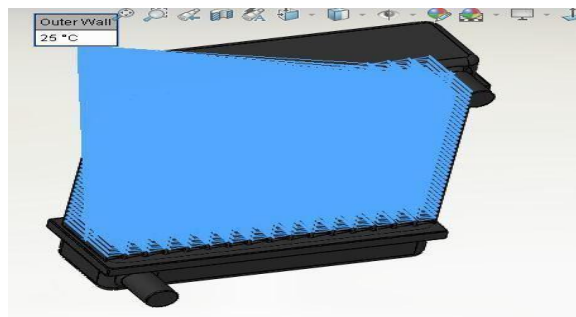


Fig 1.7:- the entire outer wall has to be selected and thermodynamic parameters should be (25°C)

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

## IV.RESULTS AND CONCLUSION

### 1) Temperature

As shown in the figure we see that variation of temperature takes place from 29.49°C-80.86°C.

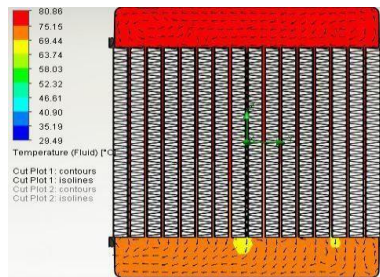


Fig1.8:- Temperature

### 2) Pressure

As shown in the figure we see that variation of pressure take place from -951232.01 to 5733935.49 pas (negative pressure is due to relative pressure).

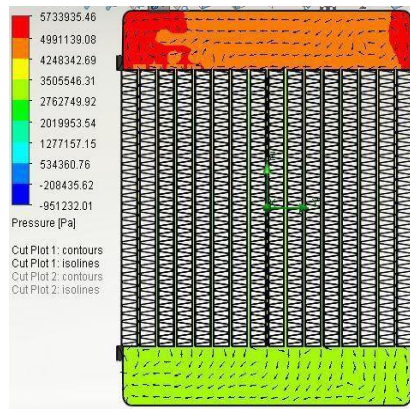


Fig1.9:- Pressure

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

### 3) Velocity

As shown in the figure we see that variation of velocity takes place from 0 to 82.560(m/s).

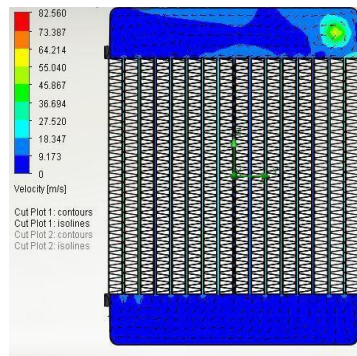


Fig1.10:- Velocity

### 2) USING WATER AND COOLANT AS MEDIUM

When we use water and coolant as a medium all the parameters are the same only medium will be changed. In 1<sup>st</sup> case, we take water as a medium and in 2<sup>nd</sup> case water and coolant are used for the simulation (1:4) ratio.

#### General Settings CFD analysis

In the Solidworks when we performed the CFD analysis some general setting has to be selected. In this software, two types

of analysis have to be done internal and external analysis is done. It depends on the model what type of analysis has to be selected. After selecting the analysis type the physical features has to be selected.



Fig 2.1 – Internal analysis and heat conduction has to be selected

#### Water as fluid

In the Solidworks when we performed the CFD analysis some general setting has to be selected. After selecting the analysis

the medium has to be selected and flow characteristics have to be selected.

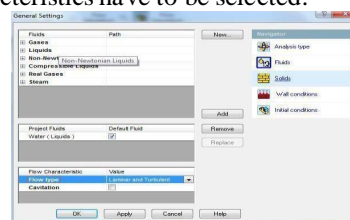


Fig 2.2- water and coolant as fluid and laminar and turbulent flow have to be selected.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

## Computational Domain

In the Solidworks when we performed the CFD analysis some general setting has to be selected. After selecting the analysis and the medium the computational domain has to be defining. The computational domain defines the area within which flow simulation has to be performed.

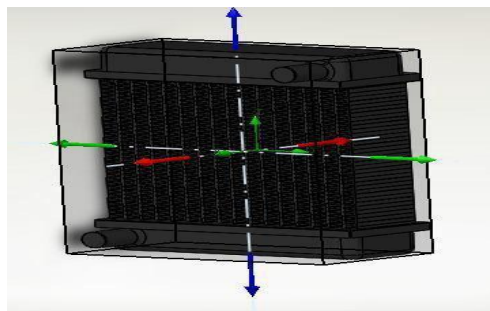


Fig 2.3- computational domain has to be selected along X-axis, Y-axis & Z- axis

## BOUNDARY CONDITIONS

In the Solidworks when we perform the CFD analysis some general setting has to be selected. After selecting the analysis, the medium, and the computational domain the boundary condition has to define. Some parameters have to be defined according to the simulation.

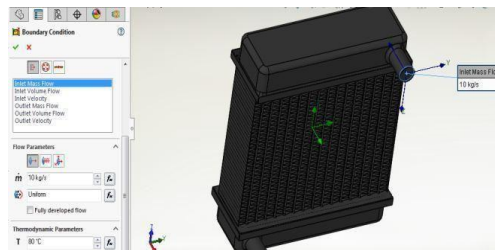


Fig2.4 – Inlet mass flow rate 10KG/S and thermodynamic parameters 80°C

## Material

In the Solidworks when we performed the CFD analysis some general setting has to be selected. After selecting the analysis, the medium, the computational Domain, and the Boudry condition. One of the most important parts is the material has to be defined. If the material is not defined the simulation will show an error during the simulation process. After selecting the material LID has to be selected for creating a vacuum.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

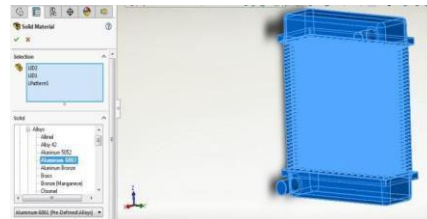


Fig2.5- LID and aluminum 6061

## Outlet

In the Solidworks when we performed the CFD analysis some general setting has to be selected. After selecting the analysis, the medium, the computational domain, the boundary condition, and the material the static pressure has to be defined. It states the force per unit area by the fluid.

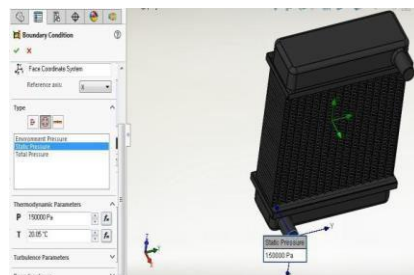


Fig2.6- static pressure has to be 150000Pa and temperature 20.05°c

## Wall

In the Solidworks when we perform the CFD analysis some general setting has to be selected. After selecting the material and the static pressure the outer wall temperature has to be defined.

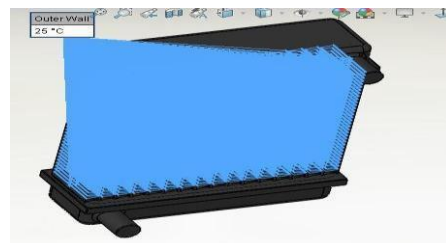


Fig 2.7:- the entire outer wall has to be selected and thermodynamic parameters should be (25°C)

## RESULTS

After filling all the data accordingly and running the flow simulation. Temperature, pressure, the velocity of a radiator will occur.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

### 1) Temperature

As shown in the figure we see that variation of temperature takes place from (26.26°C-80.30°C).



Fig2.9:- Temperature

### 2) Pressure

As shown in the figure we see that variation of pressure takes place -100962.71-1442512.26 (Negative pressure is due to relative pressure).

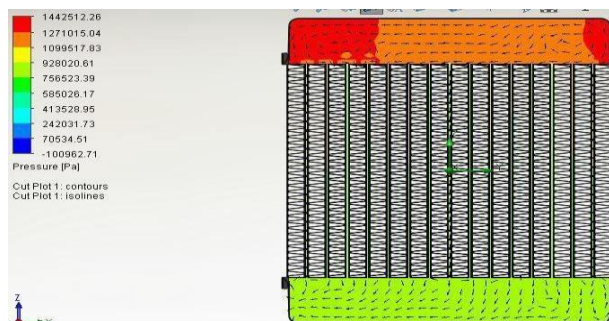


Fig2.10 Pressure

### 3) Velocity

As shown in the figure we see that variation of velocity takes place (0-20.342) (m/s).

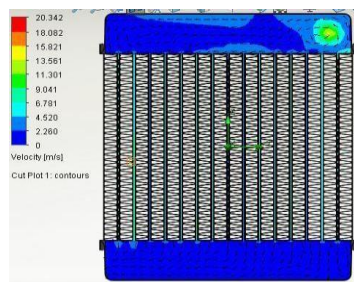


Fig 2.11:- Velocity

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

FLUID	INLET TEMPERATURE	OUTLET TEMPERATURE
WATER	80	29.49
WATRE AND COOLANT ratio (1:4)	80	26.26

FIG 5: - RESULT TABLE

In the above table, we can see that there is a large temperature difference in the simulation when we change the fluid

## V.CONCLUSION

- ✚ This project includes brief research on radiators and their operation.
- ✚ Radiator modelling is done with the Solidworks 2016 programme.
- ✚ The radiator is subjected to a CFD analysis using the solidworks Flow simulation module.
- ✚ The radiator is subjected to CFD analysis using two different fluids, namely normal water and coolant.
- ✚ Boundary conditions are set at 80 degrees Celsius for the fluid's entrance temperature, which will be cooled by convection through radiator pipe and fins at a temperature of 25 degrees Celsius.
- ✚ The temperature of the fluid flow inside the radiator will decrease due to convection; after examination, the temperature, velocity, and pressure of the fluid will be measured and tabulated.
- ✚ We can deduce from the results table that coolant and water fluids provide better convection, i.e. provide better cooling to the engine than water.
- ✚ When compared to all other fluids utilised in the analysis, WATER AND COOLANT produce the best results.

## REFERENCES

1. V.L.Bhimani, P.P.Ratho and A.S.Sorathiya, "Experimental study of heat transfer enhancement using water based nanofluids as a new coolant for car radiators". IJETAE Vol.3, Issue 6, June 2013.
2. Gaurav Sharma and Lal Kundan, "Experimental investigation into thermal conductivity and viscosity of Al<sub>2</sub>O<sub>3</sub> based engine coolant". IJRMET Vol. 3, Issue 6, May/Oct 2013.
3. Adnan M.Hussein, R.A.Bakar, K.Kadirgama, G.L.Ming "Heat transfer augmentation for the car radiator by using nanofluid". MRE, ISBN: 978-1-63248-002 doi.
4. Rahul A.Bhogare and B.S.Kotahwale "A review on applications and challenges of nanofluids as coolant in automobile radiator "International journal of scientific and research publications, volume 3, Issue 8, August 2013, ISSN 2250-3153.
5. Chavan D.K. and Tasgaonkar G.S. "Study Analysis and Design of Automobile Radiator Proposed with Cad Drawing and Geometrical Model of the Fan" IJMPERD ISSN 2249- 6890, Vol 3, Issue 2, June 2013, 137-146.
6. Deepak Chintakayala and, Rajamanickam C.S. "CFD analysis of fluid flow and heat transfer of an automotive radiator with nano fluid "978-1-4673- 6150- 7/13/\$31.00@2013 IEEE.
7. NavidBozorgan, KomalaganKrishnakumar, NarimanBozorgan " Numerical study on applications of CuO nanofluid in Automotive Diesel Engine Radiator " Modern mechanical engineering 2012, 2,130-136.
8. Paresh Machhar and FalgunAdroja "Heat transfer enhancement of automobile radiator with TiO<sub>2</sub> and water nanofluid
9. "International journal of engineering research and technology, ISSN 2278-0181, vol 2 Issue 5 may-2013.
10. [https://www.researchgate.net/publication/316200090\\_A\\_Numerical\\_Study\\_of\\_Heat\\_Transfer\\_Characteristics\\_of\\_a](https://www.researchgate.net/publication/316200090_A_Numerical_Study_of_Heat_Transfer_Characteristics_of_a)





ISSN(Online) : 2319-8753  
ISSN (Print) : 2347-6710

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Vol. 5, Issue 1, January 2016

Car \_Radiator\_Involved\_Nanofluids/link/59a6f209aca272895c169997/download.