

Design and Analysis of an F1 Car

-Rahul Koshy

Formula 1 (F1) is sometimes referred to as the pinnacle of motorsport. The entire process, from production to crossing the chequered flag, requires the best of the best to extract every bit of performance from these machines. In my attempt to further my knowledge in this industry, I have designed the rear half of an F1 car on CATIA V5 and performed an aerodynamic analysis of my design on ANSYS FLUENT. In this report, I will summarize the process behind the design and explain the results from the analysis.

Design Process

The figure below shows the reference model used for the project [1].

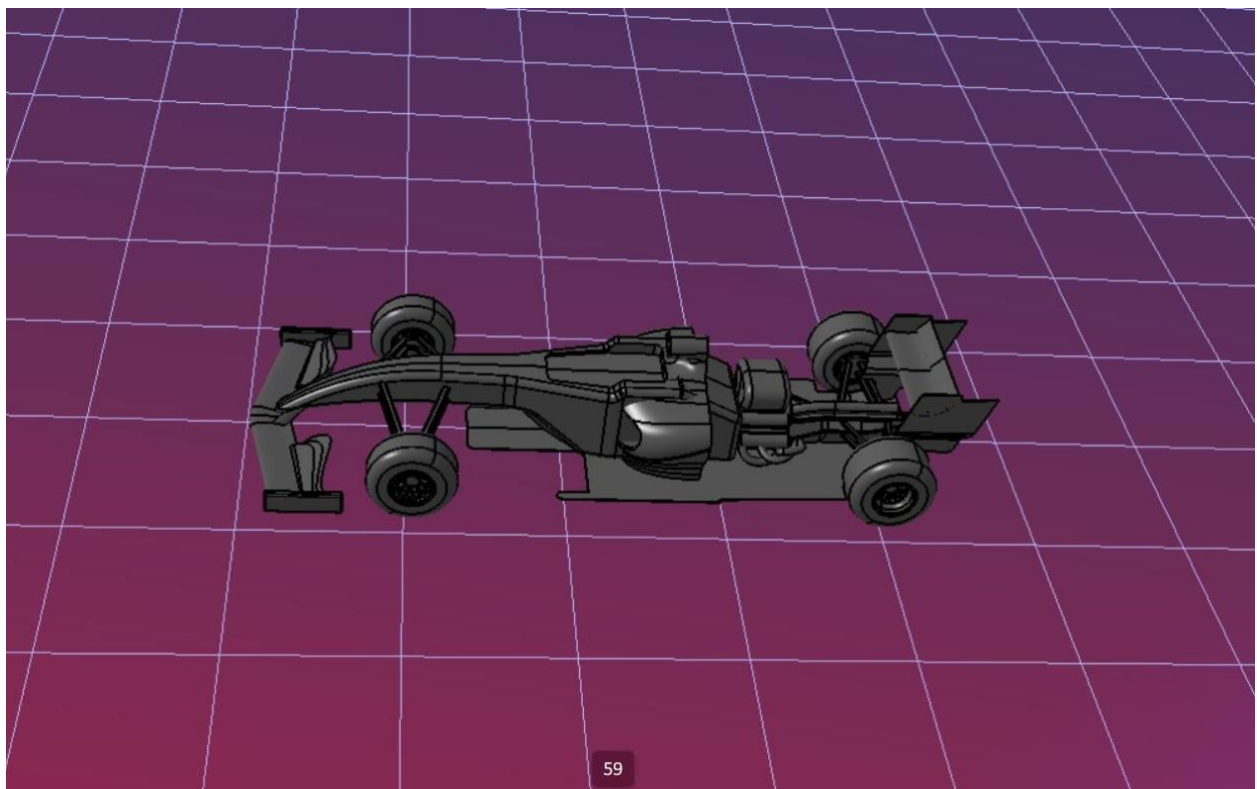


Figure 1: Reference model.

As shown in the figure above, the reference model is missing a sidepod, air intake, engine cover, and a shark fin. Each of these components play a significant role in providing cooling to the

different components of the engine while also providing the necessary downforce required. Downforce is defined as the aerodynamic force that pushes the car down onto the track, enhancing cornering speed and thus decreasing lap times.

Sidepod

Sidepods are a vital component of an F1 car, providing crucial benefits in both heat management and controlling air flow. Well generated sidepods allow the F1 car to experience less aerodynamic drag, which increases the car's overall top speed. The figures below show the sidepods that have been made for the project.

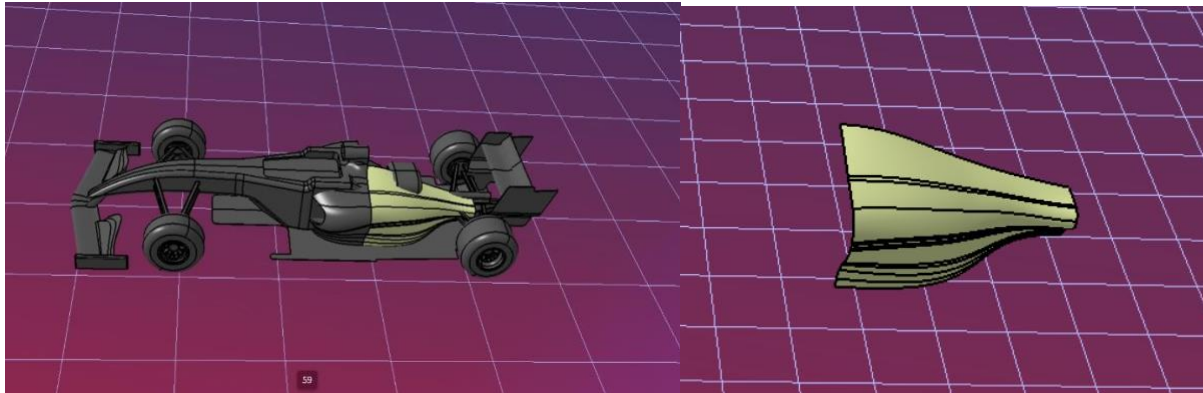


Figure 2,3: Sidepod design.

Figure 2 shows the sidepod intersecting the engine. However, this will be fixed once the other components have been made. The important thing to note on the sidepod construction is the smooth contour lines flowing from the middle of the car to the rear. As previously mentioned, this smooth transition from middle to rear decreases aerodynamic drag. The design was made by using 4 different curves on various lengths of the car. These curves were then connected to form the figures shown above.

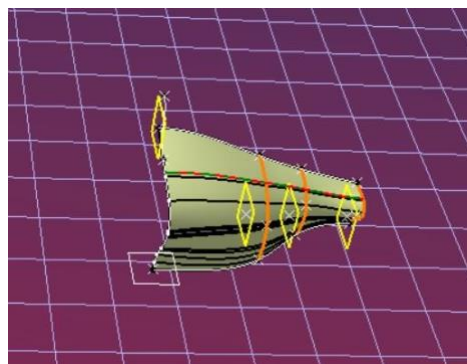


Figure 4: Sidepod construction.

Air Intake

The next step of the process was designing the air intake. As the name implies, the air intake is used to supply air to the internal combustion engine of the car. Designing an air intake for reduced drag is a component of the design process, but the main use is to supply air to the engine through the ram air effect. The ram air effect is defined as the increased volume of air that flows through the air intake when the F1 car is moving at a high speed.

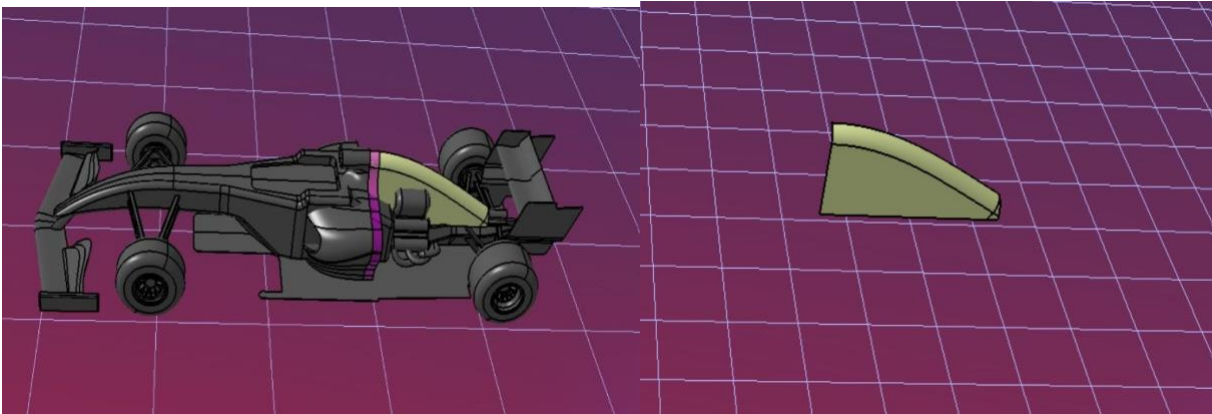


Figure 5,6: Air intake design.

The figures above show the construction for the air intake. The design was made by connecting points tangent to a plane as shown in the figure below.

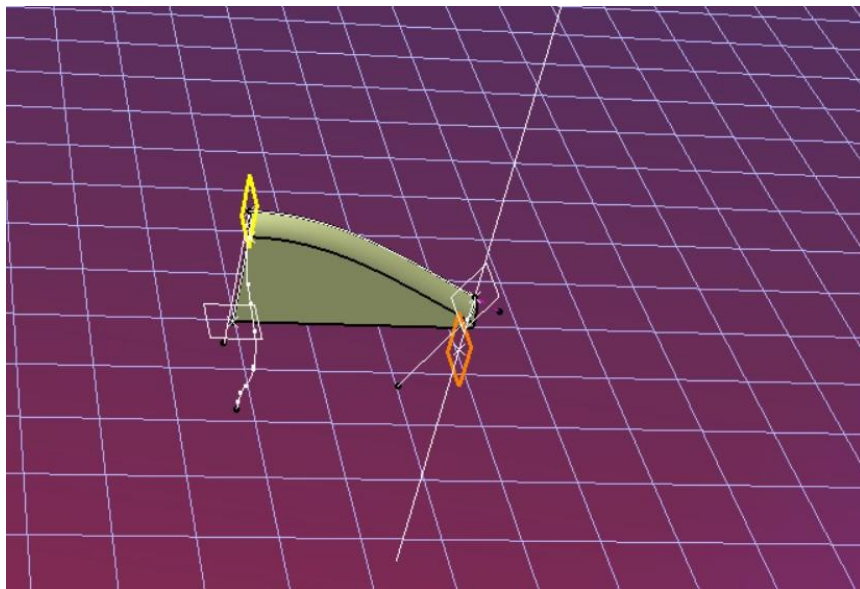


Figure 7: Air intake construction.

Engine Cover

The engine cover provides a housing for the engine to protect it from damage and shield it against debris. This part of the design also fixes the problems with the intersections between the components and the engine.

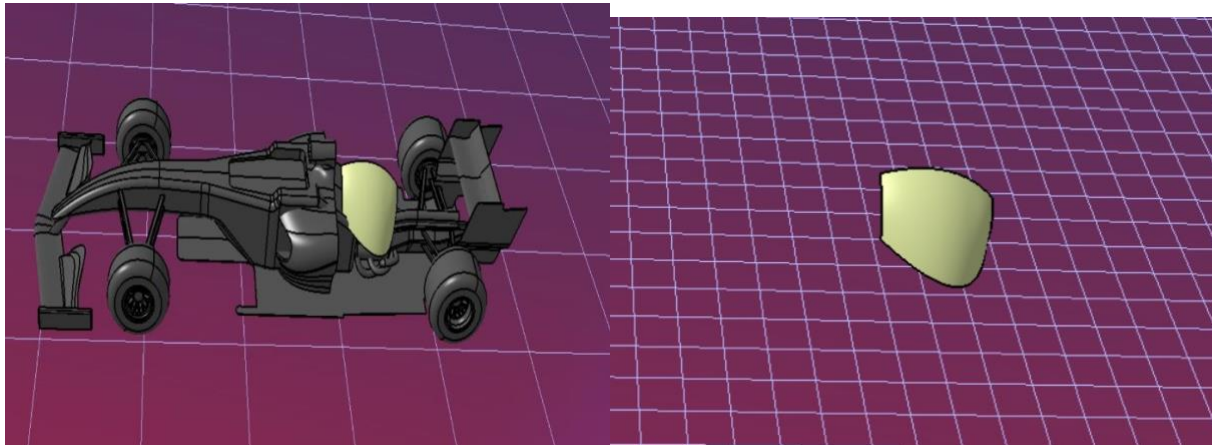


Figure 7,8: Engine cover design.

While many different engine designs can be chosen, the one shown depicts a simple housing right on the surface of the engine. A similar process to the sidepod design was taken for the engine cover.

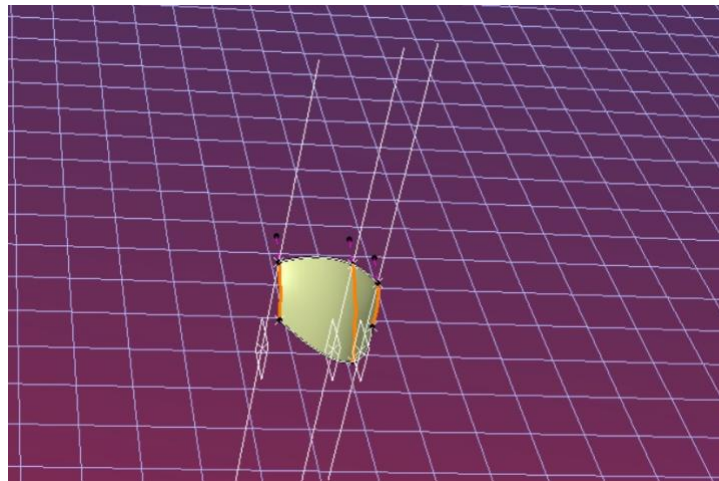


Figure 9: Engine cover construction.

Shark Fin

The last component that was constructed was the shark fin. While these components were removed after the 2017 season due to safety concerns and other factors, I decided to include it in this project to get experience with creating different components. These components were made to control the airflow on the top of the car.

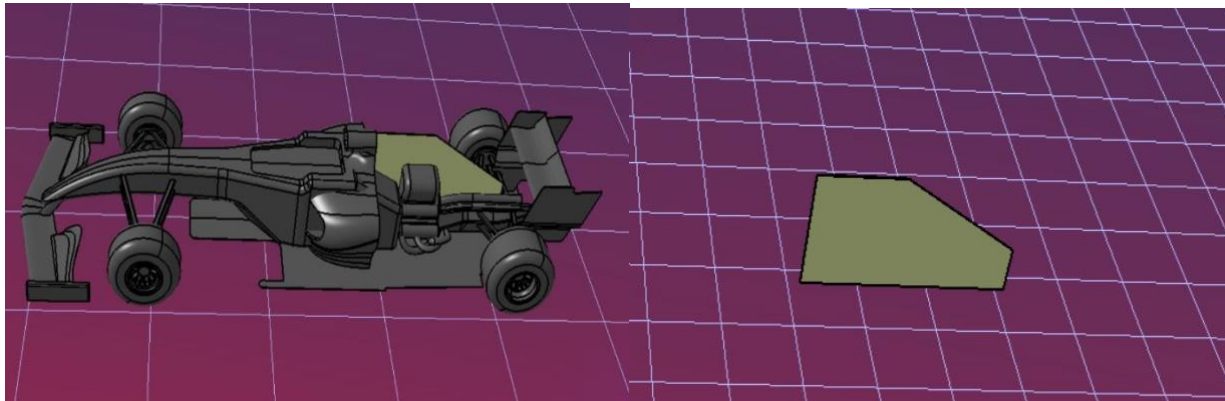


Figure 10,11: Shark fin design.

The construction of the shark fin was one of the more difficult components to design. A series of intersections were made to construct the wireframe. This was then positioned by creating a reference plane on the surface of the bodywork.

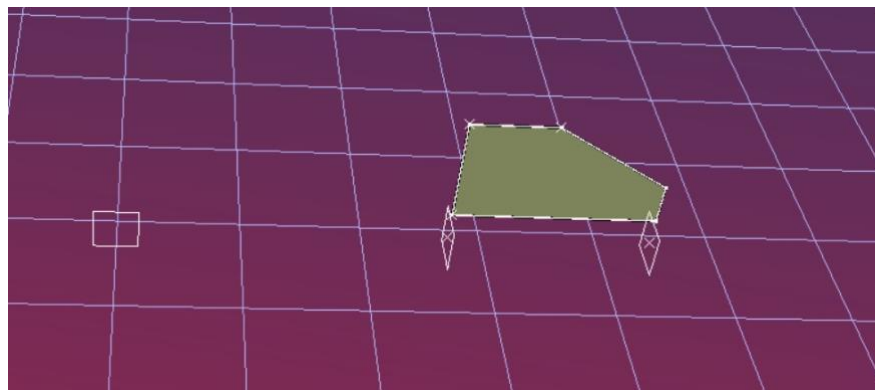


Figure 12: Shark fin construction.

Trim File

The final step of the design process is to combine all the components that have been created to make the final bodywork. This side will also be mirrored on the other side to complete the bodywork of the car.

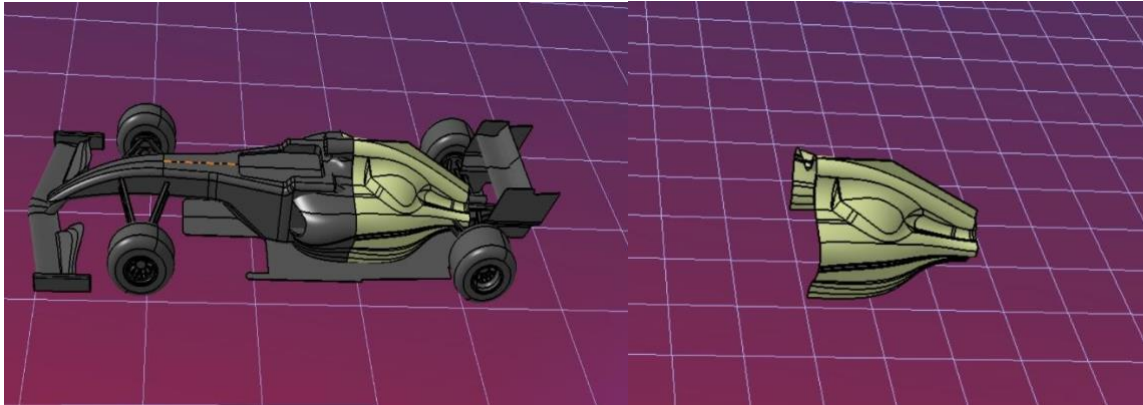


Figure 13,14: Trim file design.

The construction of the trim file included joining the different components that were created together while using edge fillets to smooth out the bodywork to make one final piece.

Design Changes

Organization is very important when creating these files. By having a part linked to a geometric set in CATIA, it allows the design to be changed and refined at any time. It also makes it easier for other colleagues to understand the design process. The figure below shows the organization used for this project.



Figure 15: File organization.

It is important to note that every main part would have a subsection similar to that of the trim file shown above. Each part would have an input, construction, and output folder.

With the design of the bodywork complete, it can be imported to ANSYS for a CFD analysis.

CFD Analysis

ANSYS FLUENT was used for the CFD analysis of this project. Computational Fluid Dynamics (CFD) is used for aerodynamic analysis. It is a simulation technique used to simulate the airflow around the car. The program works by forming a mesh around the input geometry and then using the equations of fluid dynamics such as the Navier Stokes equation on each cells of the mesh to provide a result for how the airflow works around the model. Finer meshes would provide more accurate results but would require more computational power.

There were a few factors that hindered the accuracy of this project. Since the reference model was not split into separate parts, different components were not able to be defined properly. One example of this would be the tyres, which were not able to be set as a rotating surface. This would result in the program treating the tyres as a stationary wall. Another factor that has to be considered is that the reference model was not watertight. A watertight geometry is a geometry that has no holes or leaks resulting in a more accurate flow simulation. Fortunately, ANSYS has a feature for meshing geometries that are not watertight. It is an important factor to note when analyzing the results.

The figure below shows the mesh that was created for the geometry.

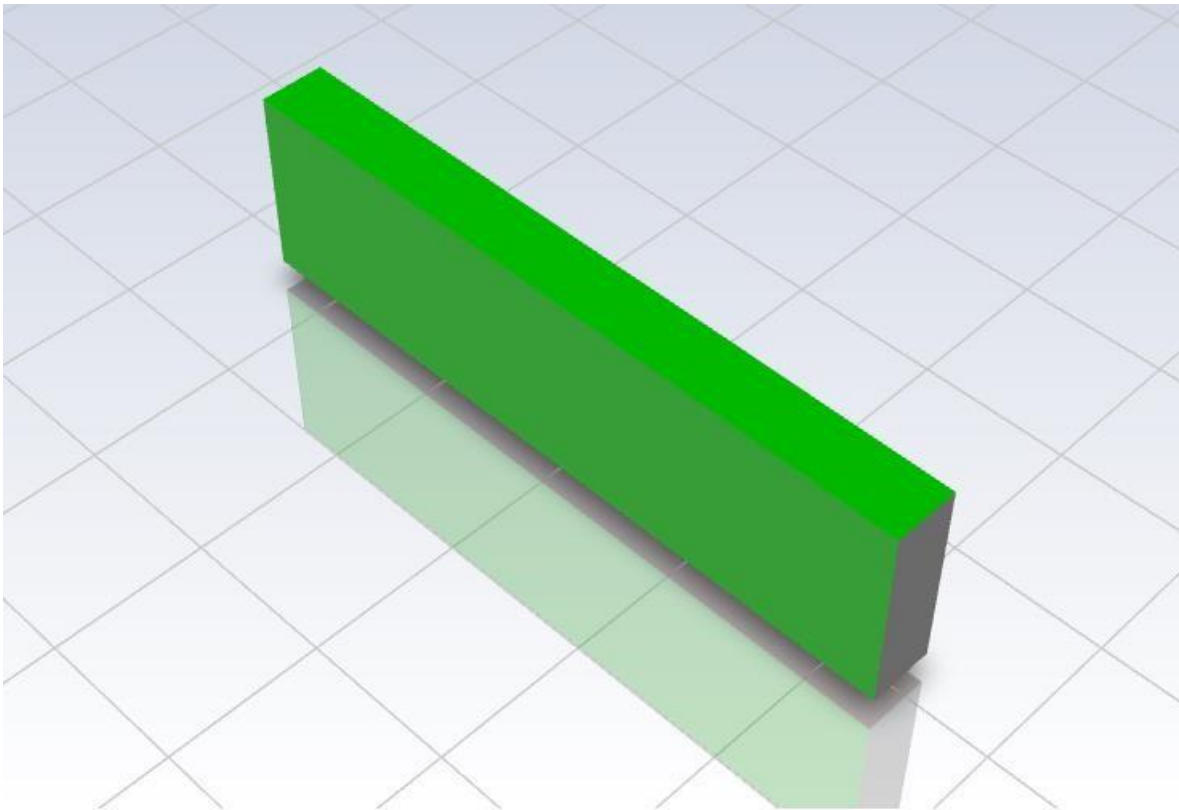


Figure 16: Generated mesh

After applying the conditions set, the block mesh reciprocates the real-life model of a wind tunnel. The air flow would be directly towards the car, much like that of a real wind tunnel.

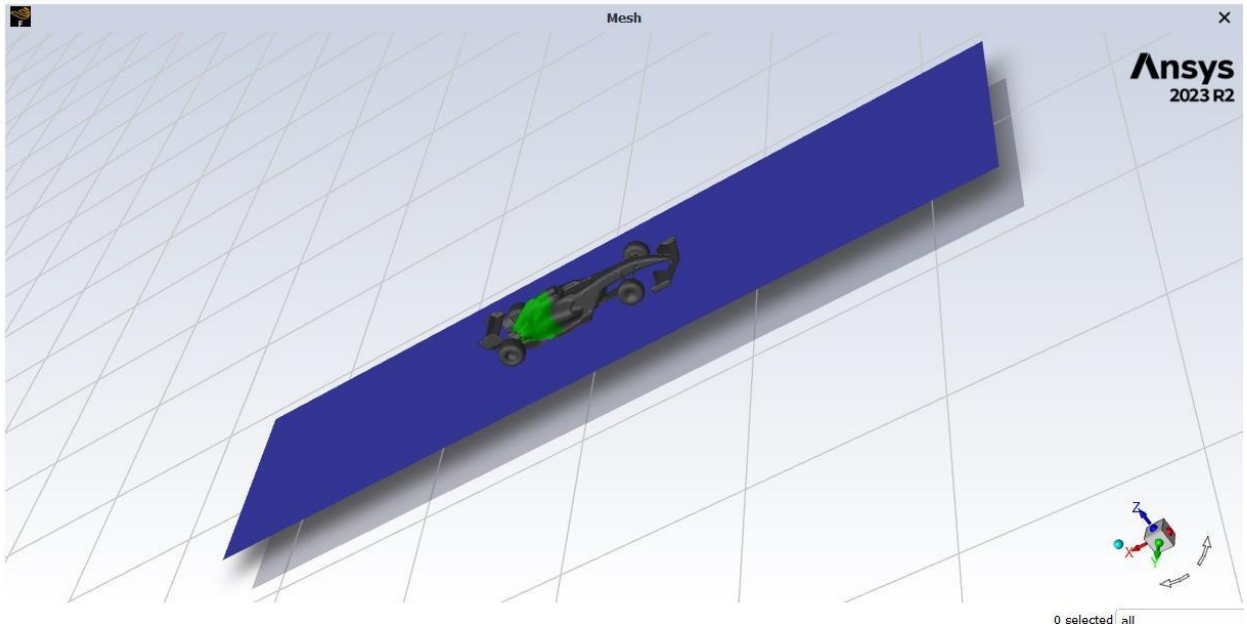


Figure 17: Generated mesh without showing the three walls.

With the mesh generated, the initial conditions such as the freestream velocity speed and cross-sectional area were input, and an analysis was done. The figures below show the drag coefficient and lift coefficient from the analysis done with a freestream velocity of 100 m/s.

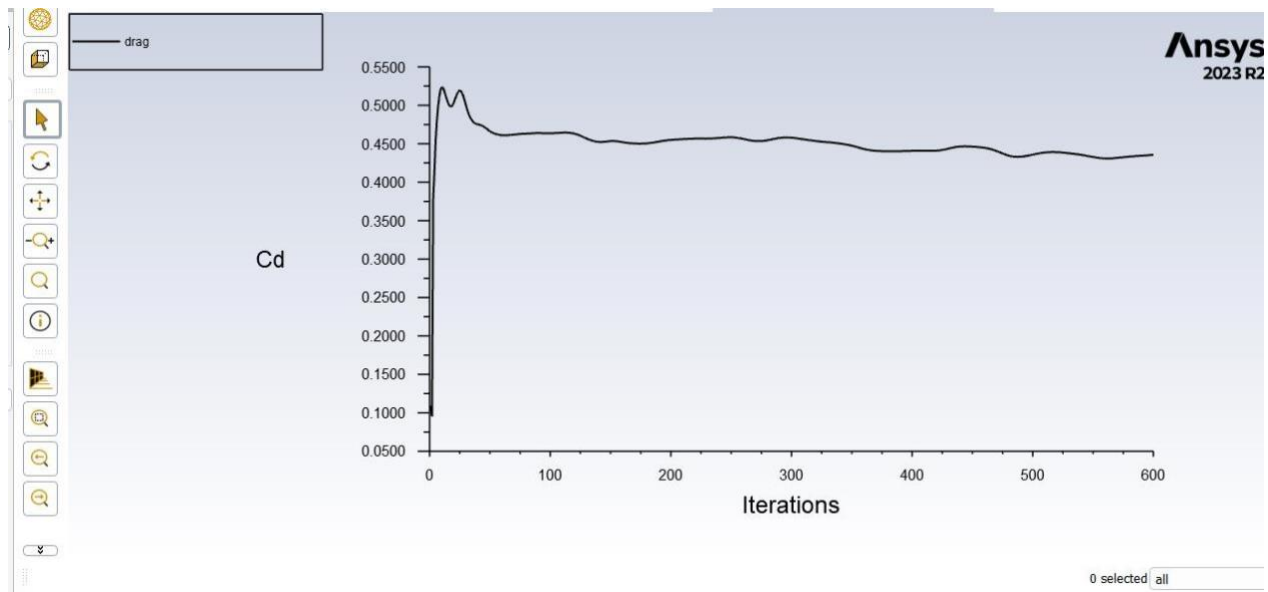


Figure 18: Drag coefficients.

As shown by the figure above, the drag coefficients seem to fluctuate and then stabilize at around 0.45. It is important to note that there would be an increased fluctuation if the different

components were set instead of it all being one body. This would also result in the drag coefficients increasing by a slight amount. The figure below shows the lift coefficients for the analysis. However, these results are not too far away from the expected value which is around 0.7 to 1.0 [2].

The figure below shows the lift coefficient for the model.

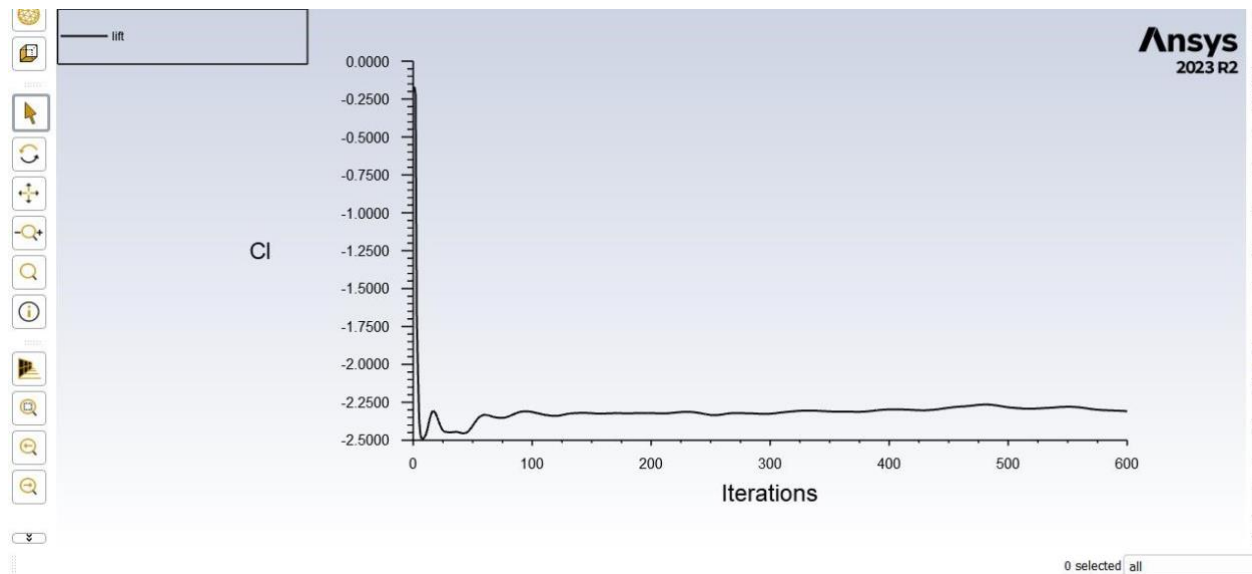


Figure 19: Lift coefficients.

The same phenomenon can be seen for the lift coefficients as the graph seems to fluctuate and then stabilize at a value of -2.25. The coefficient is negative as this represents downward lift or downforce. The downforce is higher than a typical F1 car which ranges at around -1.0 to -1.5, but these discrepancies can be attributed to the same factors that were due for the drag coefficients.

In theory, the setup of this car would be pointed towards a high downforce setup. This would provide the car with higher cornering speeds, which would be beneficial in a track where cornering speed dominates such as Monaco.

The final part of the project was to experiment with some of the post-processing features that ANSYS has to offer. The figure below shows the contour plot of the airflow around the floor of the model.

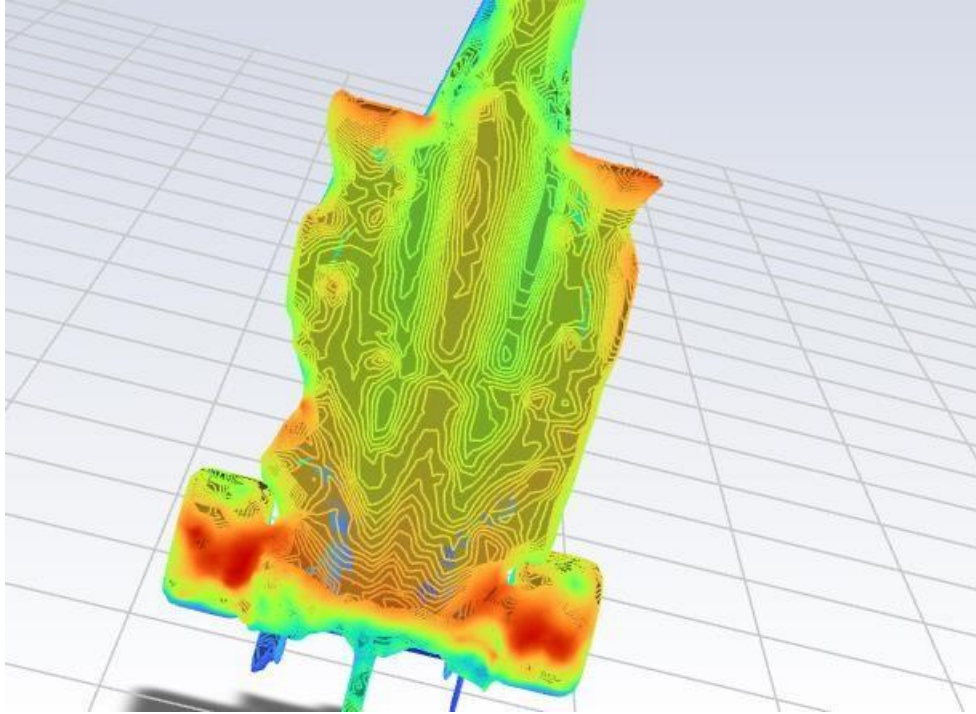


Figure 20: Airflow contour plot.

Some additions to the overall floor design of the reference model I would include would be the addition of floor skirts to the front end of the floor to guide the air and help with vortex management on the side of the car. This would help the air flow through to the back of the car to help with cooling the diffuser.

Future Additions

This project helped me understand F1 design and aerodynamics a lot more. However, there is still a lot more that is there to learn. In the future, I would want to experiment with different designs such as a shrink wrap design and compare the CFD results to the current bodywork.

References

- [1] "F1 Aero Design & Bodywork," Udemy [Online]. Available:
<https://www.udemy.com/course/f1-aero-design-bodywork/> Accessed: August 12,2023
- [2] "Title of Forum Topic," F1Technical.net Forum. [Online]. Available:
<https://www.f1technical.net/forum/viewtopic.php?t=1861>. Accessed: August 12, 2023.