International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2019): 7.583

Nose Design for Formula Student Vehicle with Aerodynamic Components

Harsh Savliya

[1]The Dwarkadas J Sanghvi College of Engineering, Mumbai, India
harshsavliya0743[at]gmail.com

Abstract: ***The paper describes the nose design of a Formula student race car. Aerodynamic package of the car is designed to produce maximum downward force within the acceptable limits of increased drag and reduced top speed. Nose is not an aerodynamic component though it plays an important role as it is the first body part which encounters air. The design is made to use optimum benefit with its shape. This paper gives detailed information about design considerations that are considered to obtain maximum benefits for the performance of a car on track. Computational fluid dynamic (CFD) analysis results, static data, graph related to analysis are also described in this paper. The result of latest design is compared with the previous year design result to quantify the improvement. By performing several changes as per the external flow analysis, we were able to accomplish the desired result. This paper will assist you to achieve the nose design with less drag and maximum possible downforce.***

Keywords: Nose, Formula student, Optimization, Aerodynamics

Nomenclature
C_d: Coefficient of Drag
C_L: Coefficient of Downforce
CFD: Computational Fluid Dynamics
FBH: Front Bulkhead
FRH: Front Roll Hoop
IA: Impact Attenuator
MRH: Main Roll Hoop
SAE: Society of Automotive Engineering
UT: Undertray

1. Introduction

Formula student is a student design competition organized by SAE International. Engineers from worldwide universities design, build and compete at formula student competition around the globe. The formula student competition features two main disciplines, the Static Events and dynamic Events total of 1000 points. Cars are judged by industry specialists and the team with highest points win the competition [3].

More than 50years of Formula style Motorsport history concerning body shape design have a major evolution with aerodynamic design and devices on vehicles [5]. Body parts comprise components such as Diffuser, Undertray, Sidepods, Wings, Vortex generators which generates downforce which pushes the car's tyres towards the track that makes cars move faster around corners [5], [7]. Nose is not an aerodynamic component though it can be utilized to improve performance of a car. The observed changes till now are done with respect to nose design from lower tip to slightly higher tip of nose design in high speed racing cars. Especially when it comes to formula student cars, the current scenario is the cars are limited at the speed which is 120-130kph. At this speed, the effect of aerodynamic forces is limited [4], [5]. So, to increase the vehicle's performance it is necessary to make an aerodynamic package as much as efficient at low speed [6]. In other words, the aerodynamic package should be with minimum amount of drag and the maximum amount of downforce. The main motive of this paper is to design nose with minimal drag force and effective utilization of ground effect to attain maximum downforce. External flow CFD analysis was used to identify pressure around the nose and based on comparison of acquire values the final design was selected.

2. Work done

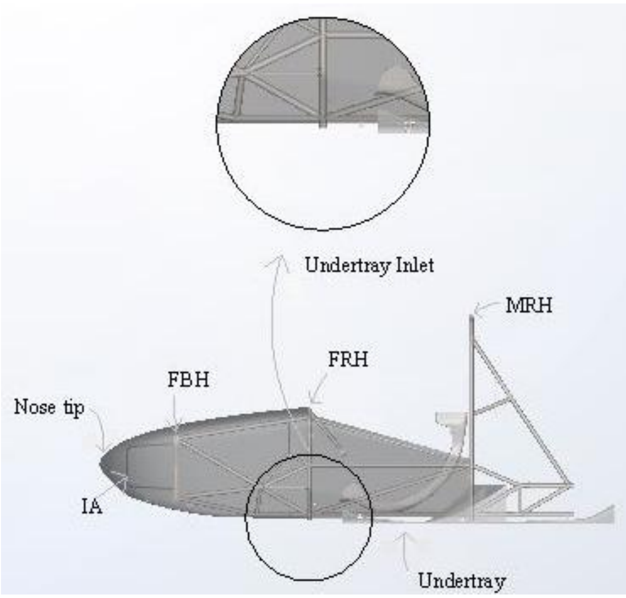

Fig. 1

Nose is part which covers chassis mainly impact attenuator and frontal area of car as shown in fig. 1, also it is the reason for the direction of flow which helps aerodynamic components to generate downforce. The nose plays a major role as it is the first body part that encounters air particles. In order to increase the performance of a car it becomes vital to manage the flow arrives to the car as it is faced by the later portion of the car. Nose separates air particles to flow over, around and under the vehicle body. Nose tip is the front most section of the nose body where this distribution occurs. The more air goes underneath the vehicle produces more pressure difference with the help of ground effect, as per Bernoulli's principle pressure below the car decreases due to decreasing area tends to high velocity flow as compared to above the car. Hence the downforce increases. Ground effect plays a crucial role for a car as it provides streamline flow to undertray which directly causes an increasing downforce as it produces major downforce (In reference frame of the car the flow beneath the car comes in a boundary layer of ground which results in streamline flow). At lower speed, the expected aerodynamic package must include the undertray [7], [8]. Our focus is to design nose with minimal drag and possible maximum downforce. We should reach a conclusion which works for lower speed cars to increase vehicles efficiency.

Nose design starts after designing of the chassis. There are multiple factors that are to be considered for the design of the nose that comprise rules as per Formula Student Competition Rule Book, Streamline design, Position of nose tip etc. The major factor that influences the design most is the nose tip position. As per the tip position differentiation is done as below.

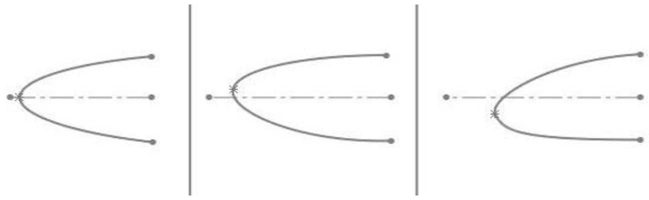

Figure 2: Types of nose tips

There are three segments in which the nose tip position is segregated as shown in above Fig. 2 shows right hand side view of nose which are respectively Symmetrical nose tip position, Higher nose tip position, and lower nose tip position. Here, the tip is the leftmost point on the curve (Frontmost portion on actual nose). Symmetrical tip separates upstream and downstream flow almost equal. As per Bernoulli's principle, to generate downforce change in tip position towards above or below the symmetry line is must. To find out which one is more beneficial, theorical comparison is made.

Bernoulli's equation describes the relation between airspeed and pressure as in equation 1 given below [5]. It shows pressure and velocity are inversely proportional. This formula can be applied to flow around vehicles.

$$\frac{P}{\rho} + \frac{V^2}{2} = \text{Constant}. \quad (1)$$

Higher nose tip:

Pros:

- More space for airflow to go underneath the vehicle as compare to lower nose tip.
- Reduce drag as the front air flows fast underneath the car as suction increase due to ground effect (The distance between the ground and lower surface of Nose).

Cons:

- Less downforce from the upper surface as the tip is higher.

Lower nose tip:

Pros:

- More downforce as more airflow above upper surface (Which tends to increase the force in -Y direction also called as downforce as shown in fig. 3).

Cons:

- More drag as compared to higher tip as the frontal area increases due to more exposure of the body.
- Less utilisation of ground effect as mass flow decreases underneath the vehicle.
- Flow gets moreover distributed around sideways.

Theoretically the most beneficial design according to the points above is the nose with the higher tip. Confirmation of the theory had achieved with CFD analysis as it provides ability to theoretically simulate any physical condition where wind tunnel testing or on track testing takes more time, cost and man power [2].

2.1 Analysis

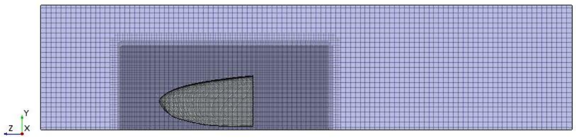

Figure 3: Meshing of final nose model

A solid model of the nose was created in solidworks using surface features. CFD simulation was performed in Star-CCM+ software. To reduce simulation time symmetry feature was used. Trimmer mesh was used for meshing. Several iterations were done to optimize the design. The reference values for respected characteristics are as per table 1 below. The External flow analysis was performed for optimisation.

Table 1

Characteristics	*Reference Values*
Static Temperature	300 *K*
Fluid velocity	15.0 *m/s*
Reference Pressure	101325 *Pa*
Density	1.18415 *kg/m^3*
Fluid Type	Turbulent

2.1.1 Iterations: There are total of four iterations that were done in star CCM+ software. The modification was made to reduce the dents on body region to make it more streamline. Iteration 1, 2, 3 are as shown in Fig. 3, respectively. The pressure counter shows significant increment in pressure difference between point A and B on comparing each iteration, respectively. from iteration 1 to 3. As shown in Fig. 6 drag and downforce coefficient data was also compared to ensure proper modification.

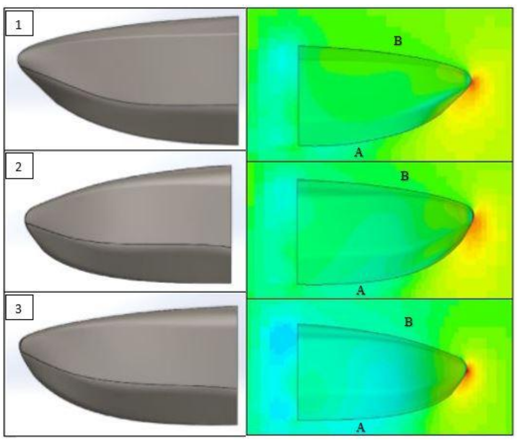

Figure 3: First three iteration respectively

2.1.2 Final Iteration. As shown in fig.4 final nose model was derived after modification according to external flow analysis in iteration 1, 2 &3. Velocity counter and pressure counter were derived as result of external flow analysis as shown in fig.5 and fig.6.

Figure 4: final nose model

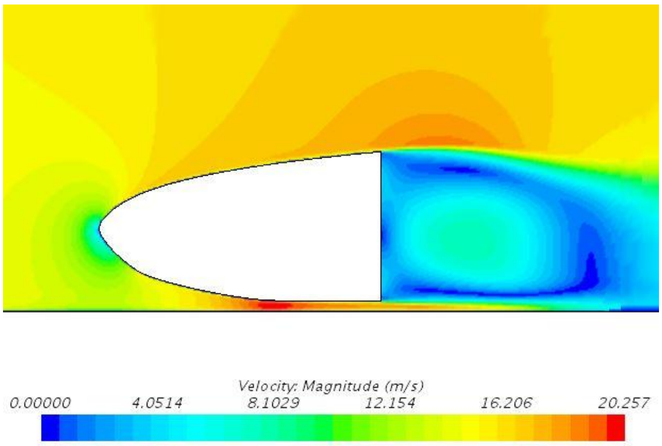

Figure 5: Velocity counter of final nose

The Velocity counter shows the difference between velocity of upstream and downstream airflow. Continuity equation is shown in equation 2 to describe the relation between area and velocity of fluid. Also, according to the law of conservation of mass the velocity of fluid here air must be higher nearby UT inlet to lower the pressure at that region. In fig. 5 region having red colour indicates increasing velocity just ahead of inlet of UT and lower portion of nose as compared to flow over the nose.

$$AV = \text{Constant} \quad (2)$$

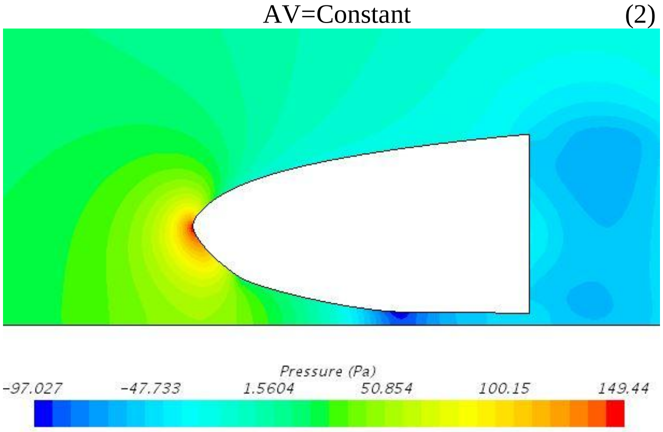

Figure 6: Shows pressure counter of final nose

Pressure counter in Fig. 6 shows significant difference between pressure around upper and lower boundary layer of the surface. After air stagnation in the boundary layer near the nose tip, air splits and flow propagate both upstream and downstream [5]. As airflow moves towards UT inlet it moves with higher velocity as area decreases as already discussed in equation 2, hence pressure decreases. At the same time airflow over the nose resulted in more pressure as compared to below region. Pressure difference increases hence downforce increases.

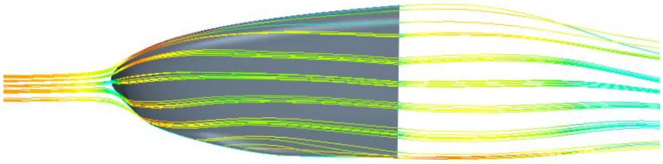

Figure 7: Shows velocity vector of final nose

Velocity vector in fig. 7 shows attached flow with high stability around the nose as a result of streamlined body shape. Coefficient of drag (C_d) and coefficient of downforce (C_L) values that vary after each iteration is shown in table 2. Previous year data, new nose design iterations and final nose design were compared for improvement. C_L and C_d is calculated by formula in equation 3. From the observation of table 2, value of C_d is decreased and C_L is increased. Obtained value of C_d for final nose design is 0.188 which is

lower than previous year value 0.524 and C_L value is increased from 0.024 to 0.08

$$F = \frac{1}{2}\rho A V^2 . \quad (3)$$

Table 2: remark C_d is coefficient of drag and C_L coefficient of downforce, downforce coefficient is denoted by C_L as it is coefficient of negative lift force

Nose design	*Coefficient of drag (Cd)*	*Coefficient of downforce (CL)*
Previous year nose	0.524	0.024
Iteration 1	0.576	0.018
Iteration 2	0.529	0.024
Iteration 3	0.512	0.025
Final nose design	0.188	0.08

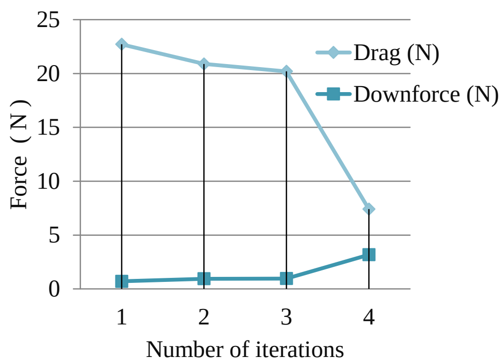

Figure 8: Shows drag force and downforce values after each iteration *(N). Remark, in above line chart final nose design is denoted as iteration 4.*

For final nose design the recorded value of downforce is 3.176N and drag force is 7.417N. Where previous year downforce value recorded is 0.948N and drag force value is 20.884N. Obtained value of downforce and drag of each iteration is plotted on line chart in fig. 8.

Over the process of four iterations, drag force decreased by **64.48%** and downforce increased by **235%** is shown in fig. 9. Thus, increasing the overall efficiency of the nose. Hence determined goal to optimise the nose for better downforce and drag was achieved.

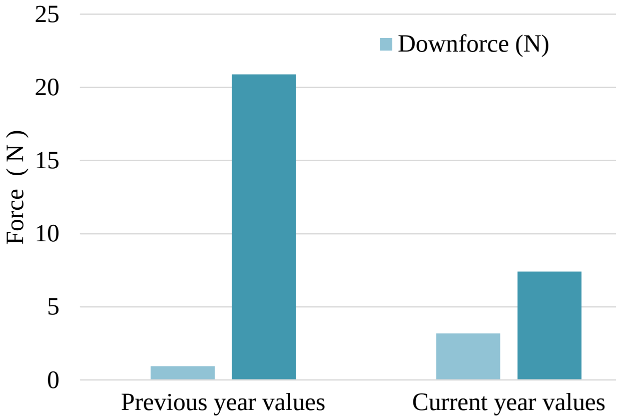

Figure 9: Shows comparison between values of previous year and current year design

3. Conclusion and Future Scope

Nearby 40 years of formula student history, aerodynamics designs have been major changes in frontal portion of vehicle. For design a Nose to improve vehicle performance it was important to conclude those changes for our research. It will provide base platform for further research.

1) The purpose of the paper is to design nose with minimal drag and maximum possible downforce was well achieved. The streamline body reduce the drag force and allows attached flow throughout rest portion of the car that was major reason for drag reduction which can be conclude based on geometrical changes done with the help of CFD analysis. Increase in pressure gradient between upstream and downstream airflow generates more downforce. Included features to the new nose design were slightly higher nose tip than horizontal symmetry line as shown in fig. 2, Inclined upper surface of the nose (from FBH to FRH) and cross-sectional area with continuous increment to make nose model more streamline.
2) Further optimisation, research, and validation (On track testing, Wind tunnel testing) of aerodynamic forces can be done.
3) Skin friction drag can be reduce by decreasing the frontal area [5]. Theoretically, downforce generated by nose is result of pressure gradient, body shape and larger upper section. However larger upper section is equivalent to larger frontal area, which is directly proportional to drag force as shown in fig. 10. Frontal area of upper section should increase, and lower section should decrease in order to avoid increment in total frontal area (Drag force is directly proportional to frontal area) of nose. And larger upper section will result into slightly more downforce (Collision of air particles with upper section at high velocity will result into drag force and downforce where collision of air particles with lower section will result into drag force and lift. Hence, increasing upper section frontal area will lead to increase in downforce with similar drag.)

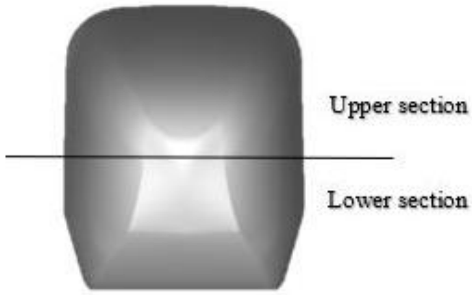

Figure 10: Front view of final nose model

4. Aknowlegements

I would like to thank our college the Dwarkadas J Sanghvi college of engineering, Mumbai and Head of department Vijay Kumar sir for providing platform to our team to build race car and participate in formula student competetion. I am thankful of my guide as well as faculty advisor Vinit Katira for their guidence and suport throughout the process.

References

[1] (n.d.). In R. K. Bansal, *Fluid Mechanics and Hydraulic Mechanics.*
[2] Doddegowda, P. (2006). Use of Computational Fluid Dynamics for the Design of Formula SAE Race Car Aerodynamics. *SAE International*.
[3] FS Rules 2020. (n.d.).
[4] Jawad, B. (2001). The Impact of Aerodynamics on Vehicle Performance in a Formula SAE Racing Style Vehicle. *SAE International*.
[5] Katz, J. (2006). *Race Car Aerodynamics-Designing for Speed*. Bentley Publishers.
[6] Ma, J. (2018). FSAE Race Car Dynamics and Trajectory Optimization Considering Aerodynamic Effects. *SAE International*.
[7] Toet, W. (2013, Jan). Aerodynamics and aerodynamic research in formula 1. *The Aeronautical Journal*, 1-26.
[8] Toet, W., & Zerihan, J. (2006). Ground Effect Aerodynamics of Race Cars. 1-17.

Author Profile

Harsh Savliya pursuing B.E (Passing year 2021) in Mechanical Engineering from The Dwarkadas J Sanghvi college of engineering, Mumbai, India.