Vehicle Design Aerodynamics

The Speedline

Kang Yang

1. Summary
2. Sketch Views
3. Initial Concept
4. CFD Analysis of the Final Concept
5. CFD Validation
6. Future Improvements

Summary:

The importance of vehicle aerodynamics is indicated in the history itself. Not only it improves the efficiency of the vehicle, but also it reduces fuel consumption.

The analysis of the sports car Speedline with different shapes and add-on aerodynamic devices were studied using Computational Fluid Dynamics (CFD) simulation in this report. The lift force and drag force are calculated by default in CFD and the drag coefficient is calculated manually through this equation:

$$Cd = \frac{2F}{\rho v^2 A}$$

Where:
F is drag Force (N),
ρ is air density (1.225 kg/m^3)
v is air velocity (m/s)
A is the frontal area (m^2)

It has been found that different add-on devices can influence aerodynamic drag. To reduce drag, it is favorable that the flow is attached to the vehicle's body as long as possible. A streamlined body would result in less flow separation, which would cause less turbulence. A lift reduction of 55% and a drag reduction of 38% are observed from the final model. A device like a spoiler much reduces drag with less additional lift force required. Diffusers on the other hand produce downforce while reducing drag. In conclusion, it is always preferable to have a streamline body with proper optimization and results in improved vehicle aerodynamics.

Sketch Views:

Early sketch: The objectives of the sports car Speedline is travelling faster with good stability. The characteristics of this initial design is to have a smooth appearance.

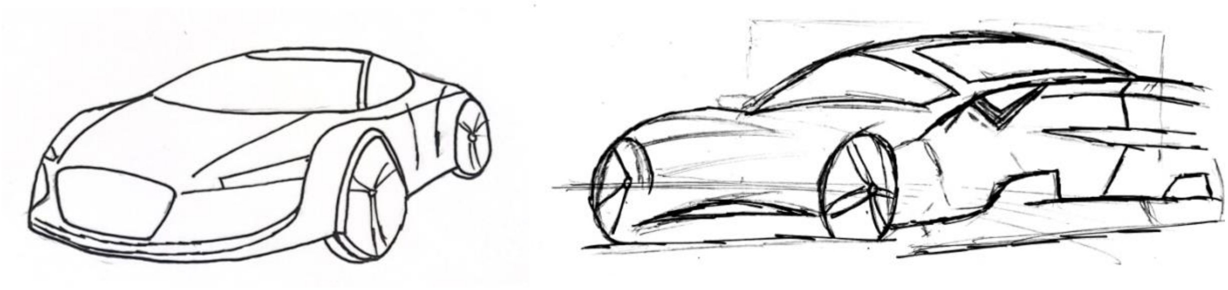


Figure1: Initial concept sketching

Final sketch: After doing CFD analysis of the initial concept (see page 3), the design is modified extensively from the inspiration of the streamlined body, which has the lowest drag coefficient among all shapes [1].

The outlook from the side of the car is similar to streamline. In order to echo this designing concept, the upper part adopts this shape to obtain better aerodynamics.

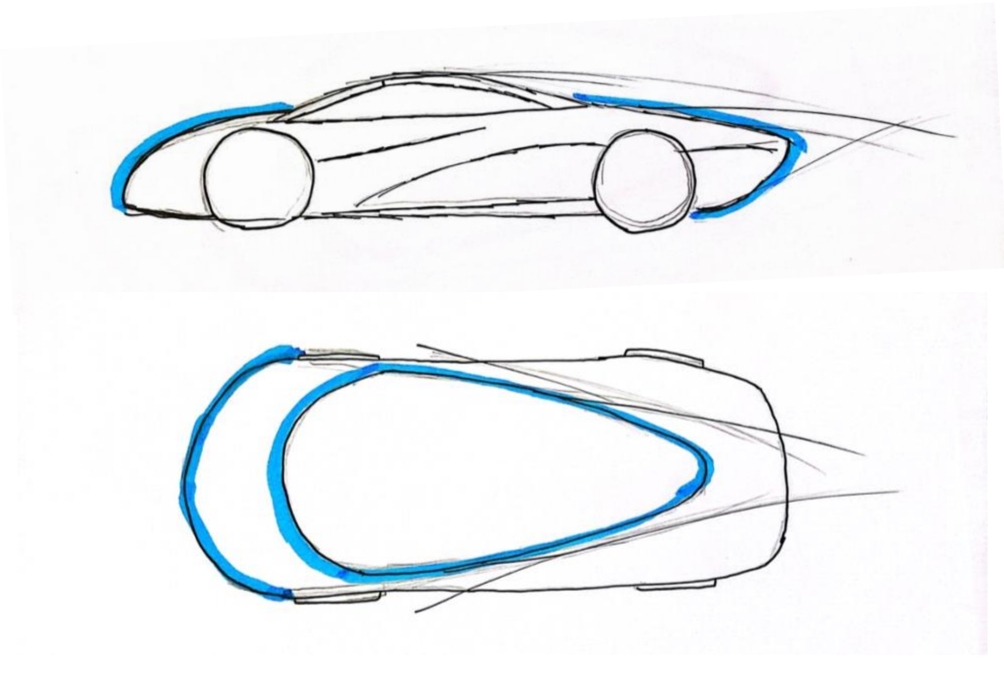


Figure2: Final concept sketching (major modification highlighted in blue)

Initial Concept:

After running a quick CFD flow simulation test (total cell count: 157870) for the initial concept, 3 goals are measured and listed below:

Name	Unit	Value
Lift Force	N	630.051
Drag Force	N	-433.763
Drag Coefficient	n/a	0.3936288

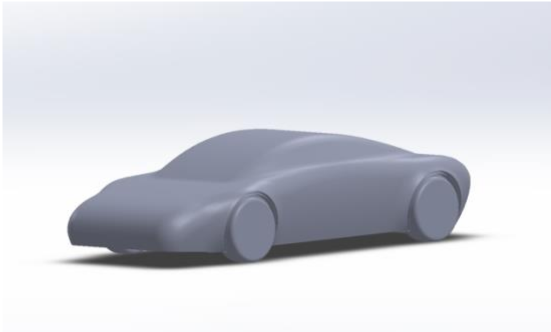
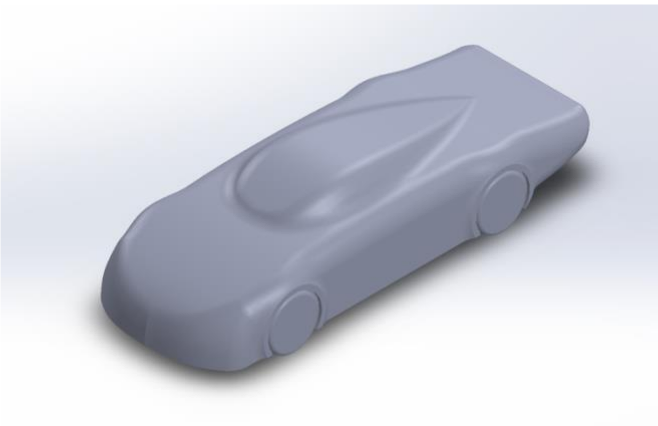


Figure3: CAD model of the initial concept

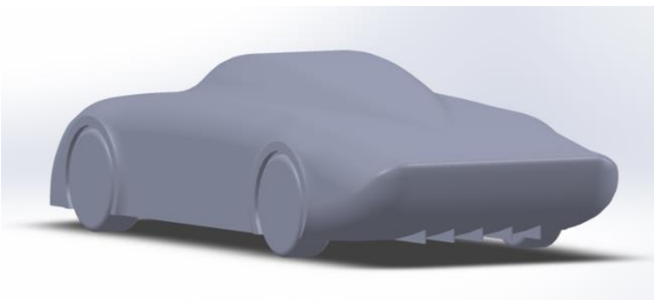
Extensive improvements need to be done to minimize lift force as well as decrease the drag coefficient within 0.3 to be qualified as an aerodynamic sports car.

Final Concept:

The spoiler is integrated into the rear of the car.



The shape of the upper body of the car is streamlined.



The diffuser is used in the car for better aerodynamical performance.

Figure 4: CAD model of the final concept

CFD Analysis of the Final Concept:

1. Vehicle Data
Frontal Area: 1.7310 m^2

2. Analysis of Mesh
Total Cell count: 777116
Fluid Cells: 777116
Fluid Cells containing solids: 7923

3. Additional Physical Calculation Options
Heat Transfer Analysis: Heat conduction in solids: Off
Flow Type: Laminar and turbulent
Time-Dependent Analysis: Off
Gravity: Off
Radiation:
Humidity: Off
Default Wall Roughness: 0 micrometre

4. Material Settings
a. Material Settings
Fluids: Air

b. Initial Conditions

Table 1 Ambient Conditions

Thermodynamic parameters	Static Pressure: 101325.00 Pa Temperature: 293.20 K
Velocity parameters	Velocity vector Velocity in X direction: 0 m/s Velocity in Y direction: 0 m/s Velocity in Z direction: -31.000 m/s
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 6.689e-04 m

c. Engineering Goals

Table 2 Velocity

Type	Global Goal
Goal type	Velocity (Z)
Calculate	Average value
Coordinate system	Global Coordinate System
Use in convergence	On

Table 3 Lift force

Type	Global Goal
Goal type	Force (Y)
Coordinate system	Global Coordinate System
Use in convergence	On

Table 4 Drag force

Type	Global Goal
Goal type	Force (Z)
Coordinate system	Global Coordinate System

Use in convergence	On

Table 5 Drag coefficient eq

Type	Equation Goal
Formula	(2*Drag Force)/(1.225*1.7310*(Velocity^2))*(-1)
Dimensionality	No units
Use in convergence	On

5. Results

Table 6 Outcomes

Name	Unit	Value	Progress	Criteria	Delta	Use in convergence
Lift Force	N	283.935	100	287.088	280.782	On
Drag Force	N	-268.626	100	-268.750	-269.441	On
Drag Coefficient	n/a	0.2578151	100	0.2585970	0.2573180	On

From the plot below, It is clearly showed there is less turbulence around the vehicle, especially at the end. It is advantageous for the flow to remain linked to the vehicle's body as long as possible in order to avoid drag. Reduced flow separation from a streamlined body would lead to less turbulence.

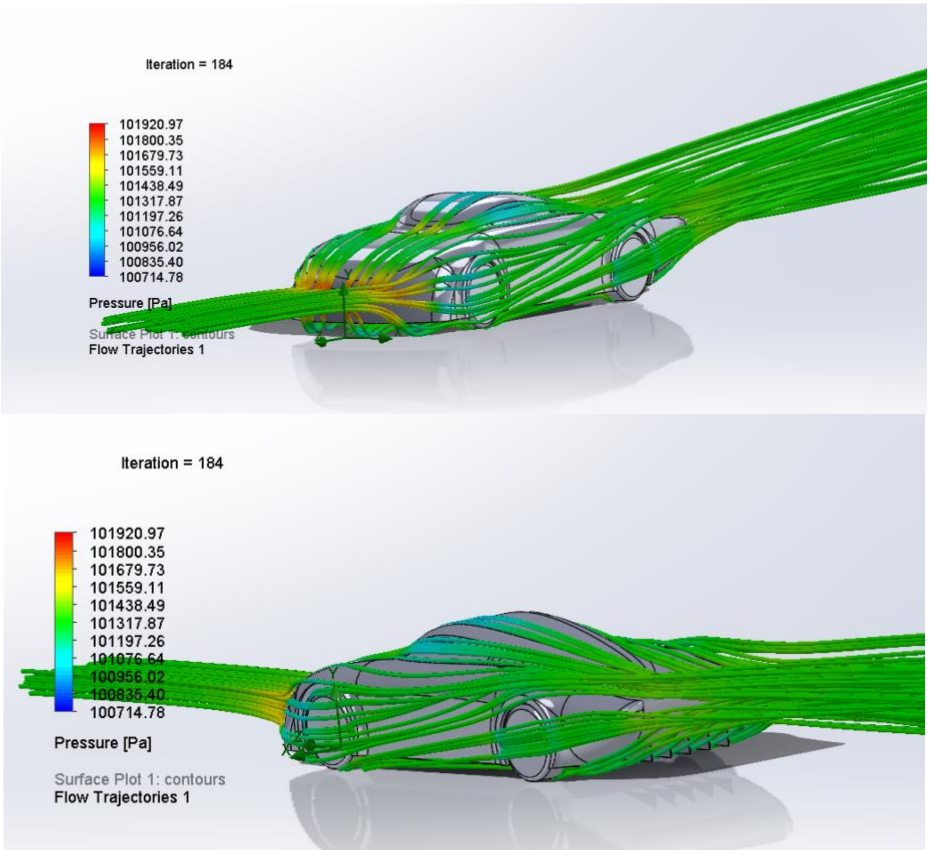


Figure 5: 3D flow trajectories plot

The recirculation zone is represented blue behind the vehicle. The less recirculation zone there is, the less turbulence is produced, which ultimately results in less drag. The recirculation zone for the final concept is much smaller than the initial model (see appendix), which is seen a nearly 38% reduction of drag force.

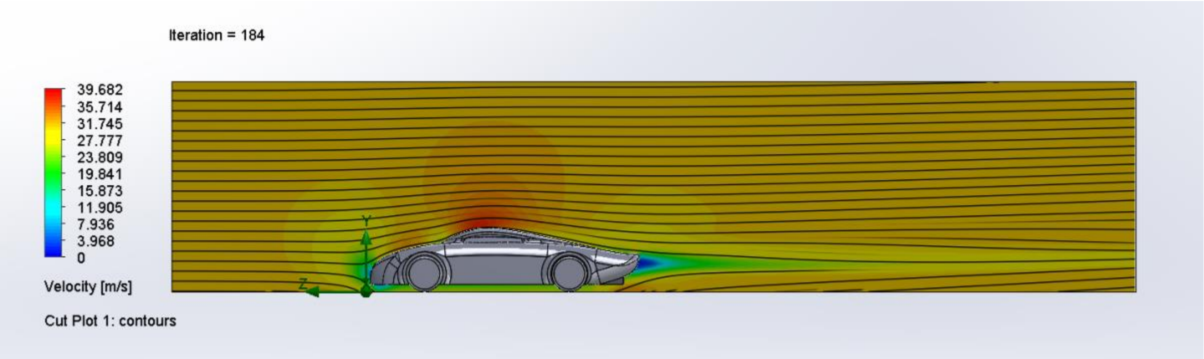


Figure 6: 2D velocity plot

The front end of the car results in the highest pressure region. The maximum pressure experienced at the front is decreased by 134 Pa comparing with the initial model.

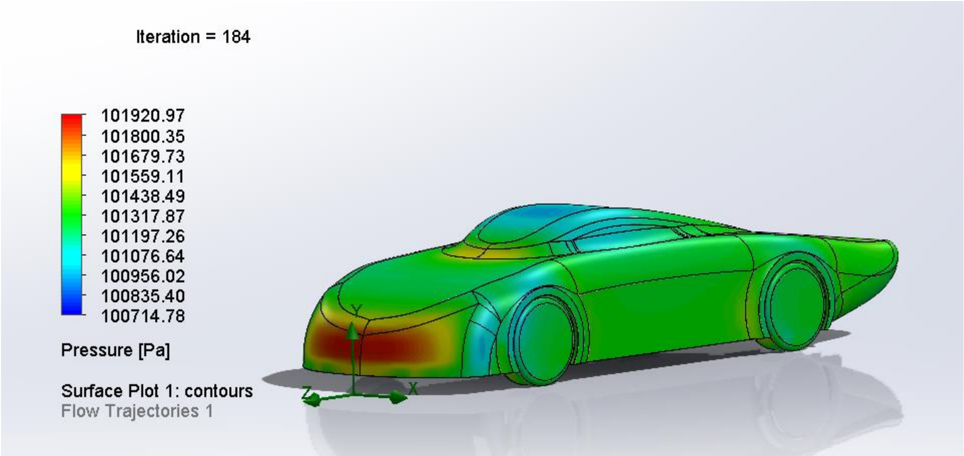


Figure 7: 3D pressure plot

The spiller is added to the rear of the vehicle (figure 8), which acts as an obstacle to air passage. This higher pressure acts upon the area of the deck to provide downforce (figure 9). The principle behind the spiller is based upon Bernoulli's principle which states that "a slow-moving fluid will exert greater pressure than the fast-moving fluid". With the corporation of the diffuser, spoiler results in lower pressure at the bottom side of the vehicle.

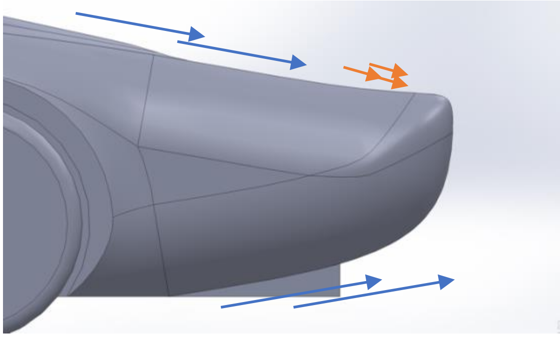


Figure 8: spoiler indication plot (high pressure in red, low pressure in blue)

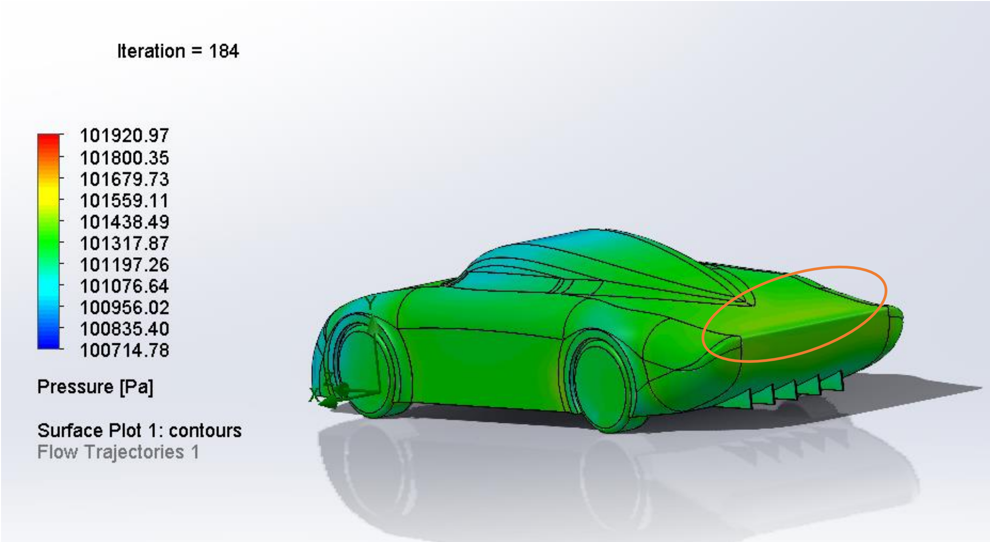


Figure 9: 3D pressure plot indicating spoiler creating higher pressure region

The diffuser has the function of accelerating the airflow under the vehicle, lowering its pressure while increasing the pressure differential between the vehicle's top and bottom surfaces, which resulting in down force. According to the article *"Drag reduction by application of aerodynamic devices in a race car"* [2], it concludes that vehicles have spoiler and diffuser generate much less drag and lift force comparing to other add on devices with the speed of 300kmph. As a result, although the advantage of adding diffuser for the vehicle is not obvious for the set speed, it will reduce air turbulence and drag at higher speed to a large extend.

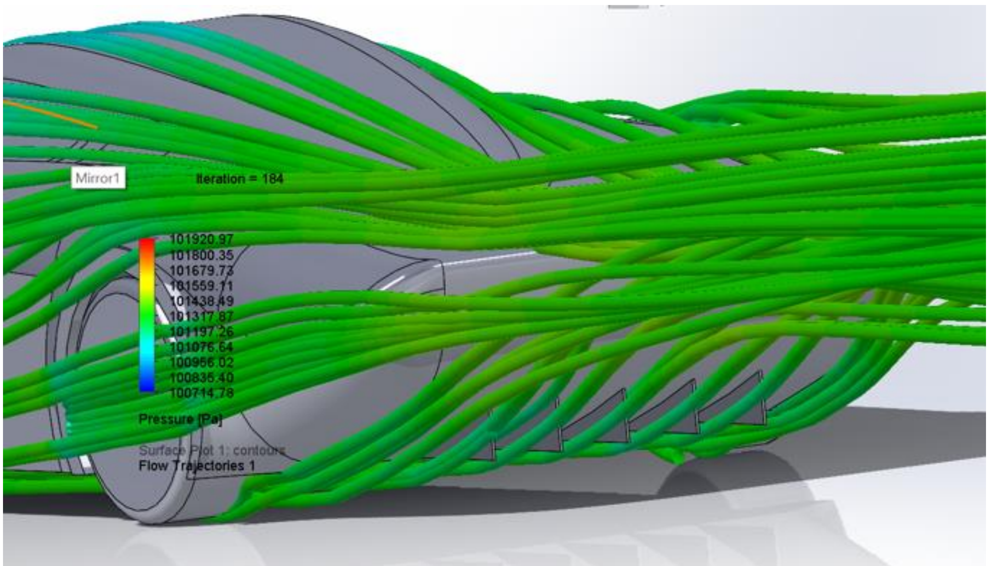


Figure 10: 3D flow trajectories plot

CFD Validation:

The CFD outputs of the Speedline are validated by comparing them to the wind tunnel test in the *SolidWorks Flow Simulation Project Sports car model report* [3].

Agreement:

The front end and roof of Speedline are much similar to the tested sports cars, which results in similar velocity distribution.

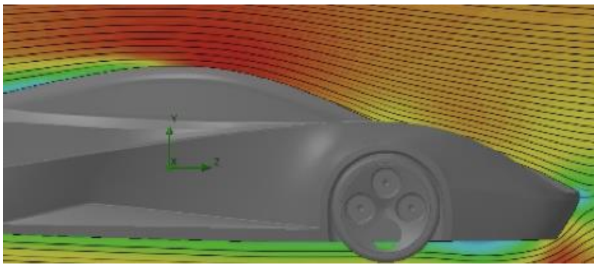


Figure 11: 2D velocity plot of the sports car

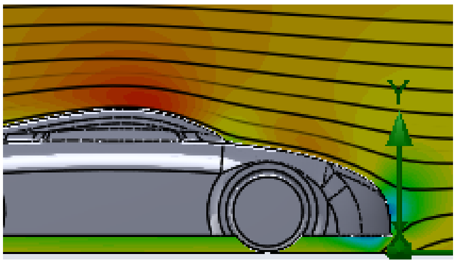


Figure 12: 2D velocity plot of Speedline

Disagreements:

The recirculation zone

From the plot below, it is clearly shown that there are some recirculation zone at the rear of the vehicle, which results in turbulence.

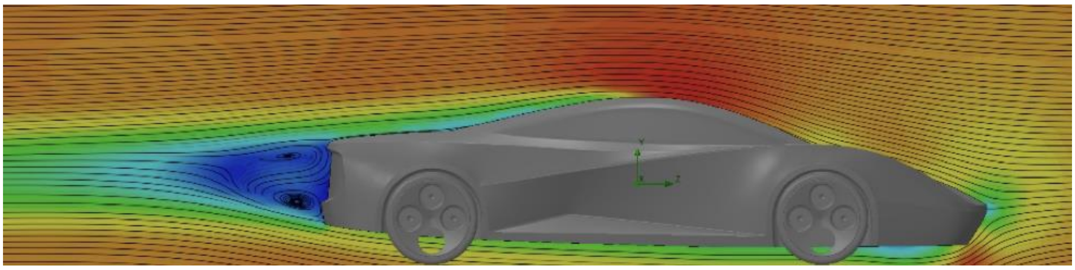


Figure 13: 2D velocity plot of the sports car

The wind tunnel test shows much more turbulence at the vehicle's rear, leading to more drag and affecting the vehicle's drag coefficient. The wind test results show that the drag coefficient is increased by 0.1 compared with its CFD results.

Figure 14: Side view of the sports car in the wind tunnel

Benefiting from the streamlined shape of the Speedline, the rear creates a relatively small recirculation zone, which will minimize the effect of unpredictable turbulence in the wind tunnel test.

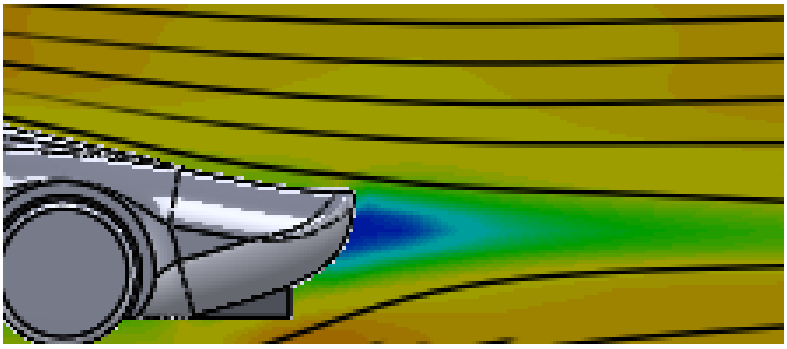


Figure 15: Close view of the real of Speedline

The chassis

Although the flow of air in the CFD is being seen as less turbulence, there is nonnegligible turbulence happening at the chassis and even more turbulence at the end of the chassis.

With the diffuser adds on to the Speedline, there will be less turbulence when conducting wind tunnel tests as it provides additional lead to the airflow.

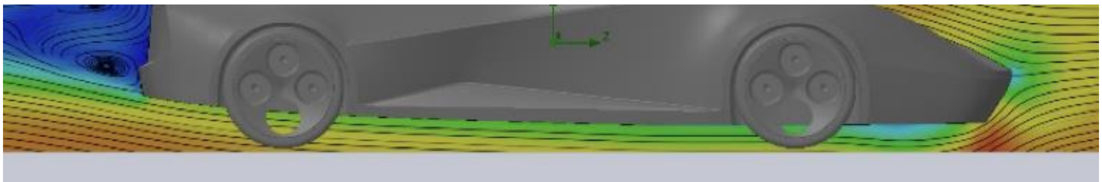


Figure 16: Close view of the chassis of the sports car

Figure 17: Close view of the chassis of the sports car in the wind tunnel

Conclusion:

The most significant difference in the drag coefficient between CFD and wind tunnel tests are results from the recirculation zone. It is unavoidable to have a certain extent of increase in the drag coefficient. Still, with the contribution of the streamlined shape and diffuser, the Speedline will have a minor increase in drag coefficient than the tested sports car in the wind tunnel test.

Future Improvements:

Changing the flat shape at the front of the vehicle to a hard edge shape will force the airflow to separate, which reducing high-pressure region and further reducing drag force.

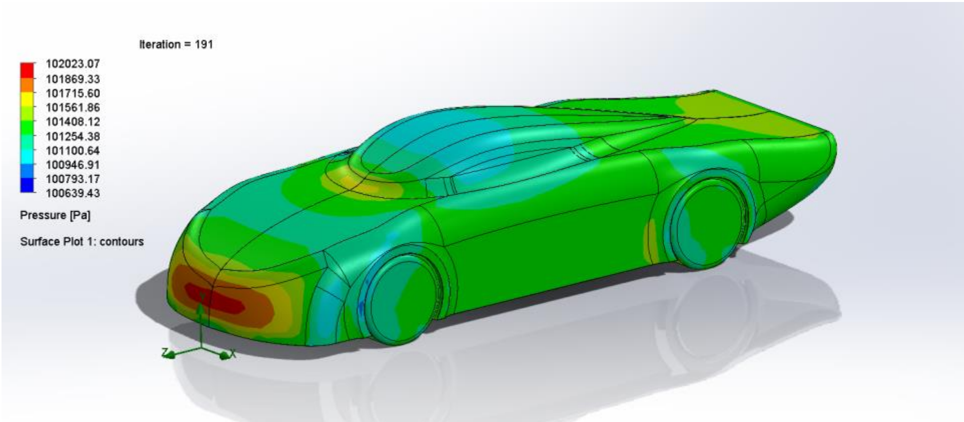


Figure 17: 3D pressure plot

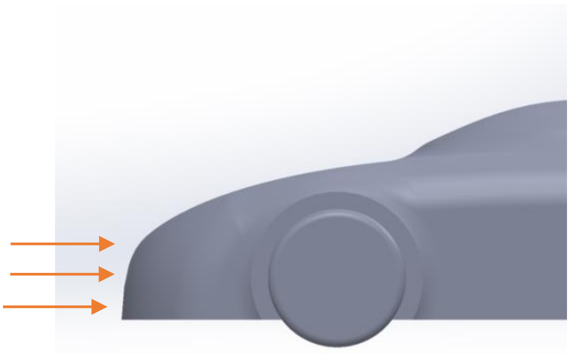


Figure 18: Current airflow direction

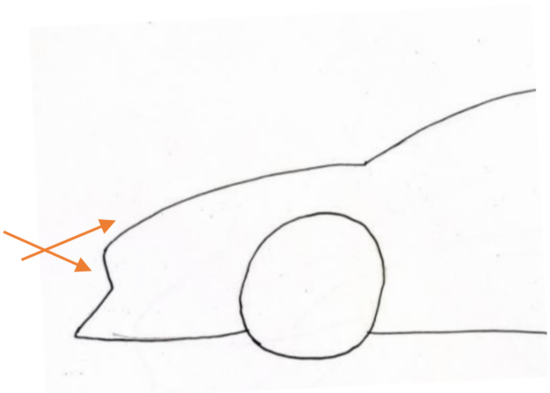


Figure 19: Airflow direction after redesigning the front shape

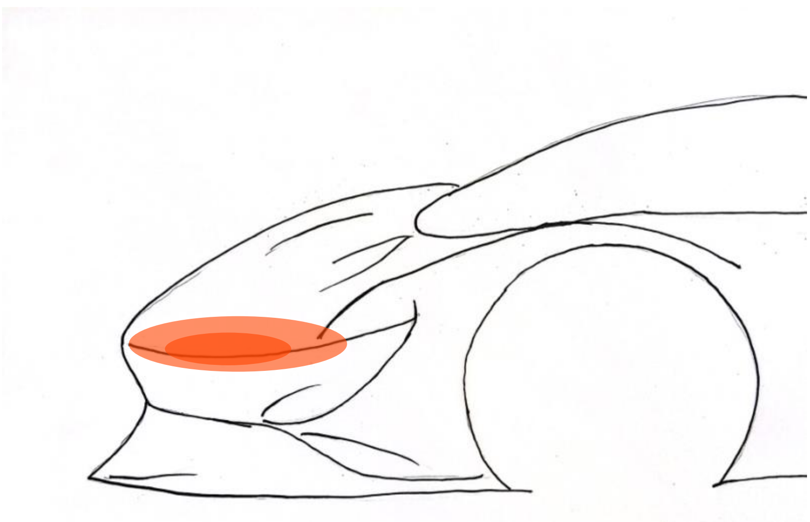


Figure 20: Future front outlook of the Speedline

Appendix:

[1]: Wikimedia Foundation. (2022, October 2). *Drag coefficient*. Wikipedia. Retrieved November 17, 2022, from https://en.wikipedia.org/wiki/Drag_coefficient

[2]: Nath, D. S., Pujari, P. C., Jain, A., & Rastogi, V. (2021, January 28). *Drag reduction by application of aerodynamic devices in a race car - advances in aerodynamics*. SpringerOpen. Retrieved November 17, 2022, from https://aia.springeropen.com/articles/10.1186/s42774-020-00054-7

[3]: Wang, J., Li, H., Liu, Y., Liu, T., & Gao, H. (2018, February 26). *Aerodynamic research of a racing car based on wind tunnel test and Computational Fluid Dynamics*. MATEC Web of Conferences. Retrieved November 17, 2022, from https://www.matec-conferences.org/articles/matecconf/abs/2018/12/matecconf_icmme2018_04011/matecconf_icmme2018_04011.html

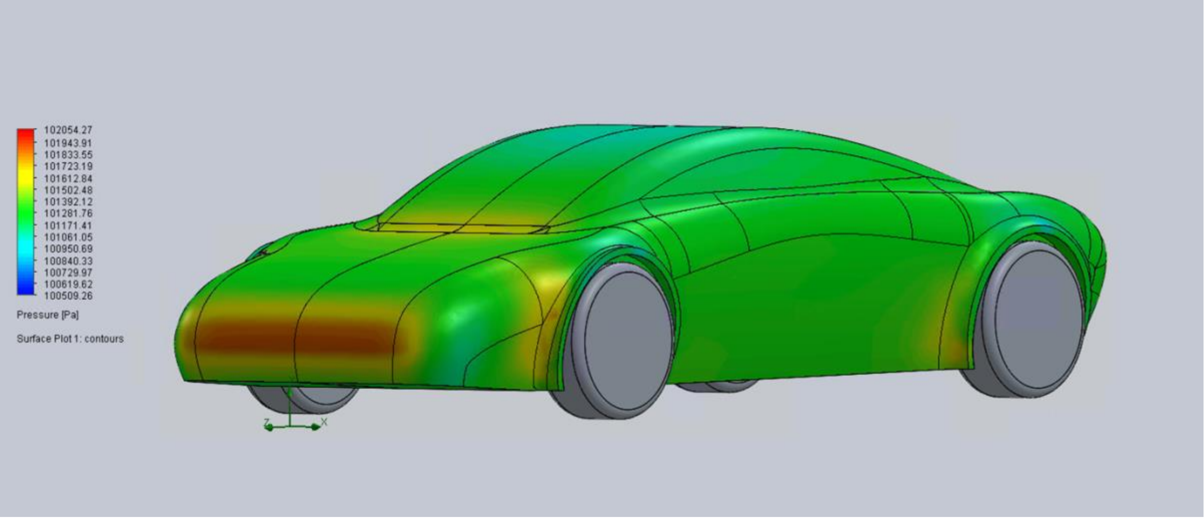
[4]: Asmedigitalcollection.asme.org. (n.d.). Retrieved November 17, 2022, from https://asmedigitalcollection.asme.org/fluidsengineering/article-abstract/137/8/081104/374123/Experimental-and-Numerical-Aerodynamic-Analysis-of?redirectedFrom=fulltext

[5]: Chowdhury, H., Loganathan, B., Mustary, I., Moria, H., & Alam, F. (2017, April 22). *Effect of various deflectors on drag reduction for trucks*. Energy Procedia. Retrieved November 17, 2022, from https://www.sciencedirect.com/science/article/pii/S1876610217302151?via%3Dihub

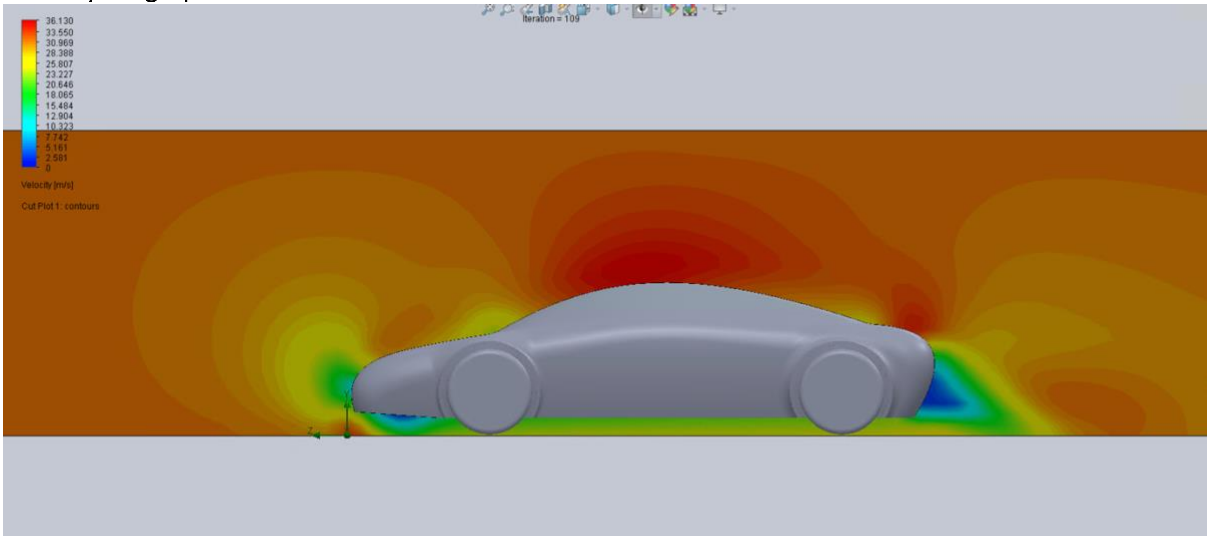
First model:

Name	**Unit**	**Value**
Lift Force	N	630.051
Drag Force	N	-433.763
Drag Coefficient	n/a	0.3936288

Pressure 3D graph:



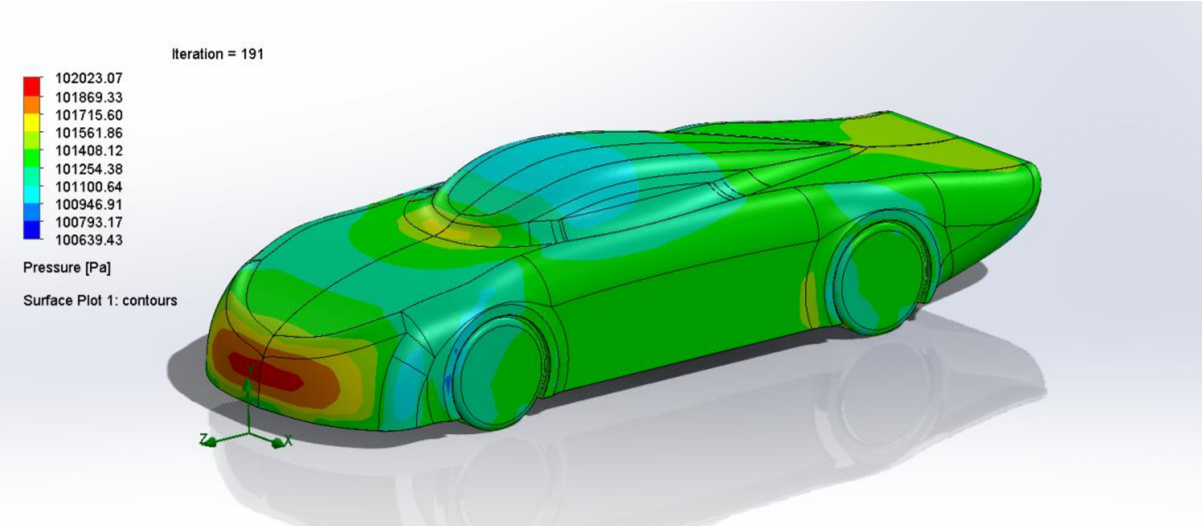
Velocity 2D graph:



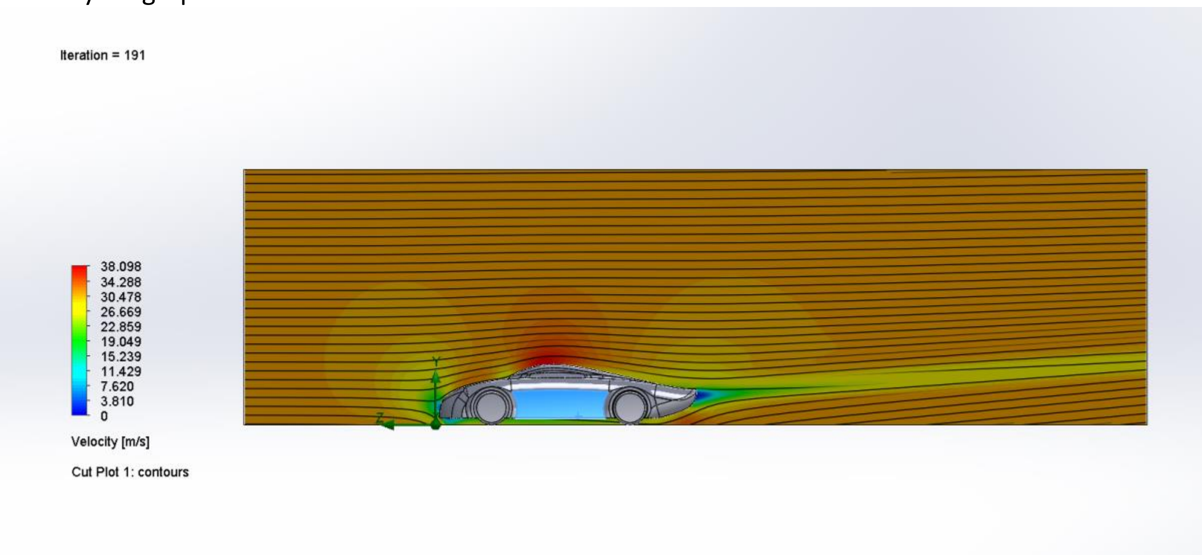
Second model:

Name	Unit	Value
Lift Force	N	285.710
Drag Force	N	-271.140
Drag Coefficient	n/a	0.2602278

Pressure 3D graph:



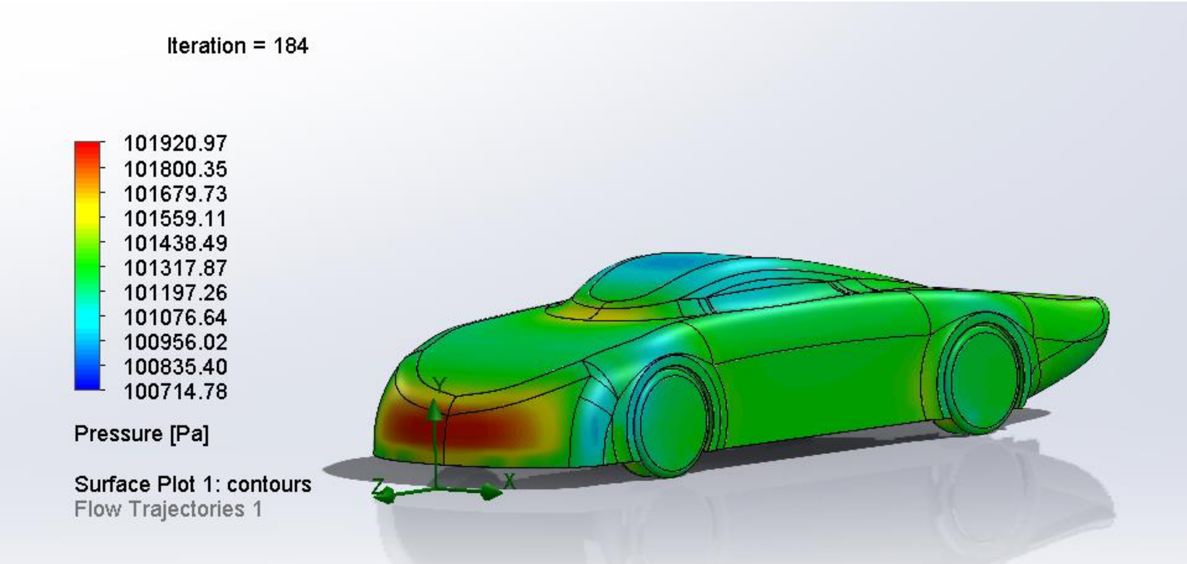
Velocity 2D graph:



Third model:

Name	Unit	Value
Lift Force	N	283.935
Drag Force	N	-268.626
Drag Coefficient	n/a	0.2578151

Pressure 3D graph:



Velocity 2D graph: