

Formula 1 Front Wing
Modeling in CATIA and Aerodynamic Study
in ANSYS Workbench



By

Abhishek Rathod

Table of Contents

| | |
|--|-----------|
| INTRODUCTION | 3 |
| BACKGROUND | 3 |
| MODELING PROCEDURE | 4 |
| CFD PROCEDURE | 8 |
| RESULTS | 9 |
| LESSONS LEARNED | 12 |
| ORGANIZATION IN GSD | 12 |
| COMBINE FEATURE TUTORIAL | 13 |
| BLEND TOOL | 15 |
| CONCLUSION AND DISCUSSION | 17 |
| APPENDIX AND REFERENCES | 17 |

Introduction

This paper presents the modeling and aerodynamic study of two Formula one front wing configurations using CATIA and the ANSYS workbench. The first configuration is a simple front wing only consisting of the base and endplates. The second is a high downforce configuration consisting of additional wing elements.

CATIA has for a long time been the choice of software for professionals when it comes to surface modeling. I have used the Generative shape design workbench to model the front wing. In the sections below I will briefly discuss my modeling process, the source of dimensions and the results aerodynamic study. I will also include the lessons I have learnt, and things I would change if I were to do this again. I believe the information will be useful to future students who want to explore this powerful workbench further.

Background

On a top speed straight an F1 car attains enough speed to take off, if not for the downforce that holds it down. A formula one car can generate enough downforce to comfortably drive upside down in a tunnel. While most of this downforce comes from the main body, 15-20% is generated by the front wing.

The front wing also happens to be the engineer's playground. This is because, compared to the design for the rest of the car the engineers are less restricted in the design of the front wing. This gives team with better engineers the edge.

Another reason the front wing is of high interest is that every track requires a slightly differently configured front wing. Tracks with long straights and fewer corners require less drag to achieve higher top speeds, and tracks with more corners require more downforce for extra grip around the bends. So the front wing is in constant development throughout the year. For example, for tracks like Monza (Italian GP) which have long straights teams use flat wings to gain highest possible speeds, and for tracks like Monaco which have narrow corners, teams use steeper wing elements.

Here is a labeled image of a generic front wing based on 2017 regulations:

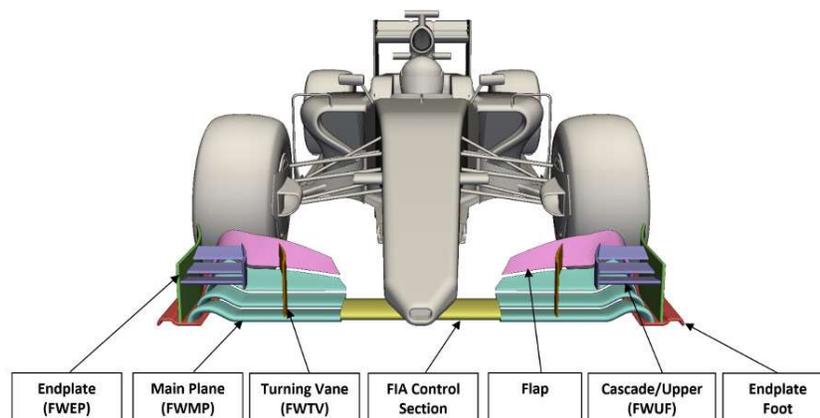


Figure 1 labeled front wing

Here are some actual wings based on 2019 regulations:

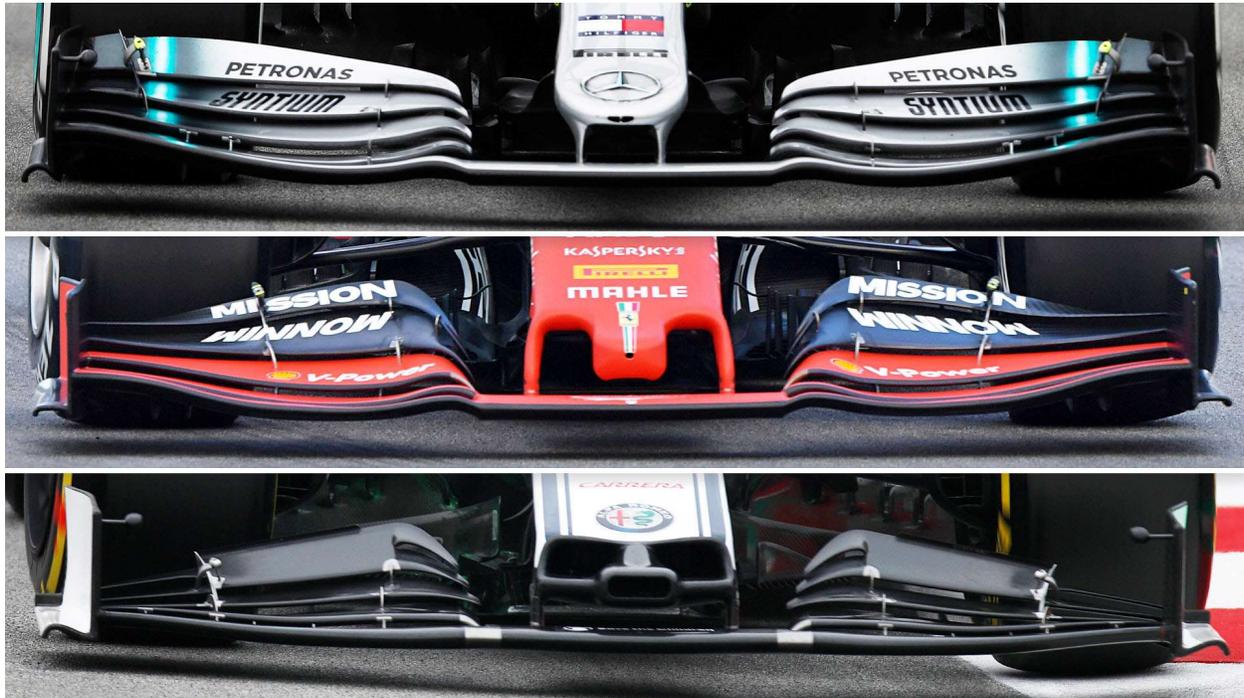


Figure 2 2019 wing examples

Modeling Procedure

My very initial plan was to use the sketch tracer and extrude those into surfaces. However, given the complex nature of the wing's curves, I chose to model one from scratch in the generative shape design workbench. I am glad I chose this option as I grew to appreciate the functionality of the workbench. It is second to none. There is a learning curve though, and I have highlighted some important lessons in a different section.

Modeling from scratch brought its own challenges. Firstly, F1 is a sport built around regulations. This means that as long as teams stay within given restrictions in their design, they are free to do what they want. This also means a lot of secrecy regarding the designs. However, I came across a few articles by an artist who, while drawing these wings, came across similar problems and so had multiple dimensions laid out in an accessible manner. These were limiting because they were only regulations and not enough for a technical model. For more dimensions, I found the F1 regulations document which had more guidelines for design. These two sources combined with google image search meant I had enough to get started.

The front wing is modeled based on the technical guidelines for the 2019 season. Some of the important dimensions are shown below, and I have provided links to the main documents.

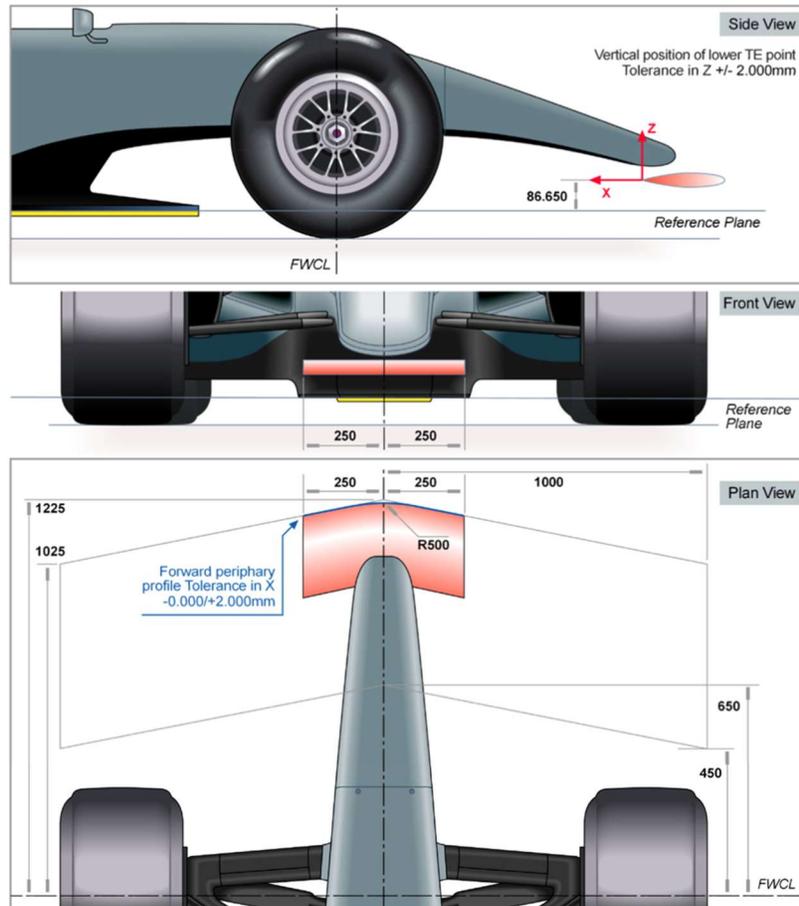


Figure 3 From the regulations document

The airfoil structure is how I started my model. The shape and size of the airfoil are crucial to the downforce profile of the wing. As noted earlier, teams change theirs depending on the track, and so I modeled mine to look the opposite of an airplane's front wing.

The features I used most are discussed in detail in the preceding sections.

Here are a few renders of the final product. The first two are the simpler model, and the second two are renders of the complex model with additional wing elements. This is the version closest to what a team would use. Compare to images above which show the ones in use currently.



Figure 4 Version 1 iso view

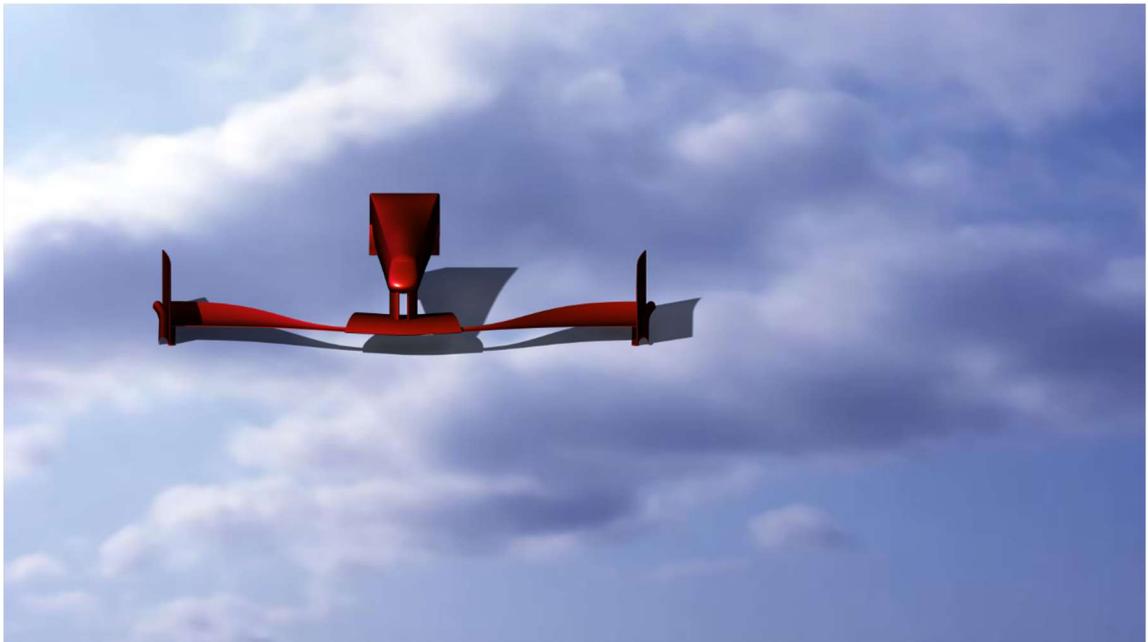


Figure 5 version 1 front view



Figure 6 version 2 iso view

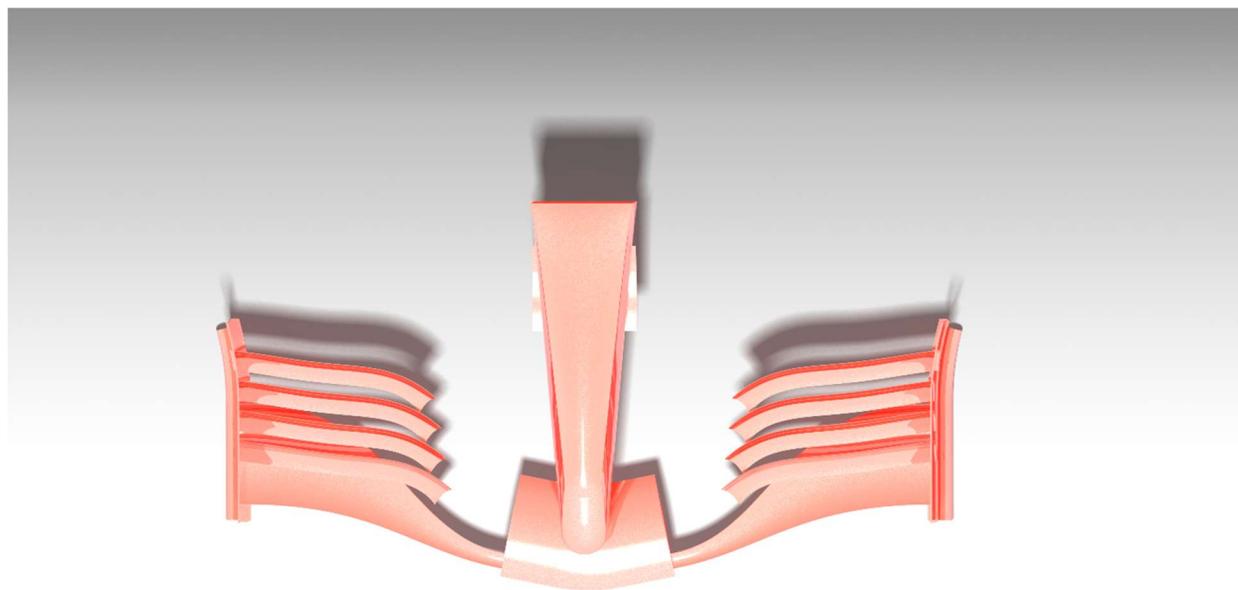


Figure 7 version 2 top view

CFD Procedure

The CFD turned out to be a little trickier than I had anticipated. This was mainly because of the meshing. I had access only to the student version of the ANSYS workbench, and it is limited to 512,000 cells. Since the geometry involved in this project, is very complex, it was easy to go beyond the limit. After a little trial and error, I was able to find the correct meshing size.

The main goal of the CFD analysis was to show how the teams use different configurations to generate more, or less downforce. The results confirmed the hypotheses that the complex one would generate more downforce. The table below summarizes the CFD procedure and can be used by future students to run similar simulations.

| CFD Process | Specification | Parameter | Values |
|---------------------|-------------------------|---|-----------------------------------|
| Geometry | 3D Rectangular Domain | Width, Height, Length | 3,2,20 m |
| Mesh | Physics preference | CFD | |
| | Inflation | Growth Rate | 2 |
| | | Transition Ratio | 0.272 |
| Sizing | Element Size | 0.92957m | |
| Setup | Precision | | Double |
| | General | Solver | Pressure Based |
| | | Time | Steady |
| | Model | Viscous | k-epsilon (2-eqn) |
| | | Energy | Off |
| | Materials (fluid – air) | Density | 1.225 kg/m ³ |
| | | Viscosity | 1.7894e-05 kg/m-s |
| Boundary Conditions | Velocity Inlet | 90 m/ | |
| | Outlet pressure | 0 | |
| Solution | Methods | Momentum | 2nd order upwind |
| | Monitors | Convergence tolerance, Drag Force (Z Direction) | 1.00E-05 |
| | Initialization | | Hybrid |
| | Run calculations | Number of iterations | 300 |
| Results | Vectors | Velocity | |
| | Contours | Velocity | Stream function |
| | | | Vorticity |
| | | | Velocity Magnitude |
| | Pathlines | X | Velocity, Particle ID, Turbulence |
| Y | | | |
| Z | | | |

Results

Here are the results of the CFD. The simpler one generates a downforce of about 1400N, while the more complex one generates a downforce of about 3100N.

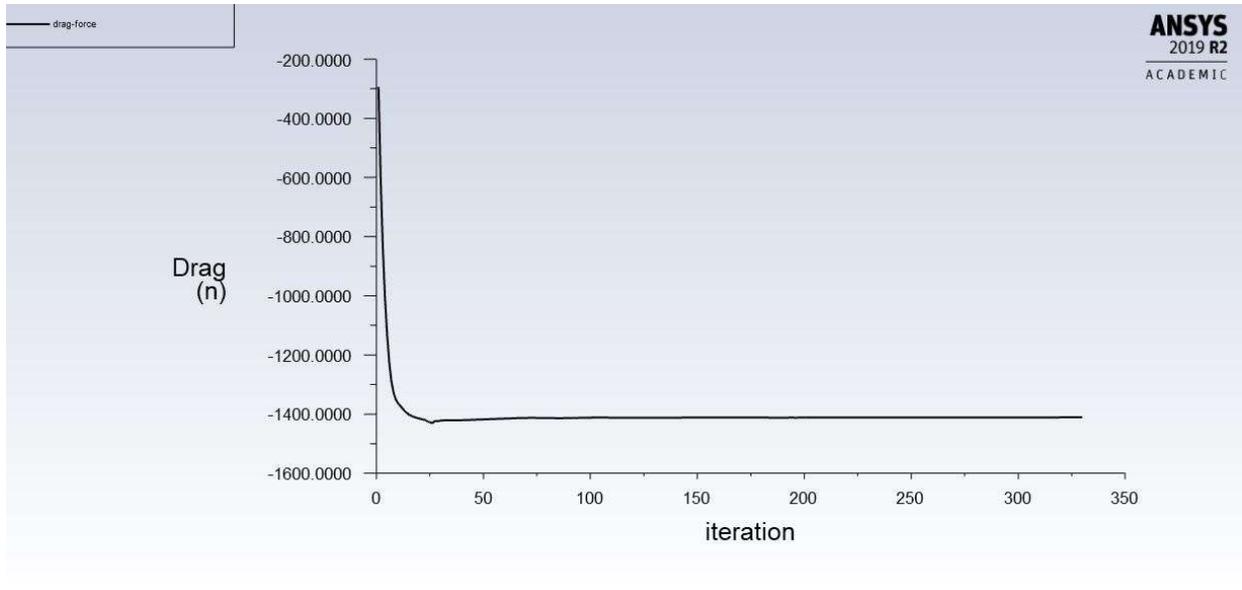


Figure 8 1400N Drag Downwards in the Z direction for the simple version

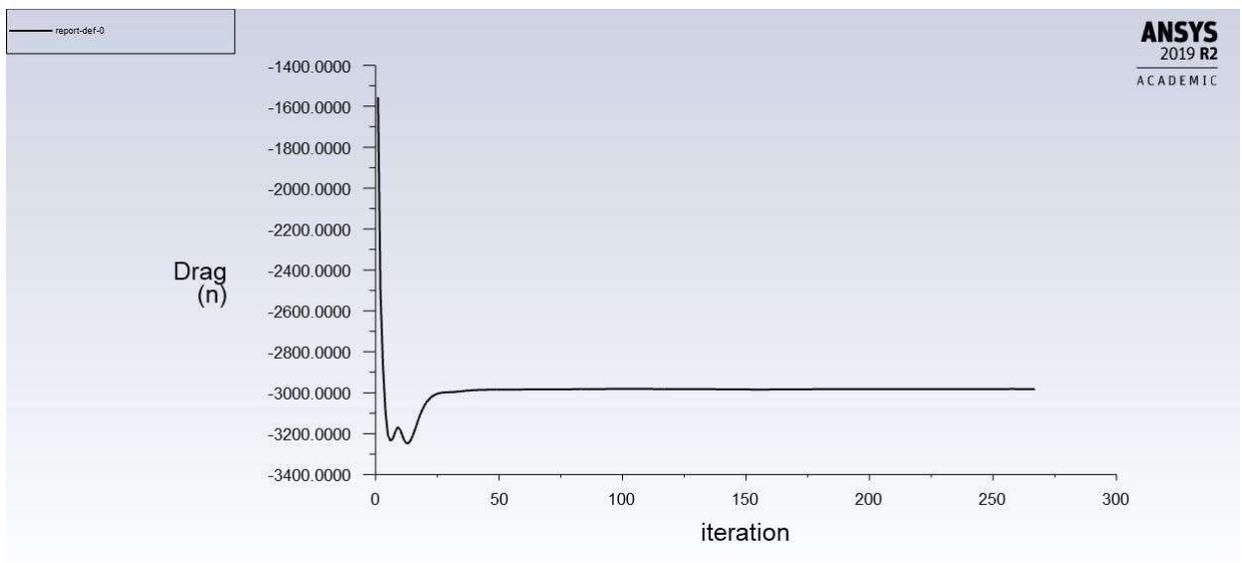


Figure 9 3100N Drag Downwards in the Z direction for the complex version

Here are simulation images:

Pathlines

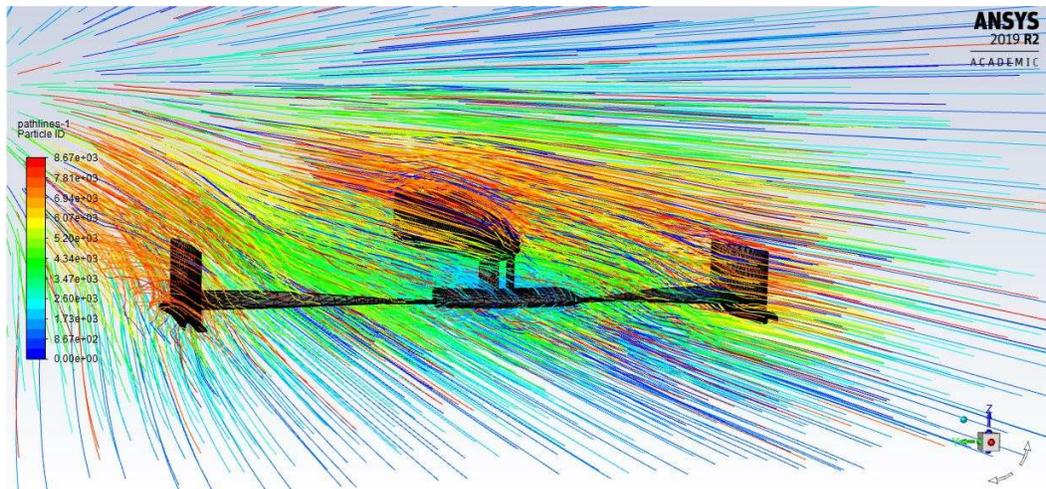


Figure 10 pathlines in simple version, no mixing of flows

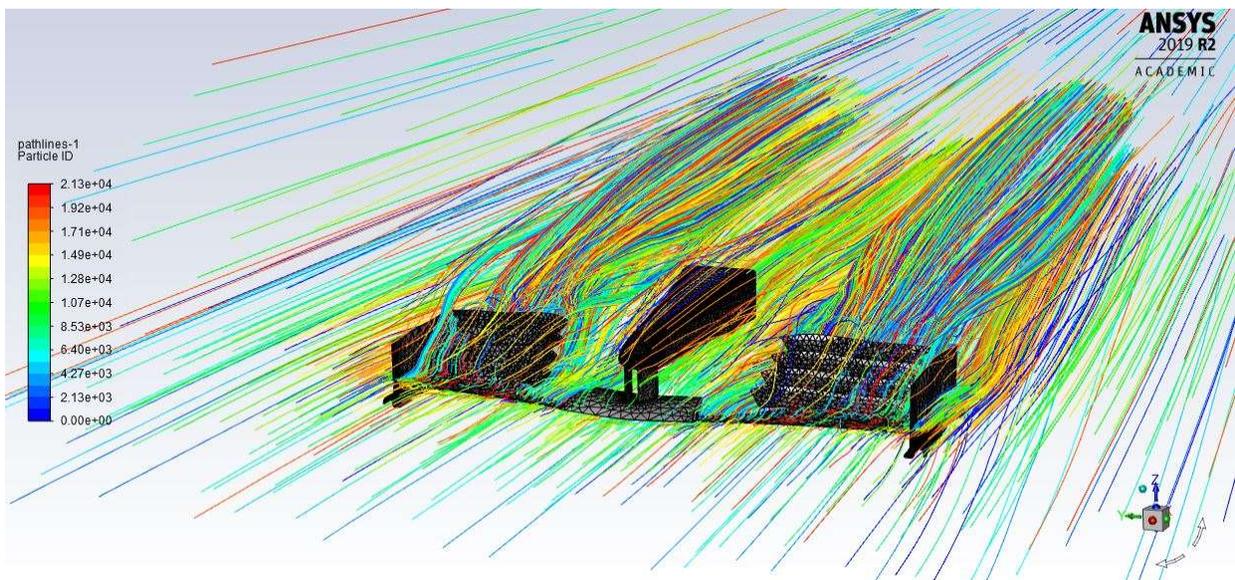


Figure 11 pathlines for complex version, notice the mixing of flows

Pathlines are the lines traveled by neutrally buoyant particles in equilibrium with the fluid motion. Pathlines are an excellent tool for visualization of complex three-dimensional flows.

The following two images are a great to visualize the eddy formation in the wake of the fluid.

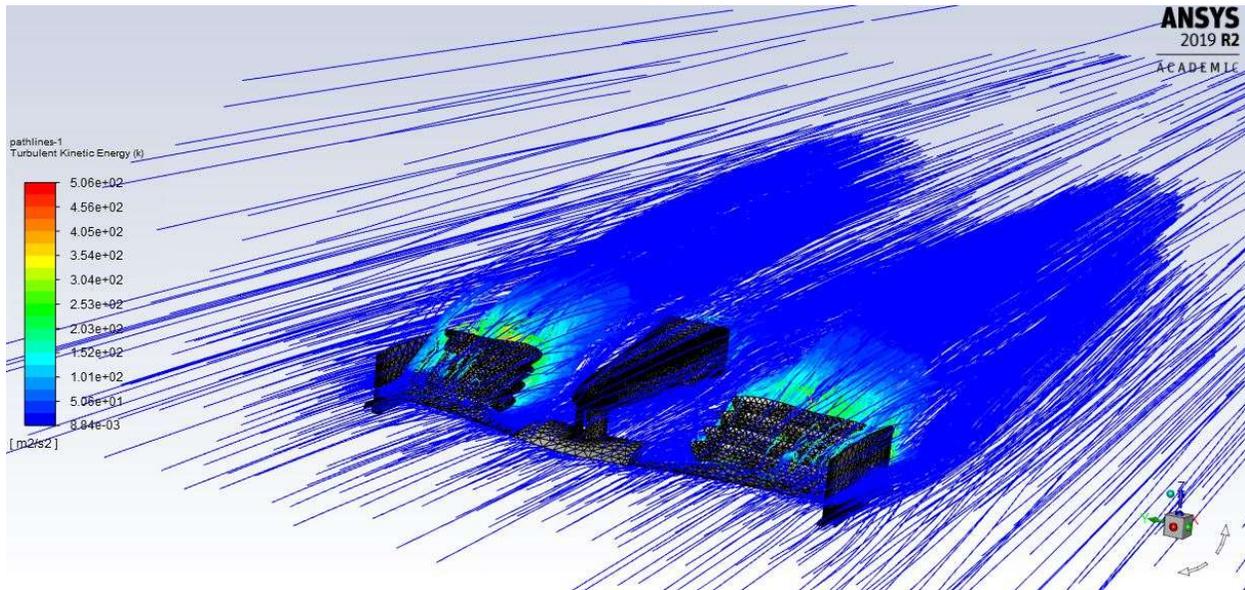


Figure 12 turbulent kinetic energy in wake

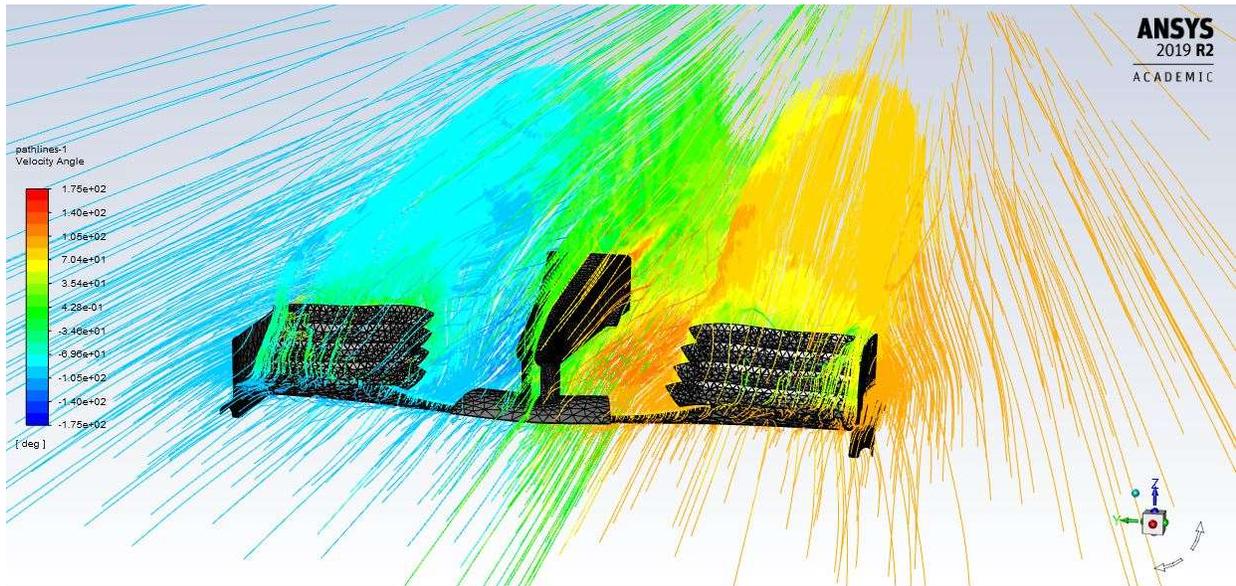


Figure 13 Velocity angle profiles

Lessons Learned

This section will discuss the most used features, some tips, and an important strategy in organizing the part while using the generative shape design workbench.

Organization in GSD

Working with such a complex part can get tricky especially because of all the sketches, curves, planes, and surfaces you make in the way.

If I were to start this project again, I would have organized my part better by using geometric bodies.

Geometrical sets enable to gather various features in a same set or sub-set and organize the specification tree when it becomes too complex or too long. You can put any element you wish in the geometrical set, it does not have to be structured in a logical way. The order of these elements is not meaningful as their access as well as their visualization is managed independently and without any rule.

For example, in my project I would create geometric sets for Sketches, Curves, Planes, and Surfaces.

To create a geometric set, navigate to the top toolbar, and click on insert. Name the set, and when you want to add to the set, right click on the design tree and click 'define in workspace'.

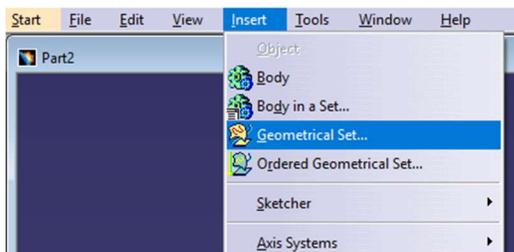


Figure 14 inserting geometrical set



Figure 15 defining in workspace to add to geometrical set

Notice below in the tutorials how I have used the geometric sets.

Combine Feature Tutorial

This was crucial in properly modeling these fins.

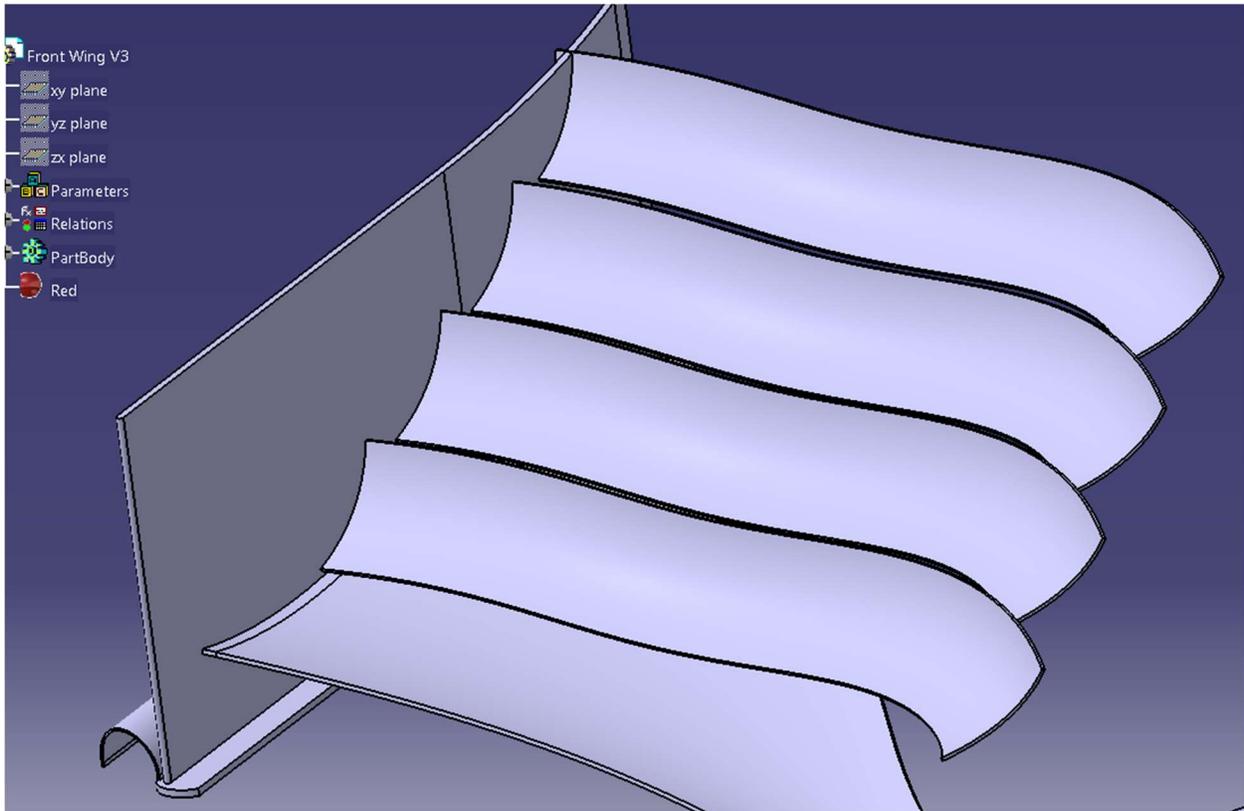


Figure 16 the fins in the wing

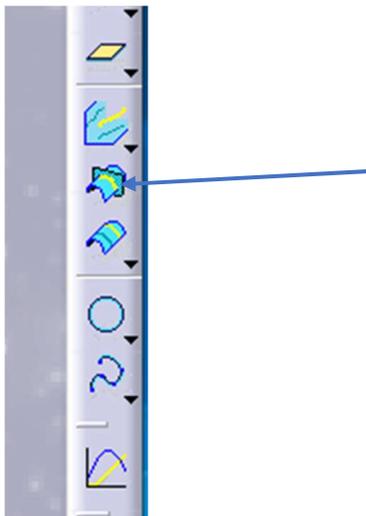


Figure 17 combine icon in the toolbar

Here I have two sketches in two different planes. By using the combine feature, I can combine them into one curve which is the intersection of the extrusion of the two sketches.

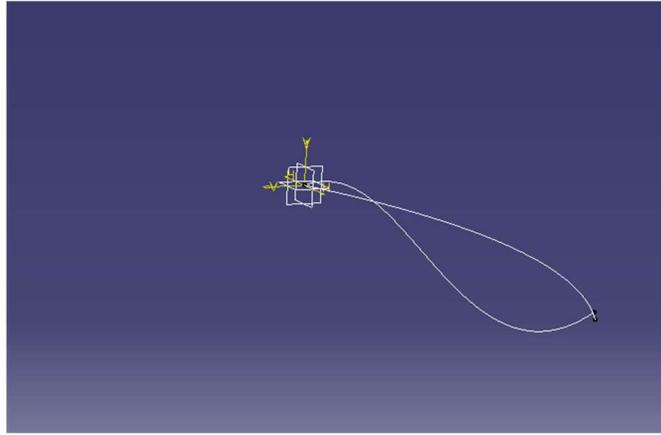


Figure 18 sketches

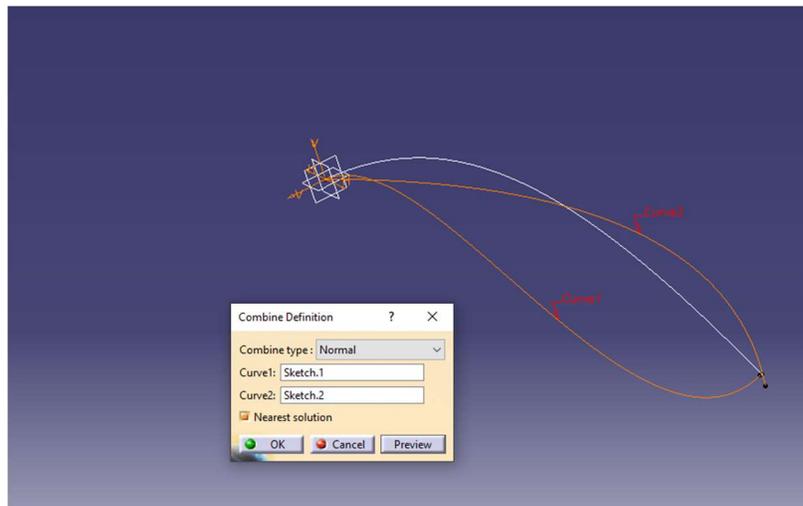


Figure 19 sketches, and curve

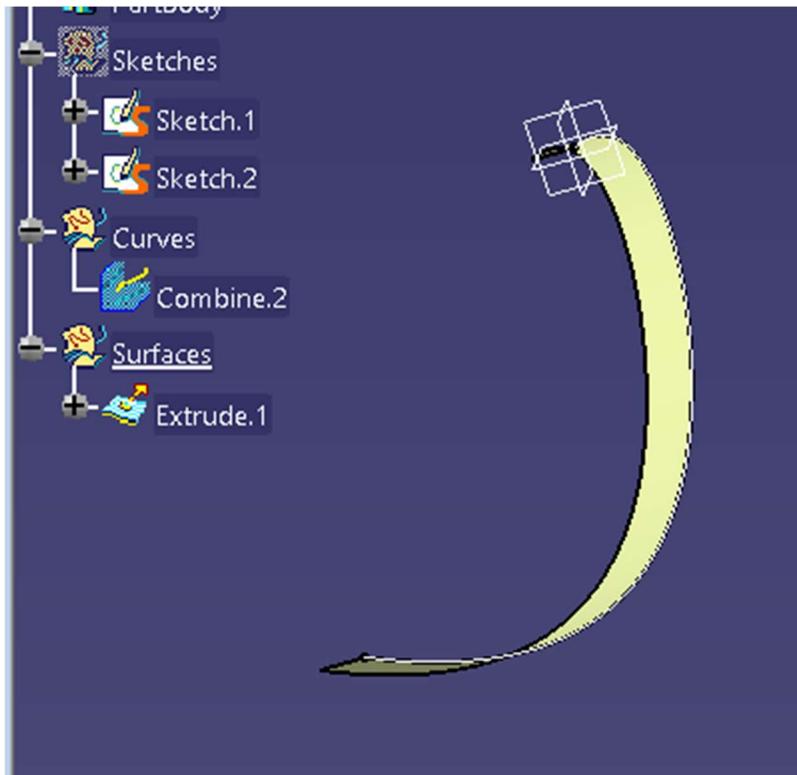


Figure 20 resultant extrude

Blend Tool

This is a short tutorial on creating blend surfaces between two curves, or surface edges. Here I start with two ellipses, which are extruded. The blend tool will allow me to create a surface between the two elements.

An important tip is to always select the larger element first. Selecting the smaller one first, often yields errors.

The curvatures can be altered using the tension options as shown in the third image.

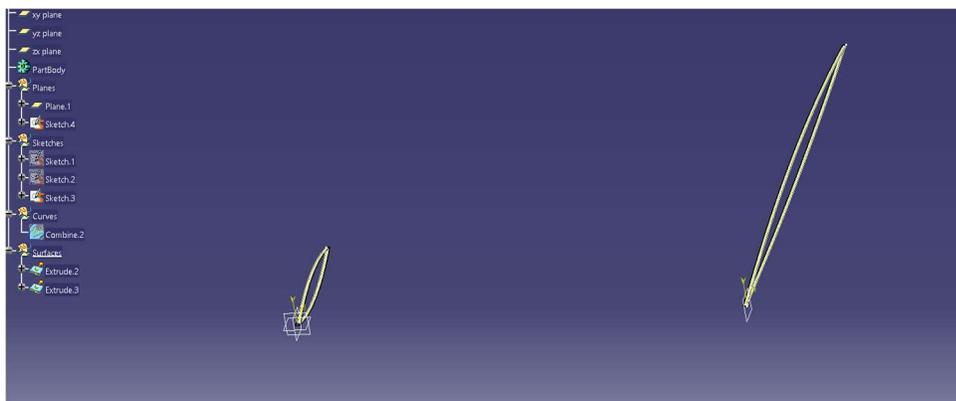


Figure 21 notice the design tree and geometrical sets used

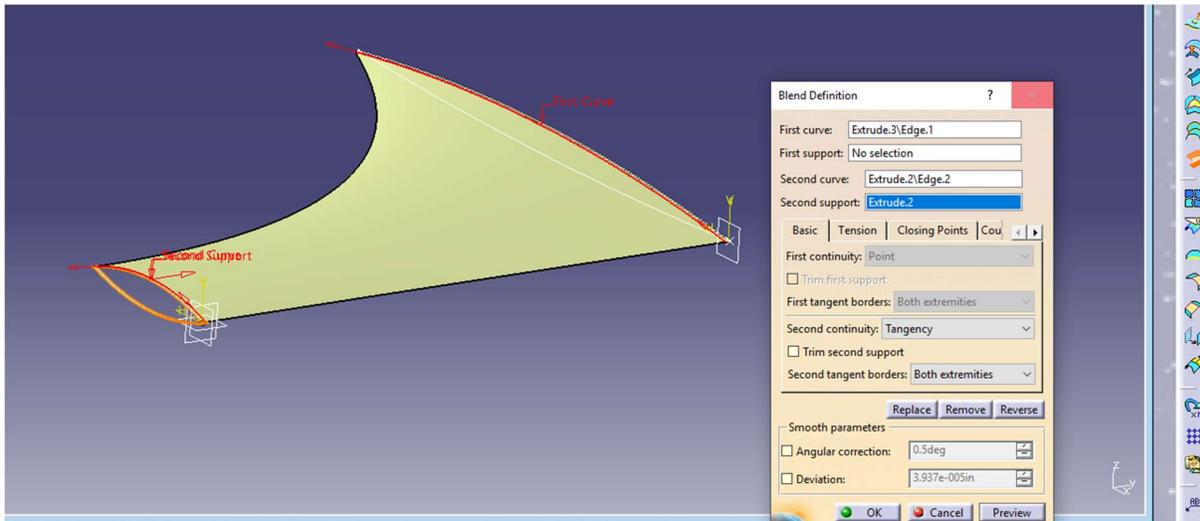


Figure 22 notice larger one is first curve

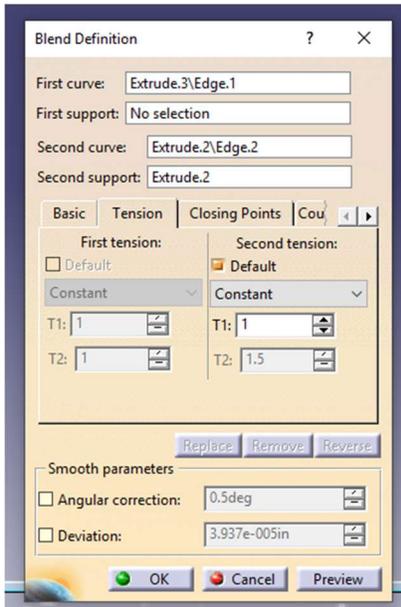


Figure 23 tension options

Conclusion and Discussion

I am glad this project gave me the chance to explore three things I enjoy: F1, Modeling, and CFD.

The CFD was very cool to say the least. The behavior of the fluid around the wing made sense, and I also learned a lot about the workbench. It is an intimidating workbench, especially with no prior surfacing experience, but worthwhile.

Appendix and References

Here are some links to more dimensions:

<https://www.racecar-engineering.com/articles/f1-mercedes-w10-testing-update/>

<https://www.racecar-engineering.com/articles/f1-mercedes-w10-eq-power/>

<https://www.f1technical.net/news/21935>

<https://www.f1technical.net/features/21879>