A Project Report on

Design and Development of a Car Spoiler

Submitted by

ALKA MALIK (40124860)
AMIT GUJRATI (40057736)
ANIRUDH PATEL (40083810)
PANKAJ YADAV (40076744)
RAHUL CHUG (40075138)

in fulfillment of the course MECH 6451 for the degree of

MASTER OF ENGINEERING (MECHANICAL)

Submitted to

Prof. Francois Tardy

Department of Mechanical, Industrial and Aerospace Engineering

Concordia University

December 2019

Table of Contents:

List of Figures:

ABSTRACT

Performance, handling, safety, and comfort of a car are significantly affected by its aerodynamic properties. Getting high power directly from the engine is just not enough to judge the performance of the car. Aerodynamic properties must be considered for the purpose of studying the drag and stability of a car. The drag force is produced by relative motion between air and vehicle and about 60% of total drag is produced at the rear end. In order to improve a car’s aerodynamic drag and its stability, an aerodynamic device is needed to perform such as the rear spoiler. Rear spoiler is an aerodynamic device that functions to ‘spoil’ the unfavorable air flow across the body or in the scientific explanation, to spoil the ‘laminar’ flow of the car’s aerodynamic movement in motion. This rear placing device creates an area of high pressure to replace the usual low pressure over the trunk resulting in increasing stability of the car. The objective is to design car spoiler on CATIA V5 then analyse it using finite element analysis and investigate its aerodynamic properties using CFD software.

ACKNOWLEDGMENTS

This report was completed in November 2019 at Concordia University, Montreal, Quebec, Canada. First of all, we would like to express our deepest gratitude to Professor Francois Tardy for providing us this opportunity to work on the course project and guide us to successfully complete this project during the period of my M.Eng. study. Under his guidance, we gained invaluable knowledge and experience of designing and analysis of the CAD model of the system in CATIA.

Secondly, we would like to thank all the lab TA's of MIAE department of Concordia University, who guided and support to overcome all the problems we came across during the completion of the project.

Lastly, we would like to sincerely acknowledge to all our friends and family members to provide the support and help, motivating us to successfully complete this challenging but interesting project.

1 CHAPTER 1

1.1 INTRODUCTION

The high power generates high aerodynamic instabilities. This result high speed vehicle to lose traction with the road and generates the lift. The overall performance, handling, safety, and comfort are affected by these instabilities. Modified parts are added to the vehicle body to reduce drag and other instabilities like spoilers, front bumps etc. Aerodynamically efficient vehicle also decreases fuel consumption and make the car overall more efficient.

Spoiler is a component to increase down force for vehicle. The main function is to create laminar flow over the surface which disperse the drag over the vehicle and help increase downforce on the vehicle. Handling of cars at high speed is affected as the car loses traction with the surface and while taking a corner the car might lose contact and slip off road.

1.2 OBJECTIVE

The objective of this report is to design and development of a car spoiler:

- design the surface of a spoiler using standard airfoil equation and aerodynamic concepts
- Modeling the spoiler and mounting assembly on Catia V5
- Analysis of the part design using finite element analysis
- Investigate aerodynamic properties of the part using CFD tool

2 CHAPTER 2

2.1 LITERATURE REVIEW

2.1.1 Airfoil

Airfoil is an aerodynamic part which uses the lack of symmetry across the camber line of airfoil. This creates a pressure gradient on the suction and pressure surface of the airfoil which causes an upwards lift of the airfoil. The underlying principal is Bernoulli's theorem where the pressure energy is converted into kinetic energy giving the flow speed over the airfoil surface.

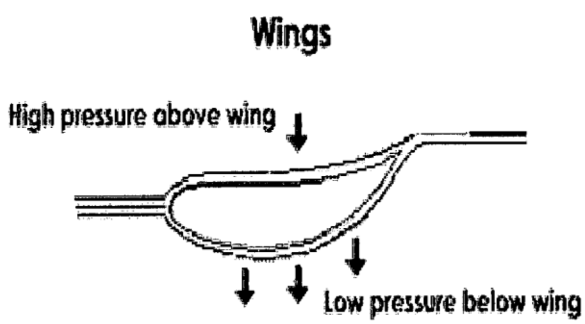

Figure 2.1: Pressure Separation on Wings

2.1.2 Concept of Spoiler

A spoiler is an airfoil with an upside-down orientation which creates a lift force in the downward direction. This lift force is referred to as down force through out this report. Spoiler also diffuses the drag over the vehicle. Helping us improve cornering ability, acceleration performance and stability in handling the vehicle. Higher the coefficient of lift higher the down force. There is always a trade off between drag and down force because as you increase the speed, the downforce increases but so does the drag. Optimisation is needed to find minimum value of drag and maximum value of down force.

2.2 SELECTION OF SPOILER PROFILE

The NACA 4-digit airfoils are widely used for aerospace and automotive industry for wing and spoiler design. Equation of the NACA profiles can be used to generate the given profile in a modelling software.

The NACA four-digit wing sections define the profile by [6]:

- First digit describes the maximum camber as percentage of the chord.
- Second digit describes the distance of maximum camber from the aerofoil leading edges in tens of percentage of the chord.
- Last two digits describe the maximum thickness of the aerofoil as percent of the chord.

NACA 0018 [8]

Table 2.1: Value of parameters for Aerofoil NACA XXXX

Angle of attack (α)	Coefficient of lift (cl.)	Coefficient of drag (cd)	Lift force (L) N/m	Drag force (D) N/m
20	1.100	0.0320	2012	72.4
15	1.363	0.0196	2465	43.5
10	1.207	0.0122	2132	36.7
5	0.625	0.0084	1987	32.4
0	0.000	0.0072	0	28.9
-5	-0.625	0.0084	-1987	32.4
-10	-1.243	0.0123	-2264	36.9
-15	-1.850	0.0196	-2899	43.5
-20	-2.439	0.0319	-3269	70.2

NACA 2412 [4]

Maximum lift force occurred at an angle of –15° and is equal to 2679.55 N/m acting down wards,

Table 2.2: Value of parameters for Aerofoil NACA XXXX

Angle of attack (α)	Coefficient of lift (cl.)	Coefficient of drag (cd)	Lift force (L) N/m	Drag force (D) N/m
20	1.409	0.0368	2511.96	65.6
15	1.495	0.0228	2665.28	40.64
10	1.309	0.0135	2333.68	24.06
5	0.857	0.0089	1527.86	15.86
0	0.263	0.0088	468.87	15.68
-5	-0.332	0.0089	-591.89	15.86
-10	-0.923	0.0128	-1647.52	22.81
-15	-1.503	0.0204	-2679.55	36.37
-20	0	0.0357	0	63.64

NACA 2412 is selected over NACA 0018 as NACA 2412 has a higher lift coefficient compared to NACA 0018. The modelling will be done based on the NACA 2412 profile as our main purpose is to obtain good downforce.

3 CHAPTER 3

3.1 DESIGN OF SPOILER

The main parameters while designing spoiler is the smoothness of the profile and downforce exerted by the profile. Angle of attack plays an important role in the performance of the spoiler.

NACA 2412 IS SELECTED

Considering the parameters, 2 designed has been created on CATIA V5 design application:

1. Design Intent 1: Using Cross-Section Solid
2. Design Intent 2: Using Surfaces

Following predefined parameters were considered while designing the spoiler and further analysis.

- Air velocity = 250-350 km/h
- Angle of attack for the spoiler = 15 degree

(The AOA choice was made based on the fact that this angle resulted in max downforce)

- Width of the car = 1910 to 1930 mm (Designed for Mercedes Benz S-Class)

3.1.1 Design Intent 1:

First approach for creating a design was making a cross-section solid throughout for the span of the spoiler, considering the following dimensions of the spoiler.

- Airfoil-1 NACA 2412: Chord length= 250mm, Thickness=65 %, radius= 6000 mm
- Span of spoiler = 1500 mm
- Width of spoiler = 250 mm

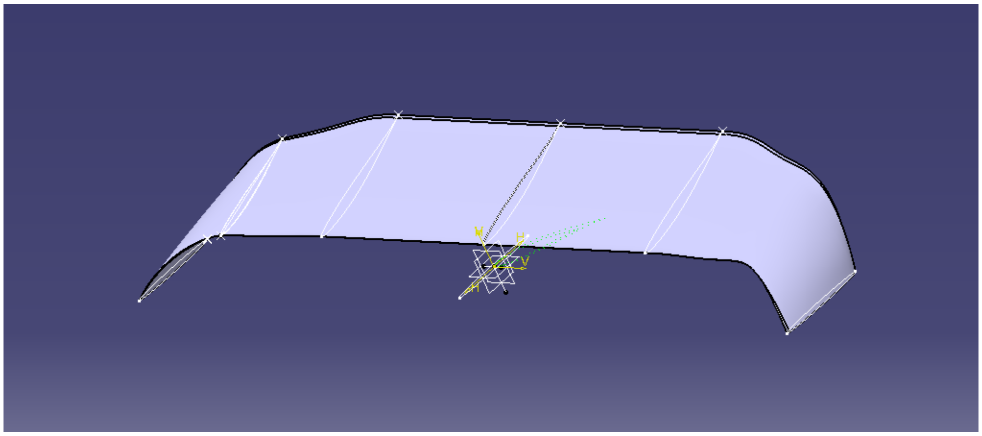

Figure 3.1: Design of the Spoiler using Cross-Sectional Solid

Fig 3.1 shows the design of the spoiler using cross-sectional solid. The design shows the irregularity in the smoothness on the surfaces of the spoiler. Also, the curves and the edges of the spoiler were not very smooth.

Hence for the better-quality smoothness of the profile Design Intent 2 has been created that resolves the problem of Design Intent 1.

3.1.2 Design Intent 2:

Second approach for the designing the spoiler has been done by surface creation. Considering the fact that spoiler has 3 faces, the design is been carried out by creating 3 surfaces (top, bottom and edge surface). And finally, the solid is being created joining all the 3 surfaces in order to get the smooth profile along the surfaces and the edges of the spoiler as shown in the figure below.

Following dimensions were considered while designing the spoiler:

- Airfoil-2 NACA 2412: Chord length= 200mm, Thickness=65 %, radius= 6000 mm
- Span of spoiler = 1330 mm
- Width of spoiler = 250 mm

Figure 3.2, shows the designing of the top surface of the spoiler using surface technique. Thereafter, the solid of the top surface is being generated.

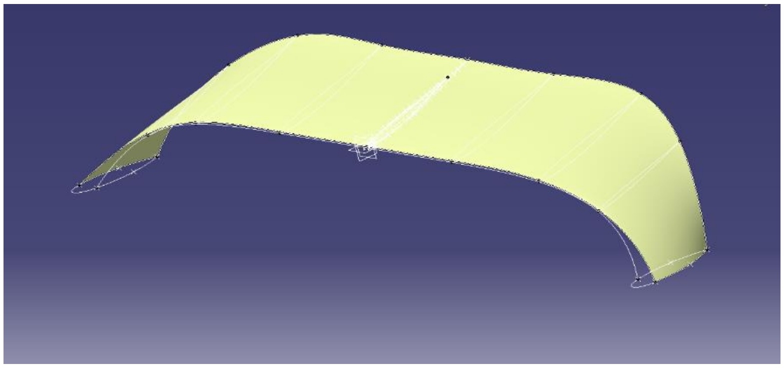

Figure 3.2: Design of the top surface of the spoiler using Surface technique

Figure 3.3, shows the designing of the bottom surface of the spoiler and further the solid of the bottom surface is being created.

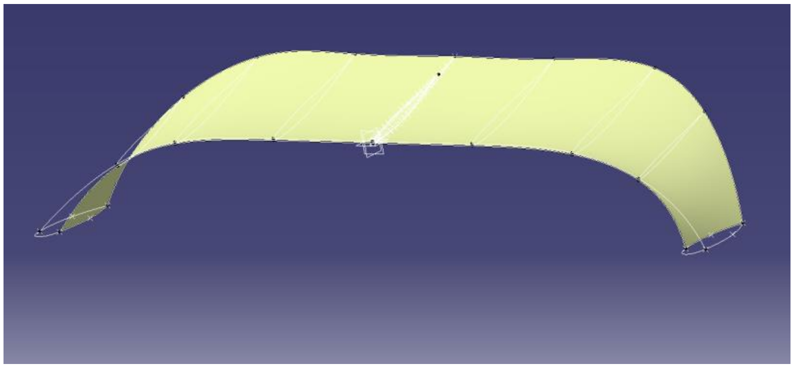

Figure 3.3: Design of the BOTTOM surface of the spoiler using Surface technique

Figure 3.4, shows the designing of the final edge surface of the spoiler and the solid for the final surface is being generated.

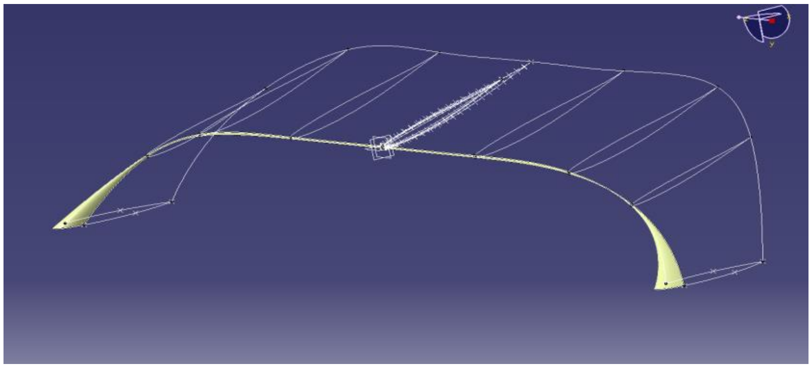

Figure 3.4: Design of the EDGE surface of the spoiler using Surface technique

Finally, after designing and generating the solids for all the three surfaces, final model of the spoiler is created by joining all the three surface solids.

4 CHAPTER 4

4.1 METHODOLOGY

After designing the CAD model of the spoiler on CATIA, the model was imported to ANSYS for a CFD analysis.

Following steps were carried out to setup the model on ANSYS.

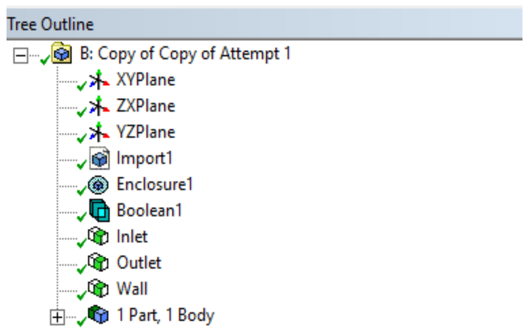

Figure 4.1: Tree Outline for Setup of the model on AYSYS

i. Import of the CAD model to IGS
ii. Enclosure was created and Boolean operation applied to subtract the model from the enclosure to analyze the liquid flow.
iii.

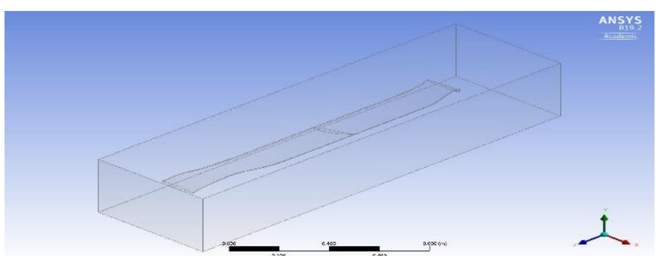

Figure 4.2: Enclosure design around the model

iv. Inlet – Outlet – Wall was defined for the enclosure as per our model and assumption.

v. Meshing was created, which divides the surface of the spoiler into large no of small meshes/elements. This will give proper analysis for each elements/meshes created and finally iterates to give the best possible results.

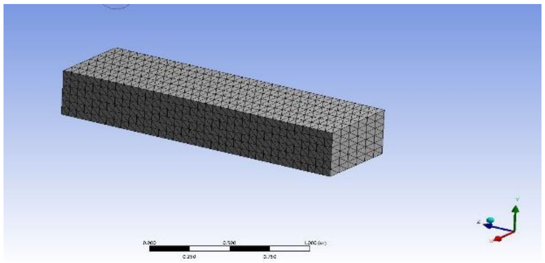

Figure 4.3: Meshing around the enclosure

vi. The mesh was set up in the solutions with pre-decided boundary conditions. The flow was taken as Laminar and the iterations were given as 1000. The solution converged at about 500. The convergence criteria were the reduction of rms values to 1e-3 or a straight line in the residuals.

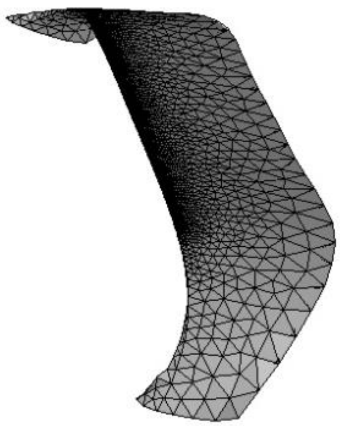

Figure 4.4: Meshing on the surface of the spoiler

After setting all the parameter definition and control inputs the model is sent for CFD analysis to determine and analyse the following variables.

- Drag Coefficient C_D
- Lift Coefficient C_L (Downward Force)
- Pressure Separation on the Aerofoil
- Velocity Separation

4.2 RESULTS AND DISCUSSION

Analysis for Design Intent 1: With boundary conditions and parameters the analysis of the variables is carried out. Following plots were plotted for the calculation of C_L and C_D

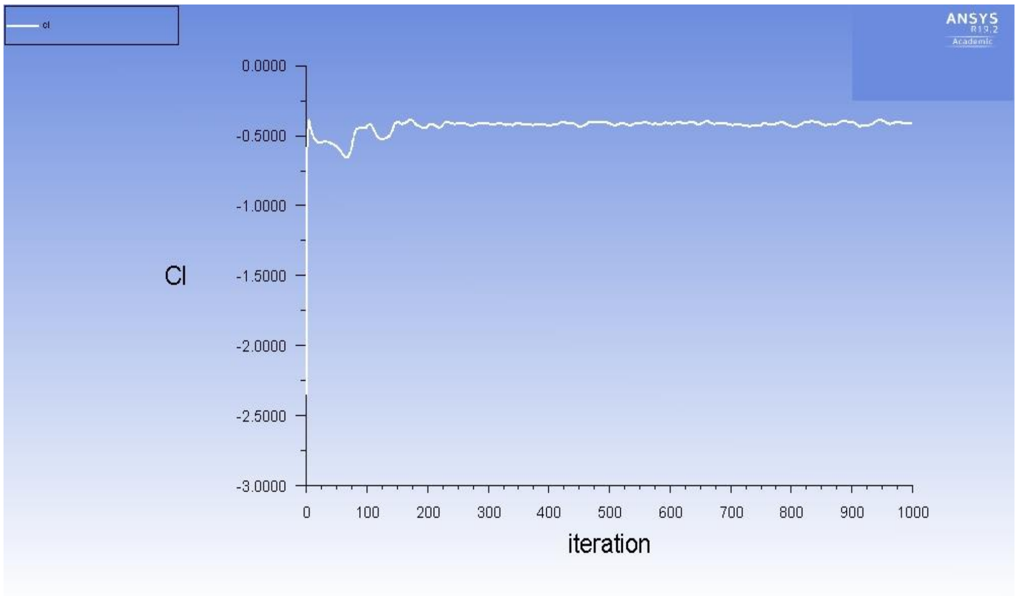

Figure 4.5: Graph of Coefficient of Lift (CL) VS Iteration for Design Intent 1

Figure 4.4 shows the graph for calculation of coefficient of lift: C_L: -0.41796356 after 1000 iteration. The coefficient of the lift was found to be a negative value as shown above which implements that the force/lift is generated downward direction instead of upward.

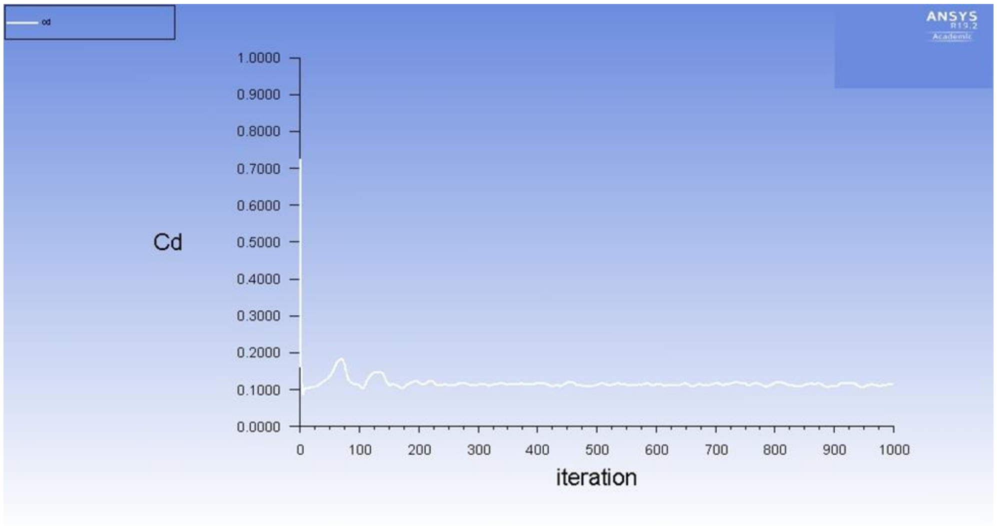

Figure 4.6: Graph of Coefficient of Drag (C_D) VS Iteration for Design Intent 1

Figure 4.5 shows the graph for calculation of coefficient of Drag given as C_D: 0.082355215.

From the graph and value, it can be implemented that final drag for the system is minimum and optimal.

Finally, from the value generation of C_L and C_D, it is considered that the system generates enough downward force and minimum drag. Furthermore, the system is feed for pressure and velocity analysis.

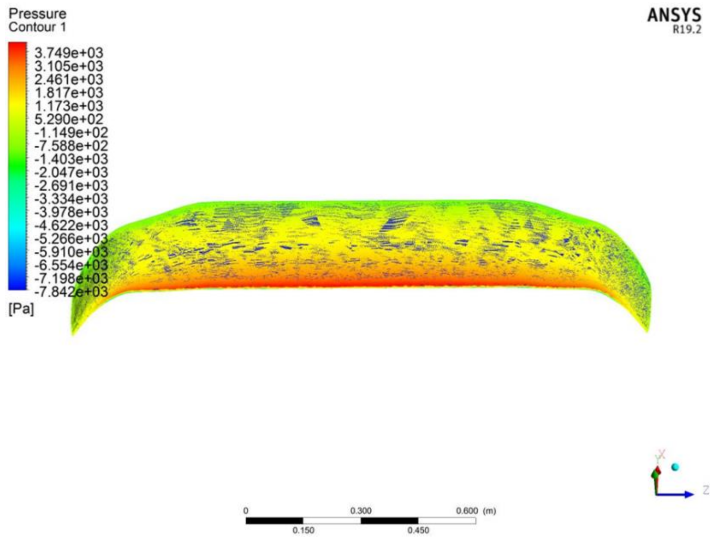

Figure 4.7: Pressure Analysis on the Surface of the Spoiler

Figure 4.6 shows the pressure on the different surface of the spoiler. The pressure at the leading edge of the spoiler were found to be maximum and the minimum at the further end as shown in the figure above. Also, the pressure on the top surface were found to higher as compared to the lower surface. Hence it can be implied that there is pressure gradient that generates the downforce.

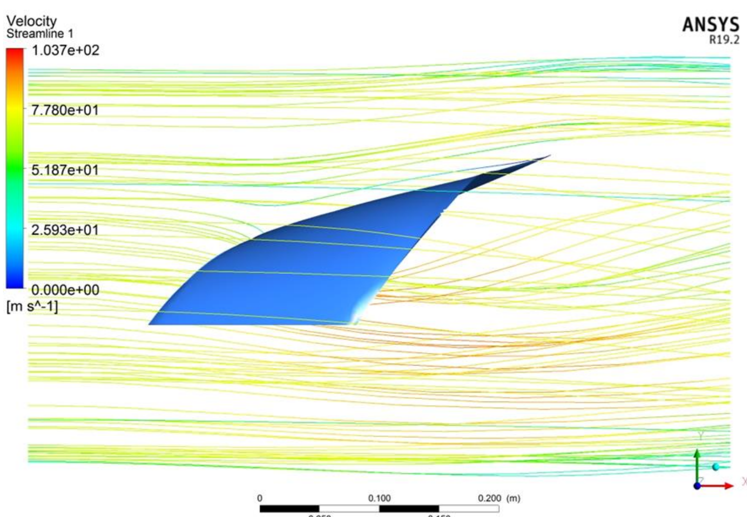

Figure 4.8: Velocity flow over the Surface of the Spoiler for Design Intent 1

Figure 4.7 shows analysis of velocity flow over the surface of the spoiler. The velocity at the lower end of the spoiler were found to greater as compared to the other surfaces of the spoiler. Hence, this confirms that pressure at the lower end of the spoiler has less pressure.

Analysis for Design Intent 2: With boundary conditions and parameters the analysis for the design intent 2 were carried out. Following plots were plotted for the calculation of C_L and C_D.

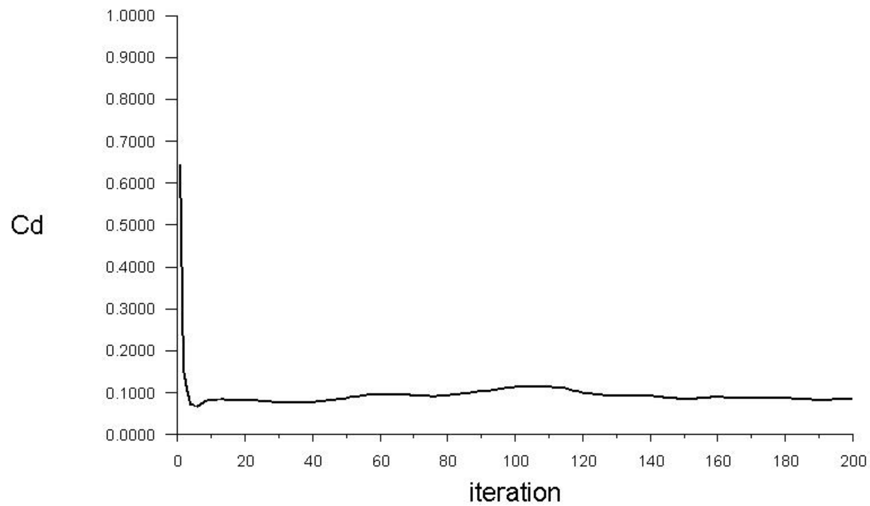

Figure 4.9: Graph of Coefficient of Drag (CD) VS Iteration for Design Intent 2

Using the similar approach as above, the calculation of coefficient of drag C_D were calculated and found as $C_D = 0.076652451$

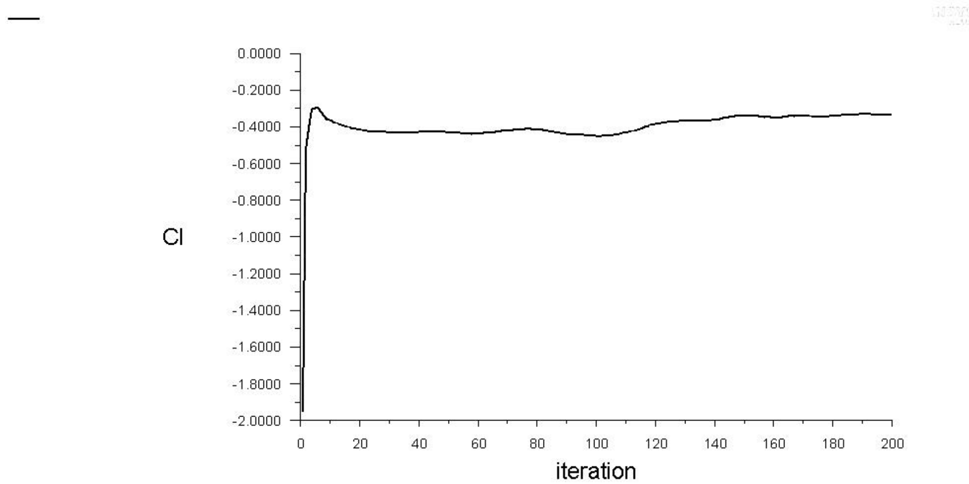

Figure 5: Graph of Coefficient of Lift (CL) VS Iteration for Design Intent 2

Coefficient of the lift for design intent 2 were calculated and found as Cl = -0.28152024 as shown in Figure 5.

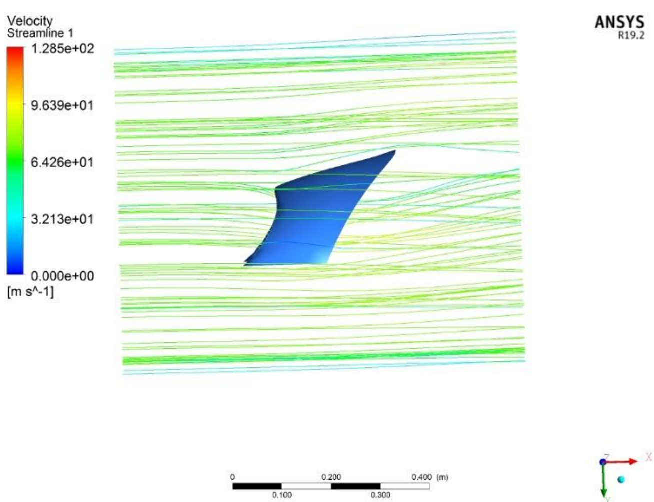

Figure 5.1: Velocity flow over the Surface of the Spoiler for Design Intent 2

Figure 5.1 shows the velocity flow over the surface of the Spoiler design 2. It can be seen that velocity at the lower end are high as compared to the upper surface.

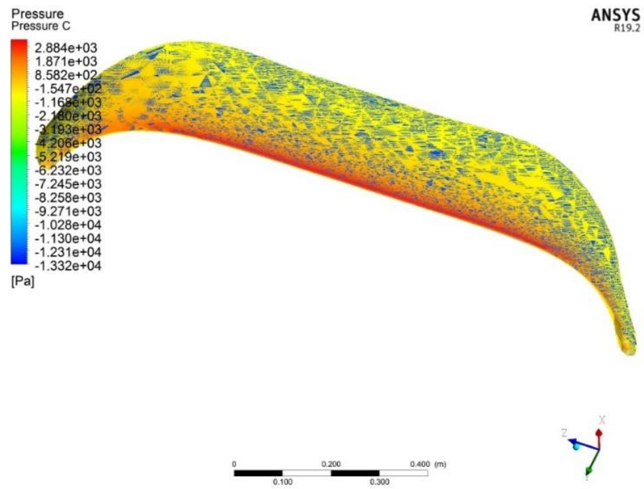

Figure 5.2: Pressure Analysis on the Surface of the Spoiler for Design Intent 2

Figure 5.2 shows the pressure analysis on the surface of spoiler for design intent 2. The pressure can be seen maximum at the leading edge of the spoiler and minimum at the back end. Also, the pressure at the top surface were found to be greater as compared to the bottom surface.

Hence it can be implied that due to pressure gradient generated on the surface of spoiler, the downward force will be generated.

The comparison of Design Intent 1 and 2 were discussed in the Conclusion section below.

4.3 CONCLUSION

The two designs for the spoiler were designed in CATIA and analysed in ANSYS using the predefined parameters. The design intent 1 will generate optimum values of coefficient of drag and coefficient of lift but also it has very high amount of pressure at leading edge of the spoiler. Also, the design contains irregularities on the surfaces. Hence in-order to improve the design, design intent 2 was created.

The new design also generates optimum values of coefficient of drag and coefficient of lift. The lower value of coefficient of drag for second design depicts the proof of smoother surface of the spoiler. The surface was made too smooth which caused the flow to flow over the spoiler with ease and the pressure gradient is not enough to cause the desired downforce resulting in the low value of C_L. The both designs generates satisfactory results and can be used according to the vehicle specific application.

For the future prospects the design can be further improved to get relatively lower value of drag with a higher amount of lift. Hence, Design Intent-1 can be chosen when, only the performance is taken into consideration while the Design Intent-2 can be chosen when performance as well as the look of the spoiler is taken into consideration.

REFERENCES

[1] H. Isa, "DESIGN & FABRICATION OF REAR CAR SPOILER FOR PROTON WIRA MODEL," Universiti Malaysia PAHANG, 2007.

[2] V. Kumar, D. Narayan, L. Rao and Y. Ram, "Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler Using CFD," *IJSR,* vol. 4, no. 9, pp. 1-8, 2015.

[3] S. Kajiwara, "Passive variable rear-wing aerodynamics of an open-wheel racing car," *Automot. Engine Technol.,* vol. 2, pp. 107-117, 2017.

[4] K. Chodagudi and T. Rao, "Analysis Of Naca 2412 For Automobile Rear Spoiler Using Composite Material," *IJERT,* vol. 1, no. 7, 2012.

[5] T. Ipilakyaa, L. Tuleun and M. Kekung, "COMPUTATIONAL FLUID DYNAMICS MODELLING OF AN AERODYNAMIC REAR SPOILER ON CARS," *Nigerian Journal of Technology (NIJOTECH),* vol. 37, no. 4, pp. 975-980, 2018.

[6] M. Kumar, C. Naiju and H. Patruni, "Experimental Study and CFD Analysis of an Aerofoil Structure for Automotive Body Design," *SAE International,* pp. 1-6, 2018.

[7] V. Kumar, V. Tomar, N. Kumar and S. Jain, "Flow Simulation and Theoretical Investigation on Aerodynamics of NACA-2415 Aerofoil at Low Reynolds Number," *SAE International*, pp. 1-8, 2015.

[8] G. Kumar, G. Reddy and K. Shekar, "Dynamic Response of NACA 0018 for Car Spoiler using CFRP Material," *International Journal of Current Engineering and Technology,* vol. 4, no. 3, pp. 1-5, 2014.

[9] V. Kumar, K. Narayan, L. Rao and Y. Ram, "Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler Using CFD," *International Journal of Science and Research (IJSR),* vol. 4, no. 9, pp. 1-8, 2015.