A simulation-based investigation on the effects of a rear spoiler in the performance of a car

[1]Rubankarthik Umapathy,[2] Siva Sankar S, [3]Abhay Prasanth , [4]Muralimohan Gurusamy[*]

[1,2,3,4]School of Mechanical engineering, Vellore Institute of Technology, Chennai Campus, Chennai 600127, India

ABSTRACT: The use of spoilers in automobiles is crucial to manage the drag and lift forces imparting aerodynamic resistance. To enhance the vehicle's traction and steering ability, spoilers can be placed to produce greater down force by diffusing the unfavourable air movement around the vehicle's body. This project emphasizes the need for spoilers in the automotive industry and employs software-based simulation for digital testing to analyse the design of a spoiler in a commercial vehicle. The project involves designing the spoiler using aerodynamic design parameters in SolidWorks and conducting a CFD study with necessary boundary conditions. The resulting aerodynamic findings are compared to those of the vehicle without the spoiler to analyse the differences in aerodynamic forces. The IPG Carmaker software is used to simulate various scenarios using cars with and without the spoiler and results are recorded for cornering stability, aerodynamic coefficients, lap time, and other performance indicators. This approach offers several advantages over traditional physical testing methods, which may be costly and less accessible. By using software simulations, manufacturers can reduce costs, increase testing availability, and improve testing quality. Additionally, the results obtained through simulation can provide manufacturers with an opportunity to improve the design elements of the vehicle prior to physical testing. Based on the predicted results from the simulations and calculations described in the methodology, an improvement in performance and an increase in stability of the test vehicle are expected. In summary, this project demonstrates the importance of spoilers in the automotive industry and presents an effective method for analysing and improving their design using software-based simulation.

Keywords: Spoiler, Drag, Lift, Drag-Lift Coefficient, Carmaker, Computational Fluid Dynamics (CFD), IPG Carmaker

1. INTRODUCTION

Spoilers are one of the most common accessories found on the rear edge of the roof not only on sports cars, but also on sedans, roadsters and some hatchbacks and SUVs. Vehicle performance is highly dependent on aerodynamic factors. At low speed you can't judge the importance, but as soon as the speed increases and the vehicle exceed his 60 km/h, about 60% of the engine power drops due to drag. Air resistance comes primarily in the form of drag and lift. Lift is generated by different airflows above and below the car body, creating a pressure difference. Drag is caused by pressure fluctuations across the vehicle. Commercial vehicle accidents are common news around the world, with highway cornering and narrow road cornering accounting for up to 30% across all ranges. A major cause of accidents is loss of traction or grip between the wheels and the road when cornering. When a car traveling at high-speed hits a curve, various forces act. Its previous movement was linear, but turning it (e.g., to the left) pulls it inwards. In other words, the friction between the road and the car's tires creates a centripetal force. Due to the high speed of the car, the inertia of the car moving tangential to the turn overcomes the centripetal force and tends to cause the car to skid.

An additional support called a spoiler was installed to improve the stability of the car. A spoiler is a symmetrical wing-shaped component that is typically attached to the rear of a car. The spoiler's intended design function is to "ruin" unwanted air movement over the body of a moving vehicle, achieving results that benefit vehicle performance. The spoiler if checked from the perspective of a design, is like how an aerofoil is designed, making them have the same characteristics in terms of aerodynamic performance. In Fig 1, an aerofoil is seen. The line connecting the front edge and trailing edge is the chord, as seen in the image. The circumference of the area above the chord is greater than that of the area below.

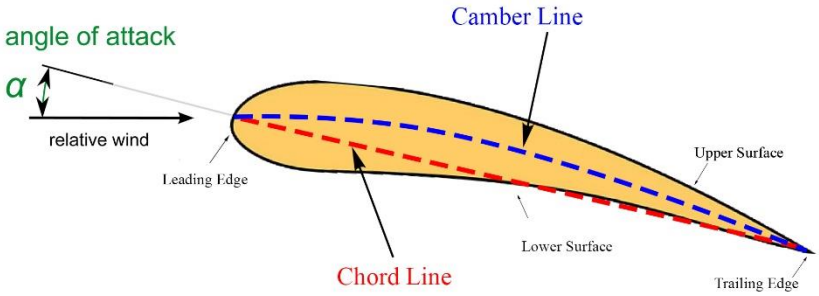


Fig 1. Aerofoil line diagram

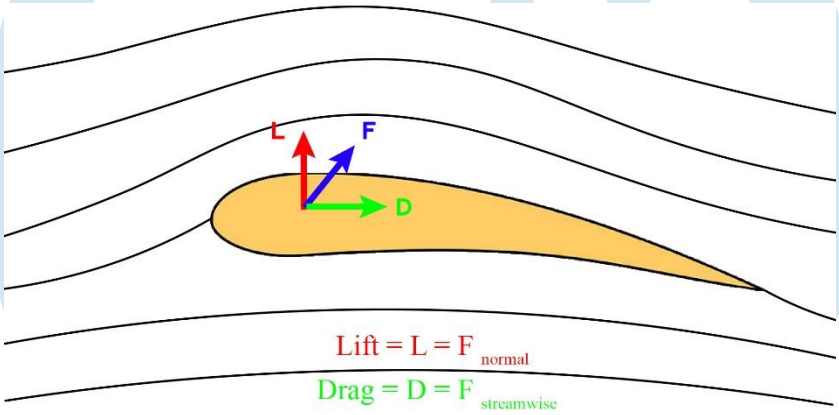


Fig 2. Airflow over Aerofoil line diagram

As a result, as the wind blows, the upper half of the wind has a higher velocity than the lower part. As long as there is a velocity differential, there will also be a pressure difference, resulting in low pressure above and high pressure below. This pressure difference will produce a force that is directed upward, or lift. As seen in Fig 2 ,

Lift = L = F_{normal}

Drag = D = $F_{streamwise}$

The angle of attack refers to the position of the aerofoil in relation to the wind direction. The quantity of wind flowing above and below the aerofoil is determined by it. A connected rear spoiler has the appearance of an upside-down aerofoil. This causes the lift force to push the automobile downward.

A well-designed spoiler will give the vehicle more downforce and increase traction while driving. However, the greater the downforce, the greater the drag. The greater the drag, the greater the resistance to forward motion, thus slowing the car. A specially designed spoiler can also reduce the vehicle's air resistance increasing the top speed of the vehicle. Higher top speed at the expense of less downforce. If the vehicle experiences more lift, leading to less downforce which makes the car tend to lose traction at high speeds leading to accidents. An ideal spoiler should be able to offer the best of both worlds.

The purpose of this study is to use IPG Carmaker's virtual driving scenario to simulate the changes in vehicle performance and efficiency when a spoiler is installed. Analyses such as computational fluid dynamics analysis can only provide static values such as drag and lift coefficients. Virtual driving scenarios allows us to find real-time values such as lap times, body roll and more. Virtual driving gives you the freedom to do countless iterations without consuming resources. The Audi TT could have avoided almost 100 accidents if it had been tested in a virtual driving simulation before production. Virtual driving not only saves time and resources, it also saves unnecessary accidents that can occur during testing and unnoticed errors and defects in vehicle design and manufacturing, preventing accidents.

The virtual driving scenarios in carmaker help us to virtually drive the car in different terrains and environmental conditions which helps us identify its effects on performance of the car. The performance variation in the car is explained using various KPIs. In Braking KPI, the spoiler reduces the stopping distance by providing necessary wake behind the trunk of the car thereby slowing down the car

with much stability. The Corner Braking KPI is essential since its effects can actually cause the vehicle to either understeer or oversteer. Spoilers helps the car to provide necessary weight transfer which provides traction and can prevent the body to roll during braking. In Lateral Acceleration KPI, spoilers may be utilised to generate the required yaw and roll moments to improve the vehicle's yaw and roll stability when performing emergency maneuvers such tight turns at high speeds. In Top speed KPI, the airflow is smoothly changed from a downward direction, which produces unwanted lift, to a horizontal direction, which produces no lift and the spoiler receives very little airflow under it that improves traction.

1. LITERATURE REVIEW

Hariantoa et al. [1], in this paper," Effect of Tapper Ratio on a Car Rear Spoiler Performance", talks about the effect of taper ratio of a car's rear spoiler that causes considerable changes in the performance of a car. The CL value is directly proportional to the angle of attack and the magnitude of the Taper Ratio, which will lead to a reduction until a certain degree due to the stall phenomenon. In high-speed cars, aerodynamics is very influential and cause instability of the car at high speeds. This is because the resulting lift is typically greater for rear-wheel-drive vehicles than for front wheel drive vehicles. The air will move faster on the upper airfoil surface than at the bottom surface. Due to differences in speed of air flow on the surface of the airfoil, a low pressure on the upper surface and a higher pressure on the bottom surface is generated. This pressure differential results in downforce on the airfoil. A NACA 6412 airfoil is be used to create a singleelement rear wing/spoiler with a tapper ratio change in order to calculate the lift and drag coefficient values. The research approach involved changing the spoiler model with five different tapper ratio modifications, including 1:1, 1:0.5, 1:0.7, 0.5:1, and 0.7:1. It is concluded that differences in tapper ratio with angle of attack (8^{o}) have an impact on lift coefficient and drag coefficient showed that the results were different. Conversely, if the surface area is smaller, the lift coefficient value will be smaller and the drag coefficient value will be smaller. After doing a simulation with various variations of the tapper ratio, the highest lift coefficient value was achieved in the 1:1 tapper ratio variation .It also produced the strongest downforce. The 1:05 tapper ratio variant yields the best CL/CD results, with a value of 12.82.

Jahanmiri Et al. [2], conducted experiment which explains the effects of spoiler height and diffuser on aerodynamic performance of a car. The experiment was conducted in the infrasound wind tunnel with 0.13 as the maximum turbulence intensity of the tunnel. In this experiment, an external balance was used to calculate forces and moments, and the signals provided by the balance were converted to forces and moments by Lab View software. The model was tested in a simple way in 4 different states with a speed range from 0 to 60 m/s using fan circuit and with the presence of aerodynamic parts such as spoilers and diffusers at the edges of the model. The drag and lift coefficient values also drop as the height of the spoiler rises. To lower the drag and lift coefficients, a 3 cm increase in spoiler height is appropriate which resulted in a 6.3% decrease in drag and 12.4% decrease in lift. The drag and lift coefficients also drop as the spoiler's height rises. Mahmoud Ibrahim Youssef [3] examines the aerodynamic properties of a car with and without a rear spoiler as well as the ideal height for the spoiler at distinct velocities. The installation of a spoiler at the 371mm upper surface of the car trunk has been discovered. Total drag coefficient reduction of 26% at this point. The Active Rear Spoilers can adjust the arflow over its vehicle based on the pressure applied on its flaps [4]. Accelerometer sensors gives signal to processor followed by stepper motors to adjust the spoiler angle that helps in achieving lower drag and higher downforce at different velocities. Active rear Spoilers can also be used to increase the lateral stability of high-speed cars in high-lateral-acceleration situations, including a fast, precise cornering maneuver. The spoiler provides higher downforce and lower drag that improves lateral stability at high-g cornering manuevers provided the the individual left and right spoilers' angles of attack are kept within the range of 10° to 15°[5]. M. Hammad [6] designed a split rear wing to control the aerodynamics of high-speed road vehicles. Based on the chosen control aim of tracking the steady-state yaw rate, an SMC technique is employed to develop the ADD controller to control the attack angle of the wings. It has been proven that the controller works better at higher speeds to increase the lateral stability of the vehicle. In place of traditional fixed wing systems, independent active rear wing systems offer high-speed vehicles better directional stability and accepted acceleration performance. Active aerodynamic control has the potential to significantly improve the lateral stability of fast-moving vehicles during rapid cornering. Hammad Et al. [7], has suggested an alternative rear wing. A favorable yaw moment may be attained to guarantee the lateral stability of the vehicle by adjusting the attack angles for the right and/or left halves of the split rear wing . This process is attained by using a yaw-plane vehicle model created in MATLAB and LQR (linear quadratic regulator)-based controller.

Based on the findings of Ipilakyaa Et al. [8], it was concluded that employing a rear spoiler with wings increased the downward force (negative lift), which in turn increased the car's drag. As a result, the car is more stable at high speeds, its braking performance and the traction was improved. Since driving safely is always of the utmost importance, having more down force may be more significant for passenger automobiles than having less drag. Additionally, the self-weight of the vehicle may be readily decreased without consequence with a downforce device as effective as this, which will make up for the higher drag in terms of fuel consumption. The findings of the study conducted by Nawam Et al., [9] demonstrated that a connected spoiler may raise a car's drag coefficient while lowering its lift coefficient. According to the data, when a spoiler is installed, the drag coefficient of the car increases by 47.20 percent. The lift coefficient value, on the other hand, deviates dramatically from 0.03957 to -0.16254. The optimal spoiler should have a significant reduction in lift and the least amount of drag increase.

To improve traction, braking, and lateral force of a vehicle, Kurec Et al., presented a variety of techniques for modelling and simulating various aspects of managing car aerodynamic properties. A solution of electronically controlled movable aerodynamic elements that

are activated and controlled in a manner to extend the driving range of fast cars can be reached through the coordinated action of experts in unsteady flow simulations, flexible material FSI (Fluid Structure Interaction) simulations, car dynamics simulations, coupled FSI car aerodynamics and car dynamics, experimental tests in wind tunnels, and road tests[10].

According to the analysis done by Prabhu.M Et al., [11] the most downward pressure was generated at an angle of 20°, while the least pressure was generated at an angle of 8°. The car model with the spoiler offers a 20-degree reduction in stopping distance. The spoiler initially at 8° shifts to an angle of 20° when brakes are engaged.When the angle of the spoiler moves downward, its influence is no longer ineffective. The pressure acting downward, has decreased the stopping distance of the car to 4.52% at a speed of 150 km/h and increased the friction between the tire and road. The number of accidents can be avoided and decreased if this method is used in day-to-day life to lower the stopping distance of the vehicle. Tomar Et al. [12], analyzed the vehicle for various spoiler angles for coefficient of drag where CFD analysis of a car was done in Ansys for different spoiler angles. The drag coefficient for spoiler at zero degree was found to be less than the coefficient of the car without spoiler, up to 12 to 13 degrees. Beyond 12 degrees of spoiler angle, the drag coefficient increased by 5% for every 10-degree increment. Chih-Chun Yang has communicated the same experiment in two different facilities. The downforce and drag forces are measured in a wind tunnel, while the velocity profile and flow field visualization are observed in a water tunnel. The most downforce without significantly raising drag was produced at an angle of attack of twelve degrees which provided greater stability. In wind tunnel tests, the downforce increases by 23.3% when the attack angle is increased from 5° to 12°, yet the drag force only goes up by 15% [13].

The rear diffuser mechanism prevents low-pressure air from the underbody from entering the space behind the trunk, instead allowing relatively high-pressure air from the sides and top of the trunk to do so [14]. This raises the base pressure. Within the range of 70 to 160 km/h, the passenger cars aerodynamic drag decreases. With an average reduction of more than 4%, this demonstrated the best drag reduction performance. Number crunching can be used to explore the aerodynamic lift, drag, and flow characteristics of a high-speed (100 mph) generic sedan passenger vehicle in both spoiler-equipped and spoiler-less conditions[15] .The aftermarket spoilers are used for the analysis attached to the roof and back. The rear-mounted spoiler design, which produced more negative lift force than the roof spoiler but less drag reduction, was demonstrated by numerical analysis.

The main goal of the study [16] is to determine the best shape for a car spoiler in terms of strength and aerodynamics while minimizing mass to help with fuel efficiency. The result of shape optimization has preserved all the material's strength and aerodynamic properties while saving 18.74% of it. This study opens a numerical tool for improving future spoiler models in terms of mass reduction of spoiler design. Hatchback model car with a roof spoiler (Wing and Strip) provides considerable changes in aerodynamics by adjusting the yaw angle [17]. Even though the drag coefficient has minimal change, the lift coeffcient reduces considerably by 16.87% for strip and 31.96% for the wing.

2. BACKGROUND

Accidents of commercial cars are common news around the world on turns in highways and lean roads accounts for up to 30% in all areas. The main reason behind the accidents is due to loss of traction or grip between the wheels and the road on the turn. The traction in turn is connected to the lower downforce on the vehicle. The Audi TT could've avoided close to 100 accidents if it had been tested on a simulation before production [18]. The airflow over Audi TT with and without spoilers is shown in Fig 3.

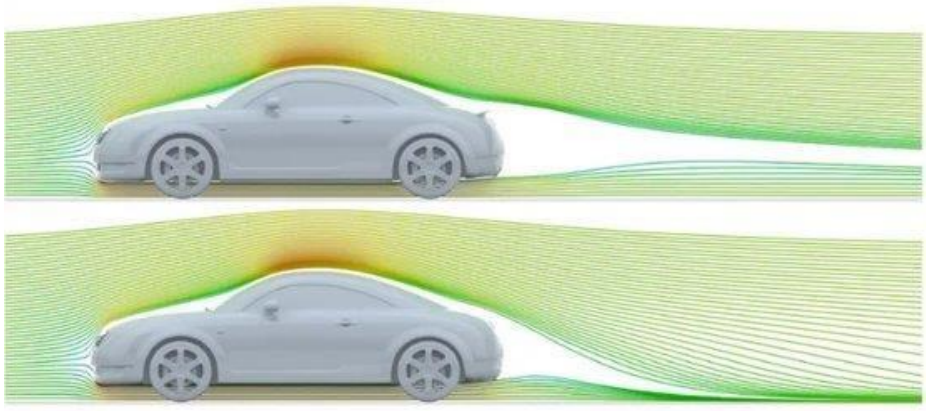


Fig 3. Airflow around Audi TT Vehicle

Physical testing of vehicles using aerodynamic tunnels (as shown in Fig.) [19] and lab systems are costlier in present life and thus access to such systems is not a part of everyone's life. Carmaker software (Fig.4) incorporates the testing scenarios without the use of physical systems thus making it cost effective and more accessible.

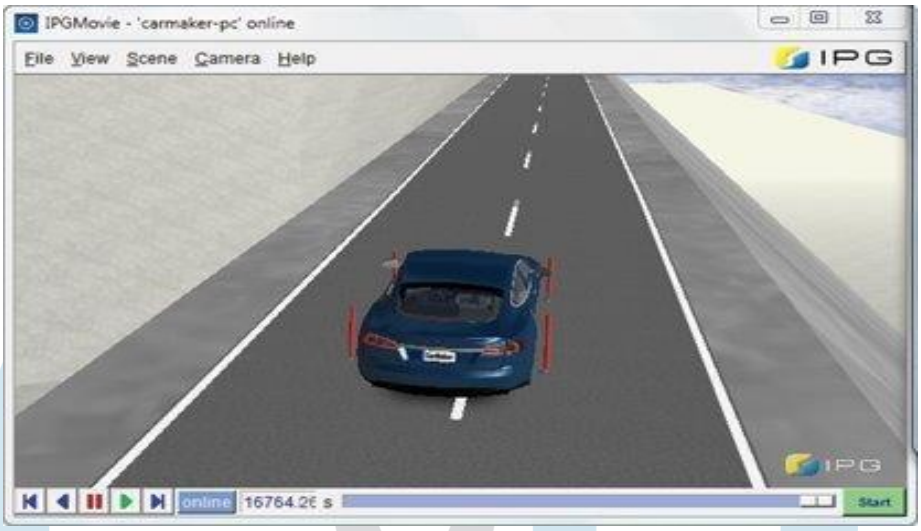


Fig 4. Virtual Simulation

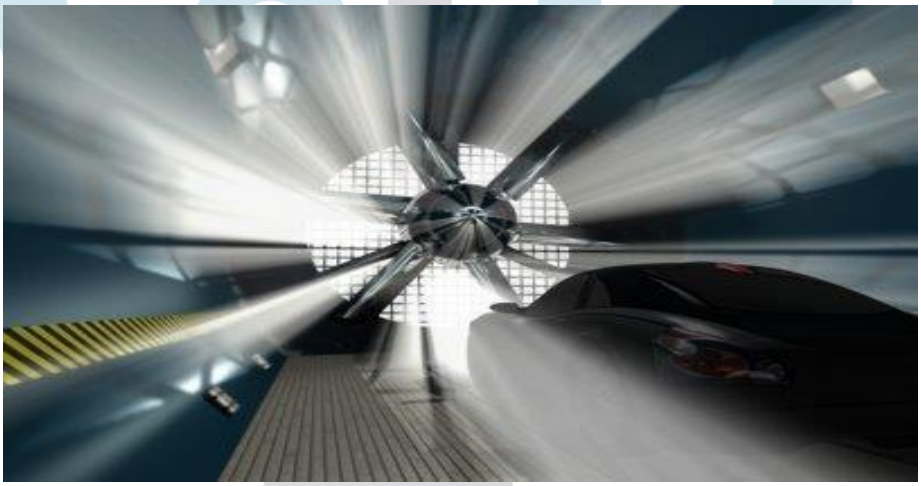


Fig 5. Aerodynamic Tunnel Testing

3. METHODOLOGY

The main aspect of methodology of this project is to make use of Computational Fluid Dynamics to simulate the changes in aerodynamics of the spoiler in the car and to validate the results using driver-in-loop simulation of IPG Carmaker. Driver in loop simulation plays key role in providing virtual simulations for automotive industry in their research and development phase. It helps us to validate crucial components such as steering system and spoiler aerodynamics with the help of virtual vehicle simulation. Fig 6 shows the flowchart of the entire methodology of the project that is being followed.

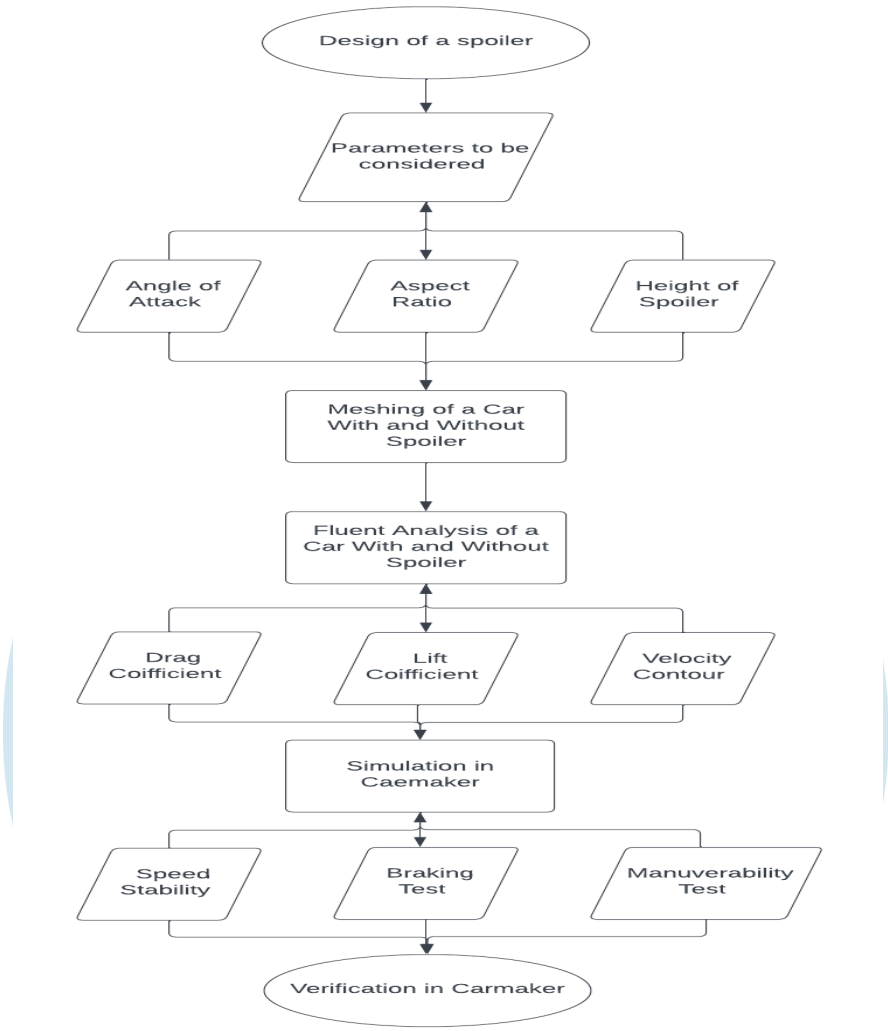


Fig 6. Flowchart

3.1 Design and Modelling of Spoilers

Computer-aided design (CAD) is the use of computer technology to aid in the design and creation of drawings and models for various applications, including architecture, engineering, industrial design, and manufacturing. This study used a 3D car model of the Tesla Model S designed in SolidWorks, with a spoiler attached to the rear of the car (Fig. 8). Initially, a dummy spoiler (Fig.7) was used to verify the aerodynamic performance changes in the car model.

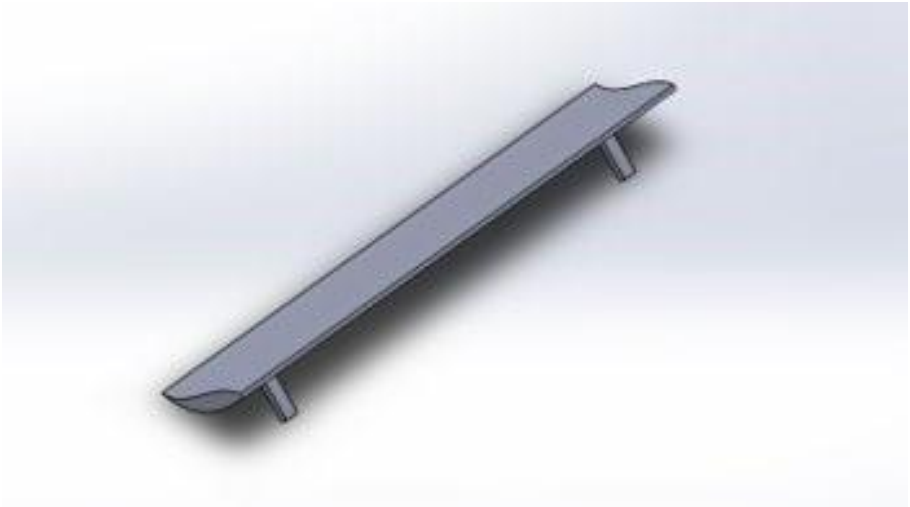


Fig. 7: Dummy Spoiler

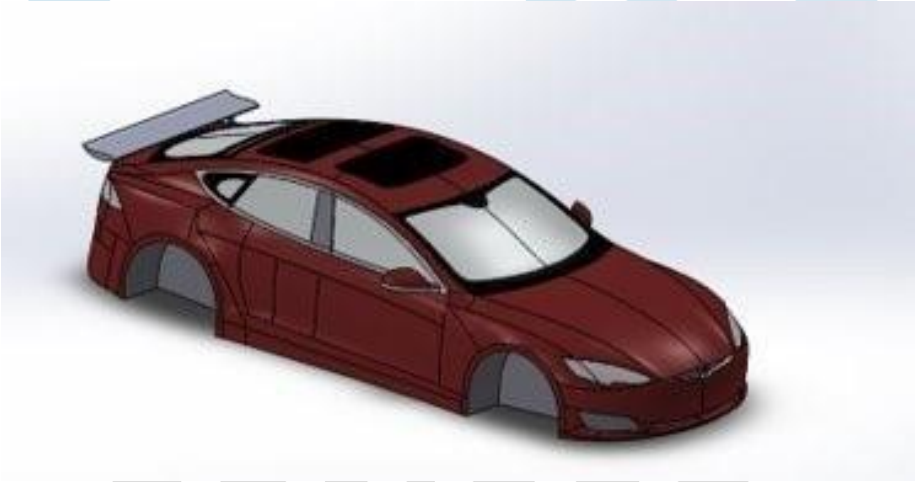


Fig. 8: Dummy spoiler attached to the Car model

Certain factors [20] were taken into consideration while designing the spoilers, which are mentioned below:

- Angle of attack: The angle of attack of a spoiler is the angle at which it is installed relative to the airflow passing over it. This angle is critical to the effectiveness of the spoiler in generating downforce on a vehicle.
- Height of the wing: The height of the spoiler can have a significant impact on its performance. Generally, the higher the spoiler, the more downforce it can generate.
- Size and Shape of the spoiler: The size and shape of the spoiler should be designed to complement the vehicle's overall
- Material Used: The material used to construct the spoiler should be lightweight yet strong, to ensure that it doesn't add unnecessary weight to the vehicle.
- Taper Ratio: The taper ratio, which is the ratio of the width of the spoiler's base to the width of its tip, can have a significant impact on a spoiler's performance.
- Aspect Ratio: The aspect ratio of a spoiler, which is the ratio of its height to its length, can have a significant impact on its performance. Generally, a higher aspect ratio (i.e., a taller and narrower spoiler) can increase the spoiler's effectiveness in generating downforce.
- Purpose: The intended purpose of the spoiler should be considered, as different types of spoilers may be more effective for different driving conditions and purposes, such as racing or street driving.

After considering the above-mentioned factors and verifying the aerodynamic performance changes with the dummy spoiler, three spoiler designs were made which were focused on increasing the downforce of the car. The taper ratio of all the spoilers is 1.05 as the best Cl/Cd results are obtained in that value [1]. All the three spoilers are made of the same material, which is ABS Plastic. Table 1 shows the values of each design parameters to their spoiler design.

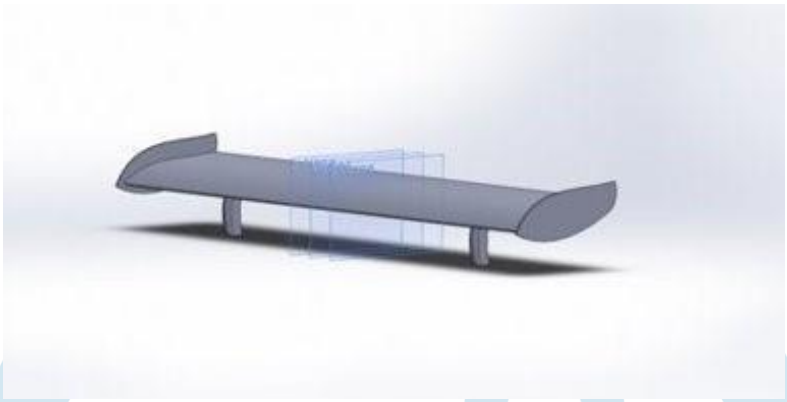


Fig. 9: Spoiler-1 with Angle of attack 12 degrees

Fig. 10: Spoiler-2 with Angle of attack 17 degrees

Fig. 11: Spoiler-3 with Angle of attack 20 degrees

Table 1. Spoilers and their Parameters

Car Model	Angle of Attack (deg)	Aspect Ratio	Weight of spoiler (kg)	Height of spoiler (mm)
Car Mod-1	12	4.83	2.4	92.92
Car Mod-2	17	5.2	2.5	62.46
Car Mod-3	20	4.24	2.9	64.88

3.2 Calculation of Drag-Lift Coefficient and Performance Analysis using Computational Fluid Dynamics (CFD)

Aerodynamic evaluation of air flow over an object can be performed using analytical or CFD method. In this work, the Car undergoes fluid simulation for different spoilers using same input parameters to analyze the variations in drag - lift coefficient values and to observe the type of airflow over the car. The ANSYS Fluent has been used for the simulation of Aerodynamics. To reproduce the air flow over road vehicles at high velocity, a cubical enclosure of specific dimensions (Table 2) is created in the Design modeller (Fig 12) by subtracting the car model from the air domain using Boolean operation. The generated enclosure is large enough to simulate the smooth airflow from the inlet surface avoiding the artificial acceleration of airflow due to the collision of air particles within the gap between car body and surface walls. The surface of the enclosure facing the car is named “Inlet” and the rear end “Outlet.” The surfaces surrounding the car are named “Outside Walls.”

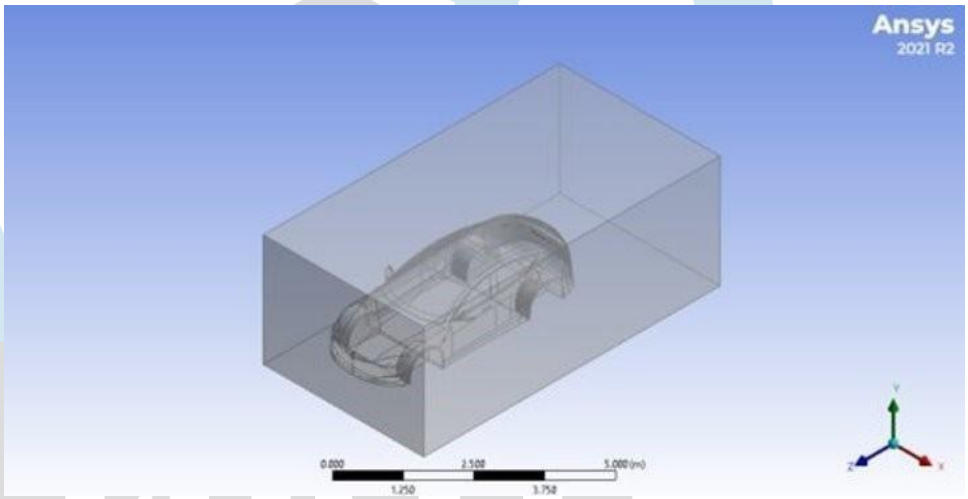


Fig. 12: Enclosure model of the Car in Design modeler

Table 2. Dimensions of Enclosure

Enclosure Dimension	Value (cm)
Length	740
Breadth	280
Height	400

3.2.1 Meshing

Meshing subdivides the model into finer elements over which a set of equations are solved to generate the results. Therefore, the finer the mesh, the more accurate and precise the results. The entire enclosure was meshed using tetrahedral elements and the ‘finite element method’ technique is used. The input parameters are defined (Table 3) and is same for every car model with different spoilers [21]. The

material of the car and spoiler is given in default as aluminum. The final mesh result of the car without spoiler is shown (Fig 13) generating a 3D grid with 52854 nodes and 281613 tetrahedral elements.

Table 3. Mesh Input Parameters

Input Parameter	CFD Value
Solver Preference	Fluent
Element Order	Linear
Face Sizing Car	10 cm
Face Sizing Enclosure	20 cm
Growth Rate	1.2
Mesh Skewness	0.9
Mesh Smoothing	Medium
Transition Ratio	0.272

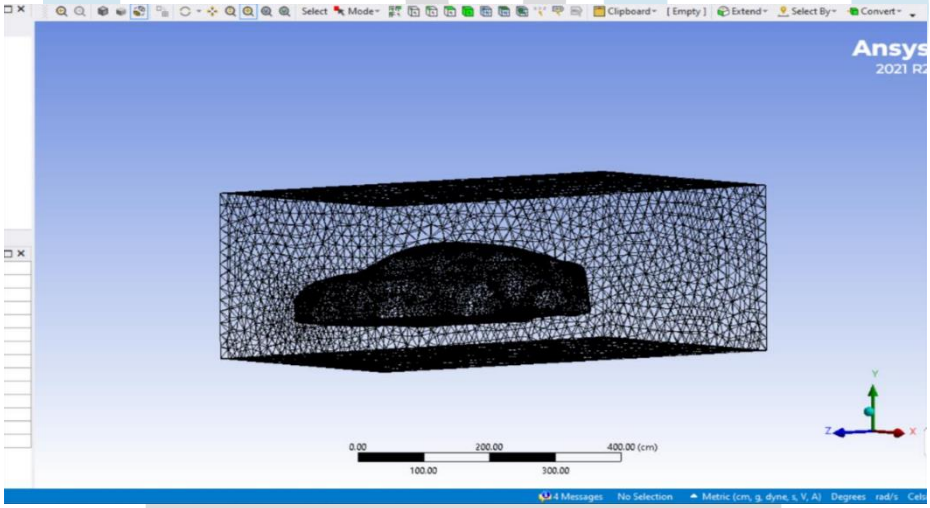


Fig. 13: Mesh of the Car and Enclosure made using ANSYS

3.2.2 Boundary Conditions

In Fluent Analysis, a coupled equation system with a preconditioning method is used to improvise the convergence performance. The realizable k-ε turbulence model with nonequilibrium wall functions is used for the analysis at steady state since it accurately predicts the flow rate of planar and round jets and provides superior performance for flows involving rotation, boundary layers under adverse pressure gradients, separation, and recirculation. The boundary conditions were chosen such that it depicts the same scenario as of the car moving at high velocity on roads under normal atmospheric conditions. The fluid medium is air with inlet velocity of 120 kmph [9] and properties shown in Table 4. The outlet is considered as a pressure outlet at constant atmospheric pressure or 0-gauge pressure.

The outside walls and car are taken as 'no slip' boundary condition. The reference values are computed from inlet with a projected reference area of 11.2 m^2 and viscosity of 1.7894e-05 kg/ms. A second-order upwind difference scheme is used for pressure and momentum discretization for better convergence and stability. The Turbulent kinetic Energy and Dissipation rate follows second order upwind. The model is further initialized for fluent analysis for 100 iterations to find the drag and lift coefficients of the car [12].

Table 4. Properties of Air

Air Properties	Value
Inlet Velocity	120 kmph
Density	1.225 kg/m^3
Kinematic Viscosity	10^{-5} kg/ms

3.3 Virtual Driving Simulation using IPG Carmaker

Virtual driving simulations are computer-based simulations that replicate real-world driving scenarios for the purpose of training, research, or entertainment. IPG Carmaker is often used to create virtual driving simulations for vehicle development and testing. With its advanced modelling and simulation capabilities, IPG Carmaker allows us to create realistic virtual environments that simulate various driving scenarios, such as different road surfaces, weather conditions, and traffic situations.

Using IPG Carmaker, we simulated the virtual driving test of the car models. The simulation process is explained in the following steps:

- We start at the Carmaker main GUI window (Fig, 14), where we open a new file.
- Then, we start the simulation process by creating a scenario in IPG Carmaker using the Scenario Editor (Fig. 16). Here, we create the simulation environment based on our needs.
- We import the vehicle parameters in IPG Carmaker using the vehicle generator option.
- In the maneuver window (Fig. 17), we designate parameters like, duration of the simulation, driver settings and commands.
- The IPG Control panel (Fig.18) represents all parameters that are generated during the simulation. These are user accessible quantities like Speed, Torque, Fuel Consumption, etc.
- The simulation can be visualized using IPG Movie (Fig. 15), where we can play, pause and repeat the simulation at any given rate or time.
- After successful completion of the virtual simulation the user accessible quantities can be verified in IPG Control where they will be displayed as a graph.

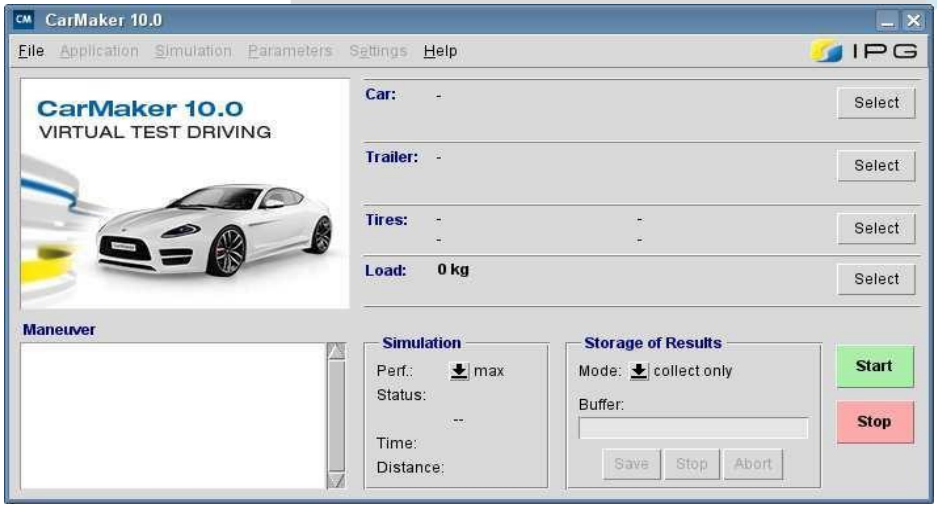


Fig. 14: Carmaker main GUI

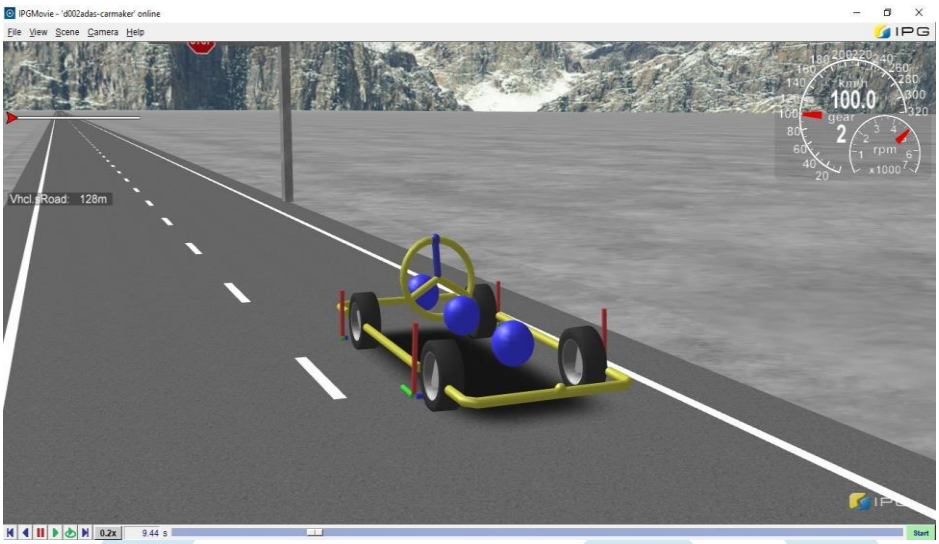


Fig. 15: IPG Movie window

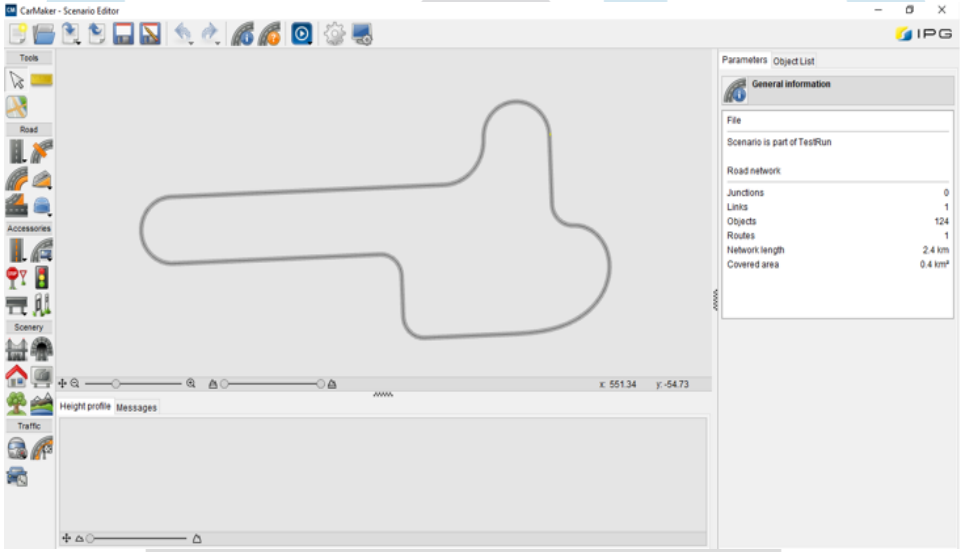


Fig. 16: Scenario Editor Window

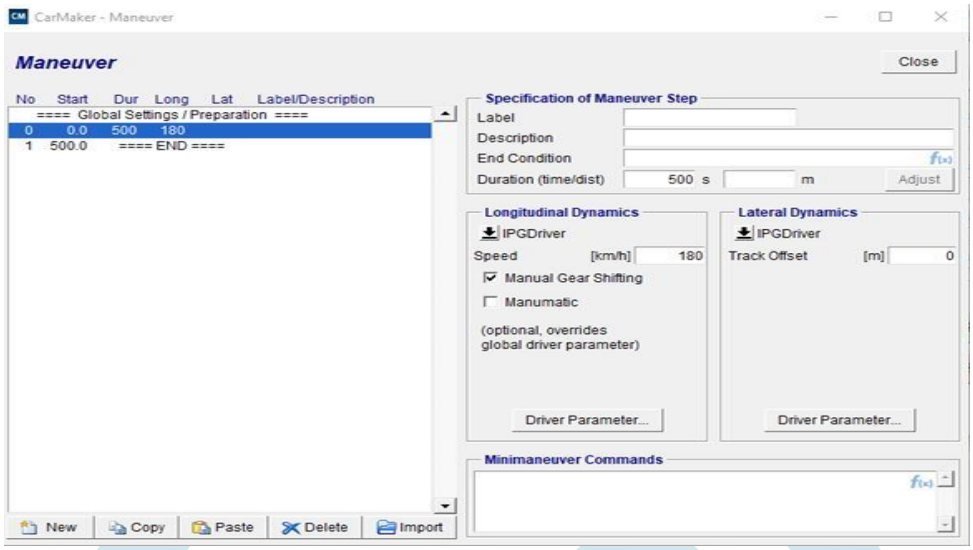


Fig. 17: Maneuver window

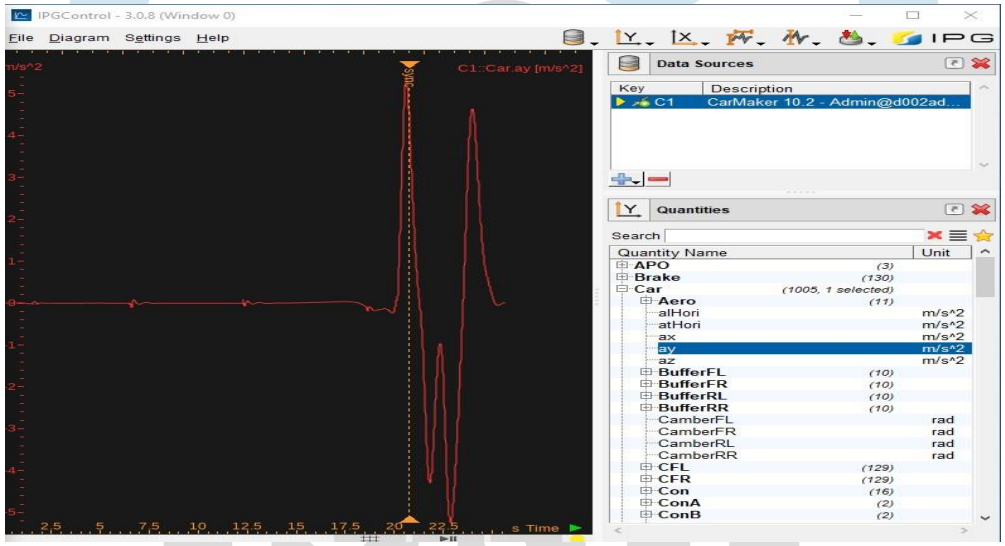


Fig. 18: IPG Control window

3.3.1 Scenarios and KPIs used for the Virtual Driving Simulations

Scenarios consist of specific conditions or events that a vehicle is expected to encounter during a simulation such as such as road surface conditions, traffic density and weather conditions, while KPIs are performance metrics used to evaluate the vehicle's performance in a given scenario such as maximum speed, fuel efficiency, braking distance and acceleration time. By simulating a wide range of scenarios and measuring a variety of KPIs, we can gain valuable insights into how the vehicles will perform in the real world and make informed decisions about design changes or performance optimizations. Table 5 shows the scenarios and its Key Performance Indicators.

Table 5. List of Scenarios and KPIs used

Scenario	KPIs
Custom Track	Top Speed & Lap time

Braking Distance	0-100 Time & Stopping distance
Corner Braking	Yaw Rate & Body Roll
Lateral Acceleration	Acceleration in Y- Axis

a) **Custom Track:** For this study, a custom track was created using Scenario Editor (Fig. 19). The Car Models were tested in this track and Top Speed & Lap Time were measured (Fig. 20).

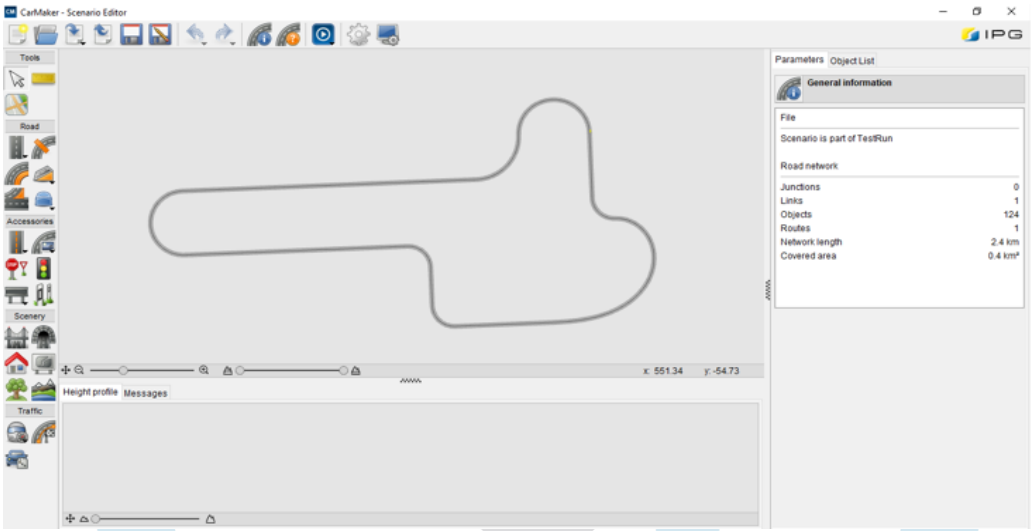


Fig. 19: Custom Track made in Scenario Editor Window

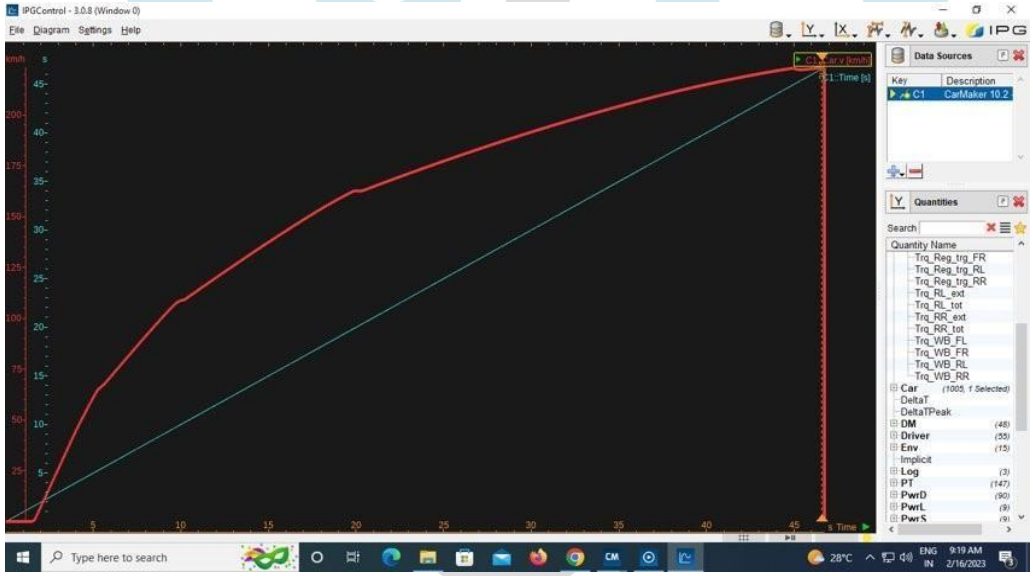


Fig. 20: IPG Control window – Custom track

b) **Braking Test**: The braking test is one of the standard test procedures for vehicles. The test is performed on a dry road surface with a homogenous friction coefficient. The vehicle test speed was set to 150 kmph. The track (Fig. 21) consists of a long straight where the time for acceleration till 100kmph of the c from a standstill is measured, which is followed by the brake test where the braking distance is measured (Fig. 22).

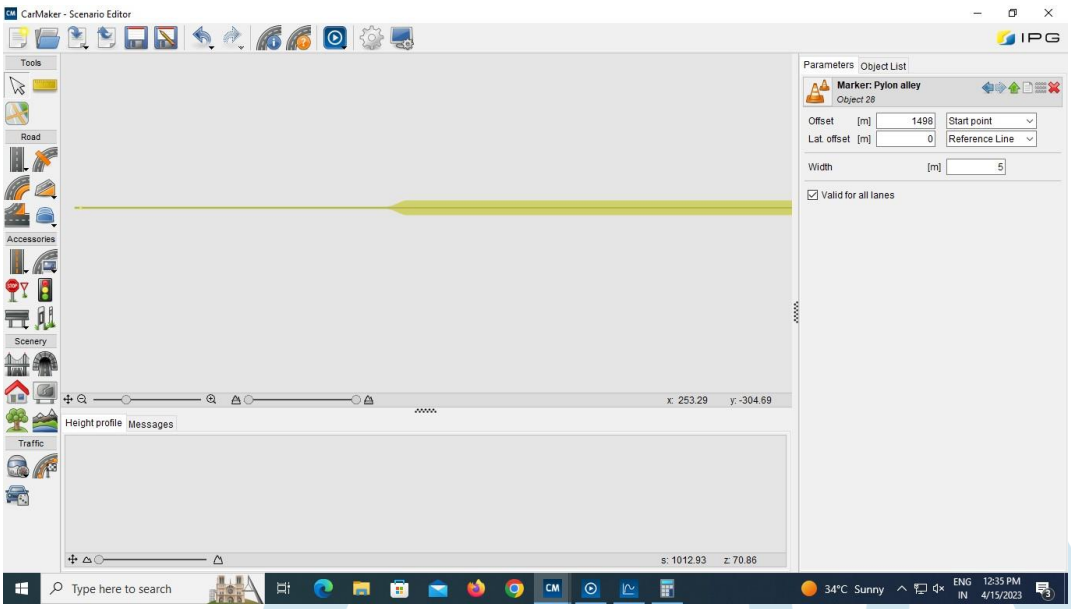


Fig. 21: Braking Test Track in Scenario Editor

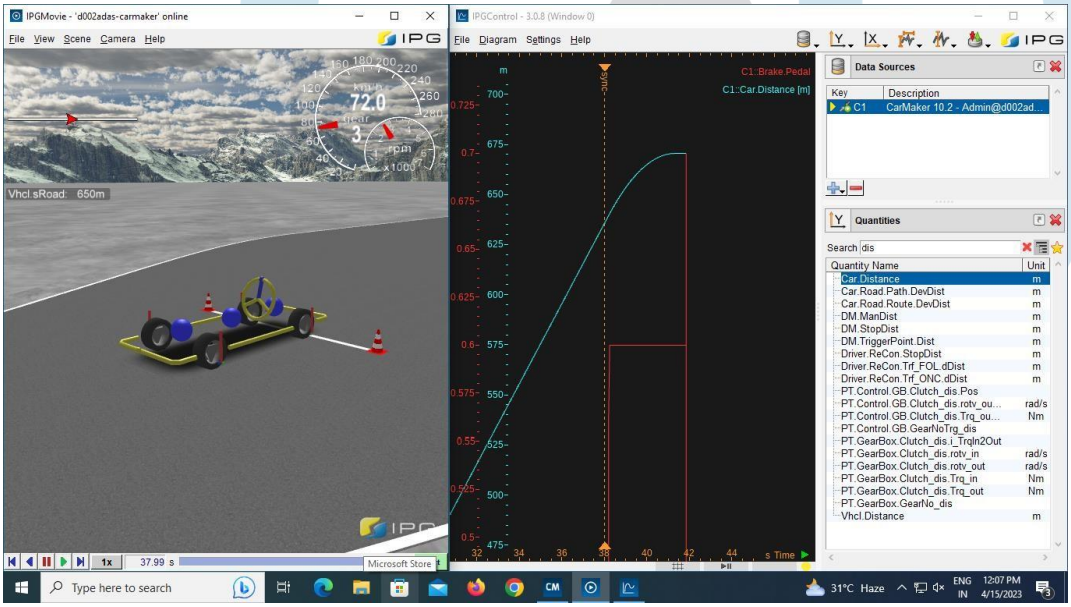


Fig. 22: IPG Movie Visualization and IPG Control Graph for Braking Test

c) **Corner Braking**: This Test Run (Fig. 23) shows a braking maneuver while driving through a corner. The Yaw-rate and Body Roll of the car are measured (Fig. 24) along the turn when the brakes are applied.

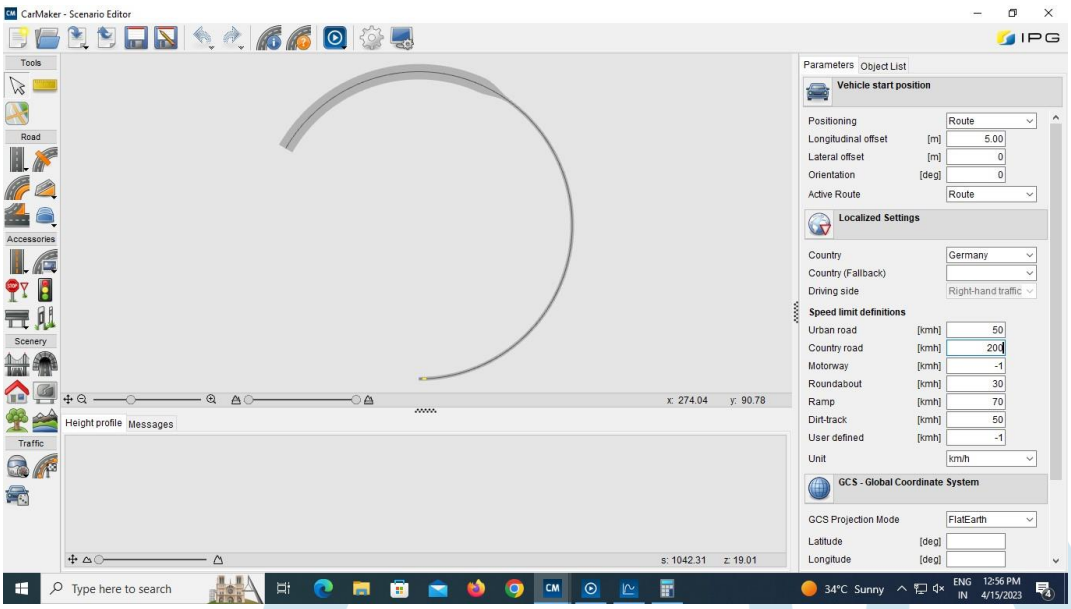


Fig. 23: Corner Braking Test Track in Scenario Editor

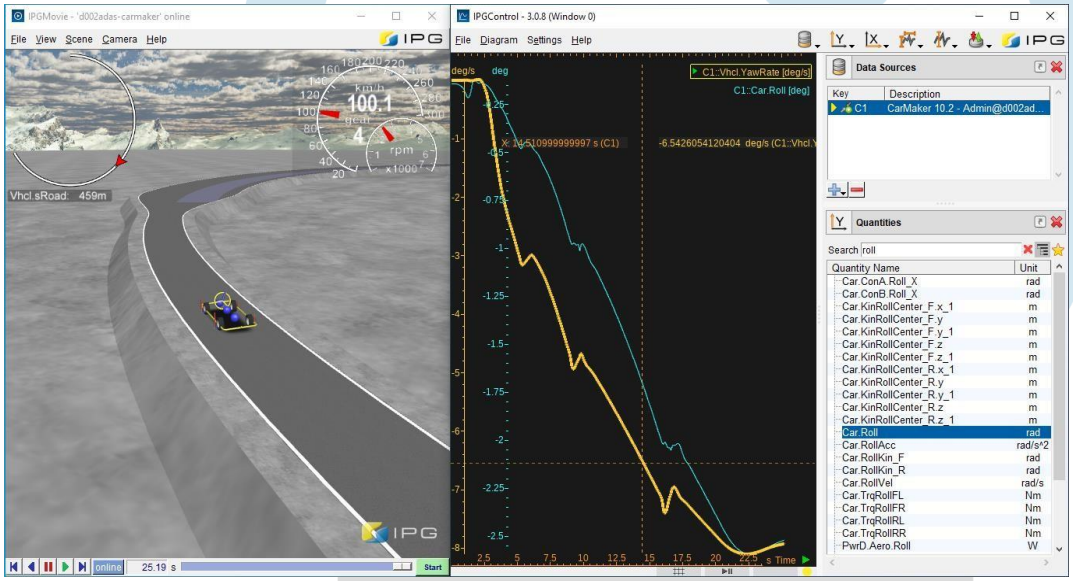


Fig. 24: IPG Movie Visualization and IPG Control Graph for Corner Braking Test

d) **Lateral Acceleration**: The small racetrack of the grand prix circuit "Hockenheim", a 2.6 km long racetrack (Fig. 25), will be used for demonstration purposes. The Lateral Acceleration is measured (Fig. 26) along this track as a factor of determining the vehicle's handling

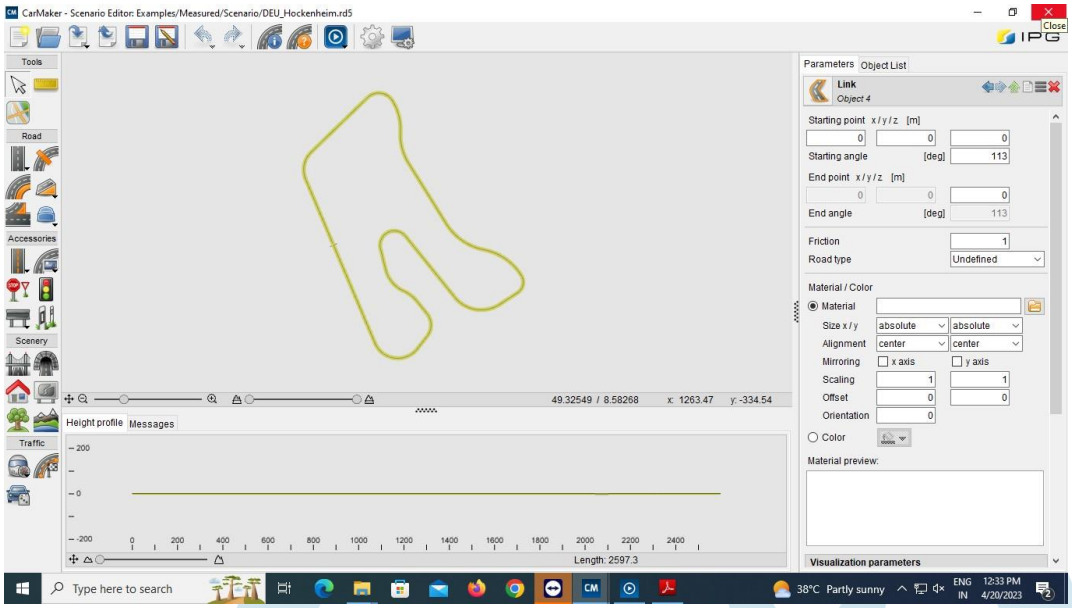


Fig. 25: Lateral Acceleration Test Track in Scenario Editor

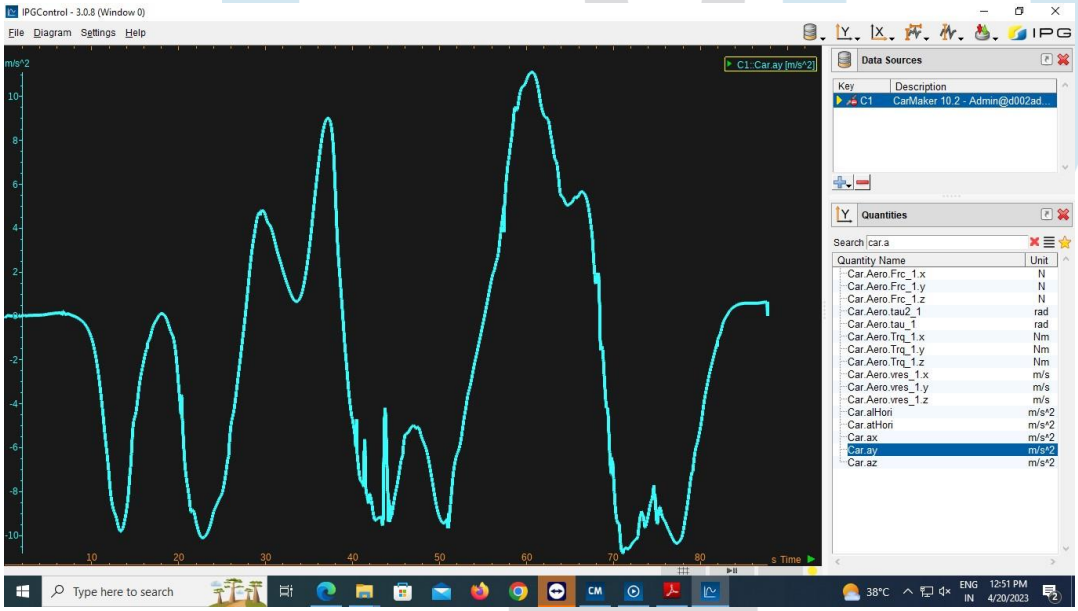


Fig. 26: IPG Control Graph for Lateral Acceleration

4. RESULTS AND DISCUSSION

The calculation of drag and lift coefficients of the car with and without spoilers has been done by solving the average Navier-stokes equation using same input parameters and the comparison of drag-lift coefficient values is shown (Table 6). It is observed that the drag coefficient of car without spoiler is 0.208 which is similar to the real drag coefficient value of the Tesla S Model Car. This verifies the

input parameters, the model and the procedure for preliminary analysis is valid [22] [23]. A graphical representation of drag coefficient (Fig 27) and lift coefficient (Fig 28) is shown to understand the efficiency of each spoiler in improvising the performance of the vehicle. It has been observed that the lift coefficient values decrease "negatively" for each spoiler. Negative lift means downforce. Spoiler 3 has higher downforce in comparison to Spoiler 2 and Spoiler 1 which indicates that Spoiler 3 provides much more stability to the cars when moving at high speeds by reducing the lifting of cars and providing adequate downforce which results in better traction.

Table 6. Drag and Lift Coefficient Values of Car with and without Spoilers

Car Model	Drag Coefficient C_d	Lift Coefficient C_l
Base Car	0.208	-0.07
Car Mod-1	0.245	-0.203
Car Mod-2	0.305	-0.309
Car Mod-3	0.343	-0.363

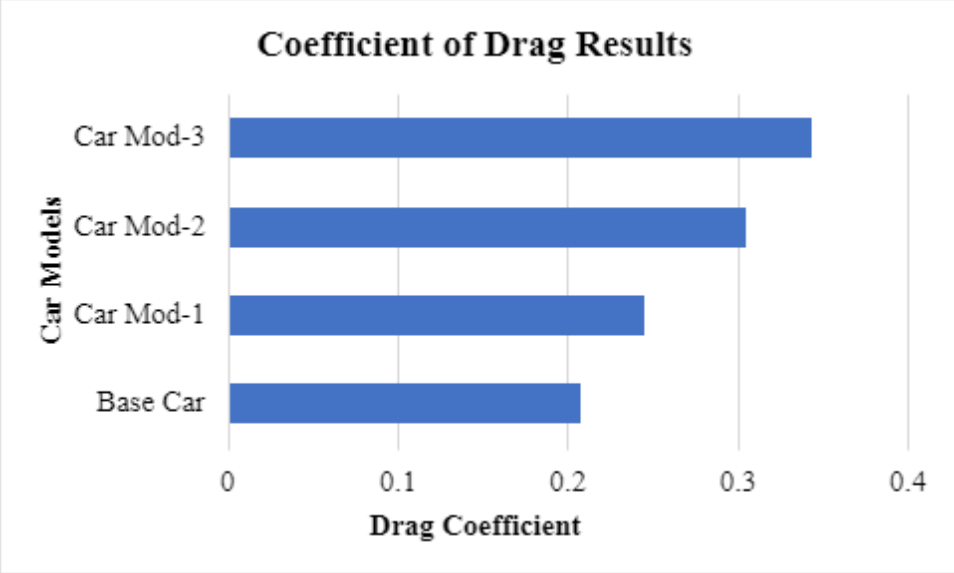


Fig 27: Graphical Representation of Coefficient of Drag Results

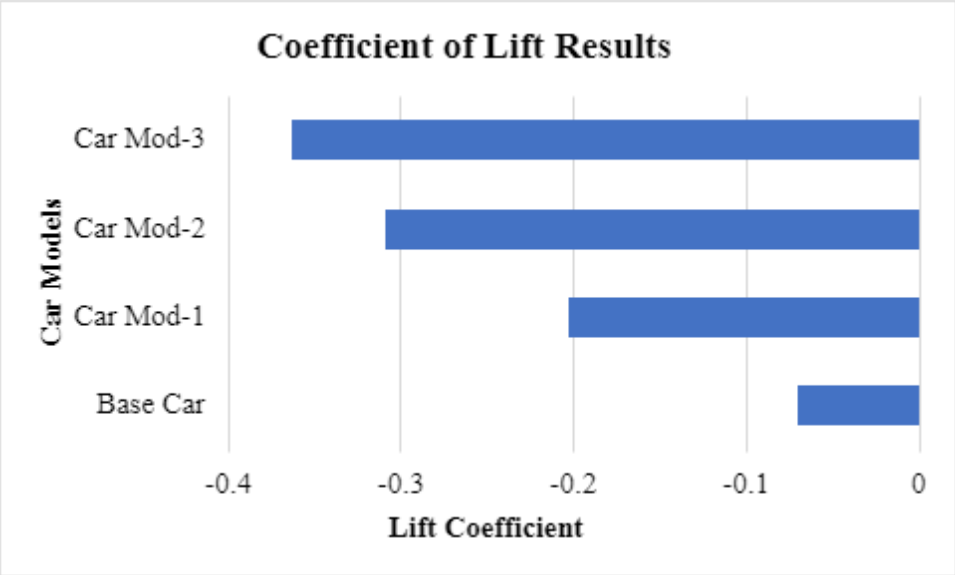


Fig 28: Graphical Representation of Coefficient of Lift Results

The graph shown in Fig 29 shows the percentage increase in Drag coefficient compared to the base car. Car Model 1 has less Drag percentage in comparison to Car Model 2 and 3. The Car model 2 has nearly 50 percentage drag which indicates that there is a good balance between drag and lift forces. The car model 3 has higher drag percentile thereby providing necessary drag for the vehicle to slow down and provide traction even for speeds higher than 120 kmph.

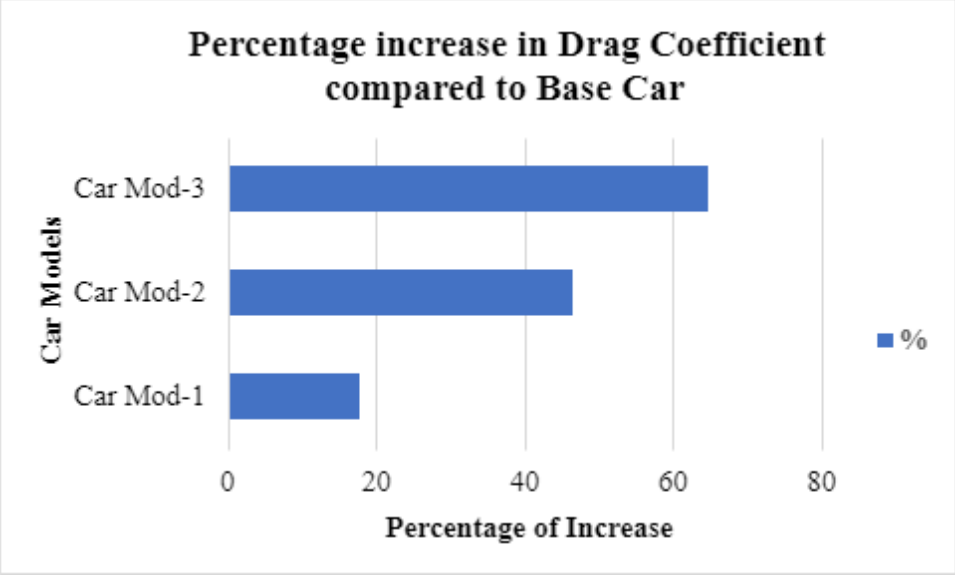
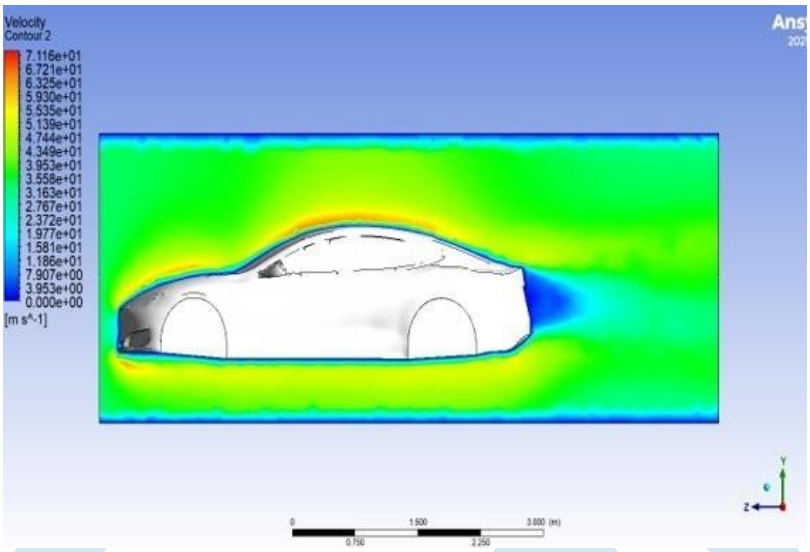
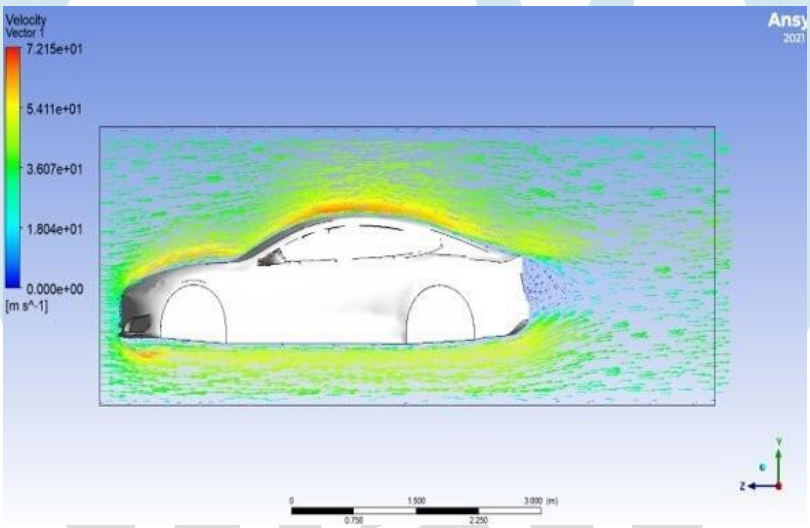


Fig. 29: Graphical Representation of Percentage Increase in Coefficient of Drag compared to Base Car

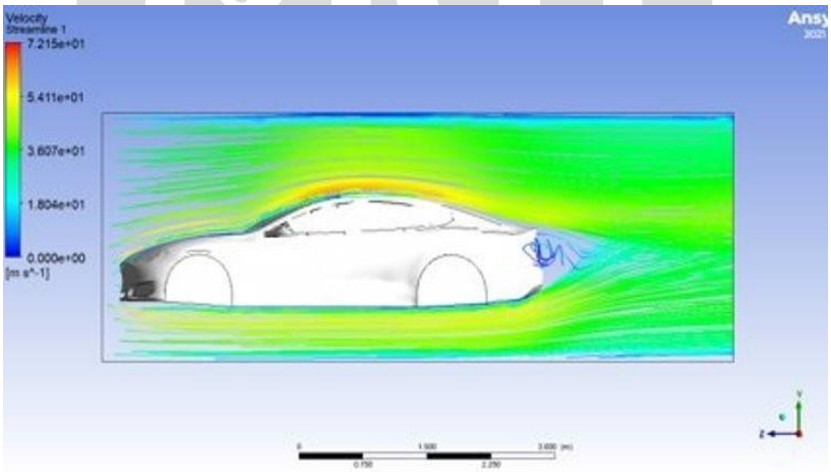
The flow of air over car's body creates a velocity distribution that generated aerodynamic loads all over the body. It indicates the amount of velocity at different portions of the car body. Figure 30 shows the velocity vector, contour and streamline flow of air over the car without spoiler. A high velocity region can be seen in front trunk of the car which indicates low pressure according to Bernoulli's theorem, causing the vehicle to lift. A high amount of air turbulence has been observed at the rear end of the car which causes aerodynamic drag. This aerodynamic drag or lower pressure region generated due to airflow separation can slow the vehicle's speed thereby reducing the performance and overall stability of the vehicle.



(a)



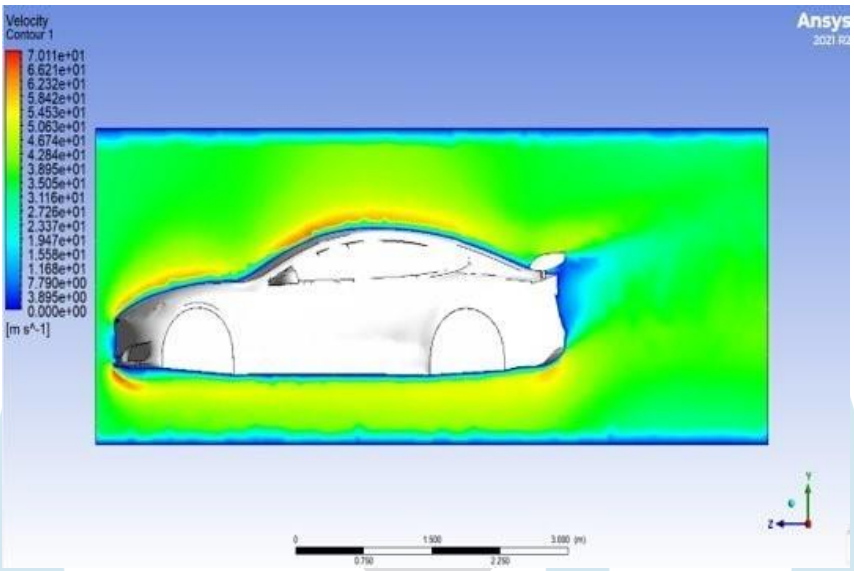
(b)



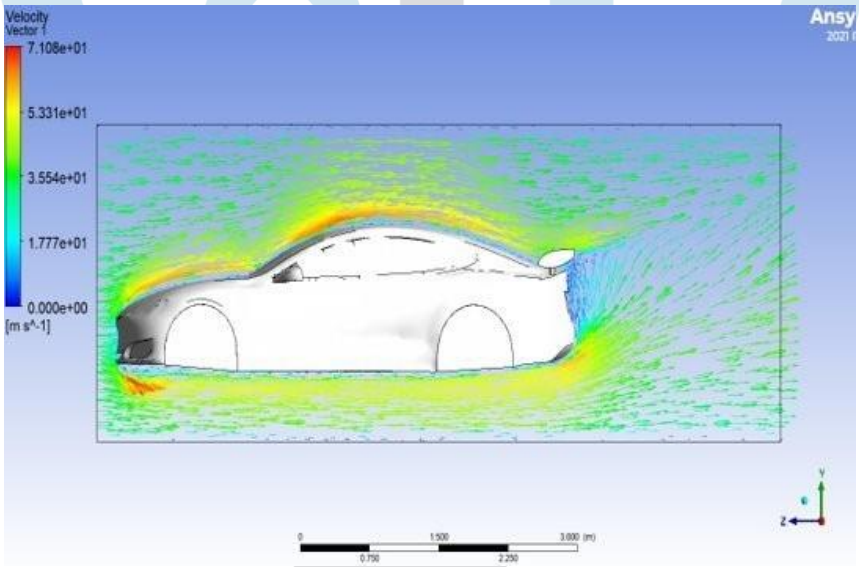
(c)

Fig. 30: (a) Velocity Contour of the Car without Spoiler (b) Velocity Vector of the Car without Spoiler (c) Velocity Streamline of Car without Spoiler

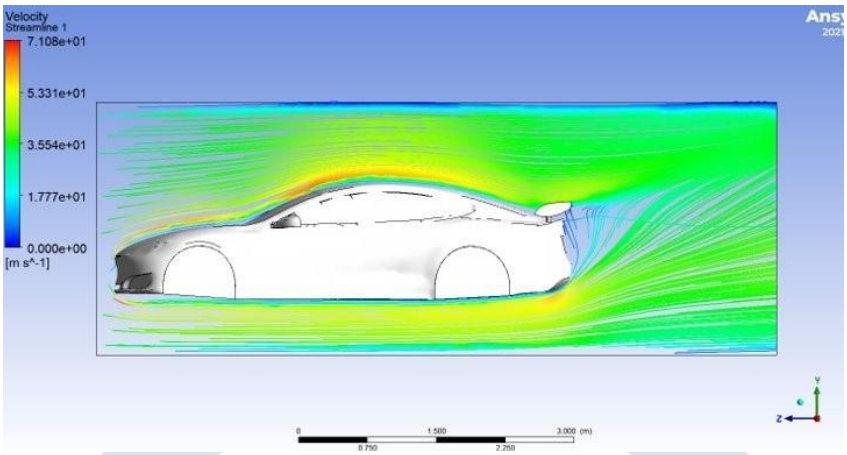
The velocity value of airflow over car is directly proportional to drag and lift coefficients of the car. The purpose of spoiler is to change the direction of airflow which causes reduction in lift force thereby increasing the performance of the car. Figure 31 shows the velocity vector, contour and streamline flow of air over the car with spoiler 1(Car Mod-1). The course of airflow at the rear end of the vehicle has been 'changed' or 'spoiled' due to the addition of spoiler and the air turbulence or drag is low in comparison with drag for the car without spoiler.



(a)



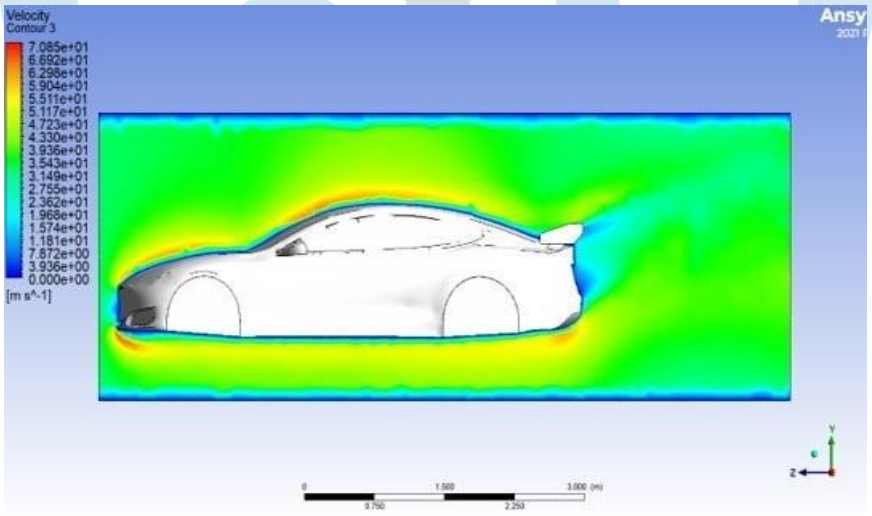
(b)



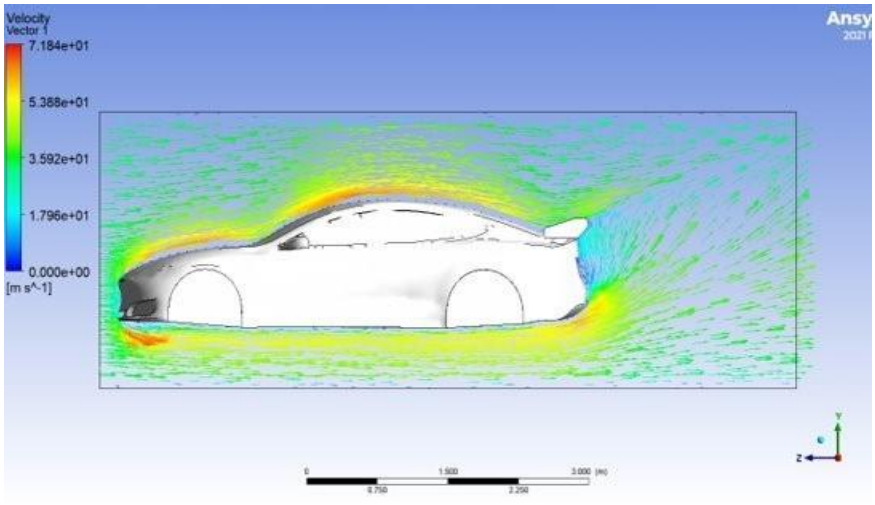
(c)

Fig. 31: (a) Velocity Contour of the Car with Spoiler 1 (b) Velocity Vector of the Car with Spoiler 1 (c) Velocity Streamline of Car with Spoiler 1

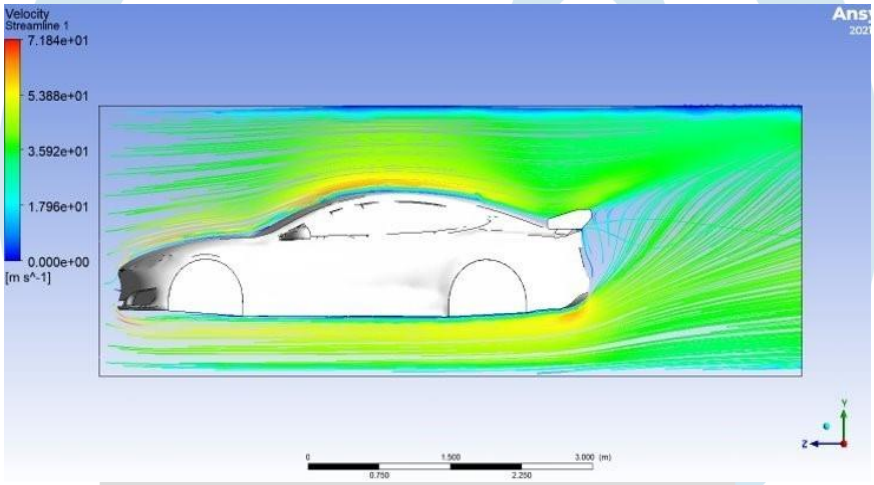
The velocity distribution of spoiler 2 (Car Mod-2) (Figure 32) is different from Spoiler 1 (Car Mod-1). The airflow is 'longer', and much smoother flow of air is observed from the roof of spoiler 2. It delays the airflow separation thereby reducing the drag which improvises stability and performance of the vehicle.



(a)



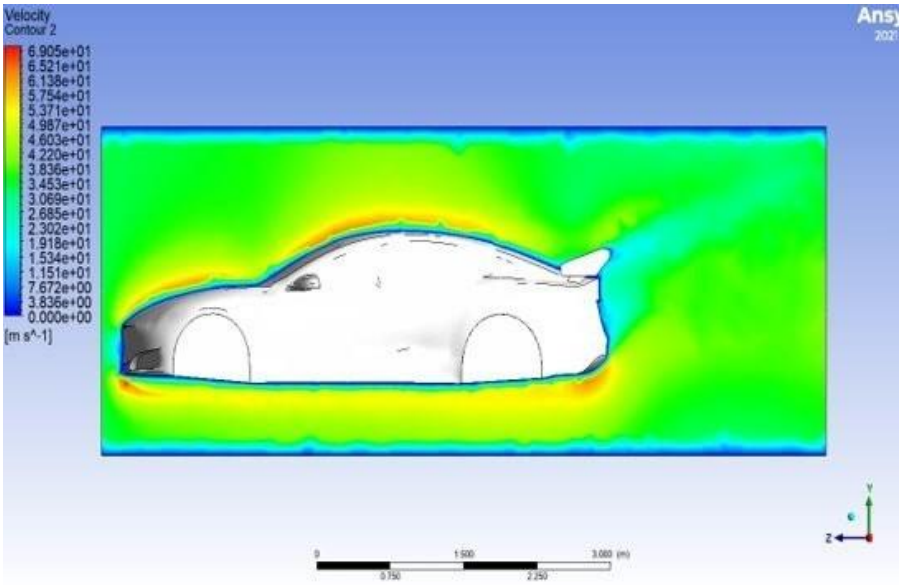
(b)



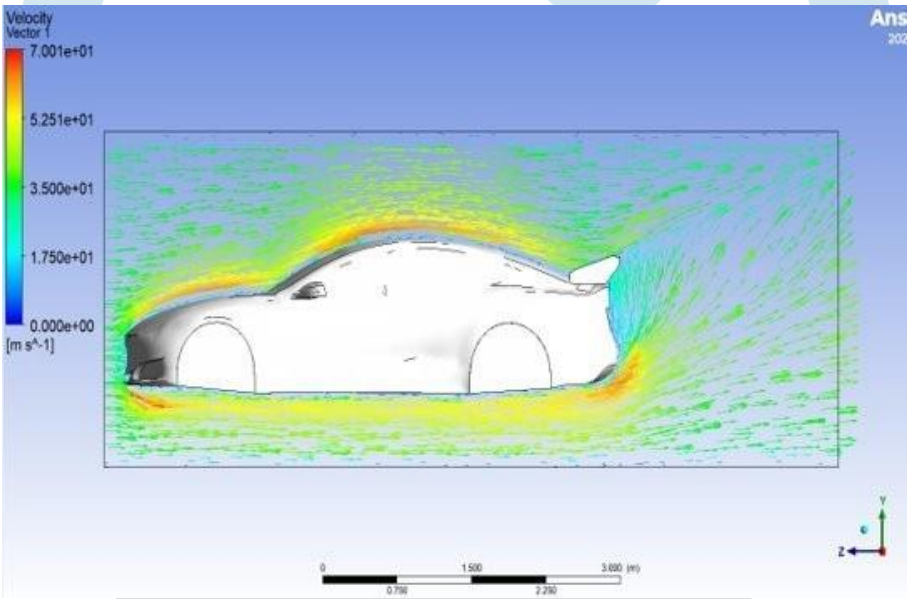
(c)

Fig. 32: (a) Velocity Contour of the Car with Spoiler 2 (b) Velocity Vector of the Car with Spoiler 2 (c) Velocity Streamline of Car with Spoiler 2

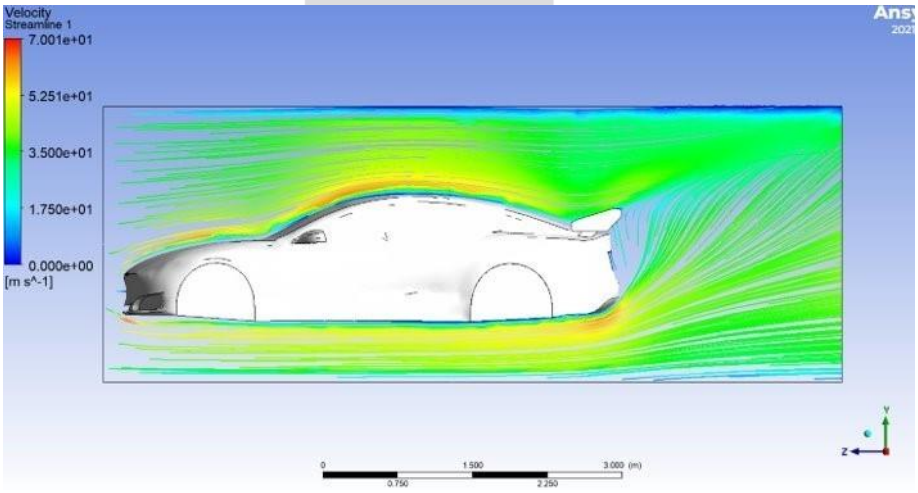
The velocity distribution of spoiler 3 (Car Mod-3) (Fig 33) is much advanced in comparison to spoiler 2 and spoiler 1. The change in aspect ratio and the angle of attack has significantly changed the course of airflow over the spoiler resulting in a thin airflow separation or turbulence at the rear end of the vehicle. This reduces the lift and improvises the drag providing better stability to the vehicle.


Velocity
Contour 2
6.905e+01
6.521e+01
6.138e+01
5.754e+01
5.371e+01
4.987e+01
4.603e+01
4.220e+01
3.836e+01
3.453e+01
3.069e+01
2.685e+01
2.302e+01
1.918e+01
1.534e+01
1.151e+01
7.672e+00
3.836e+00
0.000e+00
[m s^-1]

(a)


Velocity
Vector 1
7.001e+01
5.251e+01
3.500e+01
1.750e+01
0.000e+00
[m s^-1]

(b)


Velocity
Streamline 1
7.001e+01
5.251e+01
3.500e+01
1.750e+01
0.000e+00
[m s^-1]

(c)

Fig. 33: (a) Velocity Contour of the Car with Spoiler 3 (b) Velocity Vector of the Car with Spoiler 3 (c) Velocity Streamline of Car with Spoiler 3

From the Figure 31, 32 and 33, it can be observed that the turbulence or wake region on the rear end of the car has undergone airflow changes which avoids the streamlines from converging thereby forming smooth streamline flow over the car. Turbulence presents the disturbance that occurs behind a body as it moves around the fluid. Therefore Spoiler 1, 2 and 3 obstructs the airflow and prevents the air from being pulled downwards which would otherwise create low pressure air pockets on the trunk of the vehicle causing unnecessary pressure drag.

The results of virtual driving simulations in IPG Carmaker using the above-mentioned scenarios are as below:

a. **Custom Track**:

Table 7. Lap times and Top speed

Car Model	Lap TIme (s)	Top Speed (kmph)
Base Car	88.9	161.75
Car Mod-1	88.85	161.86
Car Mod-2	88.86	161.66
Car Mod-3	89.07	158.55

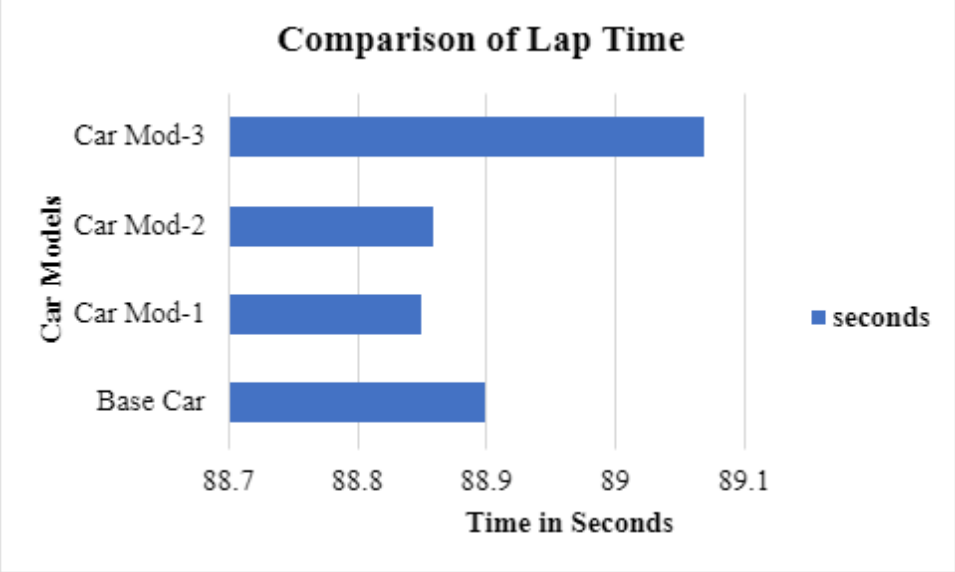


Fig. 34: Graphical Comparison of Lap times

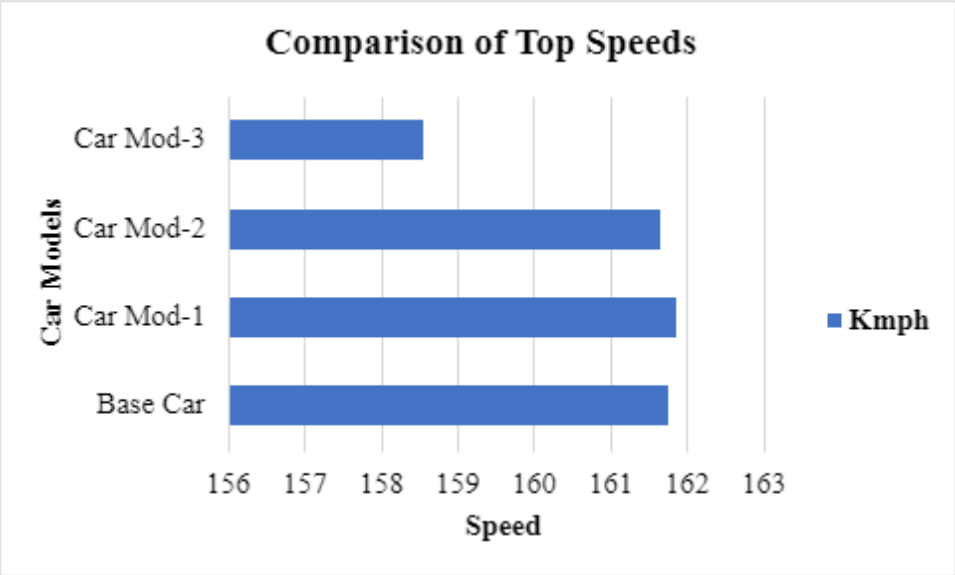


Fig. 35: Graphical Comparison of Top Speeds

From Table 7 we can conclude that the 1st Car Mod delivers the best result with improved Lap Time and Top Speed. A lesser lap time in a virtual driving simulation or real-world track test typically indicates that the car is more stable and performs better. Stability is an essential factor in a car's performance, especially in high-speed turning, as it ensures that the vehicle maintains its intended trajectory and does not slide or spin out of control. A car with good stability will be able to maneuver at higher speeds with more confidence and control, resulting in faster lap times.

b. **Braking Test:**

Table 8. 0-100 Acceleration Time and Braking Distance

Car Model	0-100 Time (s)	Braking Distance 0-150-0 (m)
Base Car	9.44	139
Car Mod-1	9.19	137
Car Mod-2	8.88	136
Car Mod-3	8.74	137

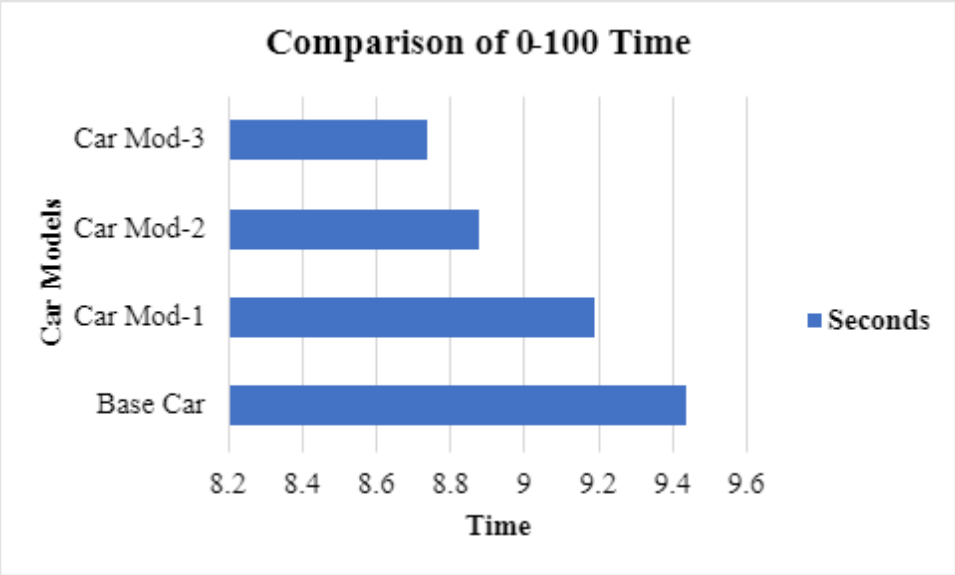


Fig. 36: Graphical Comparison of 0-100 Time

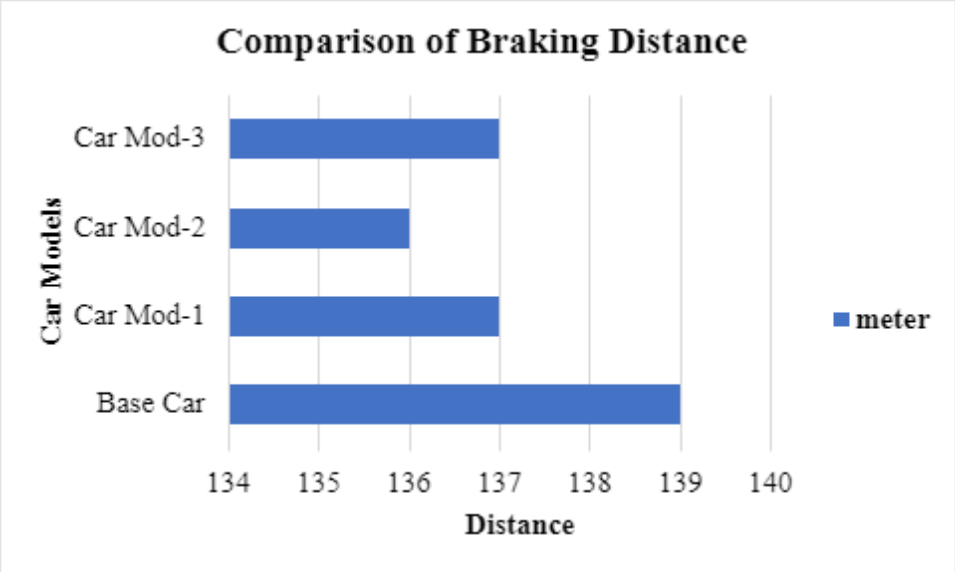


Fig. 37: Graphical Comparison of Braking Distances

Table 8 shows that the 2nd Car mod delivers better acceleration and straight-line braking. Better acceleration of a car generally indicates that the vehicle has good stability, as it suggests that the car is able to transfer power from the engine to the wheels effectively without losing traction or losing control. Good stability is essential for achieving better acceleration, as it allows the car to maintain its intended trajectory and maximize its grip on the road surface.

A lesser braking distance typically indicates that a car has good stability, as it suggests that the vehicle is able to slow down quickly and effectively without losing control or skidding. Good stability is essential for achieving a lesser braking distance, as it allows the car to maintain its intended trajectory while braking and maximize its grip on the road surface.

c. **Corner Braking Test:**

Table 9. Yaw Rate and Body Roll

Car Model	Yaw Rate (deg/s)	Body Roll (deg)
Base Car	24.64	2.69
Car Mod-1	24.201	2.62
Car Mod-2	23.14	2.5
Car Mod-3	23.1	2.44

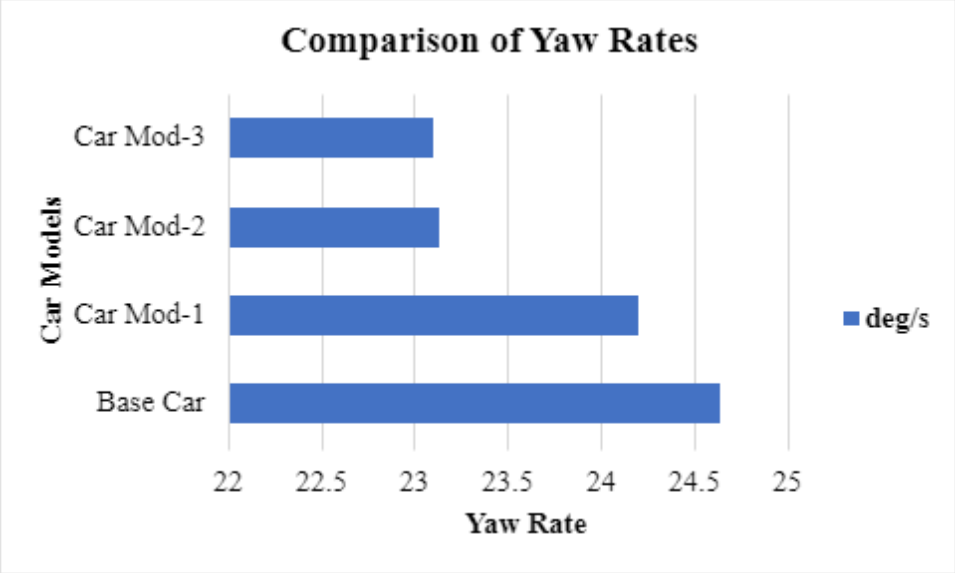


Fig. 38: Graphical Comparison of Yaw Rates

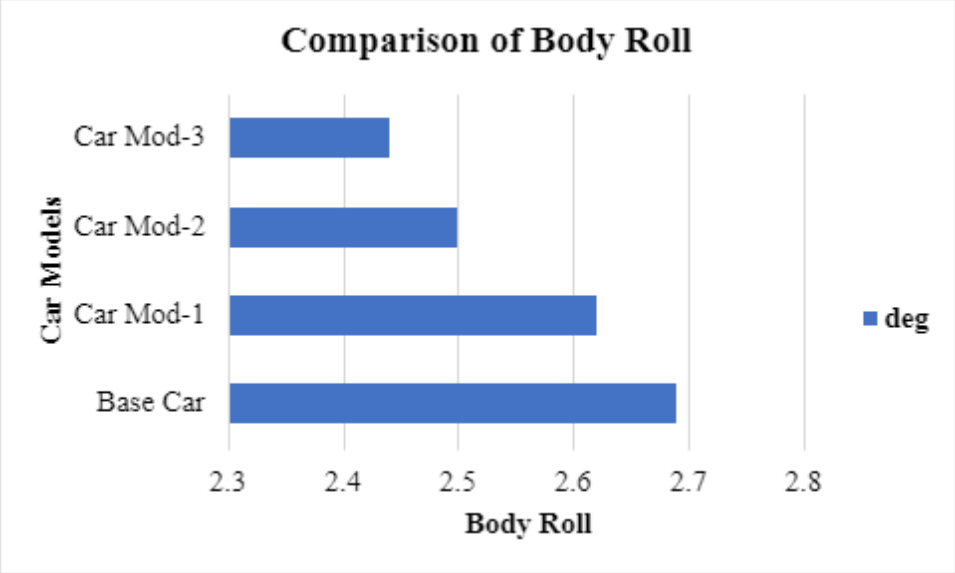


Fig. 39: Graphical Comparison of Body Roll

From the Table 9 we conclude that the 3rd Mod is the best as it has the lowest Yaw Rate and Body Roll. A car with a lower yaw rate tends to be more stable, as it is less prone to oversteer or understeer during turns. It is generally more resistant to oversteer and understeer, as it is able to maintain its intended trajectory during turns. A car with reduced body roll can improve its stability and handling characteristics as Excessive body roll can lead to a loss of traction, reduced stability, and increased risk of rollover or other accidents.

d. **Lateral Acceleration:**

Table 10. Lateral Accelerations

Car Model	Car Acceleration in Y- Axis Range while Right Turns (m/s2)	Car. Acceleration in Y-Axis Range while Left turns (m/s2)
Base Car	11.109	10.82
Car Mod-1	11.121	11.423
Car Mod-2	11.132	11.39
Car Mod-3	11.111	11.36

A car with high lateral acceleration capabilities is generally considered to be more stable than a car with low lateral acceleration capabilities. Hence, the 1st Car Mod is the ideal option in this scenario.

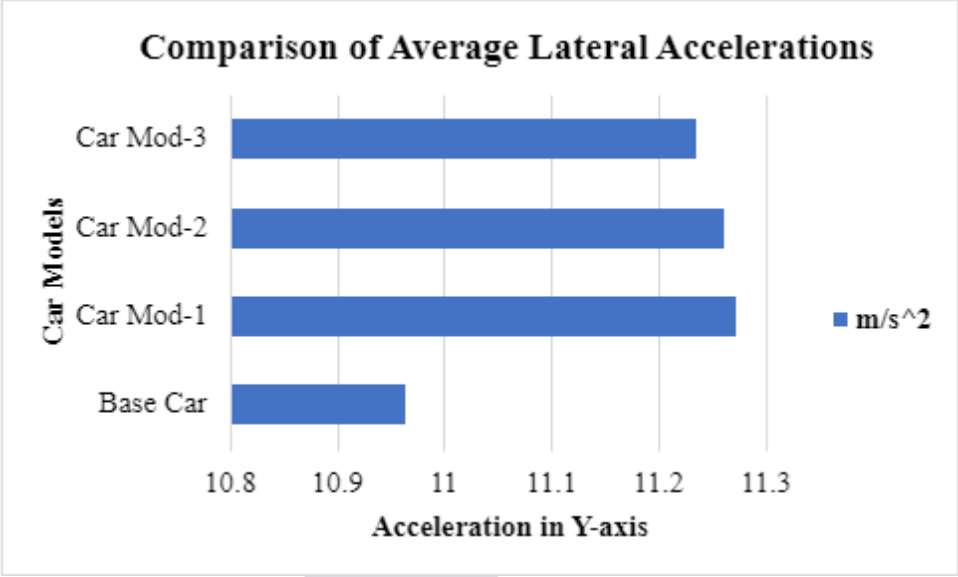


Fig. 40: Graphical Comparison of Average Lateral Accelerations

4.1 ANSYS RESULTS:

It has been observed from the fluent analysis that the airflow over Spoiler 3 is smooth with little or no turbulence in comparison with base car, Car Mod-1 and Car Mod-2. The Velocity vector flow of Car Mod-3 shows that the convergence of air molecules has been reduced and is deviated to generate a streamline flow behind the trunk of the car. This airflow reduces the wake behind the trunk thereby providing high performance and better traction to the car which increases stability. The same results are also observed from the drag-lift coefficient table (Table 6) which shows that the Car Mod-3 has higher drag coefficient and lift coefficient when compared to other Car Models thereby providing adequate downforce with proper balance between drag and lift forces. Thus, we can conclude that Car Mod-3 is efficient when it comes for better traction and stability.

4.2 IPG CARMAKER RESULTS:

We can note from the IPG Carmaker results that the overall best stability is provided by the Car Mod-3. Various scenarios and KPIs have been considered such as Body Roll and Yaw Rate in Corner Braking and Lateral Accelerations during turns. We can see that Car-Mod 3 has shown consistent results in all the tests, except the top-speed test. Due to the increase in drag of the car, the top speed of the car is expected to reduce, but that minor loss in top speed (2–4 kmph) comes with a major upgrade in the overall stability of the car. The upgrade in the stability is given by its better Braking distance and 0-100 acceleration time, better (lesser) Yaw Rate and Body Roll. Even in the Lateral Acceleration scenario, it shows better results when compared with the base car. Considering Car Mod-3's consistent performance in the scenarios created in virtual driving simulations, we can conclude that it is the most efficient in the matter of stability.

4.3 CONNECTION BETWEEN BOTH SOFTWARE RESULTS:

ANSYS Fluent was used in a study to examine the effects of various factors on the car's drag coefficient. The results of this study indicated that increasing the angle of attack (the angle between the car's body and the airflow), as well as changing the aspect ratio (the ratio of the car's wingspan to its average chord length) and height of the spoiler, led to an increase in the car's drag coefficient. However, we should note that while ANSYS is effective in providing insights into the changes in drag coefficient, it is unable to provide information on real-time performance metrics such as lap time, top speed, or acceleration. These metrics are important for evaluating the overall performance of a car in practical driving conditions.

To address this limitation, we have utilized IPG Carmaker, another software program that specializes in driver-in-loop simulations. IPG Carmaker takes the drag coefficient values obtained from ANSYS and incorporates them into its simulations. By running these simulations, IPG Carmaker can provide real-time values for metrics such as lap time, top speed, and acceleration. This allows researchers and engineers to assess the car's performance more comprehensively and understand how changes in aerodynamic properties affect real-world driving dynamics.

In summary, the combination of ANSYS and IPG Carmaker provides a powerful toolset for analyzing and simulating the aerodynamic performance of a car. ANSYS helps derive the drag coefficient values, while IPG Carmaker enables the evaluation of real-time performance metrics through driver-in-loop simulation

5. CONCLUSION

Spoilers play a critical role in optimizing vehicle performance, especially when traveling at high speeds. By carefully designing and implementing an appropriate spoiler, the car can achieve enhanced stability and improved handling characteristics.

The utilization of CFD and virtual driving simulations offers significant advantages in the design and validation process. By leveraging these simulation techniques, engineers can thoroughly analyze the airflow dynamics around the car and assess the effects of the spoiler on performance metrics such as downforce, drag reduction, and stability. This approach provides valuable insights into the spoiler's impact without the need for costly and time-consuming physical prototyping.

The integration of CFD and virtual driving simulations enables a more efficient and cost-effective design process. Design iterations can be performed virtually, allowing for quick and accurate assessment of different spoiler designs and configurations. This streamlines the overall development timeline and reduces the reliance on physical testing. Consequently, the integration of these simulation techniques proves instrumental in optimizing spoiler design and improving overall vehicle performance.

In conclusion, the successful integration of CFD and virtual driving simulations demonstrates the benefits of utilizing spoilers in car design. Spoilers enhance vehicle performance by increasing downforce and improving stability, which are crucial factors for achieving optimal handling and control, particularly at high speeds. The utilization of simulation techniques in the design and validation process enables a more efficient and cost-effective approach, minimizing the need for physical prototyping and facilitating informed decision-making in spoiler design.

REFERENCES

1. Hariantoa, Yosua Heru Irawanb, Eka Yawarac and Husni Bakhtiard (2019), 'Effect of Tapper Ratio on a Car Rear Spoiler Performance', *JEMMME (Journal of Energy, Mechanical, Material, and Manufacturing Engineering)* **4**(1), 1-14
2. Mohsen Jahanmiri and Seyed Ali Fatimi (2019),'Experimental Survey Of the Effect Of Spoiler Height And Diffuser On Aerodynamic Coefficients Of a Generic Car', *International Journal Of Innovations in Engineering Research And Technology [IJIERT]* **6**(03), 17-27
3. Mahmoud Ibrahim Youssef (2022), 'Using CFD Analysis to Investigate the Appropriate Height of the Rear Spoiler on a Car', *International Journal of Mathematics and Physical Sciences Research* **10**(1), 38-44
4. Zhaowen Deng, Sijia Yu, Wei Gao, Qiang Yi and Wei Yu (2020), Review of effects the rear spoiler aerodynamic analysis on ground vehicle performance, *in* 'Journal of Physics: Conference Series', pp. 1600.
5. Ayyagari, D.T. and He, Y. (2017) 'Aerodynamic analysis of an active rear split spoiler for improving lateral stability of high-speed vehicles', *Int. J. Vehicle Systems Modelling and Testing* **12**(¾), 217–239.
6. M. Hammad (2019),'Design and analysis of active aerodynamic control systems for increasing the safety of high-speed road vehicles'.
7. Hammad, M., Qureshi, K and He, Y. (2019), 'Safety and Lateral Dynamics Improvement of a Race Car Using Active Rear Wing Control', *SAE Technical Paper* 2019-**01-0643**, 1-7
8. T.D.Ipilakyaa, L.T.Tuleunand and M.O.Kekung (2018),'COMPUTATIONAL FLUID DYNAMICS MODELLING OF AN AERODYNAMIC REAR SPOILER ON CARS', *Nigerian Journal of Technology (NIJOTECH)* **37**(4), 975 – 980.
9. Muhammad Zaid Nawam, Mohd Afzanizam Mohd Rosli and Nor Azwar Syahila Rosli (2018),'Simulation Study on the Effect of Rear-Wing Spoiler on the Open Aerodynamic Performance of Sedan Vehicle', *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* **49**(2), 146-154.
10. Krzysztof Kurec, MichaB Remer, Jakub Broniszewski, PrzemysBaw Bibik, Sylwester Tudruj and Janusz Piechna (2019),'Advanced Modeling and Simulation of Vehicle Active Aerodynamic Safety', *Journal of Advanced Transportation,* Volume 2019, 17-37.
11. Sivanesh Prabhu.M, Arulvel.S and Mayakkannan.S (2017), 'CFD Analysis of Automobile Rear Dynamic Spoiler', *International Journal of Scientific & Engineering Research* **8**(8), 1431-1435
12. Akhilesh Singh Tomar, Anuj Prajapati, AnujSharma and Shubham Shrivastava (2019),'CFD Analysis on the Aerodynamic Effects of Spoiler at Different Angle on Car Body', *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* **8**(7), 2845-2848
13. Mohammed Amer, Chih-Chun Yang, Hung-Yu Chen, David Neugebauer and Naseem Abbas (2020), 'Experimental Investigation of a Spoiler's Impact on the Flow Pattern of a High-Speed Sport Car', *Journal of Mining and Mechanical Engineering* **1**(02), 73-84
14. Mohankumar M and Hudson E Daniel Raj (2015), 'Investigating the Effect of Rear Spoiler and Rear Diffuser on Aerodynamic Forces using CFD', *International Journal of Engineering Research & Technology (IJERT)* **3**(26), ISSN: 2278-0181
15. P. Ramasekhar Reddy, R. Lokanadham (2017), 'CFD ANALYSIS ON AERODYNAMIC EFFECTS ON A PASSENGER CAR', *International Research Journal of Engineering and Technology (IRJET)* **4**(09), 1049-1054
16. K K Singh, H Bahri, H Singh and J Singh (2019), A review of "performance analysis and optimization of car air spoilers'', *in* 'IOP Conference Series: Materials Science and Engineering, Volume 691, 2nd International Conference on Computational & Experimental Methods in Mechanical Engineering', 691 012056
17. See-Yuan Cheng, Kwang-Yhee Chin , Shuhaimi Mansor and Abd Basid Abd Rahman (2019), 'Experimental study of yaw angle effect on the aerodynamic characteristics of a road vehicle fitted with a rear spoiler', *Journal of Wind Engineering & Industrial Aerodynamics* **184**, 305-312
18. Geoff Dymott (2022), "A CFD analysis of the Audi TT spoiler", Airshaper https://airshaper.com/blog/audi-tt-spoiler
19. https://schoolworkhelper.net/wind-tunnels-how-they-work-types/
20. UKEssays. (November 2018). Design Of The Rear Spoiler For Road Vehicles Engineering ssay. Retrieved from https://www.ukessays.com/essays/engineering/design-of-the-rear-spoiler-forhttps://www.ukessays.com/essays/engineering/design-of-the-rear-spoiler-for-road-vehicles-engineering-essay.php?vref=1road-vehicles-engineering-essay.php?vref=1
21. L.Balaji, P.Deepak prasad, V.Ashok, Surendra Kumar Yadav and Sovan Samanta (2020), Design And Analysis Of Rear Spoiler To Reduce Drag Coefficient Of A Sports Car, *in* ' 8th INTERNATIONAL CONFERENCE ON CONTEMPORARY

ENGINEERING AND TECHNOLOGY', ResearchGate Carwow, UK (September 20, 2022), "10 of the Most Aerodynamic Cars Ever Made", by John Tallodi, Retrieved fromhttps://www.carwow.co.uk/best/mosthttps://www.carwow.co.uk/best/most-aerodynamic-carsaerodynamic-cars#gref

22. Inside EVs, USA (June 10, 2021), "Tesla To Highlight Its Aerodynamic Superiority At Plaid Event" by Mark Kane Retrievedfrom https://insideevs.com/news/513298/tesla-aerodynamic-superiority-plaid-event/