Development of External Aerodynamics of an FSAE Racecar using Computational Fluid Dynamics

Avinash Mohanan
Department of Mechanical Engineering
Vidyavardhini's College of Engineering and Technology
Vasai,India

Aashish Vijay Gupta
Department of Mechanical Engineering
Vidyavardhini's College of Engineering and Technology
Vasai,India

Jayesh Anil Pandey
Department of Mechanical Engineering
Vidyavardhini's College of Engineering and Technology
Vasai,India

Susana Godwin Almeida
Department of Mechanical Engineering
Vidyavardhini's College of Engineering and Technology
Vasai,India

Prof. Dipak Choudhari
Assistant Professor
Department of Mechanical Engineering
Vidyavardhini's College of Engineering and Technology
Vasai,India

Abstract—**This paper describes the procedure implemented in improving the external Aerodynamic characteristics of the FSAE racecar prototype of Vidyvardhini's College of Engineering and Technology, Vasai and development of a new Aerodynamic package for the Year 2020-21 using Computational Fluid Dynamics (CFD) packages. The objectives of the design is to increase Down force on the car with least possible induced Drag so as to improve the dynamic performance of the car in FSAE dynamic events. Also the end comparison between the Aerodynamic characteristics of the 2018-19 model and new model is presented.**

Keywords-FSAE, Formula Student, Aerodynamics, CFD.

I. INTRODUCTION

A. *Role of Aerodynamics in Race Car.*

Racecars are meant to be as fast as possible through straights and corners and at the same time, are required to be as light as possible to yield the maximum performance from the driving unit. Traction is a vital component in improving the cornering speeds which is mainly affected by the downward force available on the tires. Since increasing the Static load of the car is not an option, usage of external aerodynamic features comes in picture. Aerodynamic features on the car create downward forces when in dynamic state and hence help in achieving the required forces for optimum traction without adding on considerable additional dead load on the car.

B. *Formula SAE or Formula Student competition.*

Formula Student or Formula SAE (FSAE) is a student design competition held across the world where Student teams from various Universities and colleges participate and compete with each other. A prototype of an open wheel race car is build by the students within the set of universal FSAE constraints provided by the competition. The participating team competes under the domains of Design, Cost Analysis, Business Plan Development and Dynamic Performance of the car.

B. *Scope.*

In the paper the development of an aerodynamic package including Front Wings, Rear Wings and Underbody Diffuser is explained. The brief overview of the result iterations of the CFD analysis of the individual components and the Analysis of the complete car is presented and the overall behavior is discussed. The Aerodynamic characteristics of the 2018-19 car is compared with the latest developed Aero-kit on the basis of CFD analysis reports.

II. NOMENCLATURE

TABLE I. NOMENCLATURE

Symbol	Units	Physical Quantities
L	*N*	*Downforce (negative Lift)*
D	*N*	*Drag Force*
A	*M2*	*Frontal Area*
ρ	*Kg/m3*	*Density of Air*
Cl	-	*Coefficient of Lift*
Cd	-	*Coefficient of Drag*
v	*m/s*	*Free Stream Velocity*

III. DESIGN

A. *CFD Environment*

TABLE II. COMPUTER CONFIGURATION

Software Used	Simscale (Openfoam), Siemens Star CCM+
CFD Model	k-omega SST
Computer Spec:	3.4 GHz ,4-Core, GTX 1060
CFD Geometry	SolidWorks 2016

The inlet free stream velocity is considered at 80km/hr as suggested by the competition regulations.

B. Design Theory:

In reference to [2] and [6], the approach to consider tradeoff between the available power and induced drag was adopted. The Engine used is KTM 390cc. As per the experimental data the available power after restricting the air intake is 29 bhp.

i.e. 21.625KW. …………….[A]

As per FSAE regulations[7] the maximum speed permitted on track is 110 km/hr. The power required by the car to maintain 110 km/hr is 10.3 KW taking into consideration the rolling resistance. Therefore the power left over unused is approximated to 11.2 KW which can be utilized to overcome the drag induced when aero-kit is implemented.

In reference to the literature study from [3],[4] and [5], the effect of multi element wing configuration and effect of camber on lift and drag, Airfoils are selected from a wide database of predefined airfoils, airfoiltool.com[2].

C. Front Wings:

1) Approach:

The Front wing is used to deflect the air away from the front surface of the front wheels so as to reduce the turbulence created at the start. It creates good amount of down force due to the venturi effect created between the bottom surface of the airfoil and the ground.

The wing is divided into 3 sections namely 2 main winglets and 1 center winglet. The center portion of the front wing is ineffective due to the disturbance of the nose cone. Hence an airfoil profile is used in the center at 0^{o} angle of attack. This produces negligible drag at center profile and deflects the air downwards into the diffuser. The venturi formed accelerates the air underneath which helps in creating low pressure zone increasing the down force which in turn increases the undertray efficiency.

2) Specifications:

TABLE III. AIRFOIL SPECIFICATIONS FRONT WING

Wing Section	Airfoil used	Chord length
Main	EPPLER 421	490mm
Center	NACA 2412	380mm
Flaps	CH-10	130mm

In reference to [7], for a front wing, outside the inner face of the wheel a maximum height of 250mm is allowed and inside the inner face of the wheel maximum height of 500mm is allowed. The distance between the outer face of the front wheels is the maximum length allowed. On the basis of these rules, the cord lengths and angle of attacks are optimized.

The figure below is the CAD model of the front wings designed keeping in mind the competition regulations and mountings.

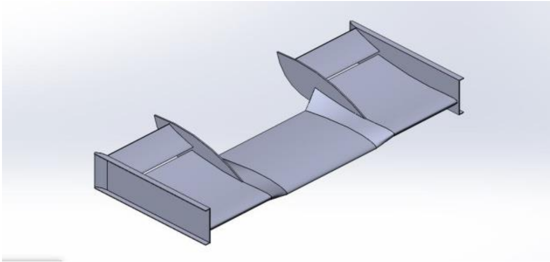

Figure 1. CAD Model of Front Wing.

3) Analysis :

Independent CFD analysis is performed using StarCCM+ considering all different variable parameters.

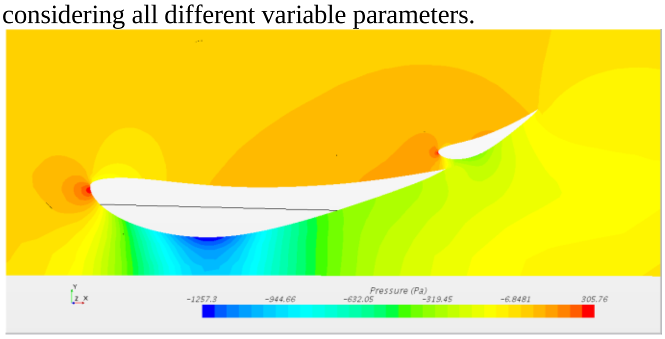

Figure 2. CFD Result of Front Wings (*Pressure Profile)*

The fact that the front wings are close to the ground enhances the downforce generated. But in reference to [7] the minimum static ground clearance of any portion of the vehicle other than the tires including a driver must be minimum 30mm. This became evident in the full body simulation results.

4) Results:

The final selected aangle of attacks is listed down in the table below.It was observed that the results were more prominent with increasing angle of attacks of the flap but due to restrictions as mentioned earlier, the final result was settled as listed.

TABLE IV. ATTACK ANGLES OF FRONT WING

Winglet	Angle of Attack
Main	5.2 Deg.
Center	0 Deg.
Flap	26 Deg.

D. Rear Wings:

1) Approach:

Rear wing contributes to the major portion of the downforce generated. A 3-element wing is designed in order to optimize the possible angle of attacks and utilize the available enclosure. The basic approach is to achieve a flow without separation at the wing surface hence reducing the Drag and increasing the Downforce.

2) Specifications:

TABLE V. AIRFOIL SPECIFICATION REAR WINGS

Wing Section	Airfoil Used	Chord Length
Main	S1223-il	450mm
Flaps	CH-10	150mm

In reference to [7], the rear wing is provided with the maximum enclosure volume. The enclosure includes the distance between the inner face of the rear tires as the maximum width, maximum height of 1200mm from ground and can extend rearwards to a maximum of 250mm from the rearmost point of the rear tires.

3) Analysis :

Within the enclosure provided, iterations were performed with variable parameters of spacing between the winglets, chord lengths and angle of attacks. Simulation results were studied and chord lengths and spacing was determined.

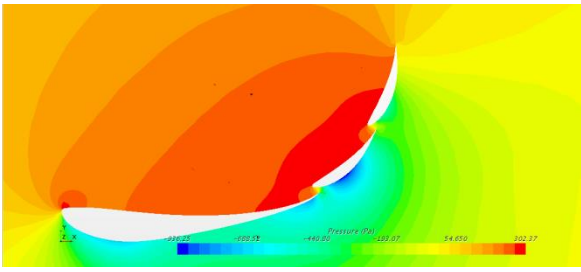

Figure 3. CFD Result of Front Wings (*Pressure Profile)*

Relative angles of attack were varied and simulations were performed. The fig.4 represents the variation of the downforce and drag value at increasing angle of attack of the flaps. It can be observed that the downforce value attains a peak in between $\alpha=35^{o}$to $\alpha=40^{o}$. Beyond that range the value of downforce drops down and drag value increases with the increase in α which was evident in the simulation results showing the flow separating at the rear.

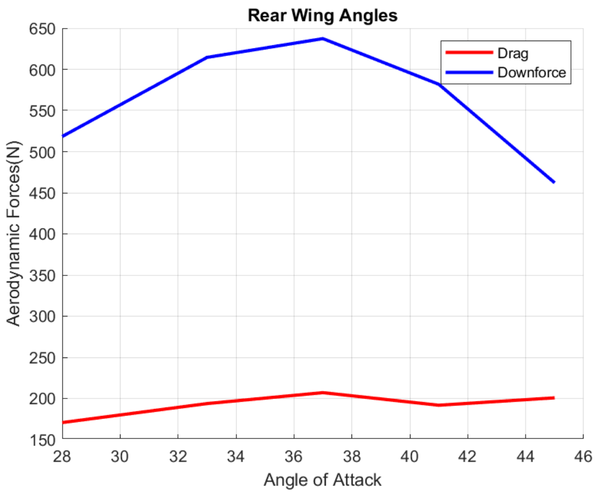

Figure 4. Graph of Downforce/ Drag vs Angle of Attack *(Rear Wing)*

Figure 5.

The graph 4 indicates the variation of the Aerodynamic forces with reference to the angle of attack of the flap 2.

Selected Configuration:

TABLE VI. ATTACK ANGLES OF REAR WINGS

Winglet	Angle of Attack
Main	4.5 Deg.
Flap 1	30 Deg.
Flap 2	33 Deg.

4) Results:

TABLE VII. RESULT (REAR WING)

Down force Generated	614 N
Drag force Generated	193 N

These values obtained are considering free stream flow at 80km/hr without turbulence which is not the case in actual car. The efficiency of the rear wing reduces due to the turbulence created by the main hoop, firewall, head restraint and the rest of the body which is evident in the full body simulation results.

E. Diffuser:

1) Approach:

The diffuser or Underbody covers the base of the car. The Ground effect produced by the diffuser accelerates the air beneath the car and hence producing a low pressure zone beneath the car. The car behaves like an airfoil and this pressure gradient pushes the car downwards producing downforce. Since only negligible amount of drag is produced and there is no surface obstructing the flow of air, diffuser is the most efficient in terms of downforce to drag proportion. The angle of the inlet to diffuser and outlet diffusing angles are designed to form an effective venturi to assist the ground effect.

2) Specifications:

TABLE VIII. DIFFUSER SPECIFICATION

Section	Angle of Diffuser
Center	38 Deg
Side	28.8 Deg

As per the regulations [7] of the competition no aerodynamic devices are allowed in the vicinity of 75mm of the tires. The maximum height is 500mm above the ground and can extend maximum up to 250mm rearwards from the rearmost point on the rear tires.

3) Analysis :

A profile of 700mm width was used for initial simulations to study the effect of varying diffusing angles. The length was kept to the actual scale.

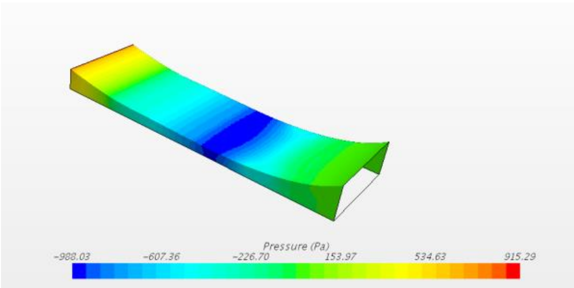

Figure 6. CFD Result of Diffuser (*Pressure Profile)*

The graph in figure 6 representing the values of drag and downforce shows the effect of the diffusing angles. This study was encorporated with design of the actual underbody.

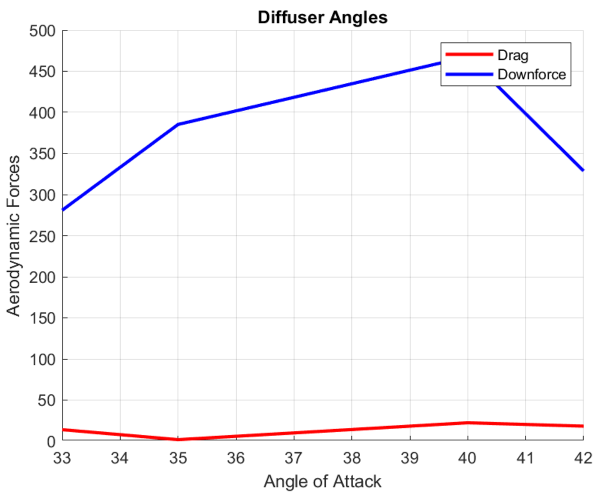

Figure 7. Graph of Downforce/ Drag vs Angle of Attack (*Diffuser*)

The actual underbody was designed in a channeled manner with 2 channels in the center and 1 on both sides. Simulations of the channeled diffuser provided better results and showcased increase in the downforce value.

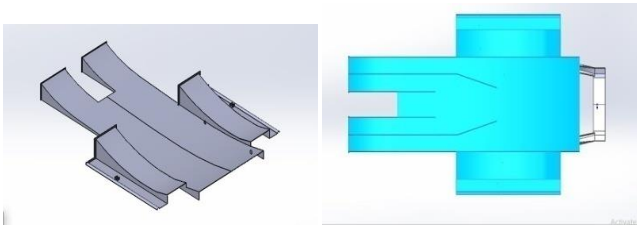

Figure 8. CAD Model of Diffuser.

The flow to the diffuser inlet is assisted by the front wings and hence the actual forces generated by the diffuser increase once simulated with the whole body.

IV. FULL-BODY ANALYSIS

A. 2020-21 New Aero-Kit (Model 1)

The newly developed Aero-kit was assembled with the chassis and a full body simulation was performed on StarCCM+. The model used for simulations was a simplified version to ease the simulation load.

Additional Aero-elements including Gurney Flap at the end flaps of the wing, Vortex generating wedges, etc were included and simulated.W

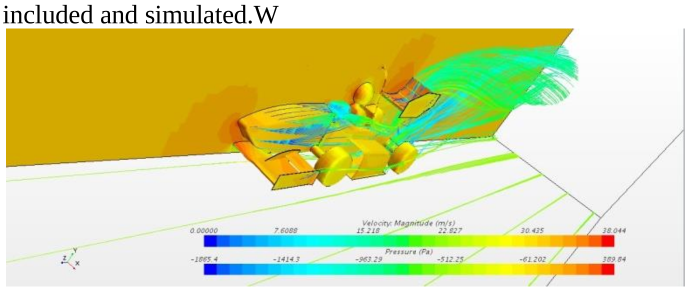

Figure 9. Full Body CFD Analysis.*(Model 1)*

Results of the Simulation:

TABLE IX. RESULT OF FULL BODY CFD(MODEL 1)

No. of Iterations	1000
Downforce	791.18 N
Drag	411.04 N
Coefficient of Lift	2.404
Coefficient of Drag	1.243
Frontal Area	1.069 m^2
Free Stream Velocity	22 m/s

The average speed on the competition tracks ranges between 60kmph to 80kmph. The power required by the car with the above mentioned characteristics to overcome the air resistance is calculated using:

$$P_{req} = \frac{1}{2} \times \rho \times V^3 \times A \times C_d \qquad \text{…[B]}$$

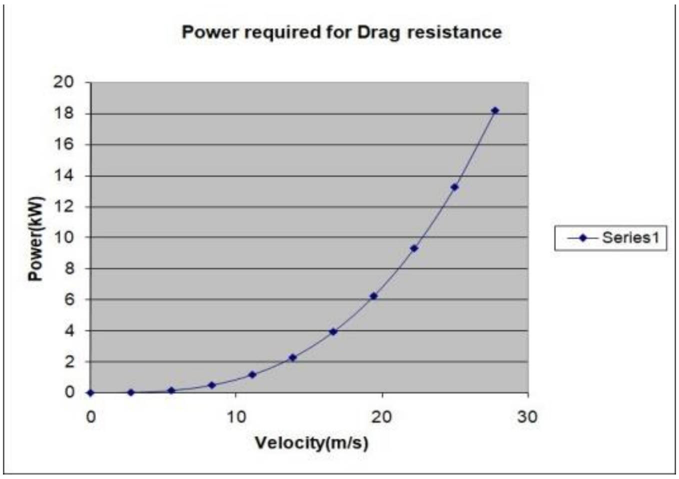

Figure 10. Graph of Power vs Car Speed.

The power required is approximately 9.2 KW at 22m/s as per the Graph obtained using the above relation [B].The graph plots the power required by the vehicle with the Aerodynamic characteristics to maintain a certain velocity.

B. 2018-19 Prototype(Model 2)

The engine specifications were kept the same as mentioned earlier. The car did not have an Aero-Kit. Simulation Results of the same are as given below:

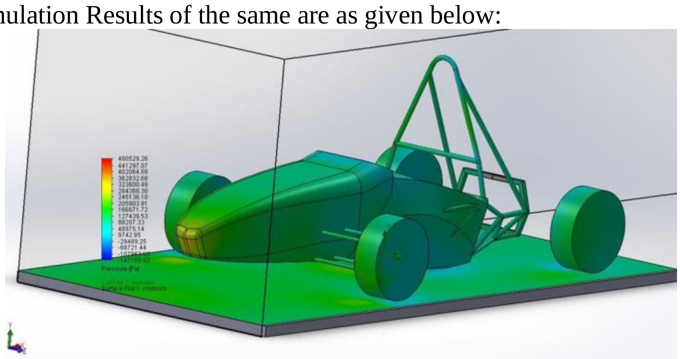

Figure 10. Full Body CFD Analysis. (*Model 2*)

TABLE X. RESULT OF FULL BODY CFD(MODEL 2)

No. of Iterations	980
Downforce	87.417N
Drag	139.314 N
Coefficient of Lift	0.3664
Coefficient of Drag	0.584
Frontal Area	0.7590 m^2
Free Stream Velocity	22 m/s

The power utilized by the above model calculated according to [A] is 3.092 KW.

V. COMPARISON AND OBSERVATIONS

The Power in case of Model 2 is underutilized in reference to available power calculated at [A]. Here the scope of implementation of aerodynamic components becomes evident.

In accordance to the design calculation [A], the power usage in Model 1 is under the available power value and hence the above characteristics satisfy the constraints and power utilization is optimized.

Even though there is increase in the drag force on Model 1 as compared to the older version but the power lost to overcome this drag can be contained within the design objectives of the car. The amount of downforce generated increased the traction between the tires and decreased the lap time. Since FSAE tracks consists of more of corners the added downforce provides an upper edge to optimize the lap timings.

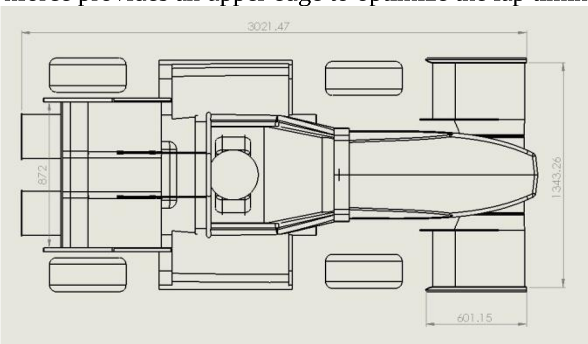

Figure 11. Final CAD drawing

VI. FUTURE SCOPE

The track testing validation using Coast-Down testing method can be done to validate the CFD design. Also methods like wind tunnel testing can be performed to validate the results. In order to optimize straight line accelerations, Drag Reduction System can be implemented to automatically reduce the angle of attack wing flaps so as to reduce drag.

REFERENCES

[1] www.airfoiltools.com
[2] Sahil Gupta* and Kishal Saxena, "Aerodynamic Analysis of a Formula SAE car," in 13th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
[3] Sneh Hetawal, Mandar Gophane, Ajay B.K., Yagnavalkya Mukkamala, "Aerodynamic Study of Formula SAE Car," in 12th Global Congress On Manufacturing And Management, GCMM 2014.
[4] Francesco Mariani, Claudio Poggiani, Francesco Risi, Lorenzo Scappaticci, "Formula-SAE Racing Car: Experimental and Numeric Analysis of External Aerodynamics", in 69th Conference of the Italian Thermal Machines Engineering Association, ATI2014 .
[5] General Aviation Aircraft Design, 2014.
[6] James Patrick Merkel, " Development of Multi-Element Active Aerodynamics for the Formula SAE car", Presented to the Faculty of the Graduate School of The University of Texas at Arlington.
[7] Formula Student Germany Rule Book 2020.