SEMARAK ILMU PUBLISHING

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index

ISSN: 2289-7879

Effects of Radiator Angle and Sidepod Profile on Aerodynamic Performance of a Race Car

Ashwin Sasikumar[1], Harish Rajan[1,*]

[1] School of Mechanical Engineering, Vellore Institute of Technology, Chennai, Tamil Nadu 600127, India

ARTICLE INFO

Article history:
Received 23 November 2022
Received in revised form 6 March 2023
Accepted 16 March 2023
Available online 30 March 2023

Keywords:
Radiator angle; sidepod profile; radiator seal; Gills; FSAE

ABSTRACT

In Formula Society of Automotive Engineers (FSAE) cars engine heat management serves as an important factor for the performance of the vehicle. Therefore, a proper design of sidepod is crucial for optimal cooling of the engine since sidepods are designed to direct the airflow through the radiator. Also, since sidepods play an important role in the aerodynamic performance of the vehicle, factors such as drag and lift must also be considered along with the cooling of the engine. This paper brings forth the methodology carried out to design the sidepods effectively. First the optimal radiator angle is determined on which the maximum mass flow rate of air through the radiator core is observed. Angles are varied from 50, 70 and 90 degrees on various orientations with respect to the car. Fixing this radiator angle, various inlet and outlet areas of the sidepod are analyzed. Choosing the model with least drag and lift coefficient the sidepod is analyzed by sealing it at various sides. Finally, the effects of gills are also analyzed for better optimization of the sidepod. The models are designed in Solidworks and the CFD simulations are carried out in Simscale. Half car simulation is performed with symmetry conditions in order to reduce the cell count. The results of the analysis showed that the radiator angled forward with a diverging type sidepod yields in the better cooling of the engine.

1. Introduction

Engine overheating has been a serious problem in FSAE cars due to the fact that most of the teams use bike engines to power their cars. Since the average running speed of the car is lower than that of a bike this reduces the air flow rate passing through the radiator. Also, the added weight of the car poses a huge challenge to the engine and thus emits more heat. Therefore, a proper design of the sidepod is crucial for the heat management of the vehicle. Sidepod acts as a duct which governs the flow of air through the radiator. It also plays a significant role in affecting the aerodynamic properties of the vehicle as well. There has been a huge discussion in determining the type of sidepod profile to be used for optimum air flow rate through the radiator. Therefore, this paper brings forth a detailed methodology to analyze and compare many different sidepod profiles which includes

* *Corresponding author.*
E-mail address: harish.r@vit.ac.in

https://doi.org/10.37934/arfmts.104.2.139152

Straight tunnel, Converging and Diverging. In addition to that to further optimize the sidepod the effect of sealing the radiator and the addition of gills to the sidepod has also been analyzed.

Extensive amount of research has been carried out in the field of airflow characteristics through an FSAE car sidepod and also approaches used for performing CFD analysis in similar cases. Wang *et al.,* [1] analyzed the effect of rear mounted radiator and concluded that the rear mounted radiator can result in a 4% improvement in standard lap time. Bahuguna *et al.,* [2] used both theoretical and simulation models to calculate the heat transfer coefficient and temperature drop through the radiator with varying parameters like core size, mass flow rate of water and air, fan configurations. Nikhita *et al.,* [3] developed a sidepod to obtain the maximum possible air flow rate through the radiator through analytical calculations, simulations and experimentation. Yadav and Singh [4] illustrated the numerical methodology to perform heat transfer analysis of an automobile radiator with varying parameters under consideration. Kamath *et al.,* [5] studied the effect of radiator angle and radiator seal using CFD and concluded that an angled radiator without a seal yields higher mass flow rate of air through the radiator. Srivastava *et al.,* [6] optimized a diverging-converging type sidepod using internal CFD simulations and concluded that a small inlet area results in increased mass flow rate of air through the radiator. Harish and Venkatasubbaiah [7] and Waman and Harish [8] worked on developing a complete aerodynamic package for an FSAE car and had also validated their results with experimental setups. Landström *et al.,* [9] observed the effects of simplifying the wheel design for CFD analysis and concluded that a simplified completely closed wheel geometry results in heavy drag reduction compared to the actual model. Bukovnik *et al.,* [10] performed CFD analysis on different rim orientational and computational methods to model the rims using various approaches. Oktavitasari *et al.,* [11] performed experimental investigation on vertical axis wind turbine and concluded that best performance turbine spacings in aligned configurations are 3D. Saleman *et al.,* [12] studied the influence of liquid film thickness on thermal energy transfer across solid – liquid interfaces and concluded that the thermal energy transfer is affected by the velocity cut – off at the contact interfaces of solid and liquid. Hassan *et al.,* [13] studied the heat transfer of car radiator with using pure water and water with nano fluid as the coolant by numerical methods and had also verified it experimentally. Budiyanto *et al.,* [14] analyzed the performance of thermoelectric coolers to decrease the temperature of electric motors and concluded that it can reduce the temperature of the motor significantly. Abobaker *et al.,* [15] performed a mesh quality study over an airfoil using structured and unstructured mesh and concluded that structured mesh results are closer to the experimental data for the calculation of drag and lift forces. Mohamed *et al.,* [16] studied the heat transfer rate of automotive radiator using an active louvered fin with shape memory alloy by varying the fin pitch, louver angle and wall temperature. They concluded that the louver fin's optimum angle is independent of it wall temperature but depends on the fin pitch and Reynolds number. Ahmad *et al.,* [17] performed a mesh optimization strategy to estimate the drag of ground vehicles by comparing the CFD results with experimental setup for varying mesh sizes. Niknahad [18] developed the vortex generator geometry of Boeing 737 from triangular profile to a circular profile and proved that the circular vortex generator decreased the drag coefficient better than the triangular vortex generator. Elfaghi *et al.,* [19] performed CFD analysis to study the effect of adding Nano fluids with water to improve the heat transfer of fluids by using Al2O3 as nano fluids in a circular pipe and concluded that the addition of Nano particle improved the heat transfer as well as the Nusselt number in the flow. Hamizi and Khan [20] performed a numerical study on oscillating delta wing with tailless, tailed and cropped configurations and concluded that the vortex energy is stronger as Reynold's number increases.

From the above literature survey, it is evident that extensive research work has been carried out on the development of the sidepod of a car. But there has not been a clear idea of the type of sidepod profile to be used and to determine the angle of inclination of the radiator. This paper brings forth a methodology to determine the angle of inclination of the radiator and also gives a comparative study of various sidepod profiles to be used. Further the effects of sealing the radiator and the gills are also analyzed.

2. Methodology

The models to be analyzed are designed in Solidworks CAD software. The complete model of the car is designed consisting of all vital components that could affect the aerodynamic properties which include the wheels, Percy, suspension components, outer body, diffuser, radiator and the sidepod [21]. The models are then converted into Parasolid format to be imported in the simscale software to perform the CFD analysis [22,23]. A virtual wind tunnel is created in simscale through which air flow over the car is analyzed. The dimensions of the wind tunnel are 2 times the length of the car in the front and the side and 6 times the length of the car in the back. The larger length in the rear of the car is present in order to capture the effect of turbulence.

Hex dominant parametric meshing is used with 75 cells in the flow direction and 13 cells in the other 2 directions. This combination of 75,13,13 has been used to create perfect cubical cells. 2 levels of body of influence are added to dense the mesh region near the car. The radiator core is meshed with cell zones to assign the porous condition. Various refinement levels have been used to refine the mesh cells near the car which are mentioned in Table 1. Also, inflation of 5 layers is added to the car with characteristic length as the car length and Y+ value of 30 with growth rate of 30 percent. Figure 1 shows the computational domain with the generated mesh.

Table 1
Mesh refinement levels

Surface/Region	Refinement level
BOI far	2
BOI near	3
Radiator	5 - 6 (With cell zones)
Edges	7 - 8
Boundary layer	Minimum thickness - 0.001018 m
Other car surface	6 - 7

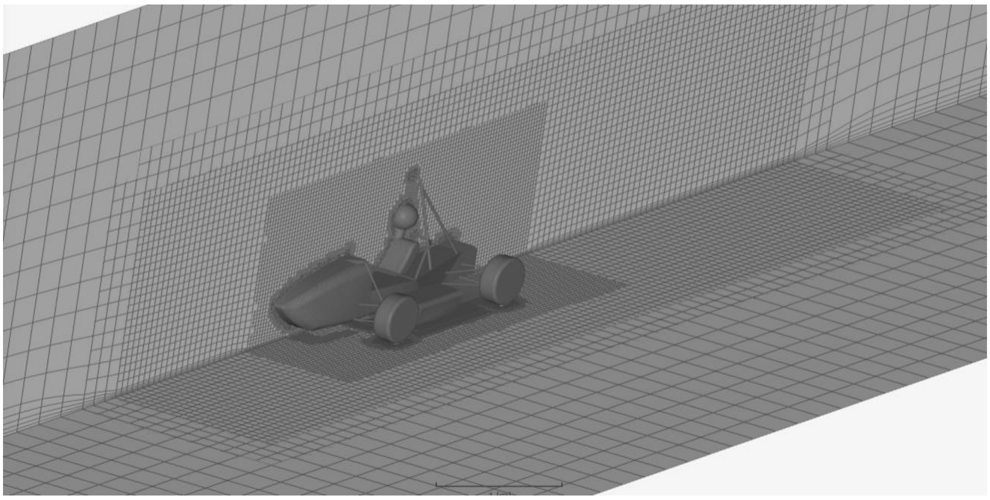

Fig. 1. Computational domain with the generated mesh

Incompressible k-omega SST turbulence model is used to capture both internal flow through the sidepod as well as the turbulence behind the car. Half car simulation is performed with symmetry conditions to reduce the mesh cell count. Inlet velocity of 40km/h is assumed considering the fact that it is the typical running speed of the car. A straight flow simulation is performed through the car and the boundary conditions used are mentioned in the Table 2. 1000 iterations with steady state flow analysis are performed to obtain better convergence.

Table 2
Boundary conditions

Surface	Boundary Condition
Inlet	Velocity inlet
Outlet	Pressure outlet
Floor	Moving wall
Symmetry plane	Symmetry
Side walls	Slip wall
Wheels	Rotating wall
Radiator	Darcy Forchheimer porous media
Other car surface	No slip wall

3. Results and Discussions

Before moving on to the sidepod analysis it is important to determine the radiator inclination angle. Therefore, the methodology is divided into a 4-step process of which degerming the radiator angle being the first step after which the sidepod profile is analyzed and then the effect sealing the radiator is studied. Finally, the effect of adding gills to the sidepod is observed. The results of each individual steps of the above-mentioned methodology are discussed below in the respective order.

3.1 Determination of Radiator Angle

Radiator is a heat exchanging device which works on the principle of convection. Hot coolant flows from the inlet to the outlet tank through the radiator tubes while the free stream air crosses the radiator tubes. Heat is transferred from the coolant to the air by means of convection. Greater the mass flow rate of air passing greater will the heat transfer rate. Therefore, a proper radiator angle must be determined to provide the highest mass flow rate of air through the radiator.

Radiator placed in the open air produces huge drag forces as it obstructs the free steam flow of air. Therefore, inclining the radiator to a certain angle helps reduce the drag forces. But inclination of the radiator reduces the frontal surface area of the radiator which in turn reduces the mass flow rate of air passing through the radiator. Therefore, a proper balance between the drag and mass flow rate of air must be maintained to better optimize the radiator.

Here 9 different iterations of radiator angle inclinations are analyzed to determine the optimized radiator angle. The first model is a straight radiator with an angle of inclination as 90 degrees. The radiator is then inclined at 50 and 70 degrees with 2 different view orientations. For the radiator with angle of inclination as 50 when viewed from the side of the car is inclined by 50 degrees both front and back side of the car which is named as Rad 50 front side and rad 50 back side. Also, the next iterations with the same inclination as 50 degrees when viewed from the top of the car is inclined in both direction front and back of the car which is named as rad 50 front top and rad 50 back top. Thus resulting 4 iterations for angle of inclination of 50 degrees. Similarly, there are 4 iterations of angle of inclination 70 degrees. All together resulting in 9 different iterations. Figure 2 and Figure 3 shows the radiator inclination of 50 and 90 degrees from top and side view respectively. The iterations are listed in Table 3.

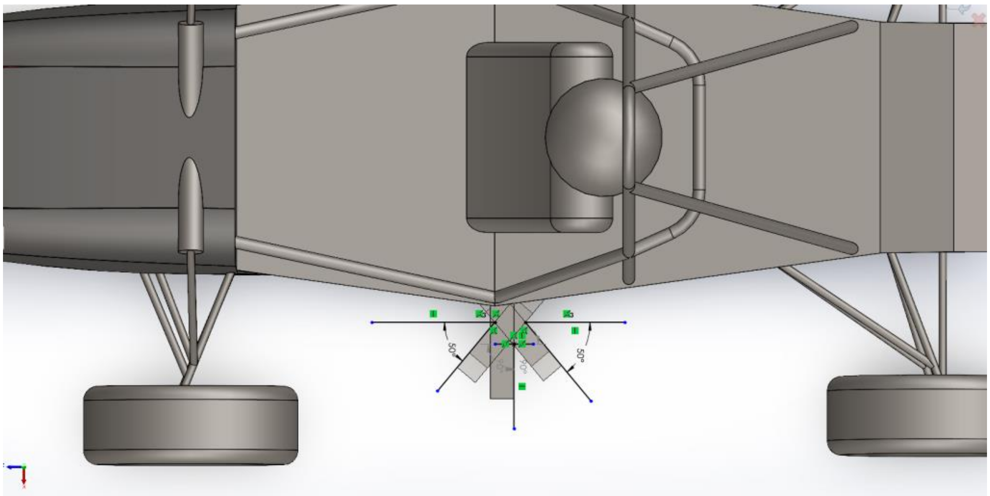

Fig. 2. 50- and 90-degree radiator angles from top view

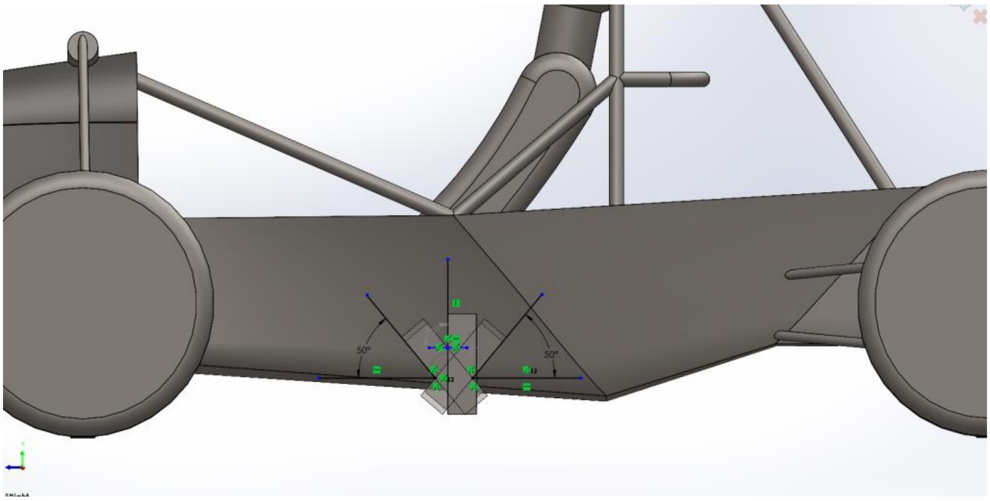

Fig. 3. 50- and 90-degree radiator angles from top view

Table 3
Models and description of iterations to determine the radiator inclination angle

Model	Description
Case A	angled 90 degrees
Case B	angled 50 degrees front from top view
Case C	angled 50 degrees back from top view
Case D	angled 50 degrees front from side view
Case E	angled 50 degrees back from side view
Case F	angled 70 degrees front from top view
Case G	angled 70 degrees back from top view
Case H	angled 70 degrees front from side view
Case I	angled 70 degrees back from side view

The result parameters considered are drag coefficient, lift coefficient and mass flow rate of air passing through the radiator. The values of which are given in Figure 4. It is observed that the radiator angled 70 degrees from the top view orientation (model F) has the maximum mass flow rate of 0.104

kg/s. Figure 5 shows the velocity streamlines passing through the radiator core of model F. However, the drag coefficient is not considerably low as compared to other iterations with a value of 0.625. Also, the lift coefficient value of 0.21 is one amongst the highest which is not desirable. However, this model will be used for further steps in the methodology as our primary concern is the mass flow rate of air. The drag and lift forces can be optimized by the sidepod profile design.

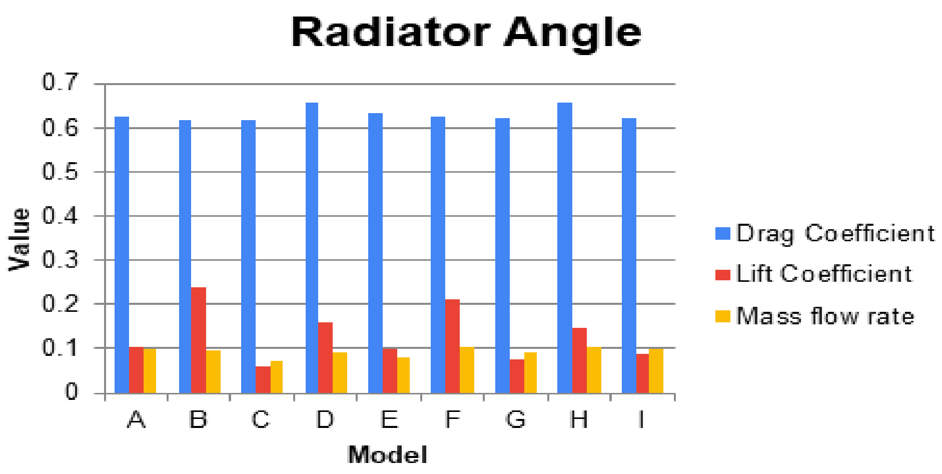

Fig. 4. Drag coefficient, lift coefficient and mass flow rate of different radiator angles

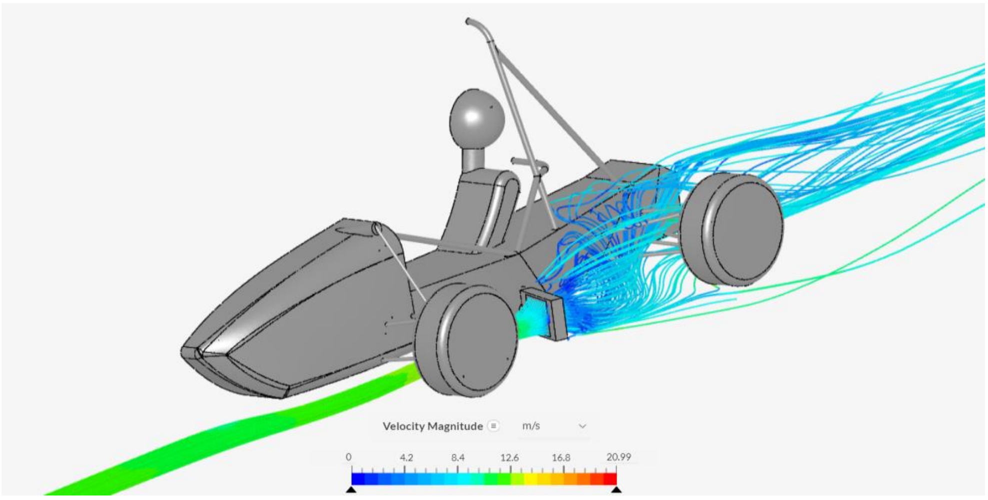

Fig. 5. Velocity streamlines passing through the radiator core of model F

3.2 Sidepod Profile Analysis

Sidepod is a duct used to regulate the flow of air through the radiator. It is designed to uniformly distribute the air flow through the radiator. A proper design of sidepod could result in reduction of drag force with minimal effect on the lift force. There has always been a huge discussion in the type of sidepod profile to be used. The 3 common types are the Straight tunnel, Converging and the Diverging type. A straight tunnel sidepod is a sidepod profile with the same inlet and outlet area and has a rectangular profile. A converging type sidepod has a larger inlet area compared to the outlet area. This accelerates the air flow at the outlet of the sidepod so as to match the free stream velocity thus reducing drag. A diverging type sidepod has a larger outlet area than the inlet area. This design lets the air to slow down from the inlet to the outlet so as to increase the time of contact between air and the radiator.

Here 9 different iterations are analyzed considering the drag and lift forces as well as the flow rate through the radiator core. 3 different sidepod inlet/outlet areas are named as Small(S), Medium(M) and Large(L) are used. All possible combinations of the 3 different areas are computed resulting in 9 different models which are listed in Table 4.

Table 4
Models and description of iterations used to determine the sidepod profile

S. No	Model	Description
1	SS	Small-Small
2	SM	Small-Medium
3	SL	Small-Large
4	MS	Medium-Small
5	MM	Medium-Medium
6	ML	Medium-Large
7	LS	large-Small
8	LM	Large-Medium
9	LL	Large-Large

Figure 6 and Figure 7 shows the inlet and outlet areas taken for consideration respectively. In this case the bottom part of the sidepod is actually the undertray of the diffuser. As the sidepod is integrated with the diffuser all the analyses are performed with the diffuser as well as the sidepod for better accuracy.

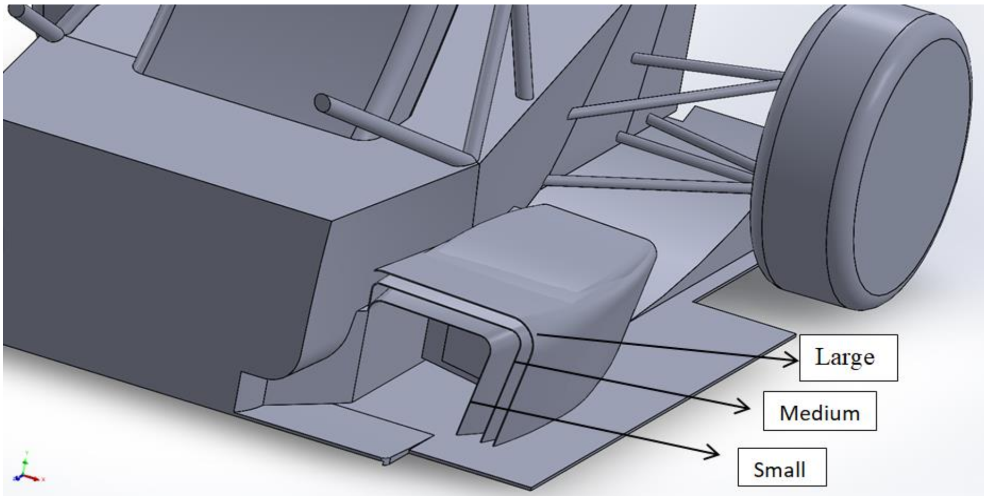

Fig. 6. Small, medium and large inlet areas

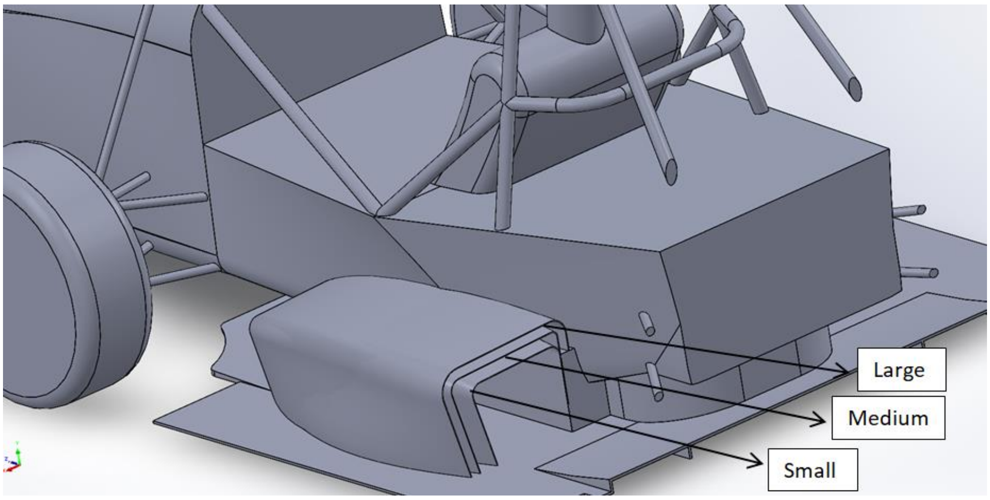

Fig. 7. Small, medium and large outlet areas

From the results it is evident that LL model has the highest mass flow rate through the radiator with a value of 0.080, SL model has the least drag coefficient of 0.566 and LS model has the highest negative lift coefficient of -0.572. The results of which are given in Table 5.

Table 5
Drag coefficient, lift coefficient and mass flow rate of different sidepod models

Model	Drag coefficient	Lift coefficient	Mass flow rate(kg/s)
SS	0.583	-0.523	0.067
SM	0.577	-0.536	0.073
SL	0.566	-0.49	0.078
MS	0.608	-0.494	0.062
MM	0.598	-0.514	0.068
ML	0.587	-0.493	0.073
LS	0.615	-0.572	0.067
LM	0.586	-0.519	0.071
LL	0.615	-0.494	0.080

It is also more important to note that SS, MM and LL models represent the straight tunnel sidepod models, SM, SL and ML represent the diverging type sidepod and MS, LS and LM represents the converging type sidepods. By careful analysis of the results, it is observed that as the sidepod's outlet area increases than the inlet area the mass flow rate also increases. That is the model with the largest outlet area has the highest mass flow rate for the same inlet area. This is also the same for drag coefficient where the drag reduces with increasing outlet area for the same inlet area except the LL model which does not follow the trend. However, the lift forces don't follow any trends with the models as in this case the lift is majorly affected by the presence of the diffuser.

As engine heat management is the top priority the LL model with the highest mass flow rate will be considered for further analysis. The streamlines of air passing through the radiator for the LL model is shown in Figure 8.

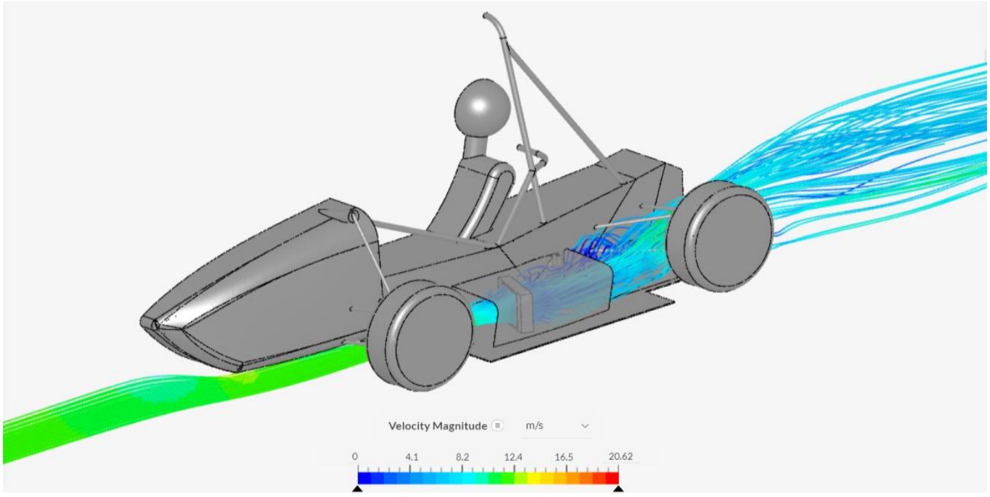

Fig. 8. Velocity streamlines passing through the radiator core of model LL

3.3 Effect of Sealing the Radiator

A radiator is usually sealed in order to concentrate the air flow only into the core of the radiator. This reduces the air escaping to the gaps between the radiator and the sidepod. However, completely sealing the radiator will result in huge back pressure which results in increased drag as well as recirculation zone in front of the radiator. Therefore, sealing the radiator to the extent up to which it is beneficial for the air flow rate is important. Here 3 different models of sealing the radiator are considered which are listed in Table 6. Figure 9 represents the CAD model of the radiator seals.

Table 6
Models and description of iterations used to determine the effect of sealing the radiator

Model	Description
Case P	Complete radiator is sealed to the sidepod
Case Q	Only the top and the bottom area is sealed
Case R	Only the sides of the radiator is sealed to the sidepod

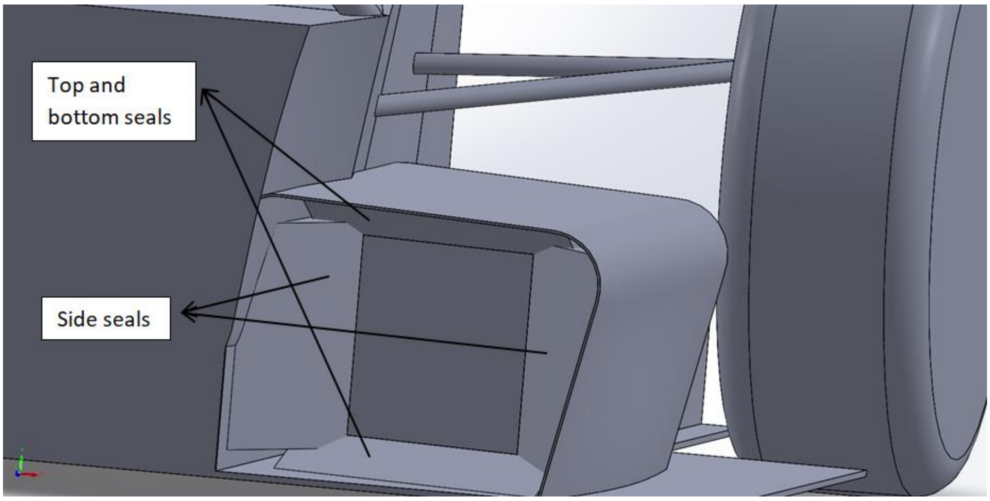

Fig. 9. CAD model of radiator seals

From the results it is evident that completely sealing the radiator results in poor mass flow rate and has the least value of all. The top and bottom seals have the highest mass flow rate of 0.081 thus increasing its efficiency. The results of which are shown in Table 7 and the velocity streamlines passing through the radiator core of case Q is shown in Figure 10.

Table 7
Drag coefficient, lift coefficient and mass flow rate of radiator seal models

Model	Drag coefficient	Lift coefficient	Mass flow rate (kg/s)
P	0.623	-0.407	0.078
Q	0.626	-0.457	0.081
R	0.599	-0.537	0.079

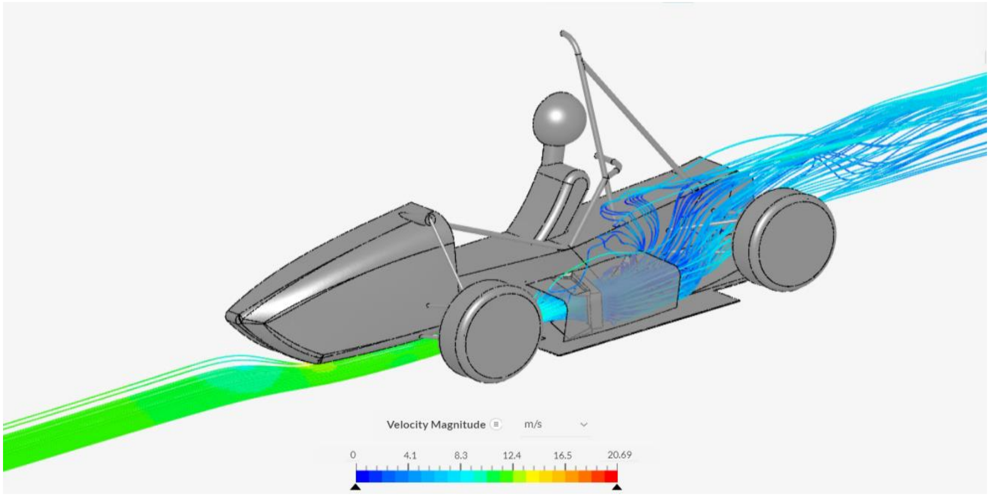

Fig. 10. Velocity streamlines passing through the radiator core of case Q

3.4 Effect of Gills in the Sidepod

Gills are the vent holes/openings present in the sidepod at the rear end of the radiator. This is usually used when the radiator is angled forward when viewed from the side. Angled radiator results in recirculation zones at the top rear end of the radiator as the sidepod acts as a blockage to the air flow direction. Therefore, the sidepod is vented at these regions to let the air out of the sidepod so as to release the blockage.

Here the radiator is angled forward when viewed from the top which creates the low-pressure region at the side end of the radiator which shows in the Figure 11. Therefore, gills are added to the sides of the sidepod to the rear of the radiator. 5 different iterations are considered with the number of gills from 1 to 5 which is shown in the Figure 12.

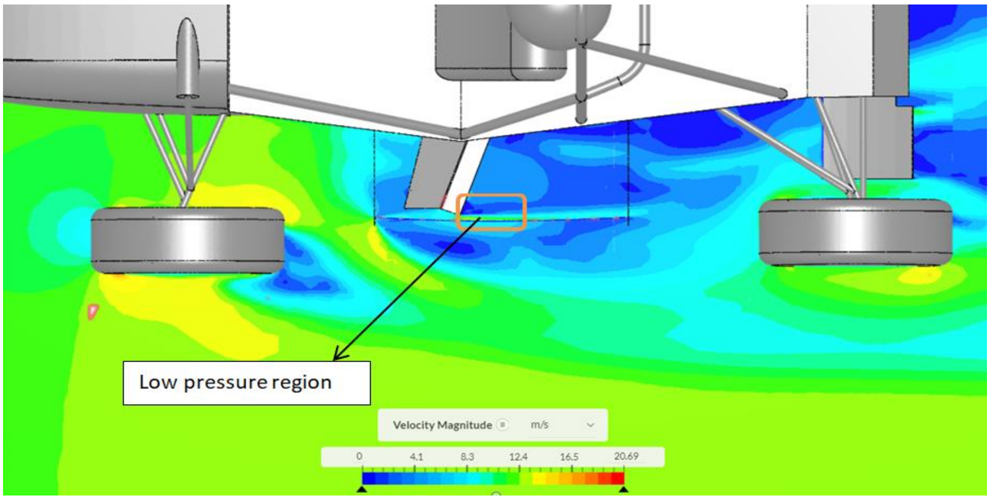

Fig. 11. Velocity contour of low-pressure region at the side end of the radiator

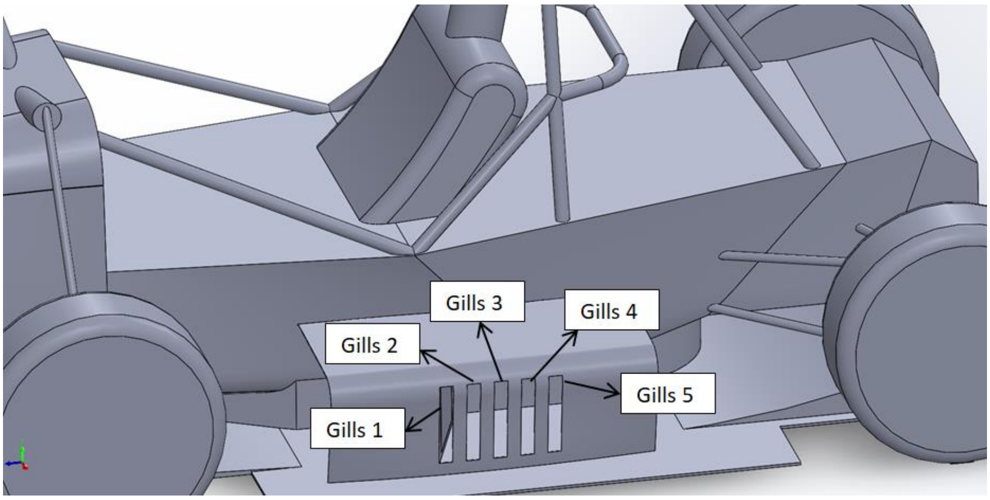

Fig. 12. CAD model of sidepod gills

From the results it is evident that none of the models were efficient enough to increase the mass flow rate than the base model. From analyzing the streamlines which is shown in the Figure 13 it is clear that the turbulence of the front wheel affects the effect of gills in the sidepod. The air which exits the gills from the sidepod is directly exposed to the turbulence region which results in recirculation zones thus reducing the mass flow rate of air passing through the radiator. The results of which are shown in Table 8.

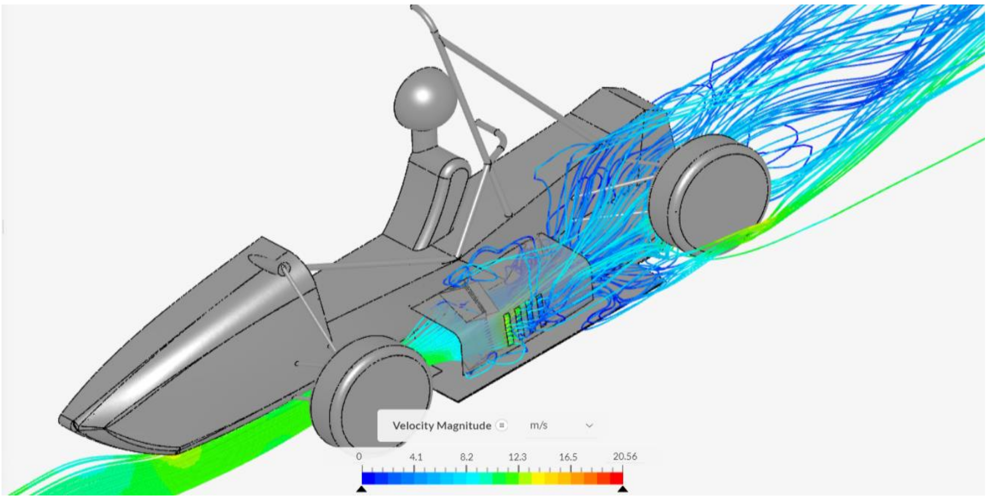

Fig. 13. Velocity streamlines passing through radiator core of gills 4 model

Table 8
Drag coefficient, Lift coefficient and mass flow rate of different models of gills

Model	Drag coefficient	Lift coefficient	Mass flow rate (kg/s)
Gills 1	0.585	-0.442	0.072
Gills 2	0.614	-0.433	0.071
Gills 3	0.595	-0.449	0.073
Gills 4	0.597	-0.456	0.080
Gills 5	0.606	-0.462	0.079

4. Conclusions

From the above research and analysis, it can be concluded that inclining the radiator helps increase the mass flow rate of air passing through the core of the radiator. Radiator angled 70 degrees from the top view had the maximum flow rate of air. Various sidepod models were analyzed which led to the conclusion that sidepods with increasing outlet area for the same inlet area yields the best mass flow rate which is the diverging type sidepod. Further sealing the radiator were analyzed at various sealing positions from which its concluded that sealing the top and bottom of the radiator increases the flow rate through the radiator. Gills were added to the sidepod models on the sides with number of gills varying from 1 to 5 which on the contrast did not increase the efficiency of the sidepod. This is primarily due to the fact that here gills are placed in the sides of the sidepod and is therefore affected by the turbulence of the wheels. Finally, an optimized cooling system for the engine is established with the overall drag and lift coefficient of the car as 0.626 and -0.457 with the mass flow rate of air through the radiator as 0.081 kg/s at a vehicle speed of 40 km/h. The future scope of research in continuation with the present work will be to consider the diverging – converging type of sidepod along with other models for a better comparison. Also, the analysis of gills on top of the sidepod can be performed to better optimize the cooling of the engine.

Acknowledgement
This research was not funded by any grant.

References

[1] Wang, Da, Fuqiu Zhou, Leqi Gu, Yuanxin Huang, Yuchen Shan, Mingjia Yang, Miaoxin Wang, and Yuqi Zhang. "FSAE Aerodynamic Design Based on the Rear Layout of the Radiator and Targeted Improvement of the Cooling System." In *Proceedings of China SAE Congress 2021: Selected Papers*, pp. 603-627. Singapore: Springer Nature Singapore, 2022. https://doi.org/10.1007/978-981-19-3842-9_48

[2] Bahuguna, Rishabh, Tanmay Prasad, Rishabh Khanna, Akshyt Bimal Kumar, Sushant Srivastava, Aagosh Mishra, and B. Ashok. *Design and Development of Cooling System for a Formula SAE Race Car*. No. 2018-01-0079. SAE Technical Paper, 2018. https://doi.org/10.4271/2018-01-0079

[3] Nikhita, K. Venkat, Pritish Bodke, and Venugopal Thangavel. *Design, Validation, and Effective Manufacturing of FSAE Sidepods*. No. 2021-01-5004. SAE Technical Paper, 2021. https://doi.org/10.4271/2021-01-5004

[4] Yadav, J. P., and Bharat Raj Singh. "Study on performance evaluation of automotive radiator." *SAMRIDDHI: A Journal of Physical Sciences, Engineering and Technology* 2, no. 02 (2011): 47-56. https://doi.org/10.18090/samriddhi.v2i2.1604

[5] Kamath, Sachin R., Prajwal Kumar MP, S. N. Shashank, Vinay Damodaran, S. R. Anand, and Prakash Kulkarni. *CFD and experimental optimization of formula SAE race car cooling air duct*. No. 2013-01-0799. SAE Technical Paper, 2013. https://doi.org/10.4271/2013-01-0799

[6] Srivastava, Prateek, Sachin Kansal, Ashish Talwalkar, and R. Harish. "CFD analysis on the influence of angle of attack on vertical axis wind turbine aerodynamics." In *IOP Conference Series: Earth and Environmental Science*, vol. 850, no. 1, p. 012027. IOP Publishing, 2021. https://doi.org/10.1088/1755-1315/850/1/012027

[7] Harish, R., and K. Venkatasubbaiah. "Numerical investigation of instability patterns and nonlinear buoyant exchange flow between enclosures by variable density approach." *Computers & Fluids* 96 (2014): 276-287. https://doi.org/10.1016/j.compfluid.2014.03.026

[8] Waman, W., and R. Harish. "Influence of volume blockage ratio on turbulent buoyant plume dispersion in mixed ventilated tunnel." *Journal of Wind Engineering and Industrial Aerodynamics* 207 (2020): 104397. https://doi.org/10.1016/j.jweia.2020.104397

[9] Landström, Christoffer, Tim Walker, Lasse Christoffersen, and Lennart Löfdahl. *Influences of different front and rear wheel designs on aerodynamic drag of a sedan type passenger car*. No. 2011-01-0165. SAE Technical Paper, 2011. https://doi.org/10.4271/2011-01-0165

[10] Bukovnik, Gernot, Wolfgang von der Linden, and Günter Brenn. *Impact of Rim Orientation on Road Vehicles Aerodynamics Simulations*. No. 2020-01-0674. SAE Technical Paper, 2020. https://doi.org/10.4271/2020-01-0674

[11] Oktavitasari, Dini, Dominicus Danardono Dwi Prija Tjahjana, and Syamsul Hadi. "Experimental Investigation on The Wake Effect of Crossflow Wind Turbines." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 85, no. 2 (2021): 44-50. https://doi.org/10.37934/arfmts.85.2.4450

[12] Saleman, Abdul Rafeq, Mohamad Shukri Zakaria, Ridhwan Jumaidin, and Mohd Nazmin Maslan. "Ultra-Thin Liquid Film with Shear and the Influences on Thermal Energy Transfer at Solid–Liquid Interfaces of Simple Liquid Methane in Contact With (110) Surface Structure of Face-Centred Cubic Lattice (FCC)." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 87, no. 3 (2021): 21-30. https://doi.org/10.37934/arfmts.87.3.2130

[13] Hassan, Qais Hussein, Shaalan Ghanam Afluq, and Mohamed Abed Al Abas Siba. "Numerical investigation of heat transfer in car radiation system using improved coolant." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 83, no. 1 (2021): 61-69. https://doi.org/10.37934/arfmts.83.1.6169

[14] Budiyanto, Muhammad Arif, Alif Hikmah Fikri, and Hanmah Ayuningtyas. "Study on the application of thermoelectric coolers inside unmanned surface vehicles." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 82, no. 1 (2021): 12-20. https://doi.org/10.37934/arfmts.82.1.1220

[15] Abobaker, Mostafa, Sogair Addeep, Lukmon O. Afolabi, and Abdulhafid M. Elfaghi. "Effect of Mesh Type on Numerical Computation of Aerodynamic Coefficients of NACA 0012 Airfoil." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 87, no. 3 (2021): 31-39. https://doi.org/10.37934/arfmts.87.3.3139

[16] Mohamed, Khobib Abdelhafeiz, Obai Younis, and Ahmed Kadhim Hussein. "Numerical Investigation Of Possibility Of Using Shape Memory Alloy In Louvered Fins Radiators." *CFD Letters* 11, no. 3 (2019): 103-116.

[17] Ahmad, Nor Elyana, Essam Abo-Serie, and Adrian Gaylard. "Mesh optimization for ground vehicle aerodynamics." *CFD Letters* 2, no. 1 (2010): 54-65.

[18] Niknahad, Ali. "Numerical study and comparison of turbulent parameters of simple, triangular, and circular vortex generators equipped airfoil model." *Journal of Advanced Research in Numerical Heat Transfer* 8, no. 1 (2022): 1-18.

[19] Elfaghi, Abdulhafid MA, Alhadi A. Abosbaia, Munir FA Alkbir, and Abdoulhdi AB Omran. "CFD Simulation of Forced Convection Heat Transfer Enhancement in Pipe Using Al2O3/Water Nanofluid." *Journal of Advanced Research in Numerical Heat Transfer* 8, no. 1 (2022): 44-49. https://doi.org/10.37934/cfdl.14.9.118124

[20] Hamizi, Ilya Bashiera, and Sher Afghan Khan. "Aerodynamics investigation of delta wing at low Reynold's number." *CFD Letters* 11, no. 2 (2019): 32-41.

[21] Harish, R. "Buoyancy driven turbulent plume induced by protruding heat source in vented enclosure." *International Journal of Mechanical Sciences* 148 (2018): 209-222. https://doi.org/10.1016/j.ijmecsci.2018.09.001

[22] Harish, Rajan, and Kondapalli Venkatasubbaiah. "Large eddy simulation of thermal plume behavior in horizontally partitioned dual enclosure." In *Building Simulation*, vol. 8, pp. 137-148. Tsinghua University Press, 2015. https://doi.org/10.1007/s12273-014-0198-z

[23] Harish, R. "Effect of heat source aspect ratio on turbulent thermal stratification in a naturally ventilated enclosure." *Building and Environment* 143 (2018): 473-486. https://doi.org/10.1016/j.buildenv.2018.07.043