International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 07 | July 2021 www.irjet.net p-ISSN: 2395-0072

Design optimization of an Undertray in Formula Student Competitions

Adarsh S. Puranik

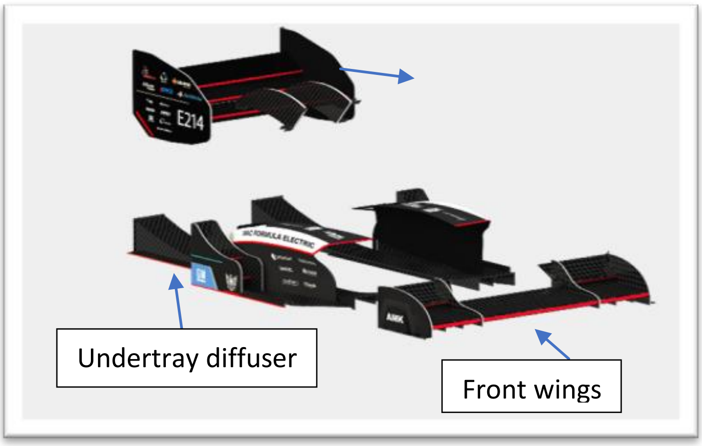
Dept. of Mechanical Engineering, SKNCOE, Pune, India, affiliated to Savitribai Phule Pune University.

Abstract - In today's automotive industry, especially in SAE, an optimum aerodynamic package is definitely a leapfrog over fellow teams. In like manner of the Rear wings and Front wings, the Undertray is also an integral part of the aerodynamic segment that conduces downforce. Perhaps, the elementary aim is to attain maximum downforce and as much low drag as possible, herewith a low pressure head is generated underneath and resultant downforce is achieved. The design report is explained further.

***Key Words*: Undertray, Diffuser, Ground effect, Aerodynamics, CFD.**

1.INTRODUCTION

'Ground Effect' is that term and phenomenon for the undertray which is responsible to create a sort of suction between the car and the ground [1]. W ell ,this was with the undertray, but it is coupled with a diffuser which drives on the basis of Bernoulli's Principle and Venturi effect. Allied to all of the above, they produce a low pressure head under the car and high pressure head above the car which results in increasing the downforce. This kicks in an added grip to the tyre with increased coefficient of friction and thus our intent-high velocity. In distinct to this, with higher acceleration, we ought to keep the 'drag' and the 'weight' low-lying because the auxiliary aerodynamic devices like front and rear wings make noteworthy or comparatively much higher drag.



2.WORKING PRINCIPLE

The cumulative target downforce value by the whole aerodynamic package has to be obtained first. That is computed by iterating various weight distributions, downforce distributions and roll stiffness distributions of the front and rear of the vehicle. Similarly, the permissible amount of drag produced from the components also needs to be noted. This is done by limiting the maximum speed of the vehicle in Autocross/Endurance using the maximum brake power offered by the engine. Skidpad, Autocross and Endurance are the three major dynamic events in the competition.

As maximum downforce from the aerodynamic package is achieved while cornering, the overall process is summarized as:

1. Aim to draw maximum target downforce in Skidpad event.
2. Aim for countering target drag in Autocross/Endurance.

The iterations performed for obtaining the target values of downforce were using tire data of Hossier 16.0 x 6.0- 10 R25B tire in MS Excel. A total of five variables were iterated in all their possible and valid permutations namely, the tyre pressure, the camber, the weight distribution of the vehicle, the downforce distribution of the aerodynamic package and roll stiffness distribution of the vehicle. All the distributions were with respect to the longitudinal axis of the car i.e. Front to Rear

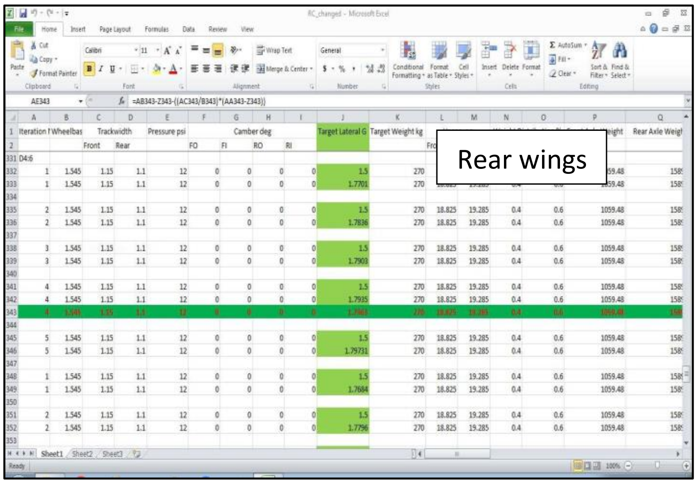


Chart -1: Excel iterations.

A peak lateral g of 1.7g was selected from all the iterations carried out. This value was selected on the basis of the factors mentioned below in the order of their priorities:

1. Achieving the maximum possible lateral g value.
2. Achieving a weight distribution which is close to 60:40 with the Front and the Rear respectively.
3. Achieving a realistic target value for the downforce
4. Achieving a realistic target value of downforce distribution.
5. Achieving a value of roll stiffness distribution such that it minimizes the chassis torsion.

3. DESIGN PATHWAY

Basically, the design protocol for the whole underbody can be divided into 3 stages:-

a. Design of the inlet.
b. Design of the flat portion.
c. Design of the diffuser.

The primitive moto in designing all these portions are as follows:

a. Taking benefit of the ground effect as discussed above.
b. Maximising the plan area as much as possible.
c. Accelerating the air at the end using a diffuser.

In the concluding stage of the design phase when the designs are finalised, Ansys and CFD reports would be reasonable to account its efficiency.

a. Design of the inlet-The undertray inlet has been designed to increase the amount of air flowing into the undertray which on moving further gets restricted between the flat portion and the ground. Due to the restriction that is created, the air will be accelerated and with the Bernoulli's principle applied, a higher velocity corresponds to low pressure, hence there is a drop in the pressure head in that area.
b. Flat portion- This is the section where we can aim to maximise the plan area sideways to an extent we can abide the FSAE rulebook regulations[2]. While the inlet has a converging structure with reference to the ground, the undertray's flat portion maintains a perpetual or constant gap between itself and ground. The air present in that portion is of a lower pressure compared to the surrounding air,this lower pressure is exerted onto the large surface area of the flat portion and in turn converted into downforce.
c. The diffuser-The diffuser is designed for two purposes, first up it generates a zone of low pressure just behind the flat portion of the undertray. This pulls the air from beneath the flat portion towards the rear of vehicle which eventually increases its velocity further. Thereupon resulting in an even larger pressure drop. Secondly, it reduces the velocity of the flowing air to complement with the velocity of the surrounding air. This results in reduction in the turbulence caused when two air-streams merge at the rear. There is no absolute or exemplary value for the angle of diffuser, so it needs to be customized according to our vehicle specifications so as to get satisfying results. The angle of the diffuser is computed considering two variables-a. ride height b. boundary layer.

a. Ride height-Depending on the ride height of the vehicle to achieve peak performance, a higher angle range is recommended for higher ride heights and a lower angle range for low ride heights[3]. This value is finalised by iterating various angles from the range in CFD considering the corresponding cl(coefficient of lift) and cd(coefficient of drag) values, the lowest of these is optimal.
b. Boundary layer- As the higher or lower angle range is determined, a value close to the lower-limit of the range may not detach the boundary layer flow, but then its velocity won't be enough to complement the surrounding air's velocity as discussed. Whereas a value close to the upper-limit of the range will separate the boundary layer flow and this phenomenon is known as "Boundary layer separation" which is not desirable as it will make the airflow turbulent. That being so, a value between the upper and lower limits that does not cause or causes negligible boundary layer separation and complements the velocity at the end with surrounding air is finally selected. This value is obtained by trial and error in CFD with iterations considering the above constraints. The profile of the undertray was flat bottomed undertray with an inlet and double-deck diffuser.

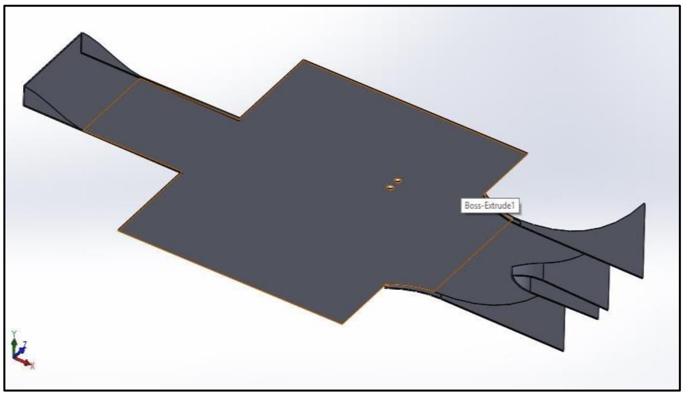


Chart -2: Undertray Bottom view

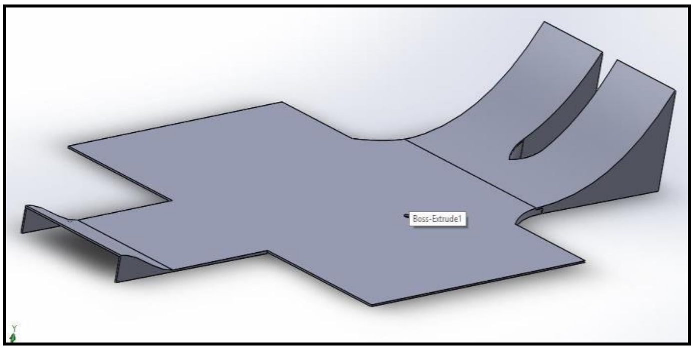


Chart -3: Undertray Top view.

4. CFD SIMULATION

Computational fluid dynamics in automotive industry helps in simulating fluid flow around the vehicle. The simulation can be with respect to pressure, drag, downforce or any other property of interest. For having a CFD model, a computer aided design(CAD) needs to be imported and it has a specific geometry. However, for testing undertray's aerodynamics, a "Wind tunnel" box is placed around the vehicle. The entrance or inlet of the wind tunnel is few car lengths away from the geometry which acts as the velocity inlet at the same time. The exit of the wind tunnel is considered many car lengths away from the behind of the geometry which acts as the pressure outlet at the same time. The simulation is done on an open wheeled vehicle, it is necessary that the tires are rotating and the ground is considered moving or frictionless. Apart from this, it is proclaimed that the tire dynamics are well simulated in order to attain satisfying results for the rest of the simulation carried out. The simulation is carried out for a turbulent flow because the airflow around the vehicle is highly turbulent, which is initiated right from the frontal area section. There are four types of simulation methods namely k-ε, k-ω, Lattice-Boltzmann and Large Eddy Simulation(LES). But the k-ε is said to be most stable according to the literature review and is widely used so that method was selected[4]. In this case, the CFD analysis was done at a velocity of 45.65kmph. The following approach was followed in the CFD analysis process-

1. Importing a solid CAD model of the aerodynamic device.
2. Generating a virtual box around the solid CAD model in order to give appropriate boundary conditions
3. Subtracting the solid CAD model from the box to generate a wall of the shape of the aerodynamic device onto which forces can be measured
4. Meshing is done by using Ansys auto-mesh, with Advanced Sizing Function set to proximity & curvature and the rest settings set to default.
5. Giving inlet boundary condition as velocity inlet to the front face of the box.
6. Pressure outlet condition was given to the rear, side and the top walls.
7. The bottom wall or the ground is kept moving at 45.65kmph for front wings and undertray whereas stationary for rear wings.
8. After initializing, the number of iterations is entered and the analysis is performed until convergence is achieved.

5. ANALYSIS RESULTS

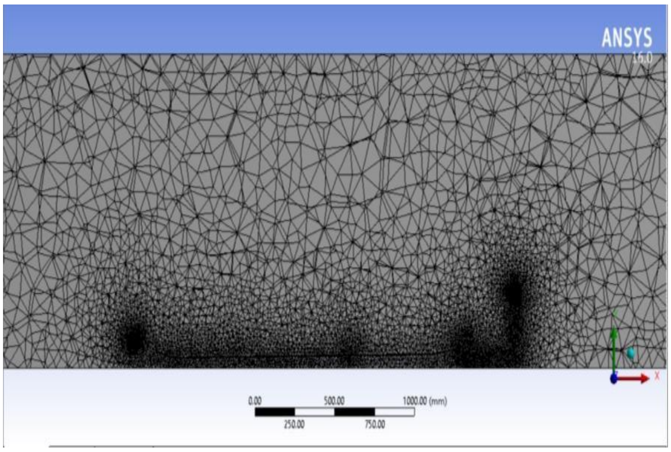


Chart -4: Undertray sideway mesh.

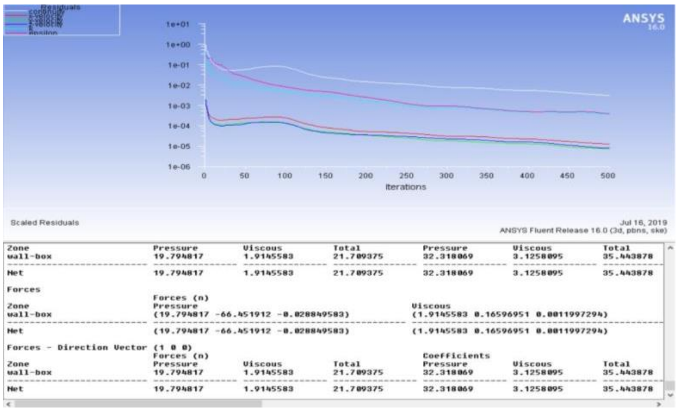


Chart-6:Equivalent forces on Ansys.

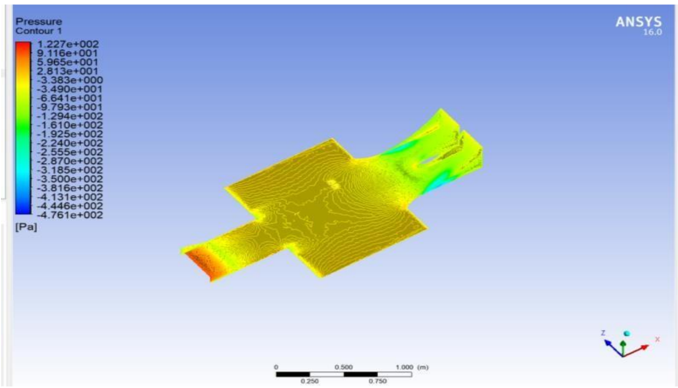


Chart-6:Undertray Pressure Contour Top.

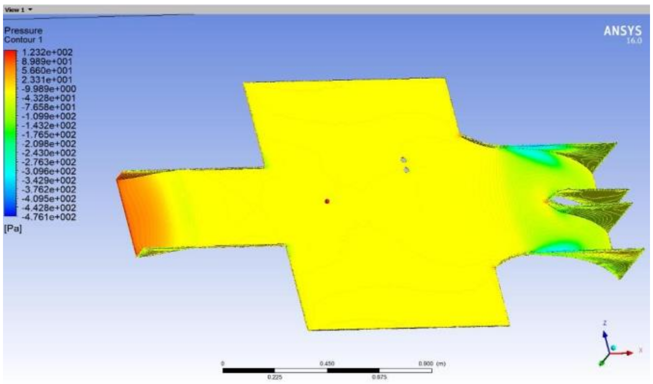


Chart -7:Undertray Pressure Contour Bottom.

Analyzing the CFD's, the inlet angle, the plan area and the diffuser angle were finalized. The inlet angle was 24 degrees where as the diffuser angle was set at 19 degrees at the start and a curve coincident to that was given until there was no boundary layer separation and negligible drag. The plan area was laterally maximized till rulebook regulations allow and longitudinally according to the mountings of the inlet and the diffuser section.

6. CONCLUSION

The resultant downforce was contributed by the undertray of 66.45N and the drag was 19.79N at 40kmph, the weight being 1484gm. There was a significant improvement in the skidpad timings compared to the timings without the undertray. Hence, a balanced and functional undertray was developed and satisfying results were obtained.

7. ACKNOWLEDGEMENT

The completion of this undertaking paper could not have been possible without the participation and assistance of people who guided me, my college faculty. I express deep gratitude and am thankful to my college Smt. Kashibai Navale college of Engineering for helping me throughout the whole process.

REFERENCES

[1] "Willem Toet explains….motorsport diffusers | Race Tech Magazine," Race Tech Magazine |, Aug. 08, 2017. https://www.racetechmag.com/2017/08/willem-toet-explains-motorsport-diffusers/ (accessed May 15, 2021).

[2] "FS-Rules_2020_V1.0.pdf." Accessed: May 16, 2021. [Online]. Available: https://www.formulastudent.de/fileadmin/user_upload/all/2020/rules/FS-Rules_2020_V1.0.pdf

[3] "Download citation of Influence of Diffuser Angle on a Bluff Body in Ground Effect." https://www.researchgate.net/publication/237900355_Influence_of_Diffuser_Angle_on_a_Bluff_Body_in_Ground_Effect/citation/download (accessed May 15, 2021).

[4] K. Jensen, "Aerodynamic undertray design for formula SAE."