Design and Optimization of the Diffuser for the Formula SAE Car for Improved Performance

Sadjyot Biswal, Aravind Prasanth, Vishal G. Naranje, *Member, IAENG*

***Abstract*— The focus of this paper is to study and analyze the governing factors in designing the diffuser for the under tray of a Formula SAE car. The topics that will be covered are relationship between diffuser angle and downforce, the importance of ground effect in designing a diffuser. The paper will also focus on the relationship between velocity and downforce for a diffuser, the link between the dimensions of the diffuser and downforce and drag and finally to create a design and to further optimize it giving the best possible design for the diffuser.**

***Index Terms*— ANSYS, diffuser, downforce, SOLIDWORKS, Optimization**

I. Introduction

THE Formula SAE competition consists of a track in which the focus is more on cornering rather than on the straight line speed. To give the extra added advantage during cornering it was decided to design an under tray with a diffuser which gives added downforce and significantly increases the performance.

The Formula SAE rules state that:

In plain view, no part of any aerodynamic device, wing, undertray or splitter can be:

- Further rearward than 250 mm (9.8 inches) rearward of the rear of the rear tires
- Further forward than a vertical plane through the rearmost portion of the front face of the driver head restraint support, excluding any padding, set (if adjustable) in its fully rearward position (excluding undertrays).
- Wider than the inside of the rear tires, measured at the height of the hub centerline.

These rules were to be kept in mind while designing the undertray.

II. Analysis

The diffuser is a component, which gives the highest amount of downforce while minimizing drag. The first step in designing the diffuser was to analyze the bluff body. The buff body is a blunt body whose outer dimensions are similar to the FSAE car. This helps in determining the optimum size, angle of the diffuser as well as helps in studying the effect of ground height in designing a diffuser. The analysis on the bluff body is conducted in ANSYS Fluent.

.

A. Geometry

The first step was to design the bluff body geometry for simulation in ANSYS. The total length of the car was taken as around 3 meters and the width around 1 meter. The height of the car was assumed to be 400 mm. The undertray ahead of the diffuser had a length of 1.9 m. The diffuser length was to be varied and analyzed to determine the length that would produce maximum downforce. The enclosure for simulation was made to be 1.5 and 3.5 times the length of the car fore and aft the bluff body in the direction of the length of the car respectively. The width of the enclosure on both the sides was equal to 1 car length while the enclosure height was equal to twice the car length.

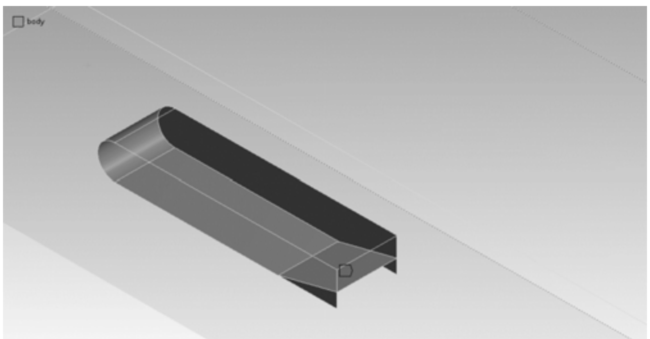

Figure 1 Geometry of bluff body

B. Meshing

The mesh count was about 1,000,000, which helps give highly accurate values for the simulation. In the meshing stage the named selections are also created which are to be used later on in the CFD analysis. The named selections that are to be created are for the inlet, outlet, body and ground.

Manuscript received March 19, 2016; revised April 10, 2016.
Sadjyot Biswal is studying at the Birla Institute of Science and Technology Pilani, Dubai, PO. 345055 UAE (sadjyotbiswal9@gmail.com).
Aravind Prasanth is studying at the Birla Institute of Science and Technology Pilani, Dubai, PO. 345055 UAE (phone: +971 552140408;
e-mail: aravind.prasanth@gmail.com).
Dr. Vishal G. Naranje is teaching at the Birla Institute of Science and Technology Pilani, Dubai, PO. 345055 UAE (azan.barodawala3@gmail.com).

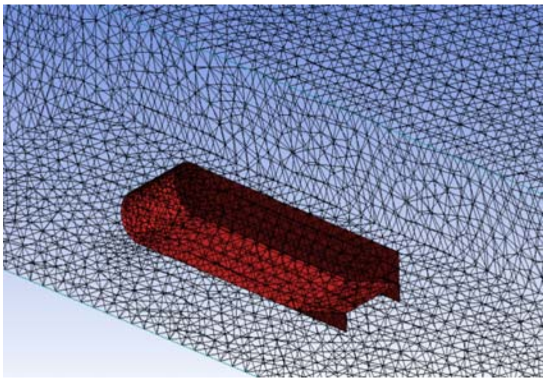

Figure 2. Meshing of enclosure

C. CFD analysis

For the solution in ANSYS CFD the simulation was done using k epsilon eq. with RNG selected. The RNG helps give accurate results for swirls and strained flows. The ground is defined as moving with the same velocity and direction as that of the inlet. Since the main concerns for the diffuser design are lift and drag, monitors to analyze them are also created.
These settings created in ANSYS CFD are used to simulate and study the results for various factors like ground effect, diffuser length and diffuser angle.

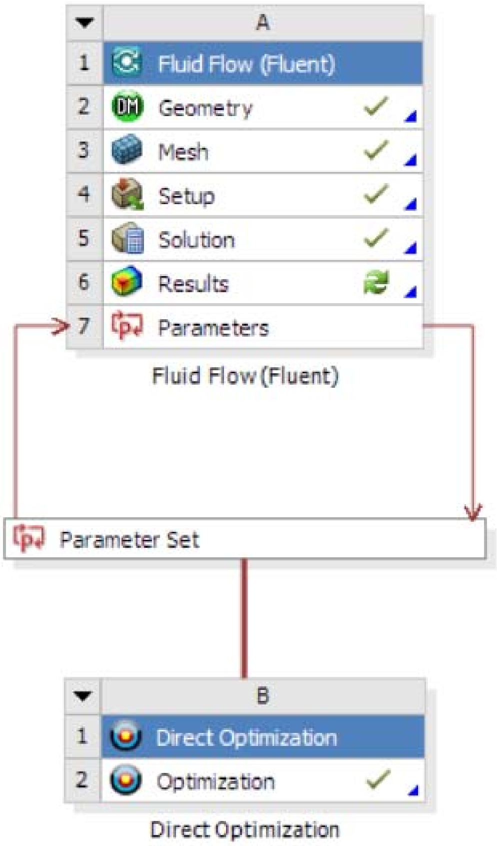

Figure 3. Direct optimization

III. RESULTS

A. Study of Ground Effect

Ground effect plays a very crucial role in deciding the design of the diffuser. The downforce increases proportionally as the ground clearance needs to be lowered until the downforce starts decreasing. Therefore, simulation was conducted extensively in ANSYS to study the ground effect and its relationship with the downforce.

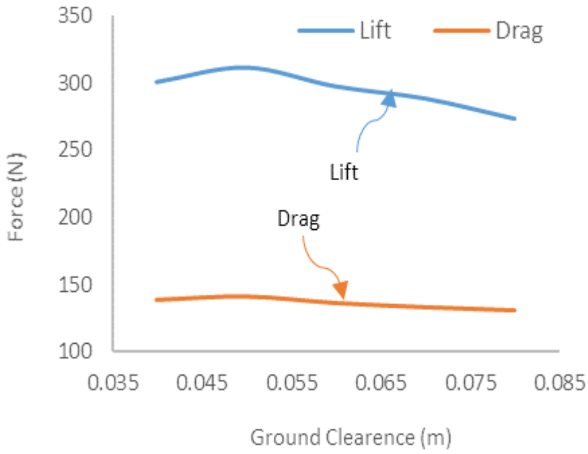

Figure 4. Graph btw Ground clearence and Force

B. Study of diffuser length

The length of the diffuser part of the undertray was varied to get the best results for downforce. The results can be seen below.

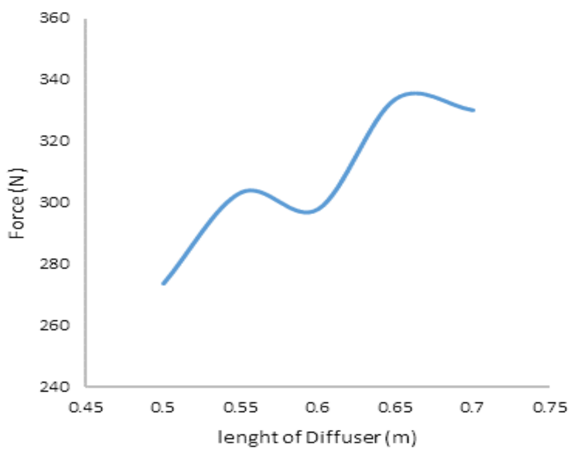

Figure 5. Graph btw length of diffuser and force

C. Study of diffuser angle

The diffuser angle is one of the main factors that determine the downforce produced by the diffuser. Narrowing down the angle for optimum diffuser performance is of the utmost importance because the downforce will be maximum only at a particular angle.

At higher angles flow separation occurs increasing drag and having highly unfavorable results.

ISBN: 978-988-14048-0-0
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

The results are as follows.

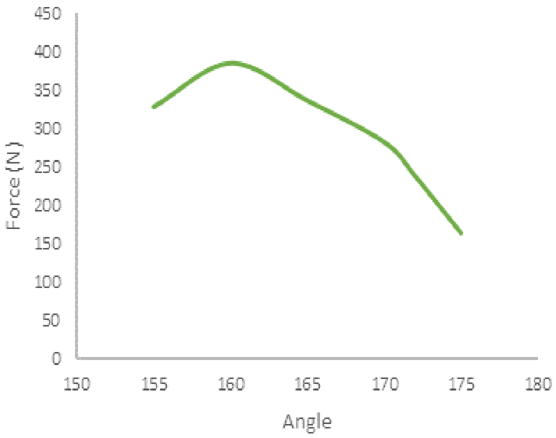

Figure 6. Graph btw angle of diffuser anf force

D. Study of velocity

The effect of velocity on the downforce and drag of the diffuser was studied to find out if change in velocity due to car speed or weather conditions on the track had any impact on performance of the diffuser.

The results can be seen as follows.

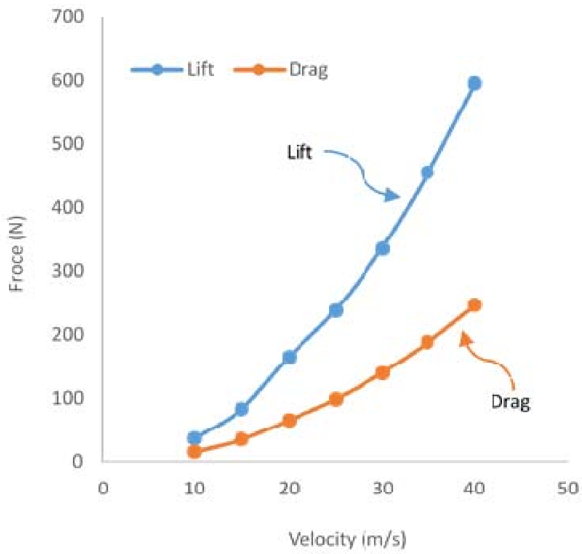

Figure 7. Graph btw velocity of the car and downforce

E. Length of the car

The effect of the length of the car on the downforce and drag was studied to see how change in car length impacted the performance of the diffuser

The results can be seen as follows.

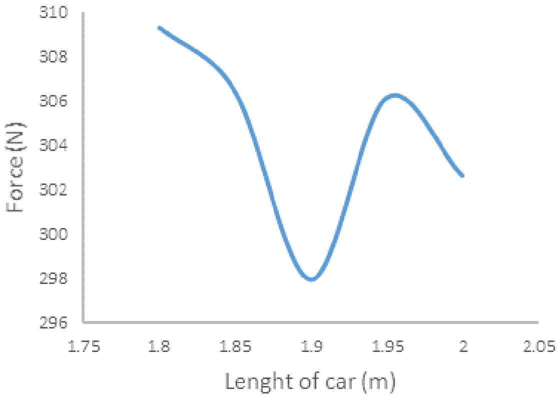

Figure 8. Graph btw length of the car and downforce

F. Pressure Contours

It is a representation of pressure gradient over the body. The contours help in refining and optimizing the design by removing any abrupt changes in pressure over the body.

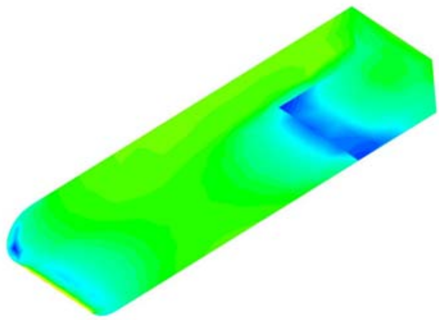

Figure 9. Pressure gradient over the body

G. Streamlines

The streamlines give an insight into the flow pattern of the wind around the body. They show various stall and flow separation conditions helping the designers realize both favorable and unfavorable flow conditions.

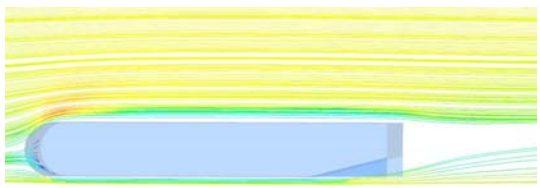

Figure 10. Streamlines around the body

IV. INFERENCE

The following conclusions were drawn from the simulations:

Decreasing the ground clearance enhances the downforce until approx. 3.5 cm from the ground. A longer diffuser also enhances the performance significantly. The angle at which the bluff body generates highest downforce is 160 degrees.

ISBN: 978-988-14048-0-0
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

At higher velocities the car tends to experience greater downforce but the drag also increases significantly.

V. Final Design

The final design was designed in Solidworks after simulation in ANSYS. The rendered view of the diffuser in Solidworks can be seen in the image below. The diffuser is the best design possible keeping in mind packaging constraints and downforce required.

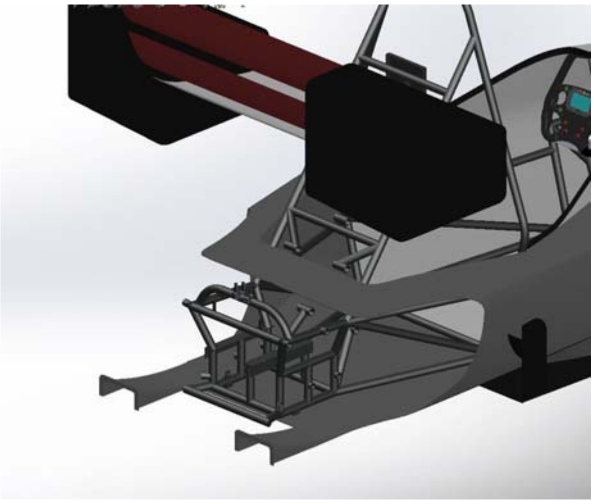

Figure 11. Final design of the diffuser

VI. Conclusion

The under tray for the Formula SAE car is designed keeping in mind the key constraints like ground height, length of the diffuser and angle of the diffuser. The aim is to optimize the diffuser for downforce while also minimizing drag. The diffuser design would help to drastically improve performance and lap time while also producing the required results in the skid pad event.

Acknowledgments

Firstly, I would like to express my heartfelt gratitude to the Director of BITS Pilani, Dubai Campus, Dr. Prof. R.N. Saha, who has ushered a new light on our college.

Also, I would like to thank the Department of Mechanical engineering, BITS Pilani Dubai campus for their motivation, support, guidance and encouragement throughout the course of this project

References

[1] Wordley, S.J., and Saunders, J.W., Aerodynamics for Formula SAE: A CFD, Wind Tunnel and On Track Study, SAE Paper 2006-01-0808, 2006
[2] J. Katz, Race Car Aerodynamics, Bentley Publishers, 1995.
[3] McBeath, S., Competition Car Downforce, Haynes Publishers, Somerset, 1998 11. Liebek, R. H., Design of Sub
[4] Ross, J.C., Storms, B.L., and Carrannanto, P.G., Lift-Enhancing Tabs on Multielement Airfoils, Journal of Aircraft, Vol 32, No. 3, pp 649-655, 1995
[5] Airfoil Coordinates Database http://airfoiltools.com/
[6] X. Zhang, J. Zerihan, Aerodynamics of a Double Element Wing in Ground Effect, AIAA Journal, Vol. 41, No. 6, pp 1007-1016, 2003.
[7] SAE, 2016 Formula SAE Rules, US Comp Edition, Society of Automotive Engineers, USA, 2004
[8] Race Car Vehicle Dynamics by William F. Milliken and Douglas L. Milliken.
[9] Smith, A.M.O., "High Lift Aerodynamics," J. Aircraft, Vol. 15, No. 9, 1978