International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 07 | July - 2022 Impact Factor: 7.185 ISSN: 2582-3930

DESIGN AND ANALYSIS OF COOLING SYSTEM OF FORMULA STUDENT CAR

B. Praveen Kumar[1], E. Nikhil Reddy[2], K. Ravi Kumar[3], B. Chandrashekar Reddy[4], S. Venkat Dilip[5]

[1]*Assistant Professor, Department of Mechanical Engineering, Guru Nanak Institute of Technology, Hyderabad, India.*

[2,3,4,5]*B. Tech Student, Department of Mechanical Engineering, Guru Nanak Institute of Technology, Ibrahimpatnam, Hyderabad, India.*

Abstract- This study takes a look at the design process of the cooling system of the Formula Student vehicle. The vehicle was built keeping in mind all the rules and regulations of the formula student Competition. Among numerous rules, one rule was to use air and plain water as the only coolant for radiator and no other substance. Though the car is dynamically fit and performed decently during the event, there was a heating issue in the engine which needed to be solved. The problem has been approached using both theoretical and simulation models. Firstly, Number of Transfer method will be used to calculate the effectiveness of the radiator and later the rate of heat transfer is calculated. Various parameters like radiator core size, mass flow rate of water and air etc., are taken into consideration. Ansys software will used to perform full body analysis to study the positioning of the radiator on the race car. The results obtained from these models will be validated experimentally on the vehicle using data acquisition. The study not only results in designing an efficient cooling system but also laying out a systematic approach for further development.

Keywords-Cooling System, Formula Student, FSAE, Sidepods, Radiator, Ansys.

1. INTRODUCTION

Internal combustion engine cooling uses either air or liquid to remove the waste heat from an engine. For small or special purpose engines, cooling using air from the atmosphere makes for a lightweight and relatively simple system. Cooling system works by passing liquid continually through the passages in the engine block. Powered by the water pump, the coolant is pushed through the engine block. As the solutions travels through these passages, it absorbs heat from the engine. Most internal combustion engines are fluid cooled using either air (a gaseous fluid) or a liquid coolant run through a heat exchanger (radiator) cooled by air.

The radiators are used in water cooled petrol and diesel engines to keep water circulation in the cylinder heads of engines at relatively constant temperature and thus prevent overheating of the engine. In Formula car, the engine is placed at the rear side, so the radiator section needs to be located near to it. Due to this less amount of air flow to the radiator can be seen as compared to the vehicle with radiator at front. So, there is a need of designing a air duct around the radiator for passing the good amount of air so that proper cooling of the engine can be obtained.

2. METHODOLOGY

2.1 Problem Statement

Formula Student or FS is a design-based competition for engineering students. Students build a single seat formula racecar. The competition is not won solely by the team with the fastest car, but rather by the team with the best overall package of construction, performance, cost and sales planning. This competition challenges the team members to go the extra step in their education by incorporating into the intensive experience in building and manufacturing as well as considering the economic aspects of the automotive industry.

The problem is to solve the heating issues that a formula student team faces by designing a cooling system that removes all the heat rejected by an engine to the coolant. This study helps the teams to design their cooling system and improve their cooling efficiency, increasing the reliability and performance of their powertrain package, which helps them in performing better in the competition.

Some rules related to cooling system from Formula Bharath Rules book:

T 7.2.1 Water-cooled engines must only use plain water.

T 7.2.4 Any cooling or lubrication system must be sealed to prevent leakage.

T 7.2.7 Catch cans, their mountings and all cooling or engine lubrication system hoses must be made of material that is permanently rated for temperatures of at least 120ºC or the temperatures the respective fluid may reach whichever is higher.

The above rules are taken into consideration while designing the vehicle.

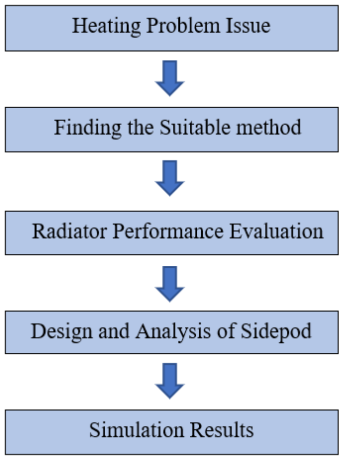


Fig. 1. Methodology

2.2 Design Goals

- To design and optimize the cooling system for a formula student car to remove excess heat from the engine.
- To maintain the optimum working temperature inside the engine.
- To design a air duct that directs and increases the flow of air at the radiator core, which results in improved cooling efficiency.

2.3 Design Approach

According to theory of heat exchanger, radiator performance can generally be determined using either one of the following approaches;

- The Log Mean Temperature Difference method (LMTD)
- The Effectiveness – NTU method (ε-NTU)

2.4 Effectiveness-NTU

In comparison with the LMTD method, this method is more effective when the radiator geometry, the coolant flow rate and the inlet temperatures of both fluids are known for a particular radiator. The outlet temperatures in association with the heat transfer rate of the radiator are subsequently calculated. This approach involves several dimensionless parameters to be calculated it consisting of Number of Transfer Units (NTU), Effectiveness(ε) and Capacity Ratio (C_{min}/C_{max}). In this method each parameter affects the performance of the heat exchanger and has its own physical significance.

A general relationship between the parameters is expressed by the following formula,

$$\varepsilon = f(NTU,\ C_{min}/C_{min},\ flow\ arrangement)$$

Once the radiator effectiveness (ε) is determined, the heat dissipation rate (Q) from the radiator can be obtained by applying the following equation;

$$Q = \varepsilon C_{min}(T_{h,i} - T_{c,i})$$

Where, T = Temperature;

subscripts h = hot fluid (coolant), c = cold fluid (air), i= inlet.

Since we don't have the outlet temperatures of hot and cold fluid, we have chosen Effectiveness -NTU method for our problem.

To begin with the design process, we need the dimensions of the radiator. We started the design taking into consideration of the stock radiator that came with the bike. Since there we have several issues with that radiator, we tried different options available in the market and finalized a best radiator that suits for our application.

The whole process is divided into two steps,

1) Calculating the heat rejected to the coolant.

2) Calculating the heat dissipated from the Cooling system(radiator).

Once both the values are known we can select the appropriate design, which effectively maintains the engine at the optimum working temperatures. We cannot afford to run the engine very cool or too hot, both show adverse effects on the performance of the engine. So, running the engine at optimum working temperatures is very crucial.

3. CALCULATIONS

3.1 Calculating the Heat Rejected to the Coolant

According to van Basshuysen and Schäfer (2004: 556), the amount of heat rejected to coolant from engine is 50–60 % of the brake power. Heywood (1988: 674) presents that 25–28 % and 17–26 % of fuel's heating value transforms into brake power and heat dissipated to coolant, respectively. Besides these estimations, Lahvic (1986) has presented an empirical correlation for thermal load according to his work in Ford Corporate Vehicle Simulation Program, which is shown as

$$Q_c = \frac{8.66V_dN + 108.93T_e + 1119.74P_b - 1010V_d + 2890}{3412.2}$$

Where,

Q_c is the heat rejected to the coolant [kW]

V_d is the displacement of engine in litres [L]

N is the RPM of engine [rpm]

T_e is the engine torque [N-m]

P_b is the brake power of the engine [kW]

Table 1. KTM RC 390 engine specifications

Bike	KTM RC 390
Year	2016
Engine	Four stroke, single cylinder
Capacity	373.2
Bore x stroke (mm)	89*60
Compression ratio	12.6:1
Cooling system	Air cooled
Max power	32 kW @ 9000rpm
Max Torque	35 N-m @ 7250rpm

By considering the Lahvic's empirical formula,

Where, V_d=0.373lit., N=10,000(say), T_e=35Nm, P_b=32kW

The heat dissipated by engine to coolant is found to be;

Q_c =21.82 kW

All these theories and studies gives us almost same value of heat rejected to cooling, so the maximum of all is rounded and taken into consideration.

Therefore, the heat rejected to the coolant is taken as, Q_c =22 kW

3.2 Calculating the actual amount of heat dissipated with our cooling setup.

Total heat capacity, $C = \dot{m}C_p$

Where, C_p is the specific heat capacity (J/Kg)

Mass flow rate, $\dot{m} = \rho\dot{V}$

Where, $\dot{V}$ is the volumetric flow rate [m^3/s]

ρ is the density of the fluid [Kg/m^3]

Table 2. Radiator Dimensions

Frontal area	29cm x16cm
Thickness	2.5cm
Total number of tubes	19
Heat transfer area	1.378 m^2

Table 3. Properties of Air and Water

	Inlet temp in °C	Density ρ (Kg/m^3)	Specific heat capacity C_p (kJ/KgK)
Air	35	1.146	1.006
Water	90	1000	4.18

For air, $C_{air} = \rho\dot{V}C_p$

Volumetric flow rate, $\dot{V} = A_c v$

Where, A_c is the cross-sectional area normal to the flow i.e, the frontal area of radiator [m^2]

v is the average velocity of fluid(air) [m/s]

v=13.8 m/s (considering vehicle speed of 60km/hr)

Substituting, the above values

C_{air}=0.743 kW/s

For water, $C_{water} = \rho\dot{V}C_p$

C_{water}=6.27 kW/K

Heat Capacity ratio, $C = \frac{C_{min}}{C_{max}}$

$$C = 0.118$$

Where, C_{min} is the smaller of C_{air} and C_{water}.

The number of heat transfer units (NTU) is the ratio of overall conductance UA to the smaller capacity rate C_{min};

$$NTU = \frac{U.A}{C_{min}}$$

The Engineering ToolBox (2003) gives the typical value of 600 to 750 W/(m^2K) for U in air-cooled heat exchangers, thus 600 W/(m^2K) is chosen.

Assuming the overall heat transfer co-efficient

U= 600 W/m^2K

A= Heat transfer area = 1.378 m^2

NTU=1.1

Using NTU- ε correlation for cross flow heat exchanger with both fluids unmixed, we have

$$\varepsilon = 1 - \left(e^{\frac{NTU^{0.22}}{C}\left(e^{-C*NTU^{0.78}}-1\right)}\right)$$

When C goes to zero, simplified relation between ε and NTU is given by

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 07 | July - 2022 Impact Factor: 7.185 ISSN: 2582-3930

$\varepsilon = 1 - e^{-NTU}$

$\varepsilon = 0.67$

Heat transfer rate is calculated using,

$Q = \varepsilon C_{min}(T_{h,i} - T_{c,i})$

$Q = 26.84$ kW

The radiator is placed on the left side of the cockpit. Considering the flow restrictions of air caused due to the suspension set-up, we are assuming 80% of efficiency to be on the safer side.

So, the overall heat dissipated by the system is

$Q_{actual} = Q * \text{efficiency}$

$= 21.476$ kW

The overall heat dissipation rate is almost equal to the heat rejected to coolant from the engine. This could be considered. However, it is not the best solution for our problem. So, we have decided to improve the heat dissipation rate by designing a air duct that directs the air towards the radiator. The design of the air duct is in such a way that it will increase the air velocity by around 20%, which is resulting in almost 10% increase in the overall heat dissipation rate.

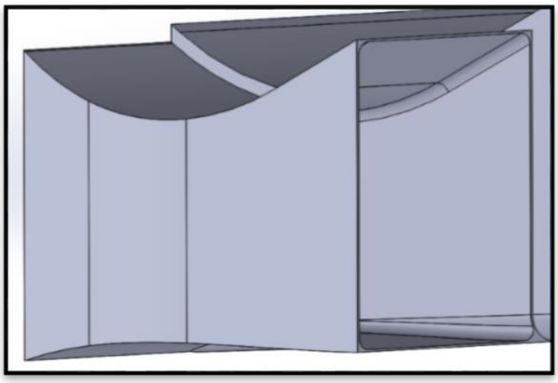
4. SIDEPOD DESIGN AND ANALYSIS

The sidepods, which house oil and water radiators for engine cooling, are responsible for providing adequate airflow to maintain stable engine operating temperatures, while increasing the downforce and reducing drag on the vehicle. By considering geometry of the vehicle and FSAE rules, position of radiator was fixed on the right of the driver cockpit at an orientation of 25deg which is not a best position for the air flow because of restriction caused by the suspension setup. To increase the air flow design of sidepod will play an crucial role. Basic design of sidepod was done by considering the dimensions of radiator as 280*160*27 mm and side impact structures.

The model is created by making the inlet area is being converged until specific point and starts to diverge until it reaches to the outlet. The outlet dimensions are such that it can accommodate the radiator with in the inlet to allow the air flow to the radiator. The model we have designed is of converging-diverging type where inlet area is 25% more than outlet area. At the position of throat radiator will be placed.

The below figure shows the design of sidepod in SolidWorks.

Fig. 2. Isometric View



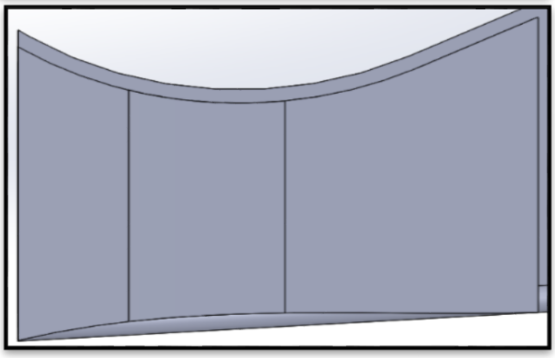


Fig. 3. Side View

For simulation of sidepod in Ansys, physical preference was set to Computational Fluid Dynamics (CFD) and fluent as solver. Tetrahedrons method of meshing with patch- conforming algorithm, fine relevance center and global setting for the element order. Set to double precision and serial processing options. Considering the viscous model with boundary conditions as inlet velocity 13m/s and outlet gauge pressure 0Pa. Coming to our simulation the solution method we used is pressure-velocity coupling based simple scheme is used with second order spatial discretization.

The following flow analysis figures shows the velocity streamlines and pressure counters in sidepod.

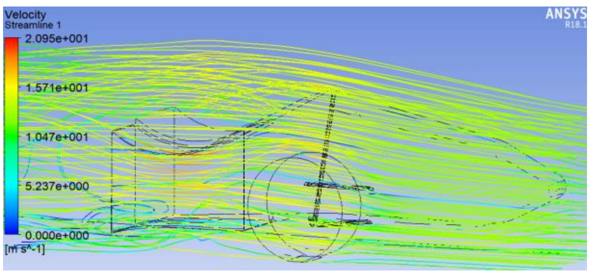


Fig. 4. Velocity Streamlines

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 06 Issue: 07 | July - 2022 Impact Factor: 7.185 ISSN: 2582-3930

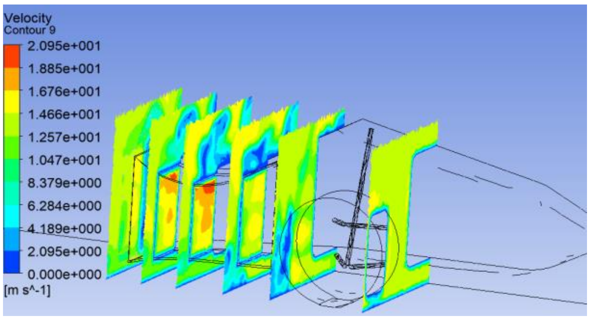


Fig. 5. Velocity Counters

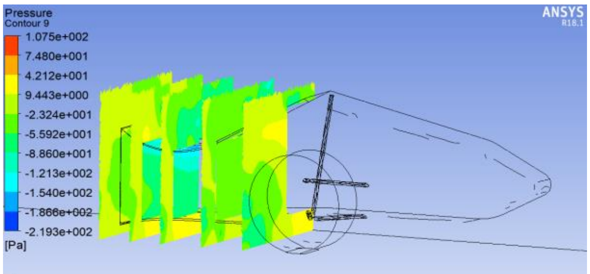


Fig. 6. Pressure Counters

In the above figures, the velocity of the air is measured by considering the region as a plane which gives the sectional view of the both velocities and pressure. The position of these planes determines the appropriate velocity and pressure results. The requirement of the cooling system is it need maximum amount of the air to flow with maximum velocity and minimum amount of pressure. From the above flow analysis results, we can consider the point where radiator need be placed has the appropriate position for optimum cooling effect.

5. RESULTS AND DISCUSSIONS

The velocity on the counter plot at the position of radiator is maximum in this model and also less pressure drop taken place. The maximum surface area of the radiator will be in contact with the maximum velocity which results in good heat exchange. So, we finalized this model as sidepod for our vehicle.

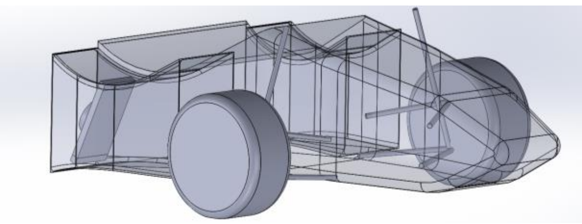


Fig. 7. Model with radiator assembly in isometric view

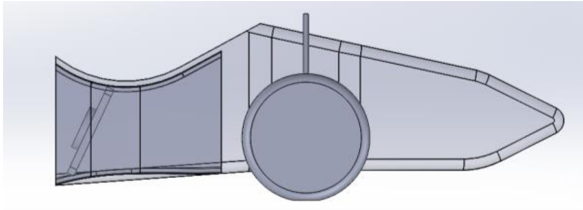


Fig. 8. Model with radiator assembly in side view

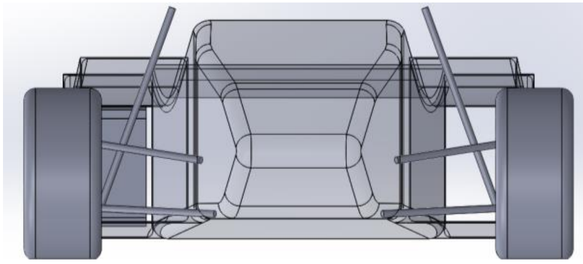


Fig. 9. Model with radiator assembly in front view

5.1 Flow Analysis Through Radiator Assembly

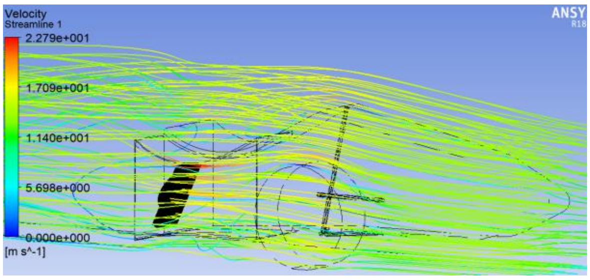


Fig. 10. Velocity Streamlines

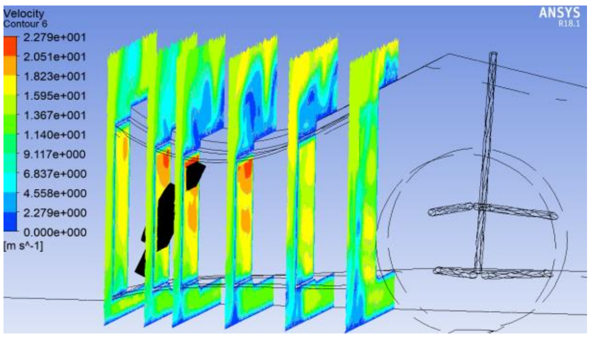


Fig. 11. Velocity Counters

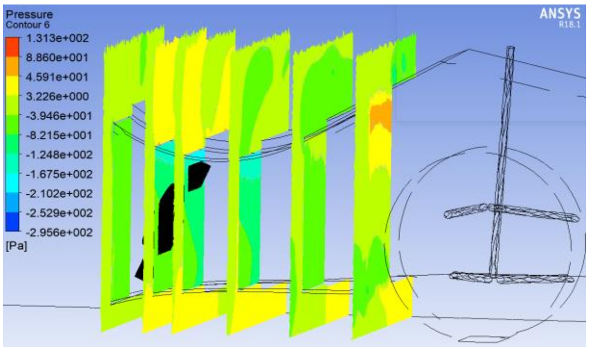


Fig. 12. Pressure Counters

6. CONCLUSION

By considering the advantages of the venturi effect, the uniform reduction of the cross-section area of the side pod will increase the velocity of the air and a corresponding pressure drop. As the velocity increases, the rate of cooling of the radiator will increase which will show the significant cooling results in the less time. The radiator is placed right after the converging section which is the better place for the placement of radiator as the maximum velocity is observed there. The design of side air vent and its analysis showed that there was improvement in mass flow rate of air at radiator core, therefore, improvement in cooling system. By the above results, we can consider that the design of the sidepod will play the vital role in the cooling system of the vehicle.

7. REFERENCES

[1] The Formula Bharat 2022 Rules Book- https://www.formulabharat.com
[2] van Basshuysen R. & Schäfer F., 2004. Internal Combustion Engine Handbook: Basics, Components, Systems and Perspectives 3 rd edition.
[3] Engineering ToolBox, (2003). Heat Exchangers - https://www.engineeringtoolbox.com/heat-transfer-coefficients-exchangers-d_450.html
[4] Lisa ven den Berg, Brendon Lofaro, 2014, “Formula SAE Cooling System Design”, Mechanical Engineering Department, California Polytechnic State University,San Luis Obispo, pp. 1-141
[5] “Designing a cooling system for a formula student race car”, Janne Kemppainen, University of Oulu, Degree Programme of Mechanical Engineering, Bachelor’s thesis 2020, 49 p.