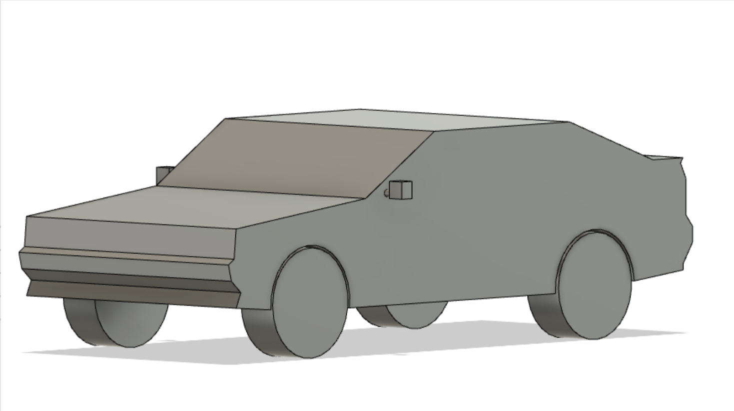
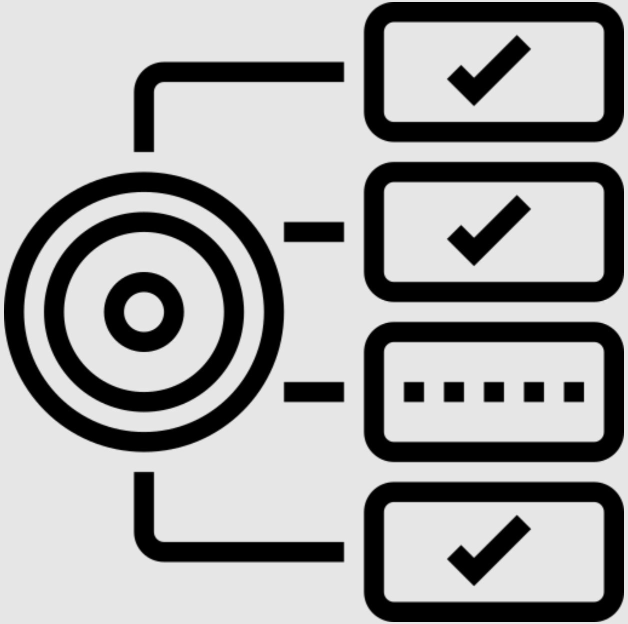
Modern Car CFD Analysis

Isis-Gabrielle Rodney-Jones (2168130)
Mahmood El-Mahalawy (2295059)
Loungo Kukama (2340697)





Project Brief and Aims

Project Brief:

- A vehicle’s overall aerodynamics are greatly influenced by its shape and design.
- Vehicle efficiency and stability are increased when air resistance is decreased via the use of aerodynamically optimised and streamlined designs.
- Modern road vehicles are continuously being designed to make flow separation occur more easily thus reducing turbulent flow. Turbulent flow causes a lot of wake and lowers the various attributes of the vehicle such as speed.

Aim:

Using the modifications there will be a concentration on lift and drag, how reducing them will help increase speed:

- To use CFD analysis to improve the aerodynamics of the car
- Ensure each version of the car is able to undergo CFD analysis

TABLE OF CONTENTS

01

Designing for CFD

01 Designing for CFD - Technical Geometry

Table 1: Parameters of the car

Mini Cooper Parameters	Values
Length	4404 mm
Width	1722 mm
Height	1346 mm
Gross weight	1840 kg
Average speed	65 km/hr

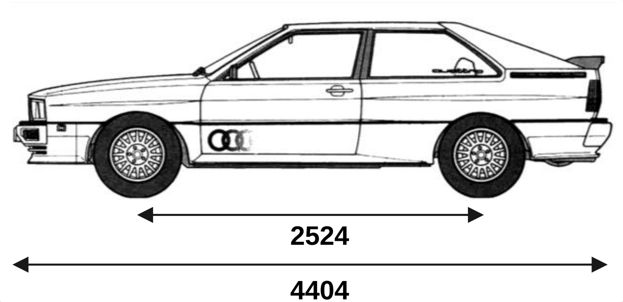


Figure 1: Side view displaying length of car

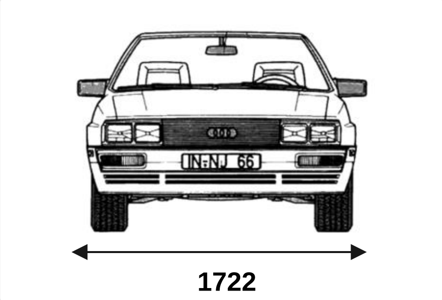


Figure 2: Front view displaying width

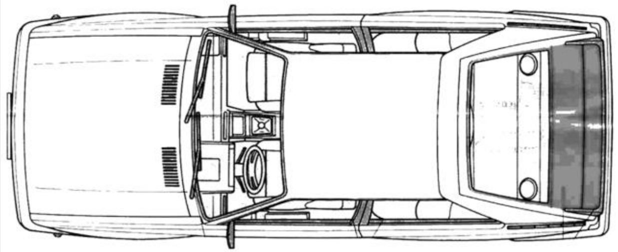


Figure 3: Top view of car

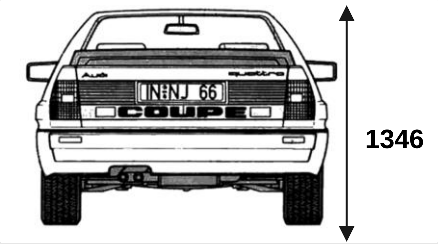


Figure 4: Rear view of car displaying height

After evaluating multiple design options, we chose a car (Audi Quattro) which fascinated us as a group and expected to deliver results from the CFD software (Ansys Workbench CFX).

[1] The parameters in figure 1 were followed to ensure that our CAD design replicates the Audi Quattro as close as possible.

[2] As aerodynamics doesn't influence the car until 60 km/hr we decided on using 65 km/hr when deciding on the speed that hits the boundary layer.

[1] Car.info. (n.d.). Specifications for Audi quattro. [online] Available at: https://www.car.info/en-se/audi/quattro/specs [Accessed 21 Nov. 2023].
[2] www.volvotrucks.com. (n.d.). How aerodynamics and rolling resistance impact your truck's fuel consumption. [online] Available at: https://www.volvotrucks.com/en-en/news-stories/insights/articles/2019/oct/how-aerodynamics-and-rolling-resistance-impact-your-trucks-fuel-consumption.html#:~:text=If%20you%20double%20your%20speed.

02

CFD Parameters and Justification

02

CFX PARAMETERS USED AND JUSTIFICATIONS

Table 2: Wind Tunnel conditions

Wind tunnel(enclosure) Parameters	Value
Total length	6 m^2
Width	5 m^2
Height	2.5 m^2
Distance between wheel and floor of tunnel	5 mm

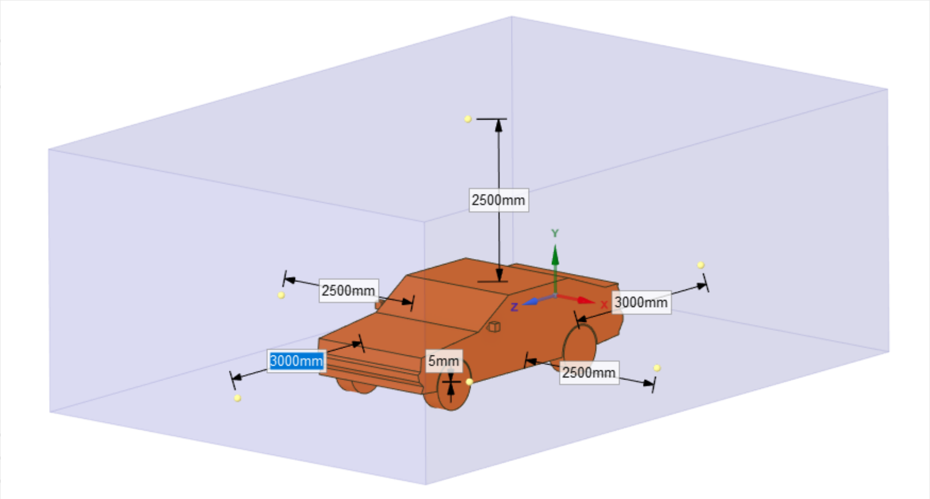


Figure 5: Wind Tunnel conditions

When deciding on making the enclosure, key design considerations were based on the boundary layers. For the front and back of the enclosure we made the lengths 3000mm as they are the inlet and outlet boundary layers, respectively.

The sides and top of the enclosure are 2500mm and are the same length as they form the opening boundary layer.

The distance between the floor and bottom of the car was set at 5mm as it reduces ground clearances, reduces the amount of air that flows beneath the car giving more accuracy on recreating how a car drives.

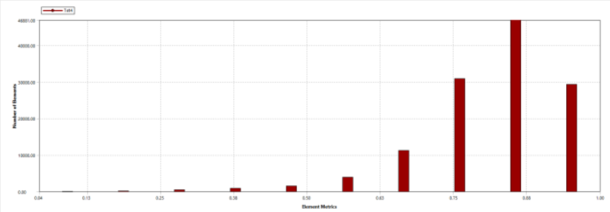
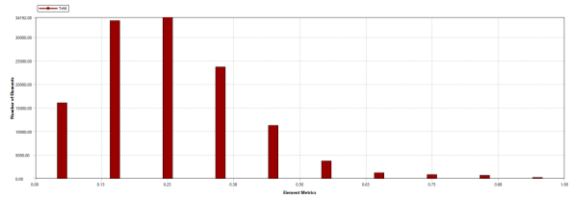
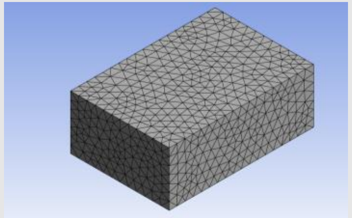
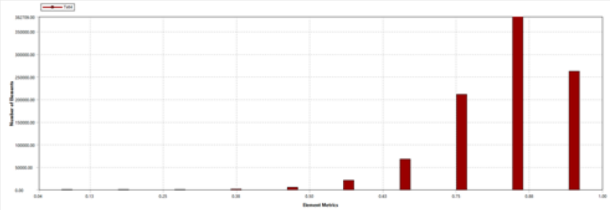
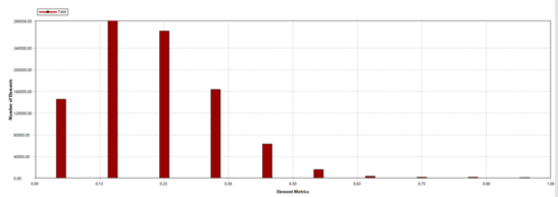
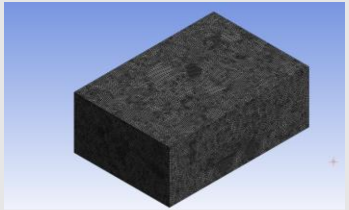
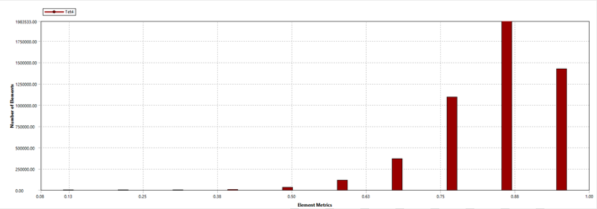
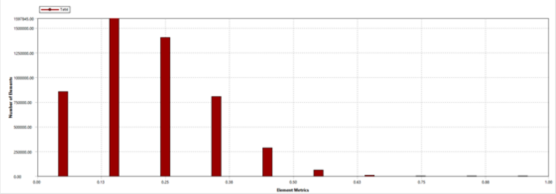
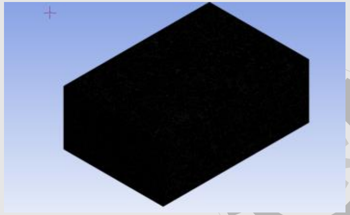
[3] We wanted to test under realistic conditions hence we used a wind tunnel. We made the enclosure to the values in table 2 to do so. The wind tunnel can recreate the action of the car driving by moving the air around the car.

Table 3: Table of fluid properties

Material properties	Value
Fluid flow	Air
pressure	1 atm
Heat transfer model	Thermal
Fluid temperatures	25 degrees
Type of flow	Steady state

02 Meshing Composition

Table 4: Histograms of element quality and skewness for each mesh size

Size of mesh	Element quality	Skewness	Isometric View
0.6			
0.1			
0.05			

https://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node167.htm

02 Boundary layer Parameters

1

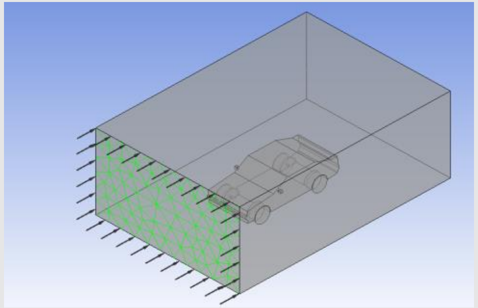


Figure 6: Isometric view of the inlet boundary layer

Boundary layer name: Inlet
Type of flow: Subsonic
Normal speed: 65km/hr

2

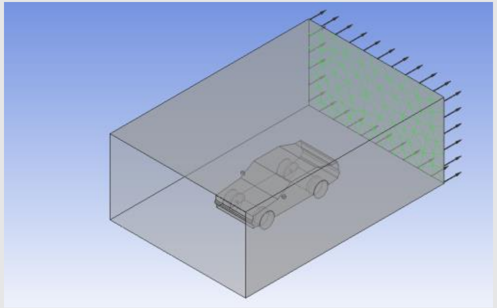


Figure 7: Isometric view of the outlet boundary layer

Boundary layer name: Outlet
Type of flow: Subsonic
Average Static pressure: 0 Pa

3

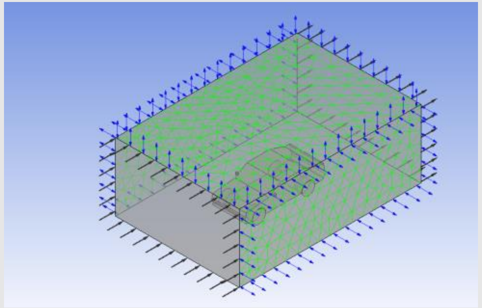


Figure 8: Isometric view of the opening boundary layer

Boundary layer name: Opening
Type of flow: Subsonic
Average Static pressure: 0 Pa

4

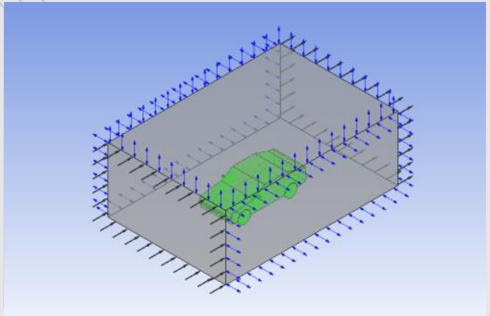


Figure 9: Isometric view of the wall boundary layer

Type of wall: No slip wall
Roughness: Smooth

5

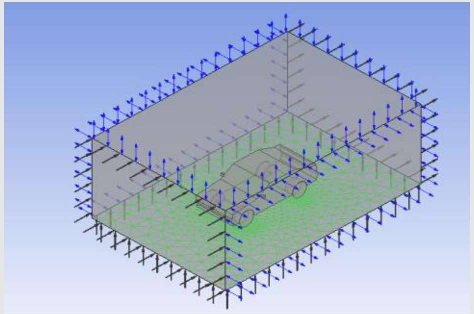


Figure 10: Isometric view of the Floor boundary layer

Boundary layer name: Floor
Type of flow: Subsonic
Speed: -65km/hr (Cartesian velocity)

6

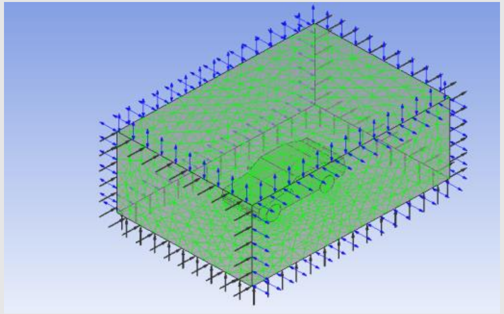


Figure 11: Isometric view of the boundary layers

It is important to use boundary layers to recreate realistic simulation of how the fluid flows around the car to determine the forces acting on the car.

03

Initial Design

03 INITIAL DESIGN

The following figures and graphs illustrate the results for the initial design using the CFD simulation and the prior sides parameters and mesh configurations.

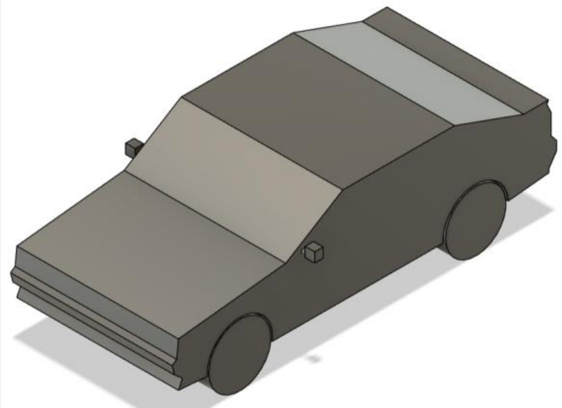


Figure 12: Isometric view of initial design

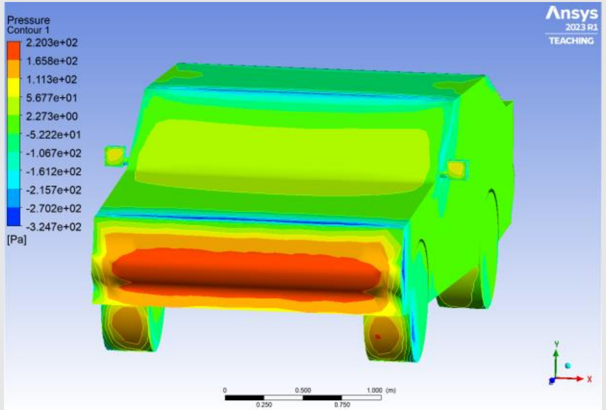


Figure 13: Isometric view of initial design

The results derived from the contour maps in figure 13 shows a large amount of pressure on the front of the car, the side mirrors, and the windows. Air should be following smoothly over the car which is shown as red/ orange from the streamline diagrams. From these results the coefficient of drag and friction can be calculated using equations:

$$C_D = \frac{2F}{\rho A V^2} \text{ (1) and } C_L = \frac{2F}{\rho A V^2} \text{ (2)}$$

F_{drag} = 343.04N (to the right)
F_{lift} = 103.76 N (going up)

C_d = 0.747
C_L = 0.226

Table 5: Parameters used for calculations

Parameter	Value
Area	2.3m
Density	1.225 kg/m3
Velocity	65km/hr

03

INITIAL DESIGN

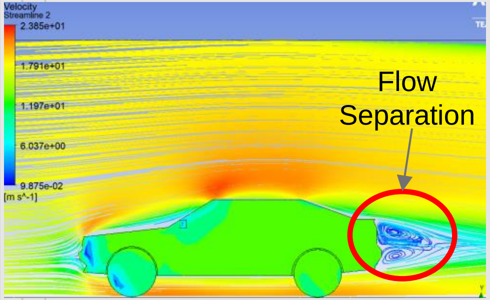


Figure 14: Pressure Contour and Streamlines (Side)

Figure 15: Velocity Streamlines (Side)

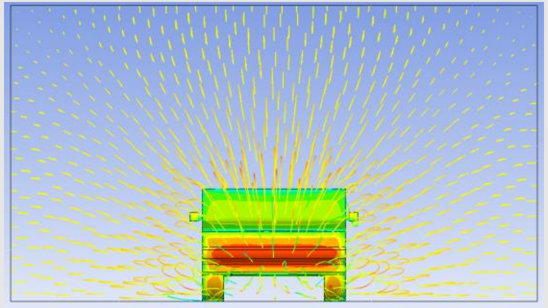


Figure 16: Velocity Streamlines (Front)

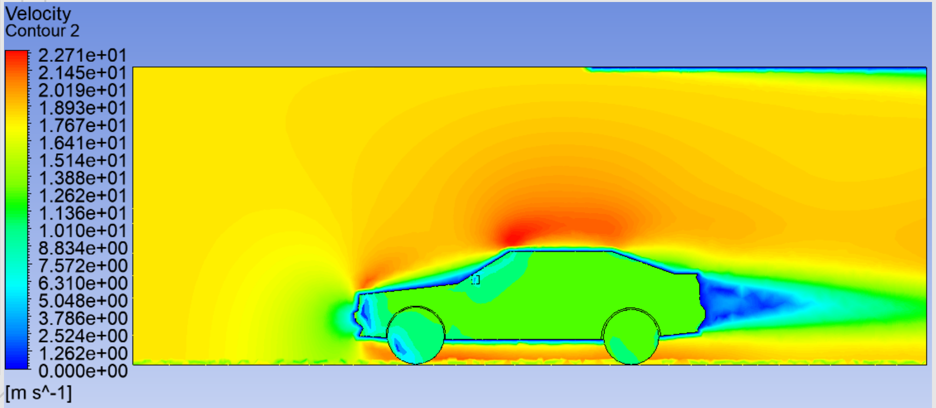


Figure 17: Velocity Contour (Side)

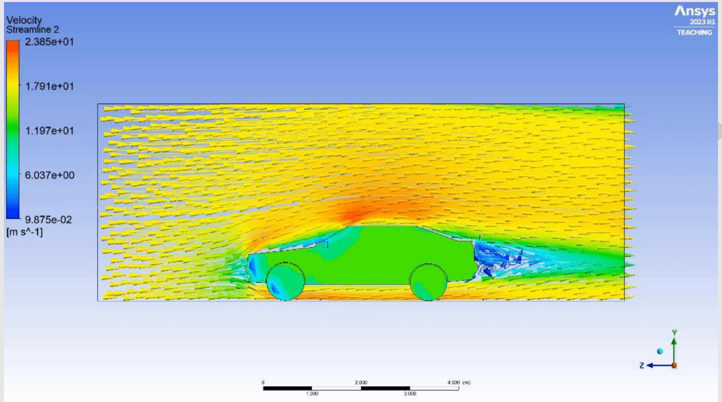


Figure 18: Animation of Streamlines

04

Improvement Designs

04

Improvement 1: Loungo Kukama Change in wind mirrors

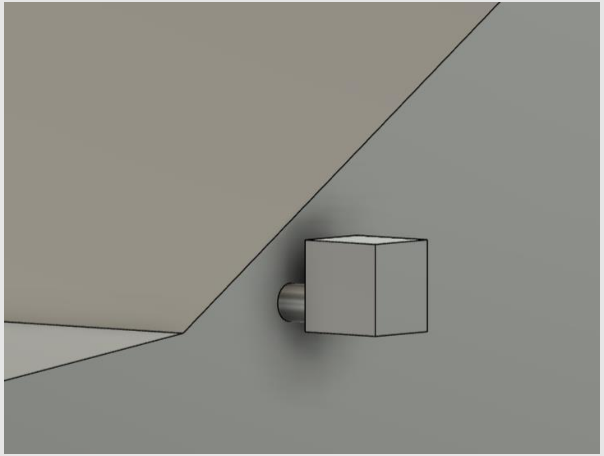


Figure 19: Isometric view of redesign 1

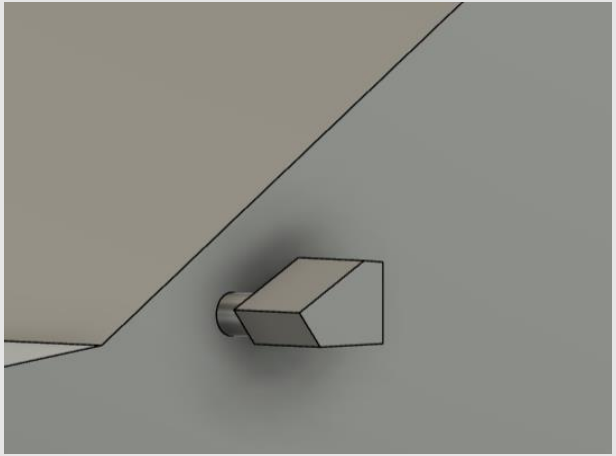


Figure 20: Isometric view of redesign 1

The first redesign of the car mainly focused on changing the shape of the side mirrors. Traditional side mirrors create aerodynamic drag due to their shape and positioning. To reduce drag we aim to change the shape of the side mirror.

To change the shape of the side mirrors, we made the front more tapered and filleted the edges to reduce roughness and increase smoothness. By doing this it reduce drag.

04 Results

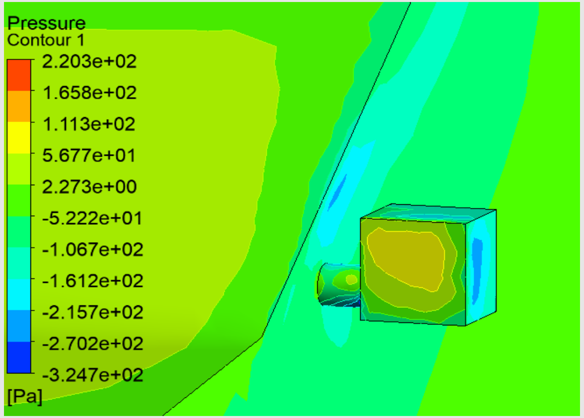


Figure 21: Pressure contour on initial design

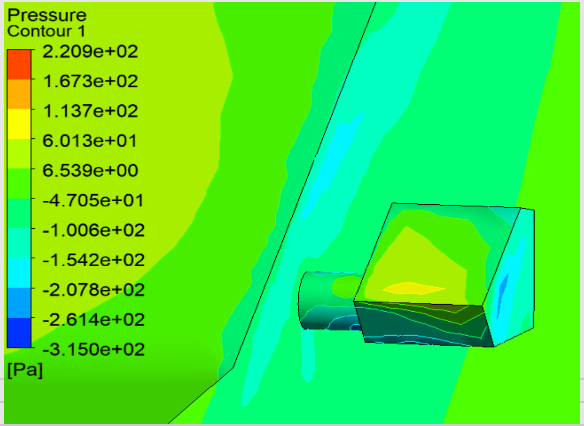


Figure 22: Pressure contour with side mirror improvement

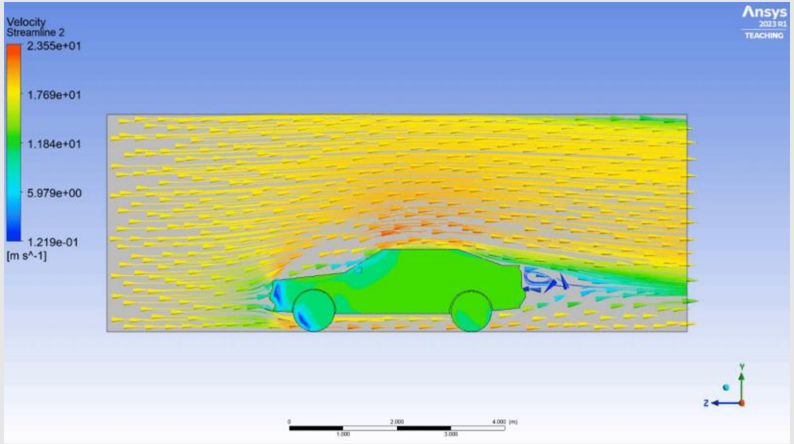


Figure 23: Animation of streamlines with side mirror improvement

F_{drag} = 336.28N
F_{Lift} = 109.77 N

C_d = 0.732
C_L = 0.239

- We analysed the pressure on the wing mirrors to see the effect of drag. The drag decreased from 343.04N to 336.28N.
- A percentage decrease of 1.97%.
- Decrease in pressure decreases drag, therefore increasing the speed.
- However lift increase from 103.76N to 109.77N due to a higher velocity above the surface of the mirror. According to Bernoulli's equation this generates more lift.

04 Improvement 2: Isis-Gabrielle Rodney-Jones
Change in Body Shape

The second version of redesigning the car focused on changing the body shape of the car from rough and sharped cut edges to a more streamline body shape. To achieve this chamfering, the front of the car to give it a more tapered end and by filleting the edges. By doing this it increase the aerodynamics as a more streamlined shape minimizes the surface area experiencing air resistance which inevitably lowers the drag coefficient.

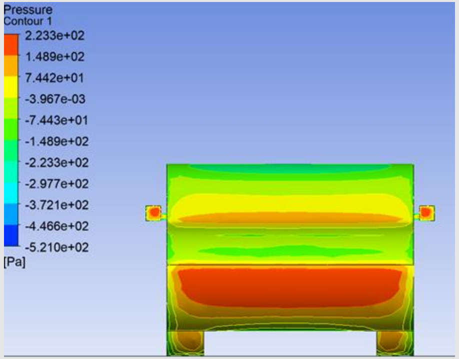


Figure 25: Front view pressure contour

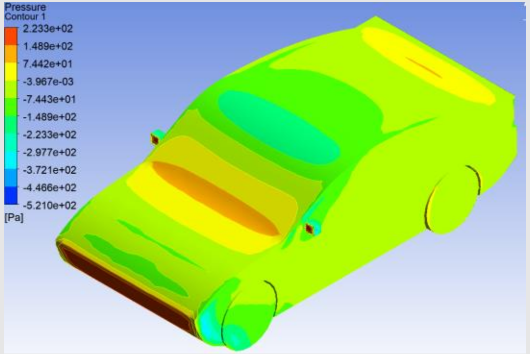


Figure 26: Isometric view pressure contour

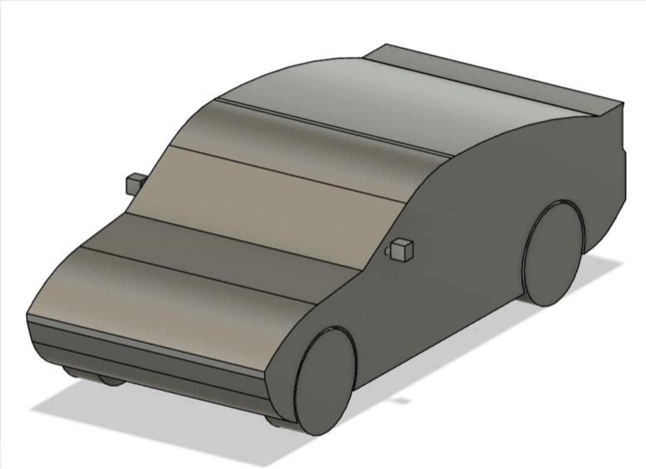


Figure 24: Isometric view of improvement 2

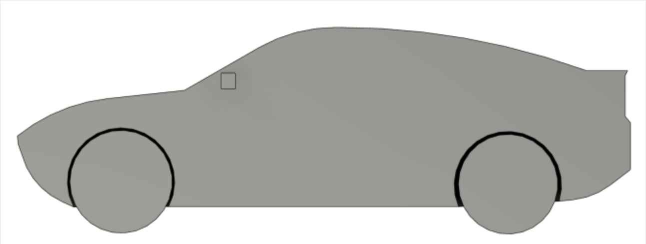


Figure 27: Side view of improvement 2

F_{drag}= 296.62 N to the right
F_{lift}= 60.42N (going up)

C_d= 0.646
C_L= 0.132

- The drag changed from 343.04N to 296.62N which percentage decrease of 13.53%.
- Decrease in pressure decreases drag, therefore increasing the speed.

04

Results

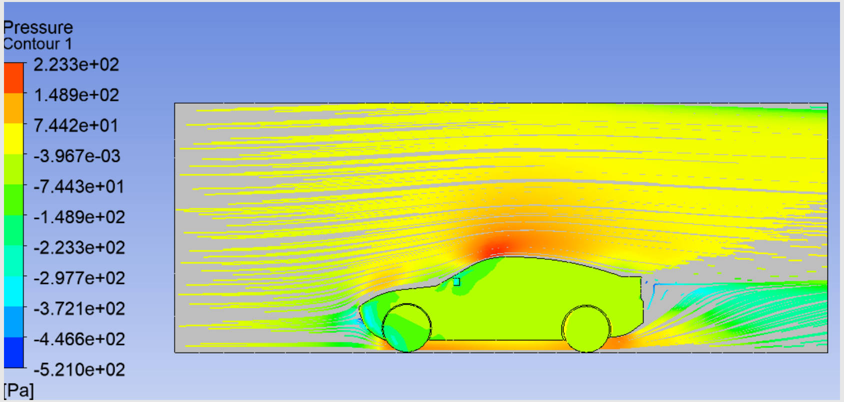


Figure 28: Velocity streamlines side view

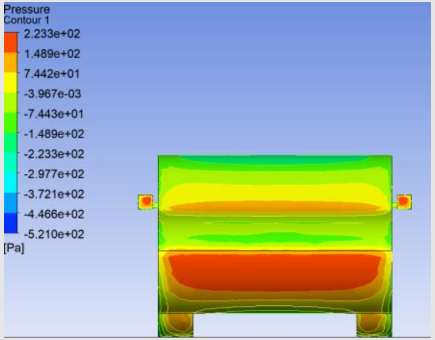


Figure 30: Front view pressure contour

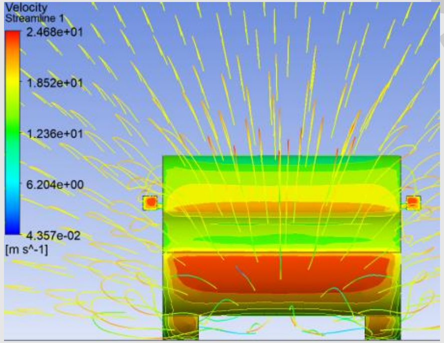


Figure 31: Front view velocity streamlines

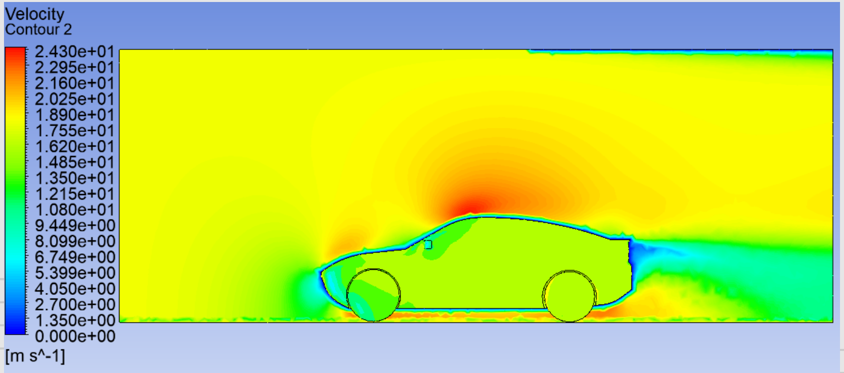


Figure 29: Side view of improvement 2

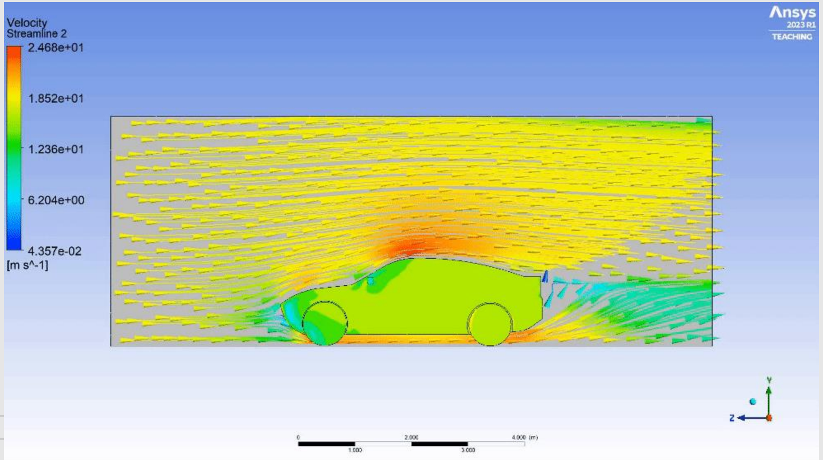


Figure 32: Animation of streamlines with body shape improvement

Improvement 3: Mahmood El-Mahalawy Addition of Spoiler

04

Improvement 3 mainly focuses on adding a spoiler to the end of the car to enhance its performance. In order to increase the aerodynamics of the car the spoiler was strategically designed and placed. The spoiler can alter the airflow around the car minimizing turbulence.

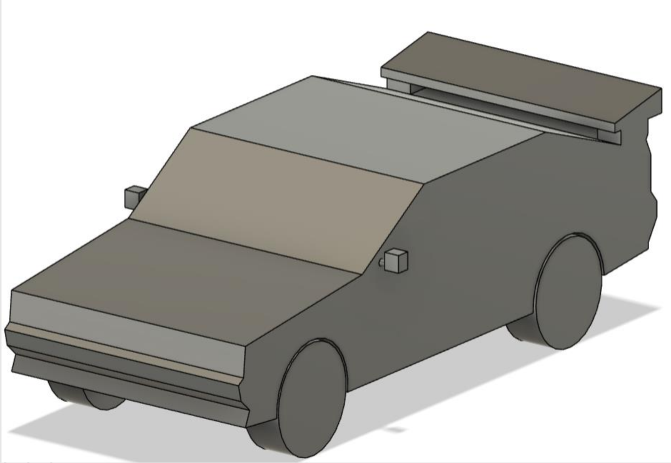


Figure 33: Isometric view of spoiler addition

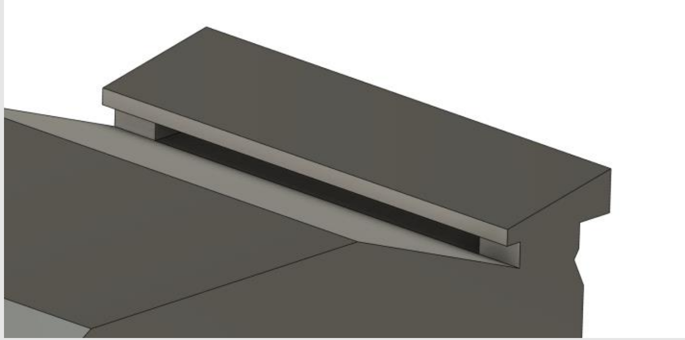


Figure 34: Closeup view of spoiler addition

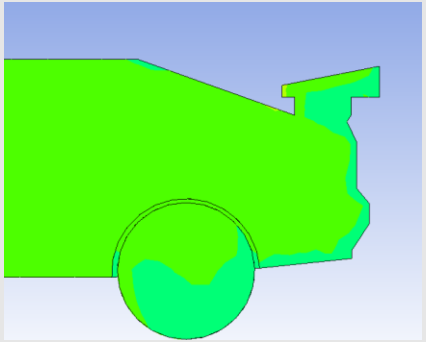


Figure 35: Side view pressure contour

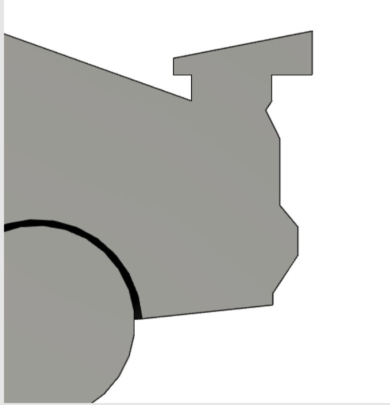


Figure 36: Side view of spoiler addition

F_{drag} = 302.21 N to the right
F_{lift} = 76.58N (going up)

C_d= 0.658
C_L= 0.167

- The drag changed from 343.04N to 302.21N which percentage decrease of 11.90%
- Decrease in pressure decreases drag, therefore increasing the speed.

04

Results

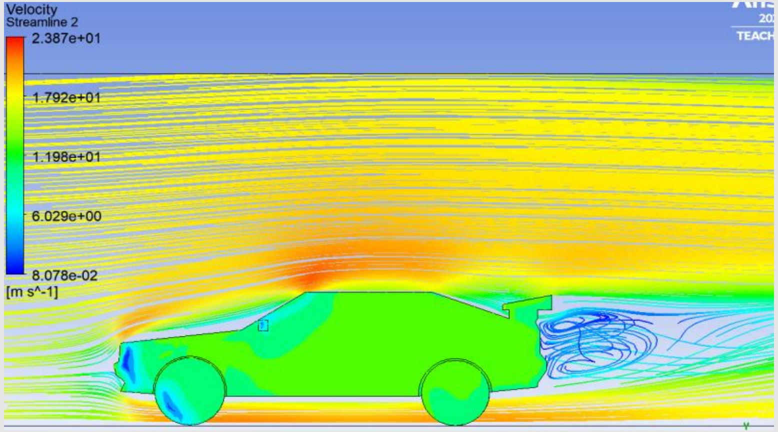


Figure 37: Velocity streamlines side view

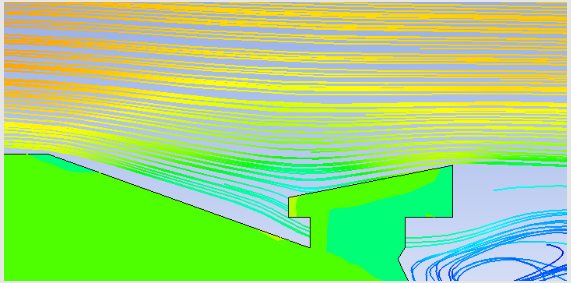


Figure 38: Closeup view of velocity streamlines across spoiler addition

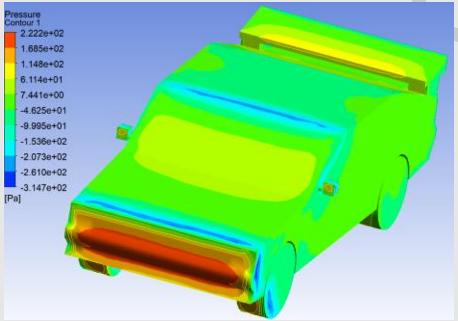


Figure 39: Isometric view pressure contour

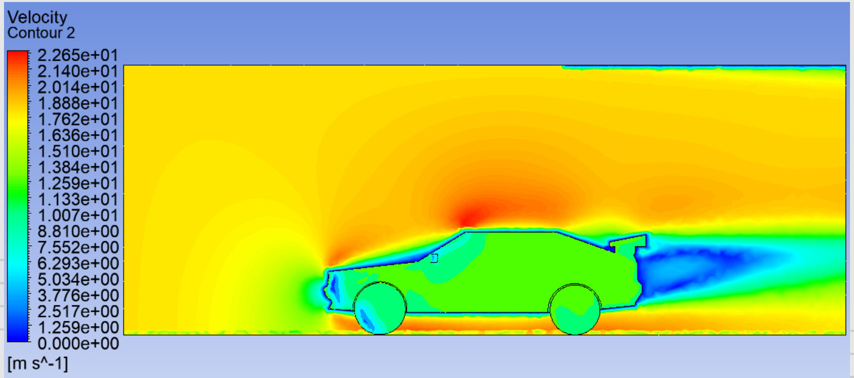


Figure 40: Closeup view of spoiler addition

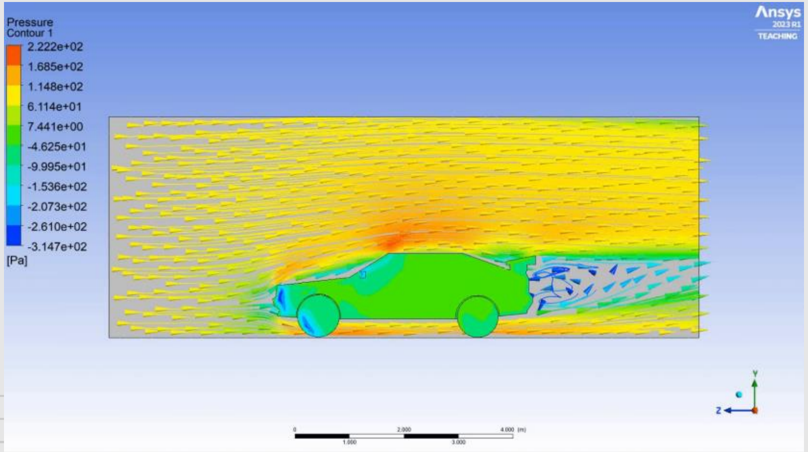


Figure 41: Animation of velocity streamlines for improvement 3

05

Final Design

05 Final Design

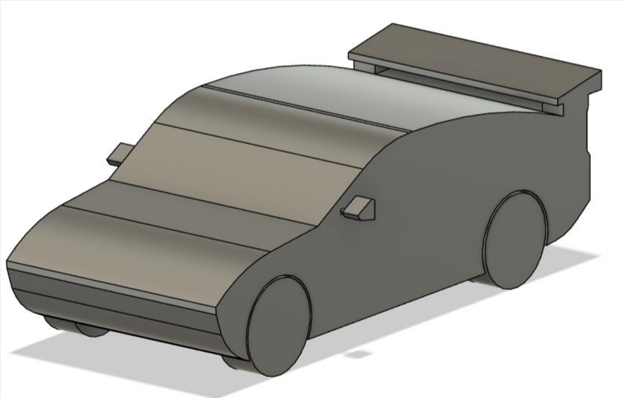


Figure 42: Isometric view of final design

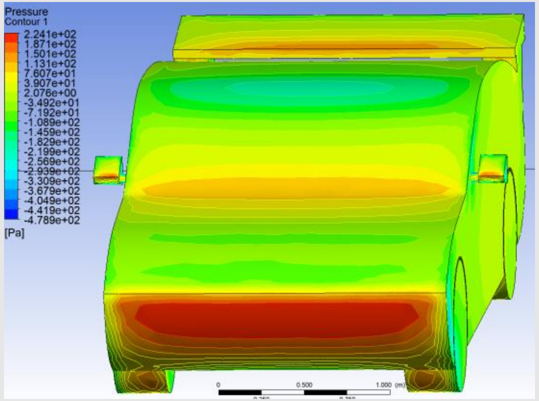


Figure 43: Pressure contour of final design

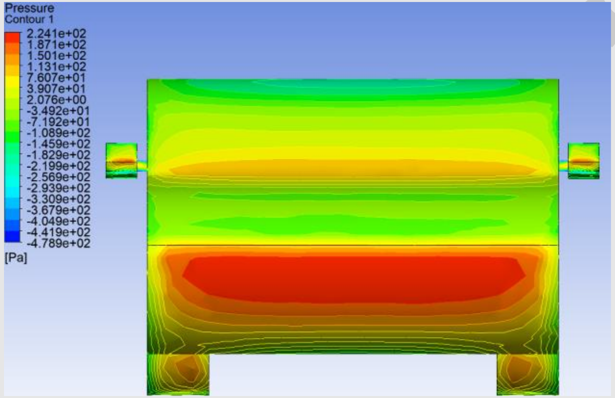


Figure 44: Front view pressure contour

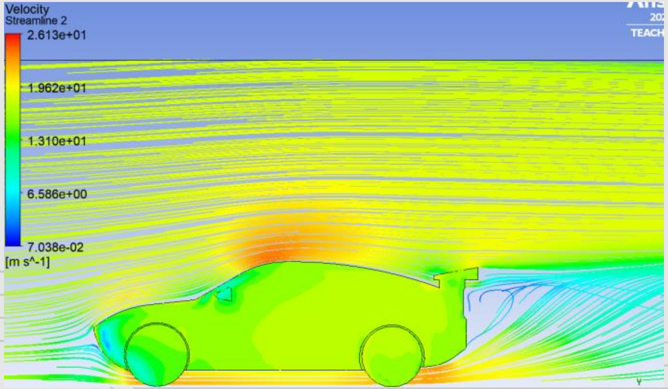


Figure 45: Side view velocity streamline

A final design was made using a combination of the elements that improved the aerodynamic efficiency for each design.

F_{drag}= 272.28N to the right
F_{lift}= 44.75 N (going up)

C_d= 0.593N
C_L= 0.098N

- The drag changed from 343.04N to 272.28N which percentage decrease of 20.68%
- Decrease in pressure decreases drag, therefore increasing the speed.

05

Final Design

Initial Design

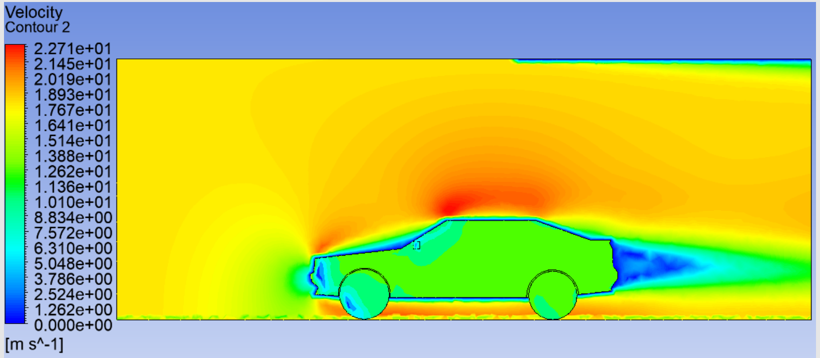


Figure 46: Side view velocity contour

Final Design

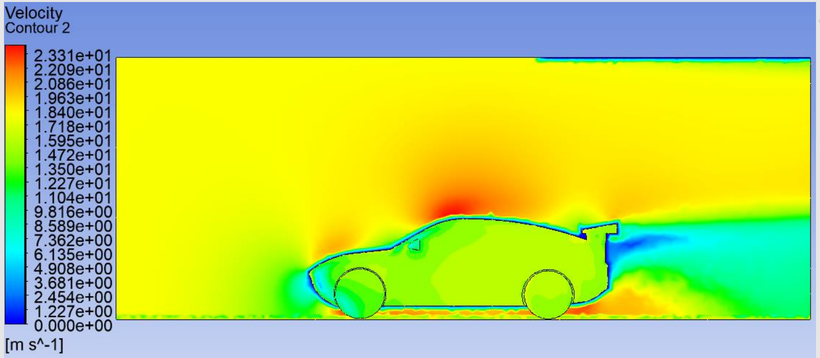


Figure 47: Side view velocity contour

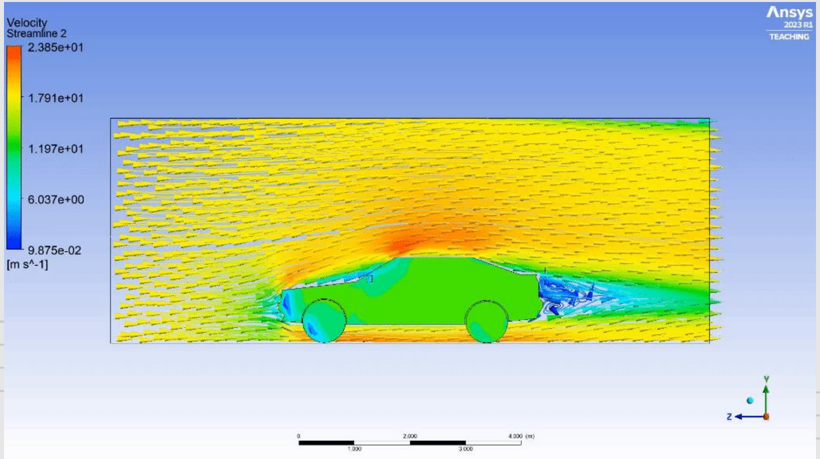


Figure 48: Animation of velocity streamlines for initial design

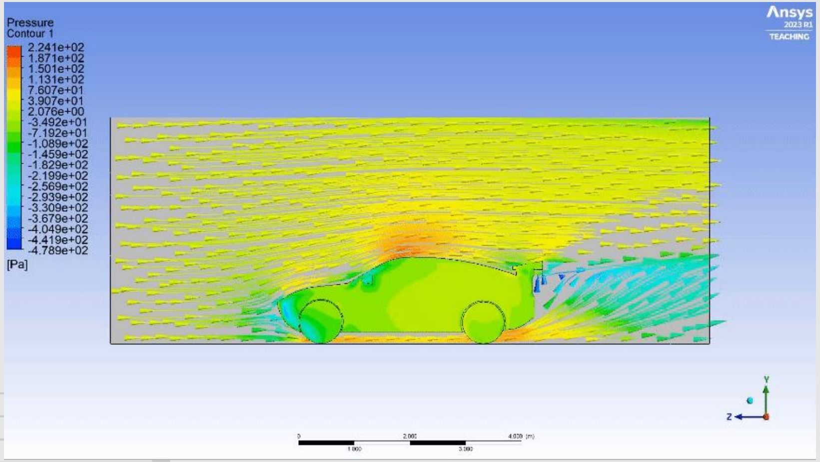


Figure 49: Animation of velocity streamlines for final design

06

Conclusion

06

Conclusion

- From the results shown we have successfully made our car more aerodynamic and increased our speed.
- As from the initial and final design there's a percentage decrease of 20.68% which shows a reduction in force drag which increases the speed.
- Additionally, there is a percentage decrease by 56.87% in the lift force from the first design to the final design.
- The design of changing the body shape has proven to be the most successful in increasing the speed of the cares it has the highest percentage decrease in the drag forces acting on the car of 13.53% whereas the spoiler and side mirrors had 11.90% and 1.97% respectively.

What can we do to improve our simulation and decrease the amount of lift and drag force in order to increase speed?

- Increase the speed of fluid hitting the inlet as it will increase the effect of aerodynamics on the car therefore generating larger results.
- Incorporate the use of different tires that encourage more airflow.
- For the spoiler strategically place and design the spoiler on the car.
- Lower the suspension of the car.

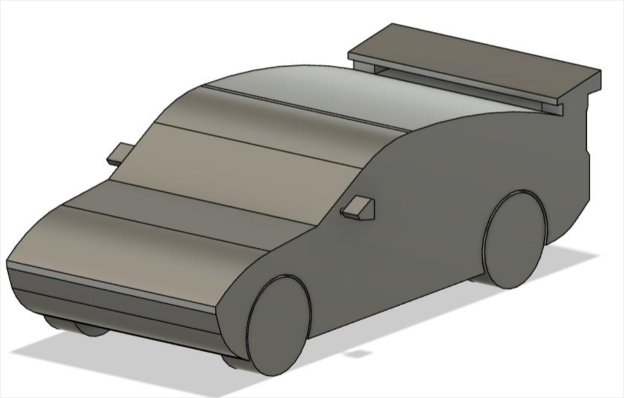


Figure 50: Isometric view of final design