Introduction to Product Lifecycle Management
Skoltech 2019

Aerodynamics Team Project Report

Mentor: Daniil Padalitsa

Team:
Anastasija Cumika
Denis Artemov
Veronika Rocheva
Puskar Pathak
Joshit Mohanty

Table of contents

1. Introduction

1.1 Team

- Anastasija Cumika - CFD simulations, airfoil design and parametrization
- Denis Artemov - active aerodynamics, rear wing design, communication with other teams
- Veronika Rocheva - undertray design
- Puskar Pathak - front wing design, car body design, 3D CFD simulation
- Joshit Mohanty - car body design, cooling

1.2 Objective

- Provide Formula Student team with aerodynamics design capable of generating downforce ~18-25 kg in at approximate speed of 50 km/h.
- The result should include:
    - Airfoil choice
    - Choose design configuration for front and rear wing through 2D simulations
    - Rear and front wing, body and undertray 3D designs
    - Implementation of actuation system for active aerodynamics
    - Make aerodynamic design to provide passive air cooling for battery packs and motors.
- Run full car aerodynamic test and provide teams with force coefficients and moments

2. Content

2.1 Workflow and Work BreakDown Structure (WBS)

Generally workflow can be described in that way: firstly we perform research of rules and restrictrictions provided by the formula student organization committee. It helps to organize aero kit design constraints and approximate dimensions of its parts. Then we split our roles in teams and start to search and study information on how to effectively develop selected parts. We researched some papers of the previous teams and got ideas from them.

First choice that must be done is the rear and front wing airfoil design. Then we do parameterization of the multiple wing design and run CFD simulations to determine the best configuration of airfoils and find a proper angles of attack and chord lengths of the wings. It leads us to modelling of the wings (rear and front) in 3 dimensions and complete CFD simulations. For better calculation of rear wing efficiency we need to provide our model with complete car body design which is used to decrease drag, and even simulate influence of aerodynamic resistance of racer presence in cockpit. Undertray makes the whole car act as inverted wing and it increases downforce efficiency and it can also be used for providing cooling for batteries and engine of the car so along with working on body and wing design we have to work on the undertray design.

We create the full car model and run simulations for different speeds and radius turns. Then we optimize the aerodynamic kit by imentic with the design and wing parametrization.

Active aerodynamics kit can be used to decrease drag influence on straights of a race track by changing wings`s angle of attack. We believe that an active aerodynamics system will improve the aerodynamics of our car. We also implement the passive cooling system into our aerodynamics design to provide cooling for the power system.

As a final result we produce our final model and obtain parameters of produced downforce, drag, momentums, cooling, power and material consumption. The workflow scheme shown in figure below.

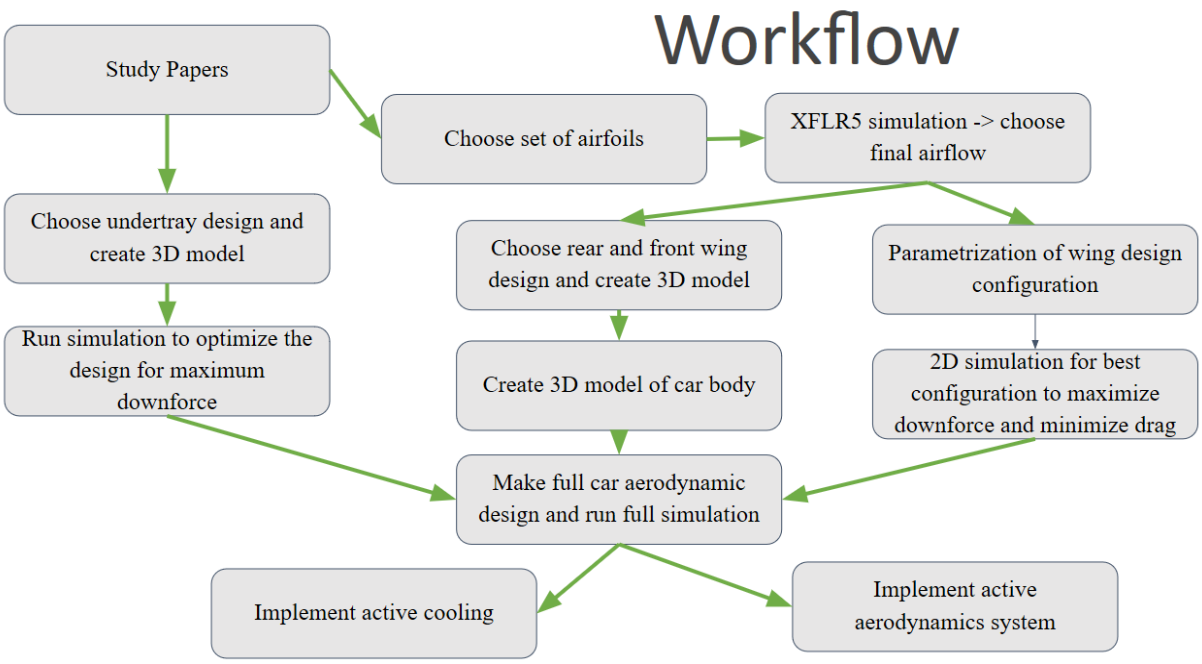


Figure 1 - Scheme of the workflow

We divided our tasks into four main parts: parametrization, 2D modeling and simulation, 3D modeling and simulation and active aerodynamics implementation. Each part has its subtasks as described in the work breakdown structure (WBS) shown below in figure 2.

2.2 Structure of subsystem (PBS), inputs and outputs

Our aerodynamic kit consists of five main parts: front wing, rear wing, body frame, undertray and active aero actuation. Each part of the kid has its subparts as shown in the product breakdown structure (PBS) below in figure 3.

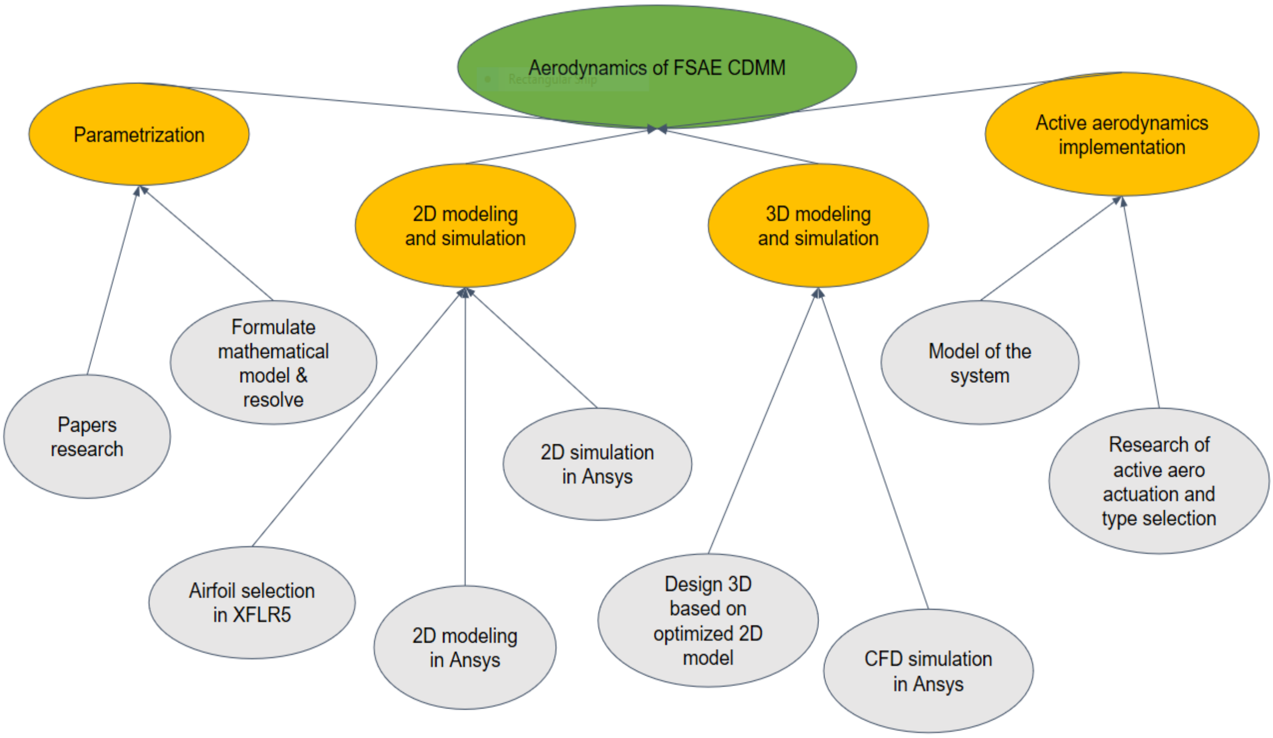


Figure 2 - Scheme work breakdown structure

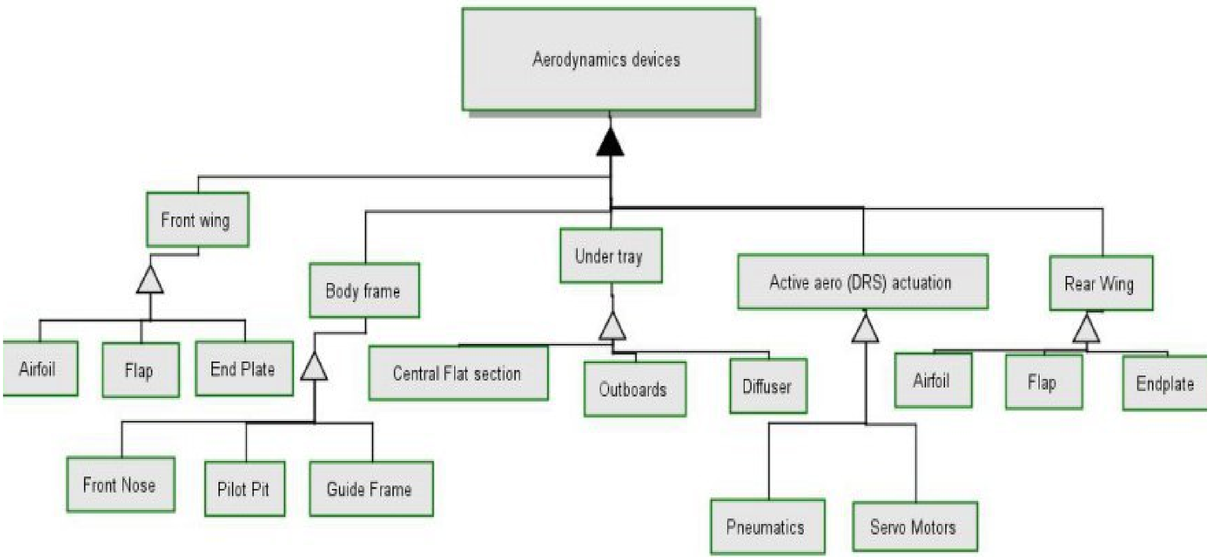


Figure 3 - Product breakdown structure

Our design is highly dependent on the design of chassis, suspension and power train teams. We make our design around their dimensions, that is why any change in their design might affect our design. Any changes in our design can change the aerodynamics coefficients therefore we have to run the simulation for every change we make. Other teams have not finalized their dimensions yet so now we can only give very rough estimations of force and lift coefficients. Our interactions diagram in terms of Inputs/outputs is provided below in figure 4.

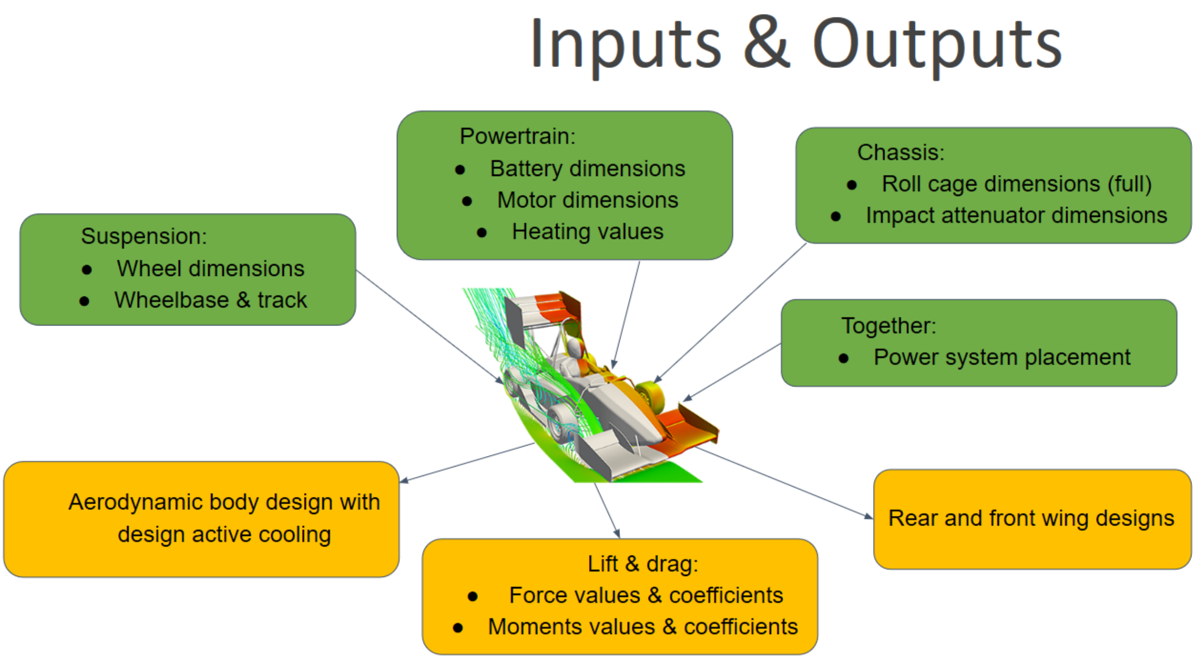


Figure 4 - Inputs/outputs scheme

2.3 Current results, numbers, current parameters of subsystem/task

2.3.1 Airfoil Selection

As it was described in the workflow section, our team started with selection of airfoil design. The main goal was to pick one with the highest lift coefficient. We studied the literature we found on the internet from other teams and found a set of airfoils that were used more often by teams and that are considered as high lift and low reynolds number airfoil. The set of chosen for test airfoils listed below:

- E423
- fx74cl5140
- fx63137sm
- s1223
- lnv109a

We downloaded the profiles dat files from the UIUC Airfoil Coordinates Database. For 2D simulation of single airfoil we used software called XFLR5 which is a free and easy to use software. Our goal was to find the airfoil that gives the highest lift with the parameters that we give. We took the average speed to be 50 km/h, sea level pressure and air viscosity. To calculate the Reynolds number we simply used y+calculator online. Putting all the required parameters in XFLR5 as shown in figure 5 on the top we get the plot for lift coefficient vs alpha (the angle of attack of the airfoil) shown in figure 5 on the bottom.

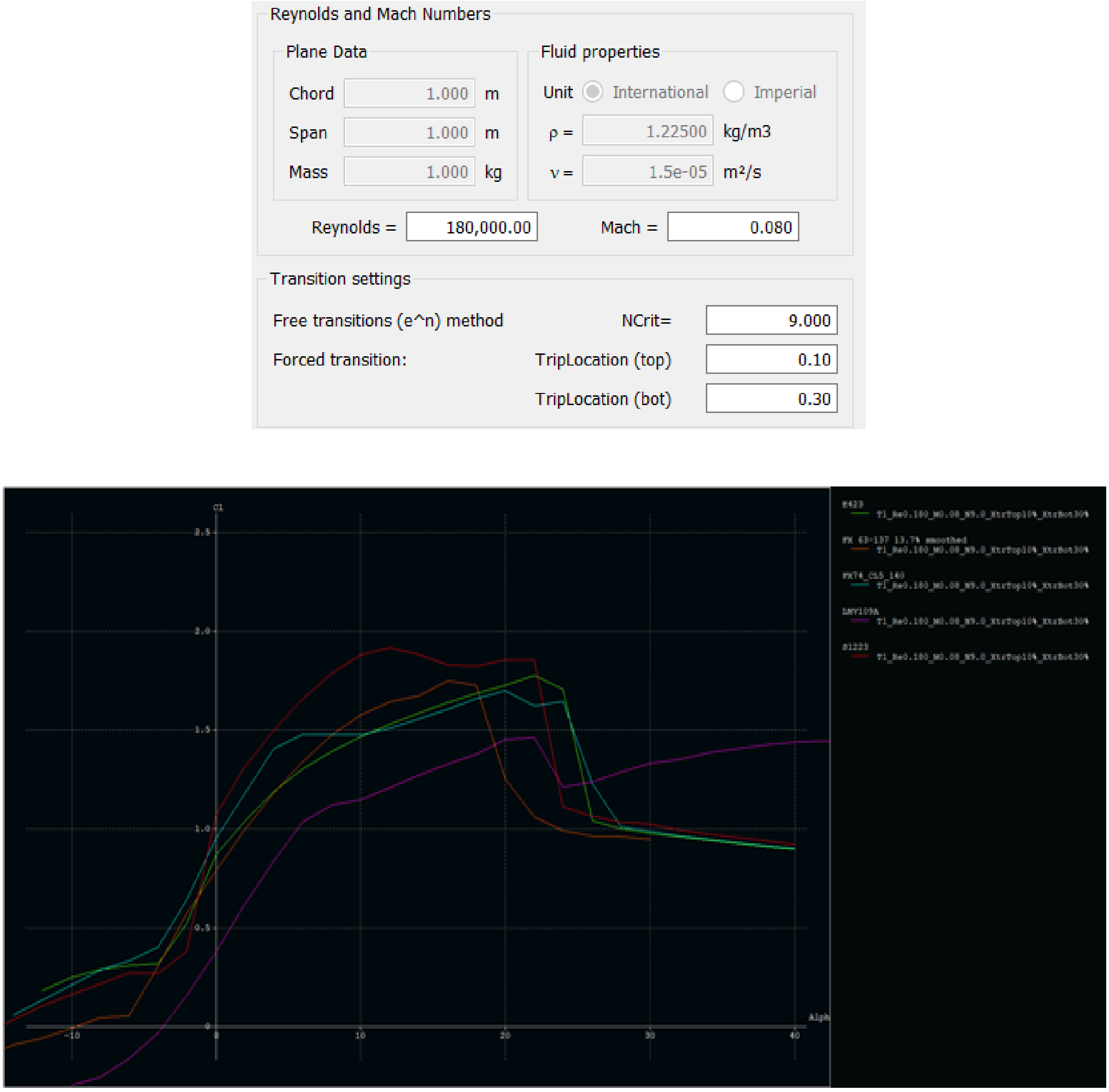


Figure 5 - coefficient we choose for XFLR5 airfoil simulations and result plot in XFLR 5

There are graphs for each airfoil. Based on that plot we can see which airfoil has higher lift coefficient at our parameters. Red graph has best performance and it corresponds to the S1223 airfoil. We choose S1223 for both rear and front wing (for simplicity since we are limited on time). The shape of airfoil S1223, shown below in figure 6.

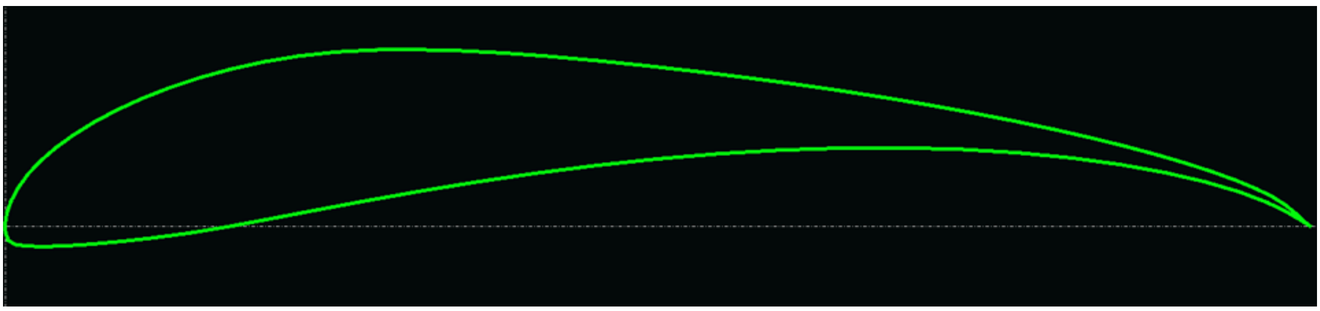


Figure 6 - Shape of airfoil S1223

2.3.2 Multiple Wing Parametrization

In order to understand what is the best placement of the wings with respect to each other, best angle of incidence and chord lengths we have to parametrize those parameters. We wrote code in Matlab (also in Maple) that takes some input parameters and gives the output parameters based on the constraints. The simple schematic of parametrization is shown in figure 7.

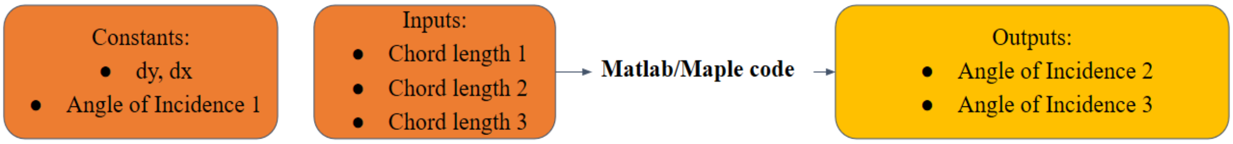


Figure 7 - multiple wing parametrization schematics

The constraints are the "box" sizes where we have to fit the wing. Constraints come from the size restrictions of the rule book as well as placement of other parts in the car (for example we wanted to have min height of front wing to minimize manufacturing cost but still lead the air around the wheel). Some constants include the separation distance between the consecutive airfoils. For simplicity now we took 1cm for both x and y direction(We want to play with it after we have a full body). We also set the angle of incidence (AOI) for the first wing to 5 degrees. As input we give a set of chord lengths of first, second and third wing. For simplicity we have second and third with the same chord length. We end up with the output of AOI for the second and third wing. The example of the code is shown in figure 8 below.

```
CC=linspace(340,360,11);
C=linspace(150,170,11);
for Ch1=CC
    for Ch2=C
        syms AOI2 AOI3
        Ch3=Ch2;
        Ch2=Ch2;
        Ch1=Ch1;
        delx=10;
        dely=10;
        X=625;
        Y=250;
        AOI1=degtorad(5);
        YY=Ch1*sin(AOI1)+dely+Ch2*sin(AOI2)+dely+Ch3*sin(AOI3);
        XX=Ch1*cos(AOI1)+delx+Ch2*cos(AOI2)+delx+Ch3*cos(AOI3);
        eqX=XX==X;
        eqY=YY==Y;
        eqns=[eqX,eqY];
        [SAOI2, SAOI3] = solve(eqns,[AOI2 AOI3]);
        radtodeg(double(SAOI2(2)));
        radtodeg(double(SAOI3(2)));
        AOI2=(double(SAOI2(2)));
        AOI3=(double(SAOI3(2)));
        if isreal(radtodeg(double(SAOI2(2))))
            if  radtodeg(double(SAOI2(2))) < 60
                display(radtodeg(double(SAOI2(2))))
                display(radtodeg(double(SAOI3(2))))
                display(Ch1)
                display(Ch2)
            end
        end
    end
end
```

Figure 8 - Parametrization code example from Matlab

We run the code for different sets of chord lengths and different Y constraints and we have chosen a set of 20 design parameters that we want to run ansys simulation for. We put the parameters in the Excel spreadsheet and obtain the parameters for ansys. Below in figure 9 one can see what our Excel spreadsheet looks like.

AOI1(deg)	AOI2(deg)	AOI3(deg)	AOI1(rad)	AOI2(rad)	AOI3(rad)	Ch1x	Ch1y	Ch2x	Ch2y	Ch3x	Ch3y	Ch2Trx	Ch2Try	Ch3Trx	Ch3Try	Wheel
7	49,8978	54,4965	0,122173	0,870881	0,951143	0,377168	0,04631	0,119812	0,142271	0,10802	0,151419	0,387168	0,05631	0,51698	0,208581	0,8698
5	43,8881	56,1959	0,087266	0,765992	0,980803	0,372577	0,032596	0,131167	0,126172	0,101257	0,151232	0,382577	0,042596	0,523743	0,178768	0,8698
5	40,8959	51,7886	0,087266	0,713768	0,903882	0,368592	0,032248	0,130015	0,112606	0,106393	0,135146	0,378592	0,042248	0,518607	0,164854	0,8698
5	33,371	41,9123	0,087266	0,582434	0,731508	0,35863	0,031376	0,13028	0,085809	0,11609	0,104207	0,36863	0,041376	0,50891	0,137185	0,8698
5	26,6195	47,8777	0,087266	0,464598	0,835624	0,354645	0,031027	0,14304	0,07169	0,107314	0,118674	0,364645	0,041027	0,517686	0,122718	0,8698

Figure 9 - Part of Excel spreadsheet with parameters for front wing

To choose the best airfoil configuration we run 2D simulation in Ansys.We import the airfoil to Ansys and create a surface. We had to cut the tail a little bit because it is too sharp and caused issues in mesh generating (it is also hard to manufacture anyways). We use the ready made parameters from Excel to create the multiple wing 2D design. In addition, we put the wheel behind the airfoils on the distance 75mm from the end of the last airfoil which is the minimum based on the rule book. We run the simulation for all 20 parameter sets and see which configuration gives us the best downforce to drag ratio. We are also looking so that we maximize the downforce because we are mostly looking for the highest downforce to increase the stability on the turns at high speeds. We did simulation reports in the way so we see what is the drag and lift force on each wing also drag on the wheel. After exporting data to Excel we calculated the total drag and lift (downforce). The results are shown in figure 10. With yellow highlighted some best configurations and with red best downforce/t coefficient. The best out of the best is in orange color.

Drag	Lift	Lift/Drag
67.28285	200.5276	2.980368
48.68204	179.5501	3.688221
48.82522	210.4065	4.309381
44.90233	181.4422	4.040819
29.10984	119.0834	4.090829
44.10589	171.5445	3.889379
48.84009	228.0169	4.668642
57.65643	259.7337	4.504853
49.07165	192.7588	3.928108
45.09209	154.3863	3.4238
44.11412	181.1034	4.105339
62.47694	233.2466	3.733323
45.35333	206.31	4.54895
33.2896	136.5795	4.102767
35.62538	146.2745	4.105907
36.8888	187.8884	5.093374
46.03369	227.1042	4.933435
51.96197	220.0411	4.234656
56.20318	213.9707	3.807093
24.56061	91.60227	3.729642
58.9827	103.2792	1.751009

Figure 10 - Simulation results for 20 parameters sets for front wing

The parameter set that got 4.93 ratio coefficient is:

Ch1 = 346 mm AOI1 = 5 degrees

Ch2 = 162 mm AOI2 = 30.04 degrees

Ch3 = 162 mm AOI3 = 42.16 degrees

We obtained the velocity distribution plot (figure 11 picture above) and velocity path lines plot (figure 11 picture below). High velocity is red and regions with high velocity have low pressure. When we have high pressure on top and low at the bottom we create downforce. In the picture we can see that we get a much higher velocity below the airfoil that creates downforce. We have very few regions with low velocity (compared to other configurations).

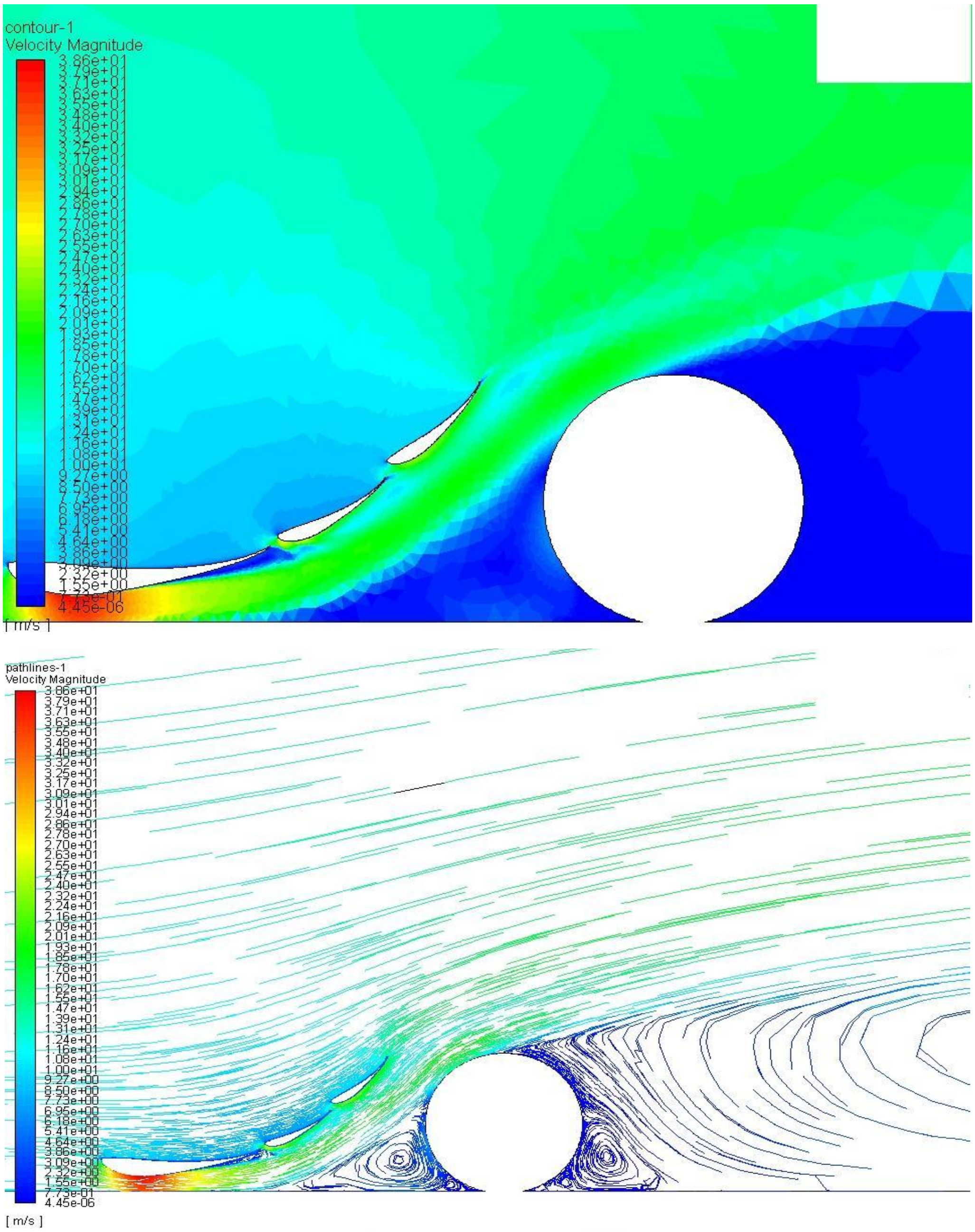


Figure 11 - Velocity plots for best airfoil configuration

Later on the wheel changed and we had to redo the simulation and choose a different parametrization for the front wing. We have also created parametrization for the rear wing. Below in figure 12 we show the various simulation results we obtained from the 2D simulation of the rear wing parametrization. On the same figure we show the representation of the 2D simulation for one of the parameters sets. With green we highlighted potential best and with orange we highlighted the parametrization that we choose for 3D model creation. It is the second best lift/drag coefficient but we chose it because we did multiple simulation runs and the 3.636 coefficient parametrization results were not stable.

The parameter set that got 2.901 ratio coefficient is:

Ch1 = 306 mm AOI1 = 7 degrees

Ch2 = 198 mm AOI2 = 45.50 degrees

Ch3 = 198 mm AOI3 = 49.91 degrees

Drag	Lift	Lift/Drag
61,06855	169,836	2,781072
58,55908	133,9037	2,286643
51,87015	188,584	3,635694
60,31811	170,3082	2,823501
36,76125	97,06683	2,640466
55,44843	113,6578	2,049794
61,86876	147,3365	2,381435
59,1557	168,327	2,845491
64,41667	136,4264	2,117874
71,90577	188,1205	2,616209
60,02766	131,7504	2,194828
77,34407	188,2361	2,433749
51,51288	121,9835	2,368019
43,39285	125,8856	2,901067
72,46675	161,9692	2,235083
62,60003	135,472	2,164088
60,20036	159,1316	2,643366
69,78694	189,6605	2,717708
53,49849	129,8392	2,42697
63,81625	169,8228	2,661122
48,46337	121,4192	2,505381
71,61577	163,2963	2,280172

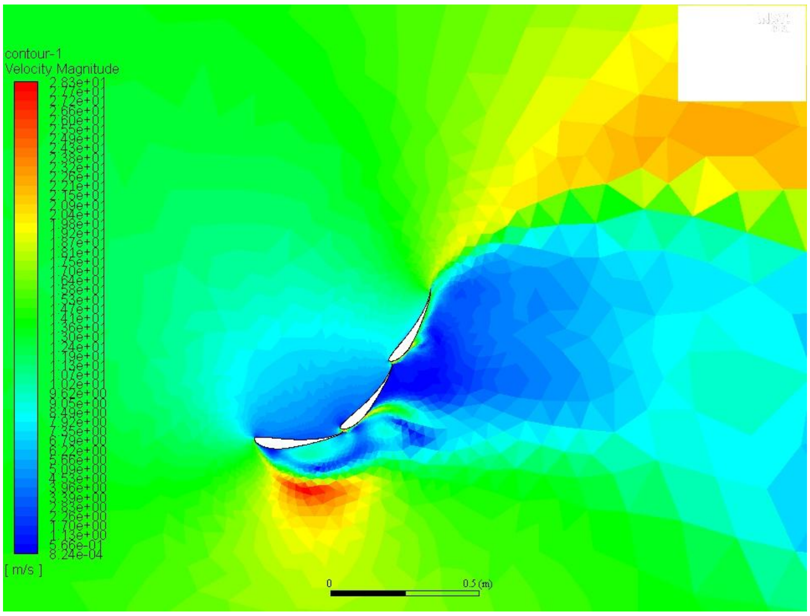


Figure 12: 2D simulation results

2.3.3 Front wing design

First front wing design, shown in figure 13, was not based on 2D simulation. We did this to start playing in ansys and get the concept of simulations in 3D.

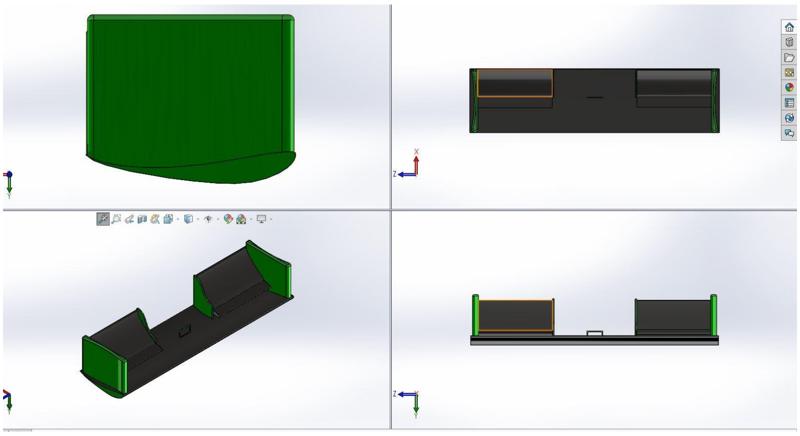
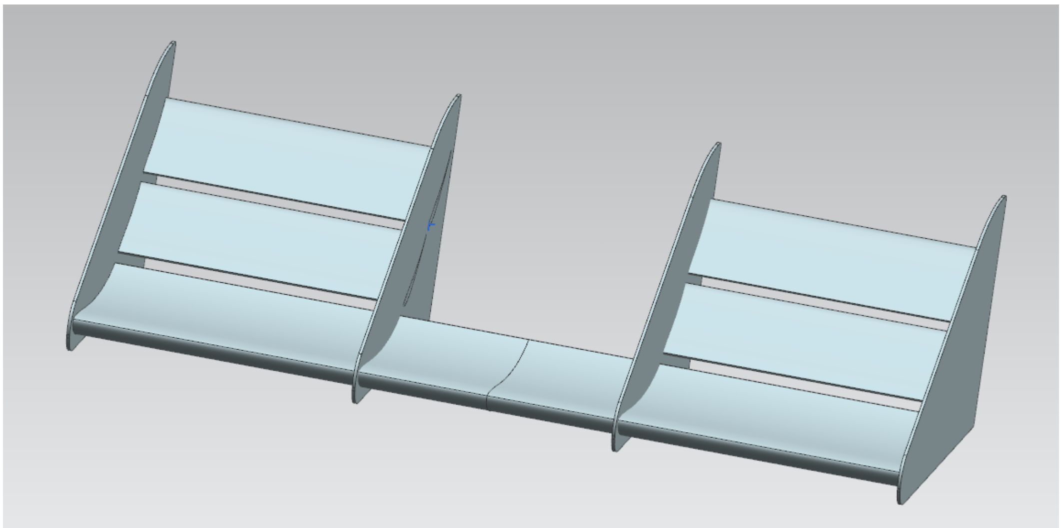


Figure 13: -3D model of front wing from Solid Work

Then we used the data obtained from the 2D simulations and designed a new front wing based on the best parametrizations results. We keep all three airfoils on the sides. And near the nose we just keep the first airfoil. The design is shown below on figure 14.

We simulated the wing in ansys and we got that our wing has a drag force of 4.75 kg and downforce of 17.82 kg. On the same figure where 3D design is shown, we show the ansys simulation representation where one can see how the pressure is distributed and how the wind flows around the wing. We also tried another design where we added little edges on the bottom and rear side of the endplate. We increased the downforce however we also increased the drag and overall downforce to drag ratio decreased so we excluded the new design.

As we started assembling the body the front wing continued to change based on the full car simulation. The discussions and results will be below in the full body simulations section.



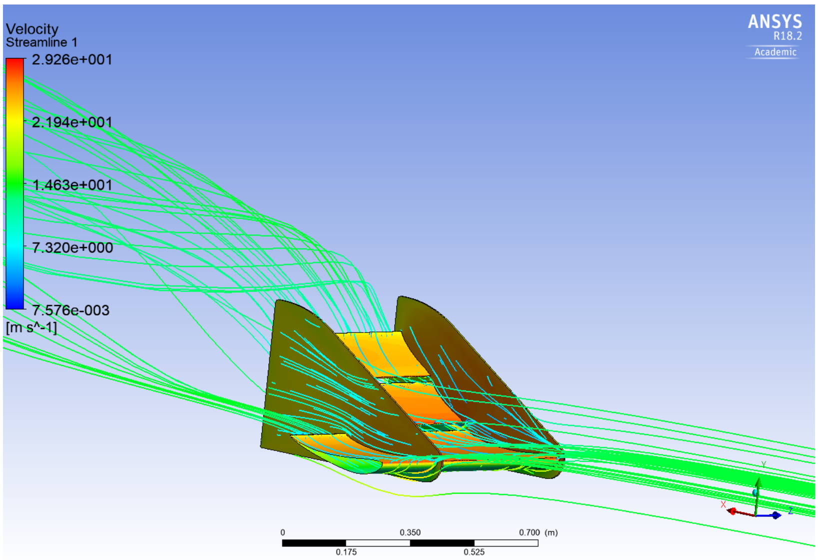


Figure 14 3D simulation results

2.3.4 Rear wing design

Rear wing design is directly based on the parameter set we obtained from the 2D simulations. We took the numbers and oriented airfoils according to the chord lengths and angles calculated. Below in figure 15 one can see the representation of the 3D simulation in ansys. For this rear wing we got 6.79 kg drag force and 17.98 kg downforce for the regular state (shown on the left). For the open state (shown on the right) we got 1.29 kg drag and 7.5 kg downforce. Open state is when we turn both rear airfoils to the 60 degree angle around the axis that is discussed in the active aerodynamics design section. We also tried turning the two airfoils on the higher angle for more effective breaking, however drag force on it was even lower than on the regular state that we simulated before. Therefore we just choose the two states discussed above. Obtained results will be used in the design of the drag reduction system (DRS) for as a part of the active aerodynamic system of the car.

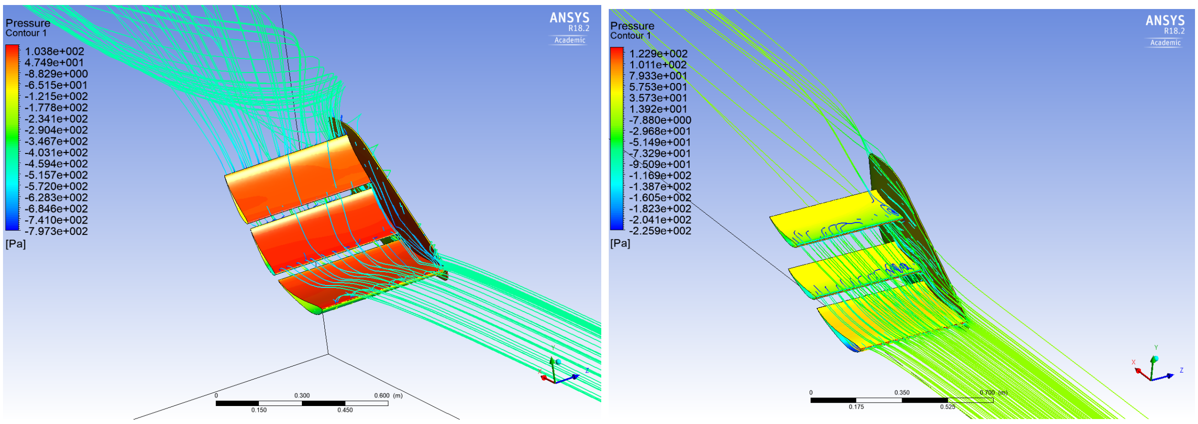


Figure 15 3D simulation results

2.3.5 Active aerodynamics design

Active aerodynamics systems are planned to be used as DRS (Drag Reduction System). It is used for changing the angle of attack of the rear wing. Two modes - the first one with the maximum downforce and the second with reduced drag. Schematic representation of this principle shown in the figure 19 below.

Figure 16 - Schematic representation of DRS work principle

In order to design this kind of a system we need to consider aerodynamic forces and momentums that affects the wings if different operation modes in order to determine suitable actuation system. For this purpose we need to find the aerodynamics focus of the wing and calculate the main pressure line of the wing. Regularly maximum momentum is about 12-25 kg/cm. Which makes possible use of electric servo motors with torque of 50 kg/cm. They have light weight (about 200 g), good control possibilities because of PWM control and built in feedback sensor. It's also a good choice because we are designing an electric car with an autopilot system, which means it will use voltage converters, microcontrollers and sensors for this system. And there is no need to install additional equipment, which may happen in case of pneumatic actuator installation.

Based on rough calculation parameters, such wheelbase, track and wheel size, wings square, basic values of downforce were obtained. It shows that approximate downforce generated by rear and front wing at a speed of 40 km/h can be about ~200-300 N in closed state for the front wing and ~300-400 N in open state, for the rear wing in open state the downforce is about ~120-200 N and ~200-250 in closed state. The aerodynamic calculator for rough calculation is shown in figure 20 below.

	A	B	C	D
1				
2		Wheelbase	1525	mm
3		Front Track	1450	mm
4		Rear Track	1450	mm
5		Tire diameter	339,6	mm
6		Tire width	335,3	mm
7				
8		Lift of device	If V max (N)	If V min (N)
9		Lift front wing (open mode)	1013	96
10		Lift front wing (closed mode)	1448	137
11		Lift rear wing (open mode)	579	55
12		Lift rear wing (closed mode)	910	86
13				
14		Calculation tuning	Value	Unit
15		C1 (front open)	1,4	Coefficient
16		C2 (front closed)	2	Coefficient
17		C1 (rear open)	1,4	Coefficient
18		C2 (rear closed)	2,2	Coefficient
19		V max	130	Kmph
20		V min	40	Kmph
21		Front wing length	625	mm
22		Rear wing length	664,6	mm
23		Rear wing width	779,4	mm
24		I (density of air)	1,225	kg/m^3
25		S front	0,90625	m^2
26		S rear	0,51798924	m^2
27				

Raw Estimation

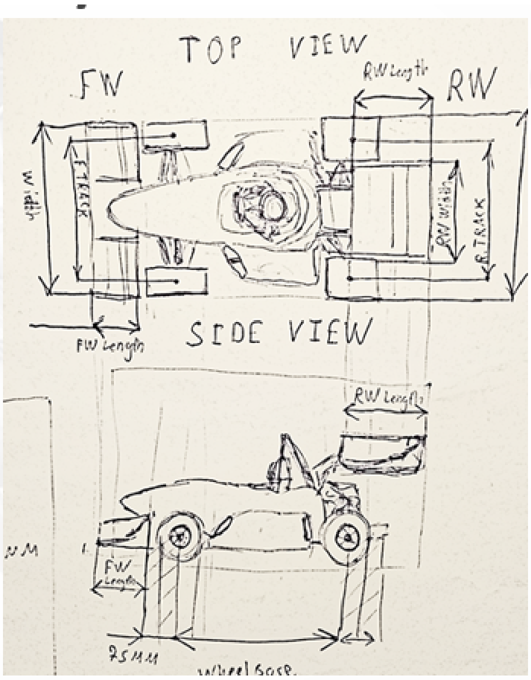


Figure 17 - Aerodynamic basic calculator

The calculator gives raw estimation of the air force applied to the rear wing. According to our last simulations, the maximum loadings of the last two rear wing segments equals less than 188 N at a speed of 50 km/h. In that case the summary momentum of 1.2 N-m needs to be produced in order to control the angle of attack of the rear wing segments. It makes possible the use of mid-size servo-motors with torque of 25 kg-cm. which equals approximately 2.4 N-m. It is sufficient for the task of DRS and also gives reserve momentum to operate at higher speeds and loads. One of the possible servos for use presented on figure_ below.

Figure 18 - Kinematic scheme of the proposed mechanism

The average speed of an unloaded servo motor is about 0.12 sec / 60 deg.

Next task is to design and construct a kinematic mechanism for wing section movement by connecting it to the servo. The desired scheme shown below:

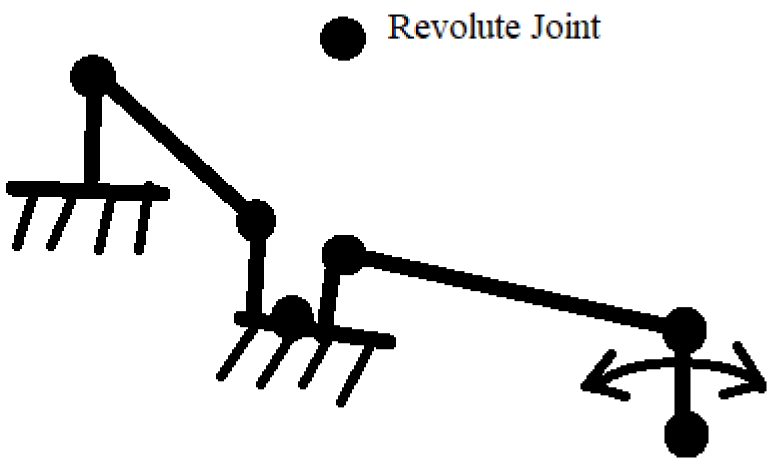


Figure 19 - Kinematic scheme of the proposed mechanism

Mechanism consists of revolute joints and straight rods connected to the wings. Mechanism actuated by rotation the rod connected to the servo motor, it marked with curved arrows.

Finally based on the kinematic scheme, assembly of the rear wing DRS mechanism has been built. Figure of CAD representation shown below:

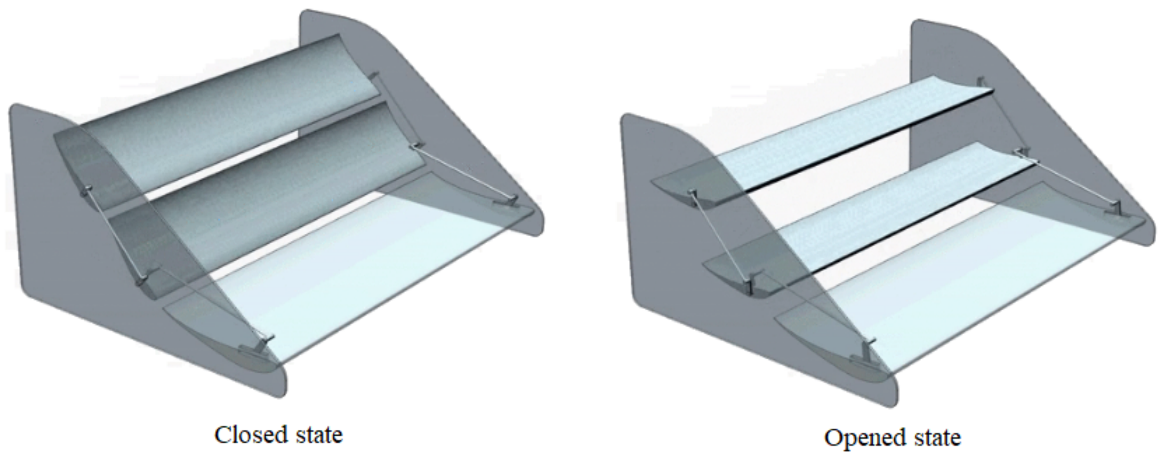


Figure 20 - CAD assembly model in Siemens NX. Closed and opened state

As a result we obtained a model capable to set the proper angle of attack for rear wing segments and calculated basic parameters as a proof of concept. It is possible to add improvements to this system: mechanical adjustments to increase efficiency of rotational pairs.

2.3.6 Undertray design

Ground clearance is the distance between the lowest point of the undertray to the ground. The equation of continuity gives an insight into how the performance of the undertray is affected by changing the ground clearance. Lesser ground clearance leads to more reduction of pressure below the under tray which leads to greater pressure difference and hence more down force. But it can't be too less otherwise the under tray could hit the ground. On the other hand if it's too large there will not be enough pressure difference to create enough down force. Due to the literature the best ground clearance is 30 mm.

The main additional part of undertray is a diffuser. The undertray diffuser is shaped section of the car underbody to improve the aerodynamic properties of the car. There is a nozzle that increases the velocity of air under the vehicle, a throat where maximum velocity is reached and a diffuser where the air is slowed back down to free stream velocity. The diffuser outlet angle is an important factor as well. The increase in the outlet angle will result in a bigger expansion area and hence greater pressure recovery is possible, hence drag will be reduced. But if the outlet angle is too large, it might lead to flow separation which will lead to a turbulent flow and hence increase in drag. Mainly it is used to generate maximum down force corresponding to minimum drag. Firstly it's needed to change the outlet angle (from 4 to 16 degrees) and looked at changes in flows and pressure distribution on the undertray using ANSYS 2D modeling.

The best results were obtained when the outlet angle was 10 degrees (Figure 17). The next step is a variety of diffuser positions.

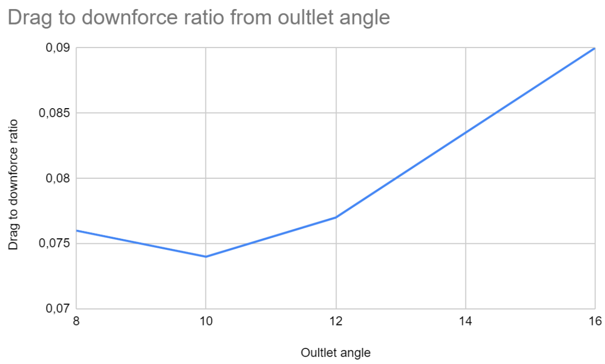


Figure 21 - Drag to downforce from outlet angle

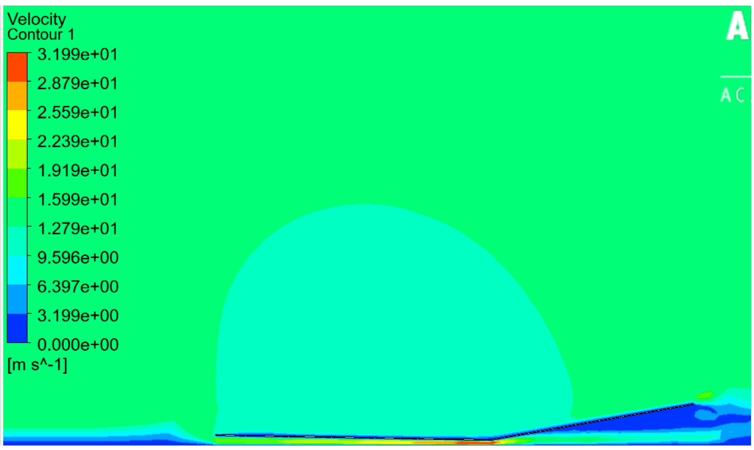


Figure 22 - CFD simulation of the undertray shape with outlet angle 10 degree and inlet angle 1 degree

As we obtained the diffuser shape we designed the 3D model for the undertray in Solidworks (Figure 18). In this design, the diffuser was added in the format of double side additions to the body. This decision was made due to the low angle of the chassis, less than the optimal 10 degrees. This solution is not the best. To improve the downforce, it is necessary to adjust the shape of the body of the machine with an outlet angle of 10 degrees. Side wings were added to distinguish between the flows and their removal from the body. And a protective lift in front of the rear wheel helps to prevent the flow of the rear wheel. We save the distance between the wheels and our rounding as 75 mm due to the rules.

Also inlet angle plays a very important role in the performance of an undertray diffuser. It acts like a nozzle and directs air below the under tray. And we need to work on it with the chassis-team. This nozzle increases the velocity of the air when it enters below the undertray. If this angle is very less, there will not be a sufficient increase in velocity. On the other hand if it is too large, it will be very difficult to match the desired conditions at the outlet as it will require the outlet area to be very large. We suppose the angle of our body is ok, but we need to work on it more.

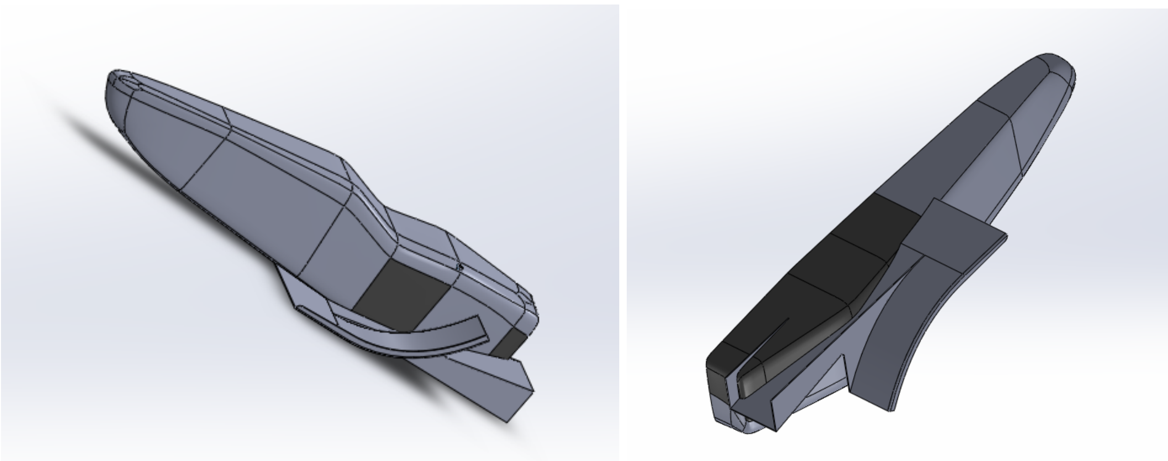


Figure 23 - 3D model of the undertray

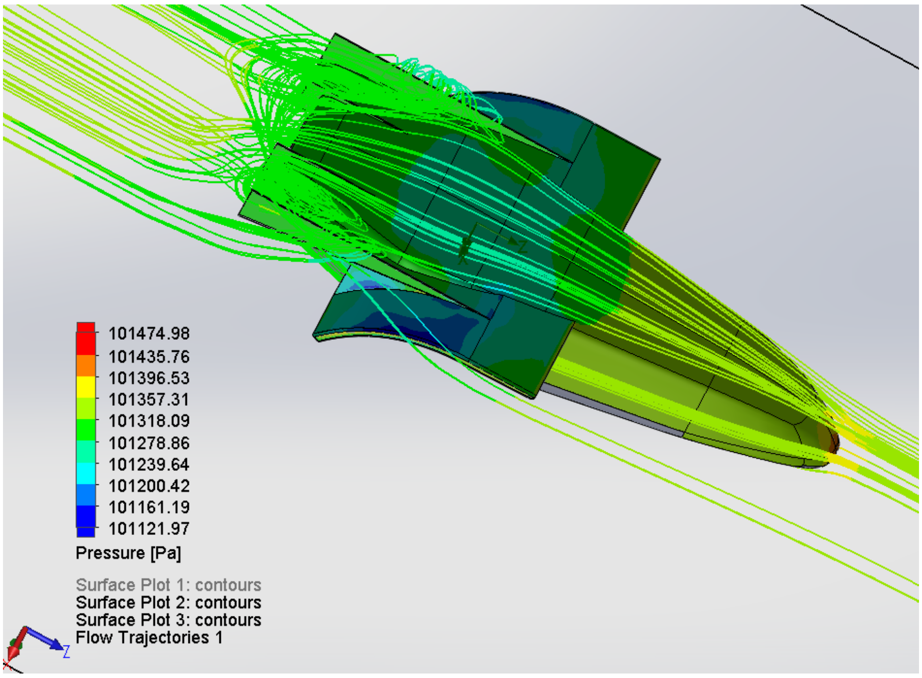


Figure 24 - 3D simulation results

The results of 3D modeling. Velocity of 50 km/h was used as velocity of the body because on average the car participating in Formula Student events travel at an average velocity of 50 km/h. Therefore it was important to understand the performance of the under tray at this velocity.

Final down force : 14.1 kg. Drag: 1.03kg

2.3.7 Vehicle Surface Modeling

The idea is to draft a model frame to perform the CFD simulation and optimize the aerodynamic devices. The 3D model is prepared based on dimensions provided by the chassis team and following the Formula Student Rule Book 2020. Refer to rules T 8.2.4.

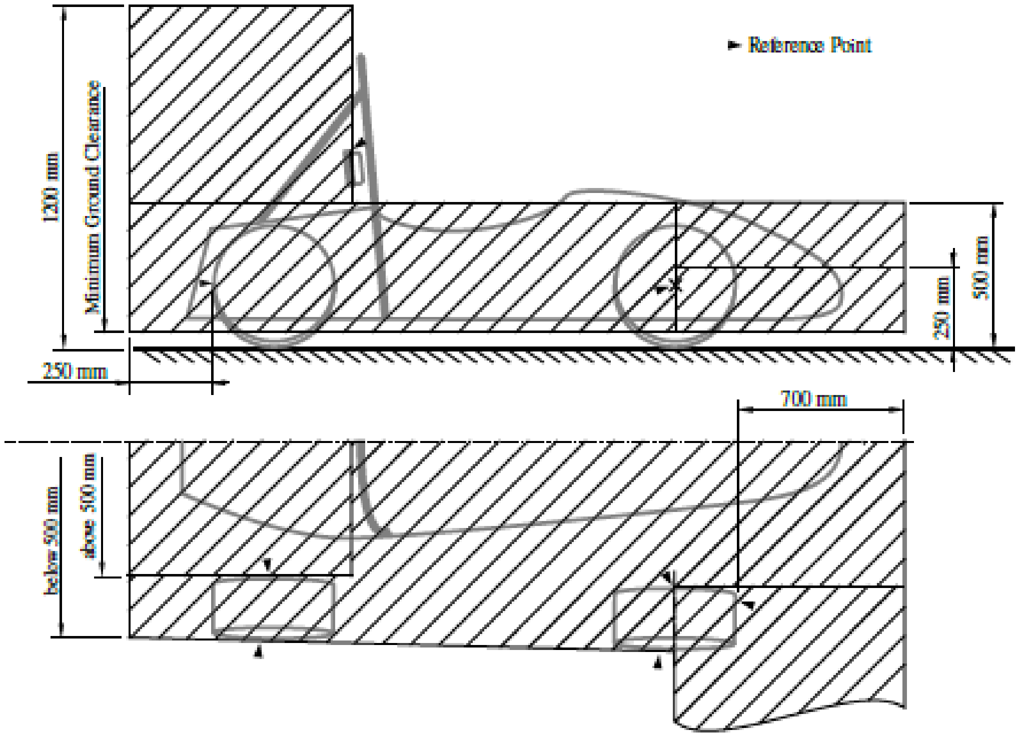


Figure 25: *Reference: Formula Student Rules 2020

Huge amount of time was spent in cooperative work with the chassis team, in order to satisfy rule constraints dimensions. We went approximately through 4 main iteration stages and dozens of little ones. Below you can see pictures of frames used to generate body design.

The main trouble was to set the dimensions for the cockpit and not to over- decrease or increase its size. As a result we successfully connected our models.

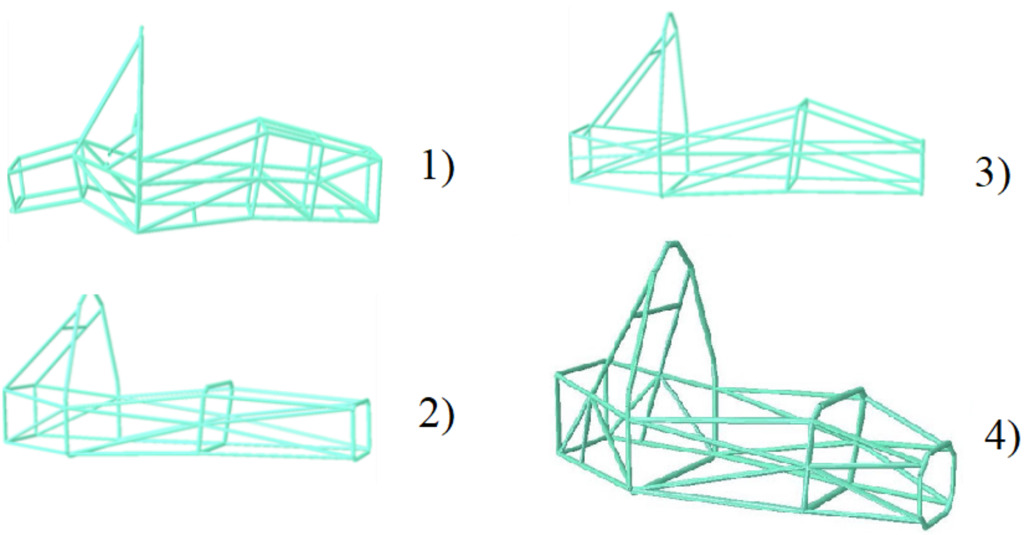


Figure 26: Chassis design iterations

Below, on figure 27 you can see the result of the design of the bodykit for the first iteration of the roll cage model. Based on the preliminary iteration by the chassis team, the surface modeling was computed in SOLIDWORKS. The dimensional limitations were considered from the rule book, and many assumptions went into the initial model. Surface weld-mets were neglected in the first run. It is clear that that iteration process was necessary.

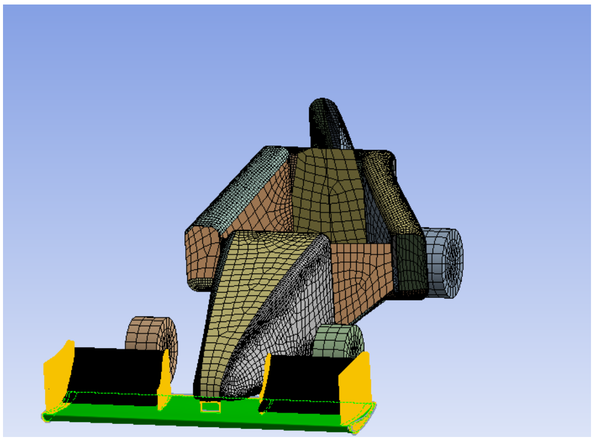


Figure 27: Preliminary Aerodynamic 3D Body

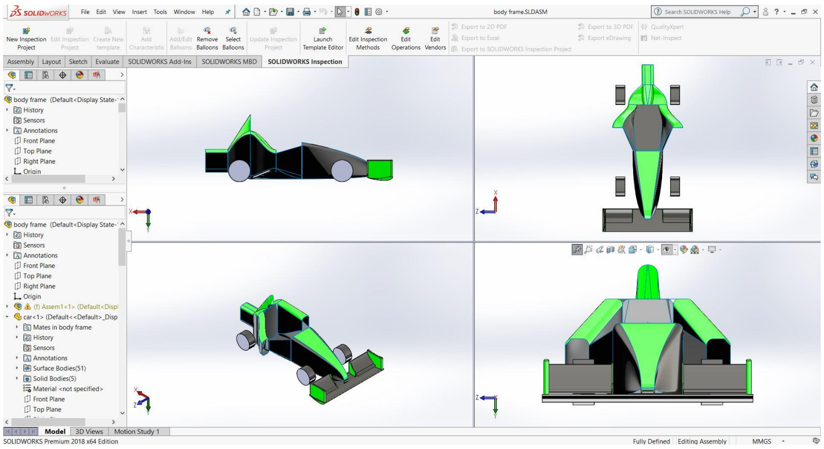


Figure 28: 3D Body frame prepared using Solidworks

After the body and chassis dimensions were finalized we assembled all the parts (body, front wing, rear wing and undertray) together in one full aerodynamics kit. For the simulation we added wheels as well as a cage that goes above the driver. Also we have updated the front wing design. We had 3 successful simulations of the full body. We started with the model shown below. One can see that the air curves right into the rear wheel and we do not like it because it creates unnecessary drag. the results for the forces on the body and both wings are shown in the table below the pictures. This model got 10.9 total drag and 32.8 total downforce. One can see in the table that body downforce is very low relative to the wings. That is what motivated us to change front wing a little bit in the next iteration.

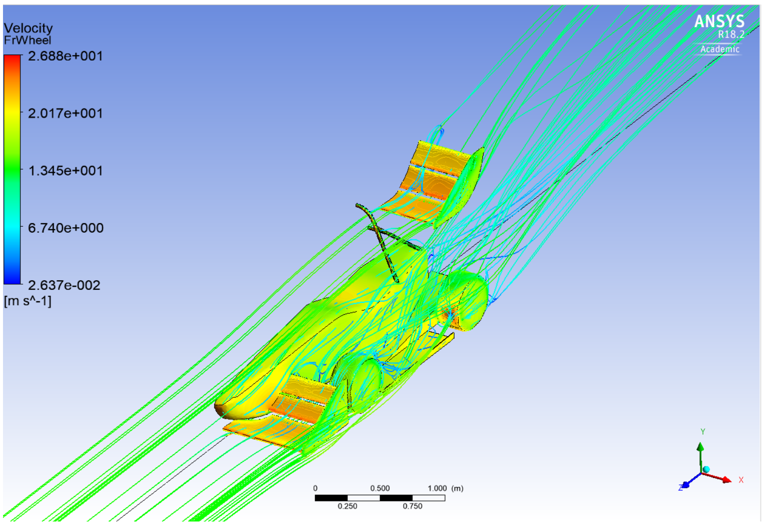
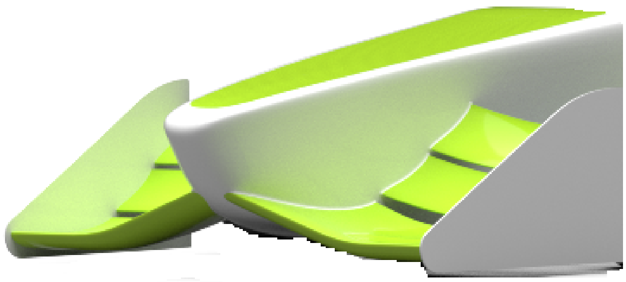


Figure 29 - 3D simulation of the whole car

In the second iteration we added a wing around the rear wheel to guide the air above the wheel. Also we made some interesting changes to the front wing as shown in the first picture on figure below. We added curvature to the end plate to guide the air around the wheels and we added curvature to the airfoils to split the air so that one part goes to the side outside the car and the other part goes close to the body. We are directing the air closer to the body because in the future we want to implement the active cooling by installing the air intakes on the sides. Air guided by the front wing to the body will be caught by the air intakes and guided to the batteries installed at the back of the car. This model has 13.3 kg drag and 35.6 kg downforce. Lift to drag coefficient is worse than in the second model, however here we have twice increased the body downforce and guided the air into the desired area.



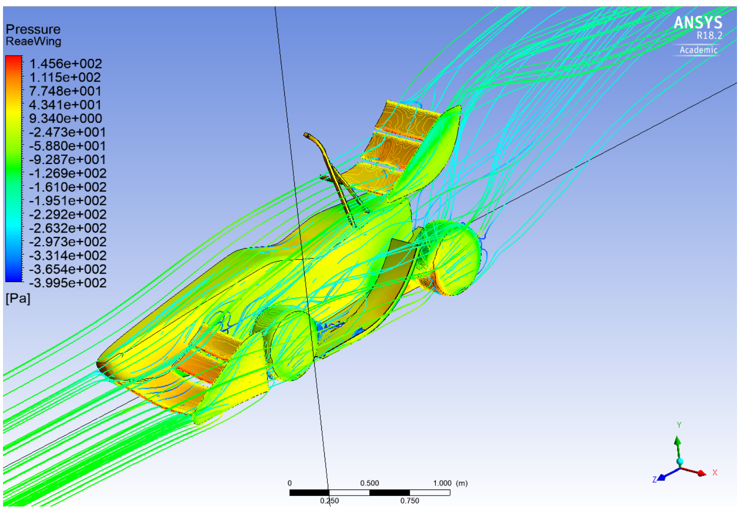


Figure 30 - 3D simulation of the whole car

In the picture below we can still see that some air hits the rear wing so we decided to add the endplate to the side wing. The final design for now is shown in the picture below in figure 27. For this model we got 13.07 kg drag and 36.02 kg downforce. We increased the lift to drag coefficient due to reducing the drag of the body. Therefore, the end plate indeed helped us.

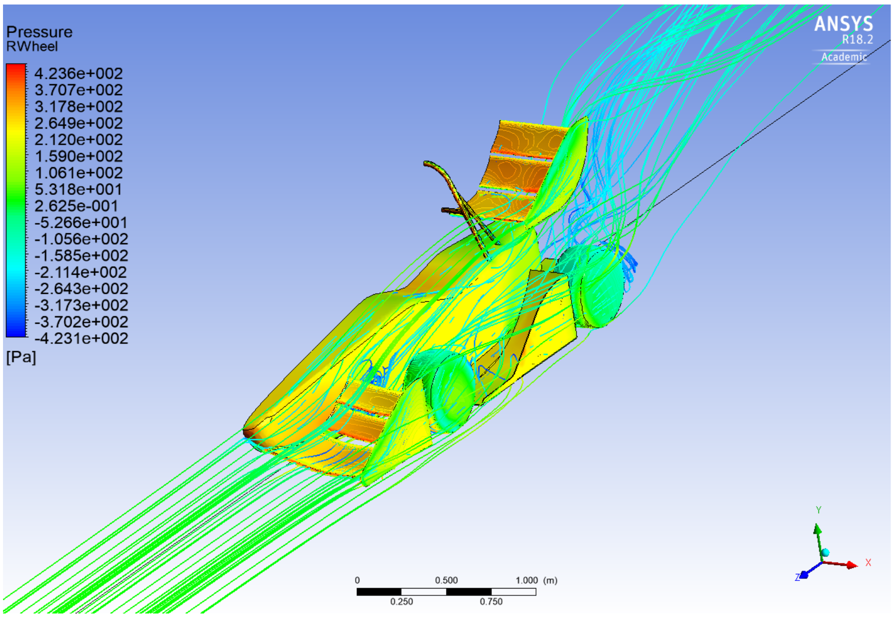


Figure 31 - 3D simulation of the whole car

There we just want to show the pretty final assembled model that our team actually assembled.

Figure 32 -Design and surface modeling SK FSAE car

3. Future Scope

- **Passive cooling system -** We are already designing our body with the idea of passive cooling for batteries and motor. However together with other teams we have to decide the placement of the power systems, because now we have no clue where to lead the air.
- **Active aerodynamics actuation -** To implement active aerodynamics we have to perform parameterization of the rear wing for 2 states (breaking and accelerating). We also create the model and simulate the system with servos. Then we try it on the full car simulation and see the effects and work on the improvement.
- **Undertray -** To improve the downforce using adjusting the shape of the body of the machine with an outlet angle of 10 degrees like a diffuser. Add vortexes in the diffuser to increase the angle of the diffuser without loss and, consequently, increase the downforce.
- **Manufacturability** - The design is in complete compliance with the aerodynamics. The team understands the complexity of such structures. It looks forward to working closely in association with the manufacturing team to reiterate some sections considering the cost and material availability.
- **Prototyping** - The team aims at prototyping the model to replicate the wind tunnel testing. Results from physical modeling will help in understanding the deviation from the computer simulations.

References:

1) Wordley, S., & Saunders, J. (2006). Aerodynamics for Formula SAE: Initial Design and Performance Prediction. SAE Technical Paper Series. doi:10.4271/2006-01-0806
2) Tyagi, A., & Madhwesh, N. (2017). Design and Numerical Analysis of an Under Tray Diffuser of a Formula Student Car for Performance Improvement. SAE Technical Paper Series. doi:10.4271/2017-01-5016
3) Desai, S., Leylek, E., Lo, C.-M. B., Doddegowda, P., Bychkovsky, A., & George, A. R. (2008). Experimental and CFD Comparative Case Studies of Aerodynamics of Race Car Wings, Underbodies with Wheels, and Motorcycle Flows. SAE Technical Paper Series. doi:10.4271/2008-01-2997
4) Wordley, S., & Saunders, J. (2006). Aerodynamics for Formula SAE: A Numerical, Wind Tunnel and On-Track Study. SAE Technical Paper Series. doi:10.4271/2006-01-0808
5) Jurij Iljaž – Leopold Škerget – Mitja Štrakl – Jure Marn, University of Maribor, Faculty of Mechanical Engineering, Slovenia. Optimization of SAE Formula Rear Wing (2016)
6) PAKKAM, SRIRAM SARANATHY. High Downforce Aerodynamics for Motorsports.
7) Aravind Prasanth, Sadjyot Biswal, Aman Gupta, Azan Barodawala Member, IAENG. Complete Design and Optimization of the Aerodynamics of a FSAE Car using Solid works ANSYS & XFLR5 (2016)