Kaunas University of Technology
Faculty of Mechanical Engineering and Design

Aerodynamic and Structural Analysis of Spoilers for Passenger Cars

Master's Final Degree Project

Dwaraganath Manimangalam Muralidaran
Project author

Assoc. Prof. Dr. Sigitas Kilikevičius
Supervisor

Kaunas, 2020

Kaunas University of Technology

Faculty of Mechanical Engineering and Design

Aerodynamic and Structural Analysis of Spoilers for Passenger Cars

Master's Final Degree Project

Vehicle Engineering (6211EX021)

Dwaraganath Manimangalam Muralidaran

Project author

Assoc. Prof. Dr. Sigitas Kilikevičius

Supervisor

Assoc. Prof. Dr. Kristina Liutkauskienė

Reviewer

Kaunas, 2020

Kaunas University of Technology

Faculty of Mechanical Engineering and Design

Dwaraganath Manimangalam Muralidaran

Aerodynamic and Structural Analysis of Spoilers for Passenger Cars

Declaration of Academic Integrity

I confirm that the final project of mine, Dwaraganath Manimangalam Muralidaran, on the topic „Aerodynamic and Structural Analysis of Spoilers for Passenger Cars“ is written completely by myself; all the provided data and research results are correct and have been obtained honestly. None of the parts of this thesis have been plagiarised from any printed, Internet-based or otherwise recorded sources. All direct and indirect quotations from external resources are indicated in the list of references. No monetary funds (unless required by Law) have been paid to anyone for any contribution to this project.

I fully and completely understand that any discovery of any manifestations/case/facts of dishonesty inevitably results in me incurring a penalty according to the procedure(s) effective at Kaunas University of Technology.

___________________________ _______________

(name and surname filled in by hand) (signature)

Kaunas University of Technology

Faculty of Mechanical Engineering and Design

Study programme: Vehicle Engineering (6211EX021)

Task of the Master's Final Degree Project

Given to the student: Dwaraganath Manimangalam Muralidaran

1. Title of the Project:

Aerodynamic and Structural Analysis of Spoilers for Passenger Cars

Spoilerių lengviesiems automobiliams aerodinaminiai ir struktūriniai tyrimai

2. Aim of the Project: The aim of the thesis is to develop aerodynamic features (such as front and rear spoilers) and to analyse their effect on the aerodynamics of a passenger car.

3. Tasks of the Project: Conduct literature survey, Conduct angle of attack trials on the different profiles used for spoiler and model spoilers based on the results, Investigate the aerodynamic parameters of the car with and without spoilers, Conduct structural analysis of spoilers with different materials using the forces derived from the flow analysis of the spoilers.

4. Structure of the Text Part: The thesis is made up of an introduction, literature survey, methodology, results of CFD simulation and structural analysis, discussions, conclusions and a list of references.

5. Structure of the Graphical Part: The various configurations used for the trials, final models of spoilers, results of all the CFD simulations and structural analysis conducted and discussions of all the results with the help of graphical tools.

6. Consultants of the Project:

Author of the Final Degree Project	Dwaraganath Manimangalam Muralidaran *(abbreviation of the position, name, surname, signature, date)*
Supervisor of the Final Degree Project	Assoc. Prof. Dr. Sigitas Kilikevičius *(abbreviation of the position, name, surname, signature, date)*
Head of Study Programmes	prof. Artūras Keršys *(abbreviation of the position, name, surname, signature, date)*

Dwaraganath Manimangalam Muralidaran. Aerodynamic and Structural Analysis of Spoilers for Passenger Cars. Master's Final Degree Project / supervisor Assoc. Prof. Dr. Sigitas Kilikevičius; Faculty of Mechanical Engineering and Design, Kaunas University of Technology.

Study field and area (study field group): Transport Engineering (E12), Engineering Science.

Keywords: Front spoiler, Rear spoiler, Angle of attack, Flow simulation, Structural analysis.

Kaunas, 2020. 61 p.

Summary

This thesis is about analysing the effect of aerodynamic features on the aerodynamics of the vehicle and to enhance the performance of the vehicle. Also, to analyse the strength of materials used for the construction of these features. The main intent of this study is to enhance the aerodynamic performance of the vehicle by using addon aerodynamic features such as front and rear spoilers. This process is enhanced by conducting aerodynamic efficiency trials on aerofoils, plate type profile and their angle of attack. The modelling for the study was done in SOLIDWORKS software, the aerodynamic efficiency of the spoilers and the car was analysed using SOLIDWORKS flow simulation. The last phase of this study is about the structural analysis of the spoilers using different materials to evaluate their rigidity and strength. This structural analysis was performed on SOLIDWORKS simulation.

A comparative flow simulation study between aerofoils such as S9026, Clark Y, S1223 and a plate type profile was conducted in SOLIDWORKS flow simulation at different angle of attack from 0° to 20° at every 5°. Based on the results, S1223 aerofoil with 10° angle of attack was chosen and both front and rear spoilers were modelled accordingly. To estimate the effect of these features on a vehicle, flow simulation was conducted on Audi A7 with and without these aerodynamic features. With both the spoilers added on to the vehicle the downforce was increased by 12.8% and drag force was decreased by 1.1%. Structural analysis was then performed on these spoilers with the help of SOLIDWORKS simulation, two materials were used ABS plastic and carbon fibre with the forces obtained from the flow analysis. Worst case scenario was also studied on the spoilers with help of forces obtained from flow analysis of aerofoil at flow velocity 70 m/s. Considering the parameters such as spoiler’s strength and cost ABS plastic was chosen as the maximum stress did not exceed the allowable stress.

Dwaraganath Manimangalam Muralidaran. Spoilerių lengviesiems automobiliams aerodinaminiai ir struktūriniai tyrimai. Magistro baigiamasis projektas / vadovas doc. Sigitas Kilikevičius; Kauno technologijos universitetas, Mechanikos inžinerijos ir dizaino fakultetas fakultetas.

Studijų kryptis ir sritis (studijų krypčių grupė): Transporto inžinerija (E12), Inžinerijos mokslai.

Reikšminiai žodžiai: priekinis aptakas, galinis aptakas, susidūrimo kampas, srauto modeliavimas, struktūrinė analizė.

Kaunas, 2020. 61 p.

Santrauka

Šis darbas skirtas aerodinaminių savybių įtakos transporto priemonės aerodinamikai analizei ir transporto priemonės aerodinaminių savybių gerinimui. Taip pat darbe analizuojamas medžiagų, skirtų šių savybių kūrimui, stiprumas. Pagrindinis šio tyrimo tikslas – pagerinti transporto priemonės aerodinamines savybes, pasitelkiant papildomas aerodinamines priemones, tokias kaip priekiniai ir galiniai aptakai. Šis procesas yra patobulintas atliekant aerodinaminio efektyvumo bandymus su aerodinaminiais paviršiais, plokštės tipo profiliu ir jų susidūrimo kampu. Tyrimo modeliavimas atliktas naudojant „SOLIDWORKS“ programinę įrangą, aptakų ir automobilio aerodinaminis efektyvumas išanalizuotas naudojant „SOLIDWORKS“ srauto modeliavimą. Paskutinis šio tyrimo etapas apima aptakų struktūrinę analizę, naudojant skirtingas medžiagas, siekiant įvertinti jų tvirtumą ir stiprumą. Ši struktūrinė analizė buvo atlikta naudojant „SOLIDWORKS“ modeliavimą.

Lyginamasis srauto modeliavimas tarp aerodinaminių paviršių, tokių kaip S9026, Clark Y, S1223, ir plokštės tipo profilio buvo atliktas naudojant „SOLIDWORKS“ srauto modeliavimą skirtingu susidūrimo kampu nuo 0° iki 20° kas 5°. Remiantis gautais rezultatais, buvo pasirinktas S1223 aerodinaminis paviršius su 10° susidūrimo kampu ir priekiniai ir galiniai aptakai buvo atitinkamai sumodeliuoti. Siekiant įvertinti šių savybių poveikį transporto priemonei, srauto modeliavimas buvo atliekamas „Audi A7“ su šiomis aerodinaminėmis savybėmis ir be jų. Pridėjus abu aptakus ant transporto priemonės, prispaudimo jėga sumažėjo 12,8 %, o stabdymo jėga padidėjo 1,1 %. Tada, naudojant „SOLIDWORKS“ modeliavimą, buvo atlikta šių aptakų konstrukcijų analizė su dvejomis medžiagas – ABS plastiku ir anglies pluoštu, naudojant jėgas, gautas atliekant srauto analizę. Blogiausias scenarijus aptakų atžvilgiu taip pat buvo tiriamas, naudojant jėgas, gautas analizuojant aerodinaminio paviršiaus aerodinaminį srautą, kai srauto greitis yra 70 m/s. Atsižvelgiant į tokius parametrus kaip aptako stiprumas ir kaina, buvo pasirinktas ABS plastikas, kadangi maksimalus įtempis neviršijo leistino įtempio.

Table of contents

List of figures

List of Tables

Introduction

Vehicle aerodynamics focuses on reducing the drag, wind noises and generating downforce instead of lift which has a huge negative effect on vehicle traction at high speeds. Nowadays vehicle aerodynamics plays a prominent role in design aspects due to various factors such as better handling and stability, economical fuel consumption and good aesthetics. According to Diamond [1], aerodynamic drag is the largest resistive force acting on a vehicle when it travels over 80 km/h and states that nearly 65% of the engine power is used to overcome drag over 110 km/h of vehicle speed.

Cheng et al. [2], explains that various other devices have been implemented to improve the aerodynamic properties of vehicles so far like deflectors at the side or top of a truck body, vortex generators at the rear section or at the end of roof, rear wing and spoilers. Among all these rear wings and spoilers, spoilers are the widely used ones in real life. Rear spoilers are aerodynamic devices that are fitted on the trunk or the trailing edge of the roof of a vehicle to produce downforce thus increasing the traction and also the fuel economy. A good downforce is very important during cornering where the vehicle will need a lot of traction to go through the curve without any slip.

A spoiler is mounted closer to the body of the car and helps in diffusing the air. It improves the overall airflow around the vehicle. Usually it is installed on the trunk or at the end of the roof of a car. When a car is moving at high speed, the atmospheric pressure affects its motion, at this time the spoiler comes into action, it disrupts the airflow over and around the vehicle. Thus, reducing the turbulence generated by spoiling the air and reducing lift and drag [3].

A designer in an automotive industry has to take extra care while designing the aerodynamics of a vehicle since it is the most crucial part in the vehicle's design. Two main things which are to be considered while designing the aerodynamics of a vehicle are downforce creation which will help the car to have good traction and handling characteristics; and reduction of drag force which reduces the speed of the car [4]. The spoiler disrupts the airflow around and over the vehicle thus dispersing the air and reducing the turbulence generated at the rear. The design of a spoiler is very important as it can affect the vehicle's performance [3].

The aim of the thesis is to develop aerodynamic features (such as front and rear spoilers) and to analyse their effect on the aerodynamics of a passenger car. Various tasks involved in this project to achieve the aim are:

1. Literature survey about the various aerodynamic features, investigation methodology, materials and their properties.
2. Comparison study between various profiles of aerofoil and plate type with various angle of attack study and at different velocities. And to develop spoilers with optimal shape and angle of attack based on the results.
3. CFD analysis of the cases such as car without any spoilers, with rear spoiler and with both front and rear spoilers in order to investigate the aerodynamic parameters of the car at different flow velocities.
4. Structural analysis of spoilers with different materials using the forces derived from the flow analysis of the spoilers.

Hence, the overview is to develop aerodynamic features for passenger car, estimate the forces acting on these features, optimise their shape and angle of attack; estimate their effects on passenger car and to find out possible economical materials with good strength.

1. Literature survey

In this section, a review about the various experiments and studies that has been done over the years to improve the aerodynamic characteristics of a vehicle is done. This section is divided into four parts:

- Theories of aerodynamics
- Aerodynamics features of a vehicle
- Investigation methods
- Materials used for spoiler fabrication

1.1. Theories of aerodynamics

Aerodynamics is a vital factor in today's automotive industries where the manufacturers are using both wind tunnels and computational fluid dynamics (CFD) for optimizing the aerodynamic characteristics of the vehicle. Road vehicle aerodynamics is nothing but the interaction between the vehicle, the road and the air surrounding the vehicle. Lot of factors are affected by vehicle aerodynamics for instance; performance and fuel consumption, control and handling, noise and vibration and vehicle aesthetics [5]. The aerodynamic characteristics of road vehicles can be classified as drag, lift and pitching moment and yawing moment and side force.

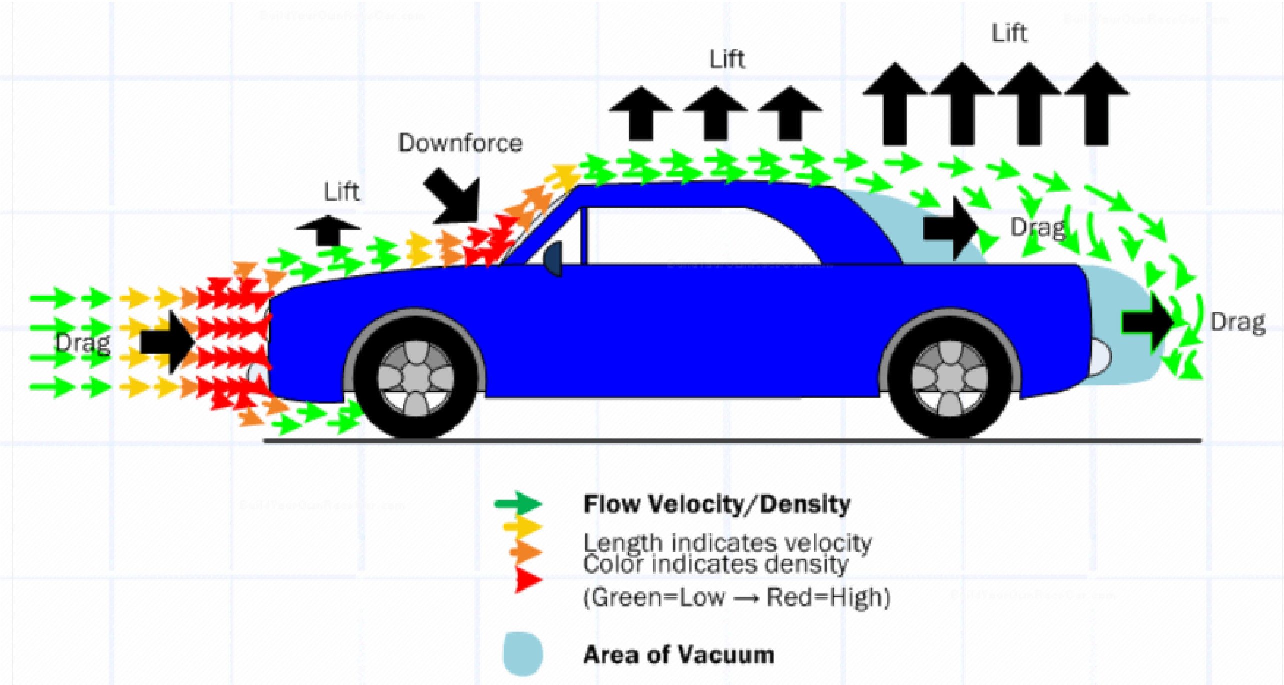

Fig. 1. Aerodynamic characteristics of road vehicle [6]

Mechanical force which resists the vehicle's motion in air is called as drag. It is generated by the interaction of vehicle's body with air. It acts opposite to the vehicle's motion [7]. Various sources of drag contribute to the total drag on a vehicle which are; Pressure drag, Lift induced drag, Skin friction drag, Interference drag and Internal flow drag. Pressure drag is due to the shape of the body. Bodies with higher frontal area have more pressure drag than streamlined bodies. It contributes around 55% of the total drag. Lift induced drag is due to the lift generated. It contributes around 7% of the total drag. Skin friction drag is due to the air passing tangentially along the body. It contributes about 9% of the total drag. Interference drag is caused due to the projections that exist on the basic body. It is about 7% of the total drag. Internal flow drag is caused when the air passes through the various components of the vehicles which requires cooling. It contributes around 12% of the total drag [8]. Drag is given by the expression,

$$D = \frac{C_D . A . \rho . V^2}{2} \quad (1)$$

Where, D is the drag force, C_D is the co-efficient of drag, A is the frontal area, ρ is the density of surrounding air and V is the vehicle speed. Hucho [9], stated that the drag increases with square of the vehicle speed and it is very much dependent on the frontal area and shape of the vehicle. He also stated that effect to reduce drag should be concentrated on reducing the drag co-efficient.

Co-efficient of drag is a dimensionless value to quantify the drag of a vehicle. It is the main contributor to skin friction drag and pressure drag. It has a pre-defined value for various basic shapes in which the streamlined body has lowest drag co-efficient value [9].

Lift is a force which directly opposes the weight of the vehicle. Lift is generated when the moving fluid is turned by a solid body. According to Newton's third law, flow is turned in one direction and lift is generated in the opposite direction [10]. Lift acts perpendicular to the axis of motion of the moving vehicle. Positive lift is created when there is low pressure over the top of a moving body compared to the underbody. This is extremely dangerous effect on road vehicles for traction and safety. Downforce is good because it pushes the vehicle downwards to the road allowing it have more traction, it is also called as negative lift [10]. Lift is given by the expression,

$$L = \frac{C_L . A . \rho . V^2}{2} \quad (2)$$

Where, L is the lift force, C_L is the co-efficient of lift, A is the frontal area, ρ is the density of surrounding air and V is the vehicle speed. Co-efficient of lift is a dimensionless value to quantify the lift of a vehicle in a fluid environment.

1.2. Aerodynamic features of a vehicle

Various studies have been conducted over the years on aerodynamic features of the vehicle to reduce drag and lift. In this section various aerodynamic features and studies conducted on them over years have been reviewed in order to obtain a clear understanding about the current scenario of the project.

Ahmed et al. [11], stated that the ground vehicles can be treated as bluff bodies moving closely to the road surface and designed a simple model of car named as "Ahmed body". The shape of the body was designed in way to imitate the large displacement flow field in front, uniform flow in middle and large wake at rear. Ahmed body has been a benchmark in automotive industry for wind tunnel and CFD testing. It has a simple geometry of 1044 mm length, 288 mm height and 389 mm width. Its rear surface has a slant of 5° – 25°. This model was tested in a wind tunnel to analyse effect of the body geometry on wake structure, drag and pressure distribution. The experiment was focused on the rear end geometry after a drag breakdown analysis which proved that most of the drag is due to flow mechanisms at the rear end. It was concluded that forebody contributed 9% of the pressure drag and the rest was due to the rear end. Based on the experimental results obtained the slant value was optimised at 12.5° with lowest drag value than the other slant angles.

Hanfeng et al. [12], developed two deflectors, and mounted them on the side edges of the slanted surface and on the top edge of Ahmed body with 25° slant angle in order to experimentally investigate the effect of deflector plate on aerodynamic drag. The experiment was carried out in a closed-circuit low speed wind tunnel, it was concluded that based in the height of the deflectors the drag values increased and decreased. At 25° slant angle, the top edge mounted deflector reduced drag about 12% and was more effective than the side edge mounted deflectors.

Fourrie´ et al. [13], investigated drag reduction using a deflector near the rear slanted surface of Ahmed body in a closed-circuit subsonic wind tunnel. Based on the results, it was concluded that deflector angle played a major role in disturbing the vortices arising on the rear lateral edges and helped in achieving drag reduction of about 9%.

Chowdary et al. [14], introduced an aerodynamic deflector on the top of a truck instead of the conventional one that is being used in South Asia, to reduce drag and fuel consumption. The experiment was conducted in a closed return circuit wind tunnel. From the results obtained, it was concluded that the aerodynamic deflector reduced the drag by 13% while the conventional deflectors increased the drag by 58%. It could be seen that the deflectors placements and geometry is very essential while wrong placement and geometry can actually increase the drag like in the above cases.

Aieder et al. [15], developed a vortex generator at the rear of a modified Ahmed body with a curved part instead of a slanted part so that the location of flow separation was not forced by the geometry. All the measurements were conducted in an in-house open wind tunnel and particle image velocimeter (PIV). Various vortex generators have been kept inline at the rear of the curved part and it appeared to be very effective in reducing the drag of 12% and lift reduction of 60%. Later, a motorised vortex generator setup was implemented at the optimal configuration based on drag and lift values.

Pujals et al. [16], investigated the drag reduction using a vortex generator at the rear end of roof of Ahmed body. Three different sizes of vortex generators and configurations were used for the investigation inside a wind tunnel. From all three configurations, the overall drag reduction was around 10%. PIV was used for the confirmation purpose which proved that the drag reduction was due to the suppression to the recirculation bubble.

Tsai et al. [17], investigated the drag and lift variation in passenger car with wing type rear spoiler using computational fluid dynamics (CFD) approach. Total of six cases and configurations were considered for the study in FLUENT, a CFD solver. Of all the six cases one case was having the lowest drag and lift values in which the spoiler had a double-wing arrangement with the leading edge of the upper wing is little downstream of the lower wing and result obtained was, lift co-efficient reduced drastically but the drag co-efficient increased slightly by 20% (from 0.4 to 0.49).

Bansal et al. [18], investigated the drag reduction of a passenger car using various add on devices such as spoiler, vortex generator, tail plates and using both vortex generator as well as spoiler. First, a car of reduced scale was tested in an open type wind tunnel to determine the drag and lift coefficients. Later using CFD the above stated add-ons were one by one. Ansys FLUENT was used for the CFD approach. From the results obtained it was observed that vortex generators have the highest drag and lift coefficients value of all and spoiler with vortex generator have the lowest drag and lift coefficient values of all. Tail plates also seem to have a very low drag and lift coefficient values but are almost on par with spoilers. With 4% of drag reduction and 19% of lift reduction, spoiler with vortex generator configuration has the most economical result of all. With spoiler alone, the co-efficient of drag was reduced by 2% and lift by 6%. It was concluded in the paper using add-on devices drag can be reduced thereby increasing the stability and fuel economy of a car.

Paul et al. [19], reduced the aerodynamic drag of a passenger car using flow control techniques using both experimental and computational methods. An array of vane type vortex generator and rear spoiler combination is used to reduce drag of a passenger car. A test model of Honda city with a reduced scale of 15:1 is used for the wind tunnel experimentation and a CAD model for CFD simulation. From both the methods, it was estimated that the co-efficient of drag was reduced by 23%. By reducing the drag, fuel consumption could be also be reduced which was theoretically estimated around 11.5% reduction in fuel consumption.

Hu et al. [20], studied the effect of rear spoiler on a passenger car using CFD approach. Two cases were taken for the study one with rear spoiler and the other without the spoiler. Ansys fluent, a CFD package is used for this study. Two types of rear spoiler were also considered for the first part of the study a plate type and aerofoil type with two different angle of attack 5° and 10°. The three configurations were numerically solved and based on the result one new type spoiler design was introduced which also acts as a diffuser with which drag has been reduced by 1.7%, lift has been reduced by 4% and fuel economy has been increased.

Bambhania et al. [21], analysed the effect of roof spoiler in a light duty commercial vehicle using CFD method. Two cases were studied in this research first one without spoiler and the second one with spoiler. GAMBIT a CFD tool is used for the analysis purpose. As the speed increases the drag force increases as it is the square of vehicle speed. After adding the roof spoiler, the drag force is much lower than that of the one without spoiler. They obtained 37% reduction in co-efficient of drag after adding roof spoiler, 0.94 – 0.59.

Hamut et al. [22], examined the effect of rear spoilers on ground vehicle for aerodynamic drag and lift using both CFD approach and wind tunnel. $1/16^{th}$ scale of an actual vehicle was tested in the wind tunnel to obtain the drag co-efficient and to compare the error band between CFD and experimental method. The error band of CFD analysis obtained was 5.8% and wind tunnel experiment was 6.1%. From the CFD results of with and without spoiler, lift was reduced by 80% after the addition of spoiler whereas drag co-efficient increased slightly from 0.31 to 0.36.

Thummar et al. [23], studied the variation of aerodynamic drag and lift in a passenger car with a rear spoiler using CFD approach. Lancer model from Mistubishi was first designed in CREO and then a spoiler was introduced to it in order to observe the variation in drag and lift coefficients. Two cases were simulated in this paper, first without spoiler and then after introducing the spoiler. After the simulation it was concluded that the addition of spoiler reduced the lift co-efficient by 25% whereas the drag co-efficient increased by 5%. Theoretically, the authors concluded that there will be approximate of 11.5% reduction in fuel consumption by adding rear spoiler.

Marathe et al. [4], analysed the effect of rear spoiler in a passenger car using CFD approach. Ansys CFD package is used for the analysis. Two cases were taken first passenger car without spoiler and then a spoiler is introduced to the passenger car and both the cases were simulated. With an increase in the air velocity, the aerodynamic drag and lift forces increases exponentially in without spoiler configuration. Whereas after adding a spoiler, a good downforce is created which will help the vehicle to hold its ground at high speeds. From the result it was stated that downforce increased by 75% but with a slight increase in drag. Still, the downforce will be helpful to improve traction and avoid skidding at high speeds.

Das et al. [24], investigated rear spoilers at various spoiler inclination angle. CFD approach was used for the analysis and SOLIDWORKS CFD was package used. Six different variations from -2° to 12° were analysed for the study, of all the various angles 12° was concluded the most optimum angle for spoiler inclination as it has the lowest lift co-efficient of all, drag co-efficient was 1.56% higher than that of 4° but it has cutback in lift co-efficient by 77% which will produce a lot of traction and better stability of the vehicle.

Tousi et al. [25], evaluated the importance of spoiler in electric vehicles using CFD approach. The authors first simulated the effect of rear spoiler in Ansys FLUENT after observing the reduced drag and lift coefficients they introduced the real spoiler in electric vehicle to test the energy efficiency after adding the spoiler. They concluded that after the addition of spoiler the drag coefficient reduced by 7% and the lift coefficient decreased from 0.008 to -0.036. The energy efficiency is also improved by adding rear spoilers which is indirectly due to the reduction of drag and lift coefficients.

Fig. 2. Aerodynamic components of a car [26]

Rear spoilers are aerodynamic devices that are fitted on the trunk or the trailing edge of roof of a vehicle to produce a downforce thus increasing the traction and also the fuel economy. Spoilers place an obstruction in the upwards airflow thus causing the upward airflow to change and leave horizontally without creating lift [27]. Rear spoilers and rear wings are interchangeable terms but still there are huge differences between these two, functionally as well. Spoilers reduce drag whereas wings tend to increase it. As stated, earlier spoiler and wings are different, rear wing are aerofoils designed to deflect the air upwards by pushing down the vehicle. This helps in the creation of downforce although drag will increase slightly. It also helps in good handling and stability of a vehicle [27].

One of the main advantages of spoiler in car is the reduction of lift and drag co-efficient. By generating downforce, it generates good traction between the vehicle and the road thereby increasing the stability and handling of the vehicle. By drag reduction it helps in improving the fuel economy of the vehicle. By creating a good downforce, it adds weight to the car by pushing it downwards without adding any extra mass and it also increases the braking stability of the vehicle. Adding spoiler also gives a very sportive look and style to the car [28].

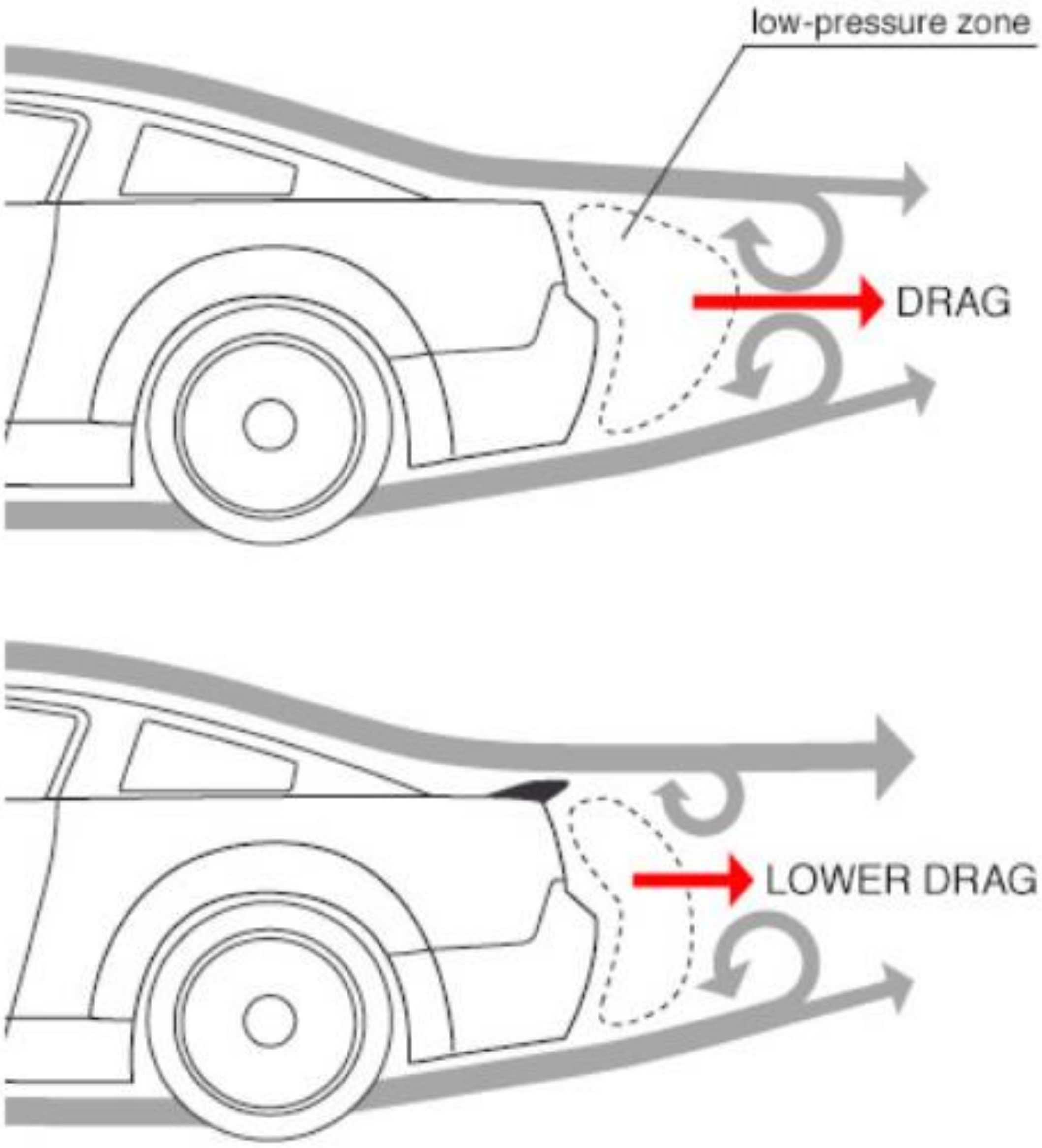

Fig. 3. Function of rear spoiler [29]

1.3. Investigation methods

Conventional wind tunnel experiments have been used by automobile manufacturers around the world to determine the vehicle's aerodynamic characteristics. This method is very expensive and time consuming hence various software packages have been developed in the recent years to numerically study the vehicle's aerodynamic characteristics. Zikanov. [30], defined computational fluid dynamics (CFD) as a set of numerical methods applied in order to obtain approximate solution for fluid dynamics and heat transfer-based problems. This definition was interpreted by Cakir [31], as CFD isn't science however it can be treated as a way of applying techniques from one discipline, say numerical analysis to another say heat and mass transfer. It incorporates not just the field of fluid mechanics and mathematics but also computer science. The governing equations of CFD are the partial differential equations of mathematics which describes the physical characteristics of the fluid motion.

According to Sayma [32], CFD provides a cost-effective means to stimulate actual flows through the governing equation's numerical solution. It utilises numerical methods and algorithms to fix fluid flow issues. Experimental methods such as wind tunnels has played an important role in the validation of governing equation so far, but it can be extremely difficult that analytical solutions might never be obtained for few practical problems. Computational methods use partial differential equations with algebraic equations which makes it easier for the computer software to solve. With CFD it is also possible to get analytical solutions for problems which are very difficult or almost impossible.

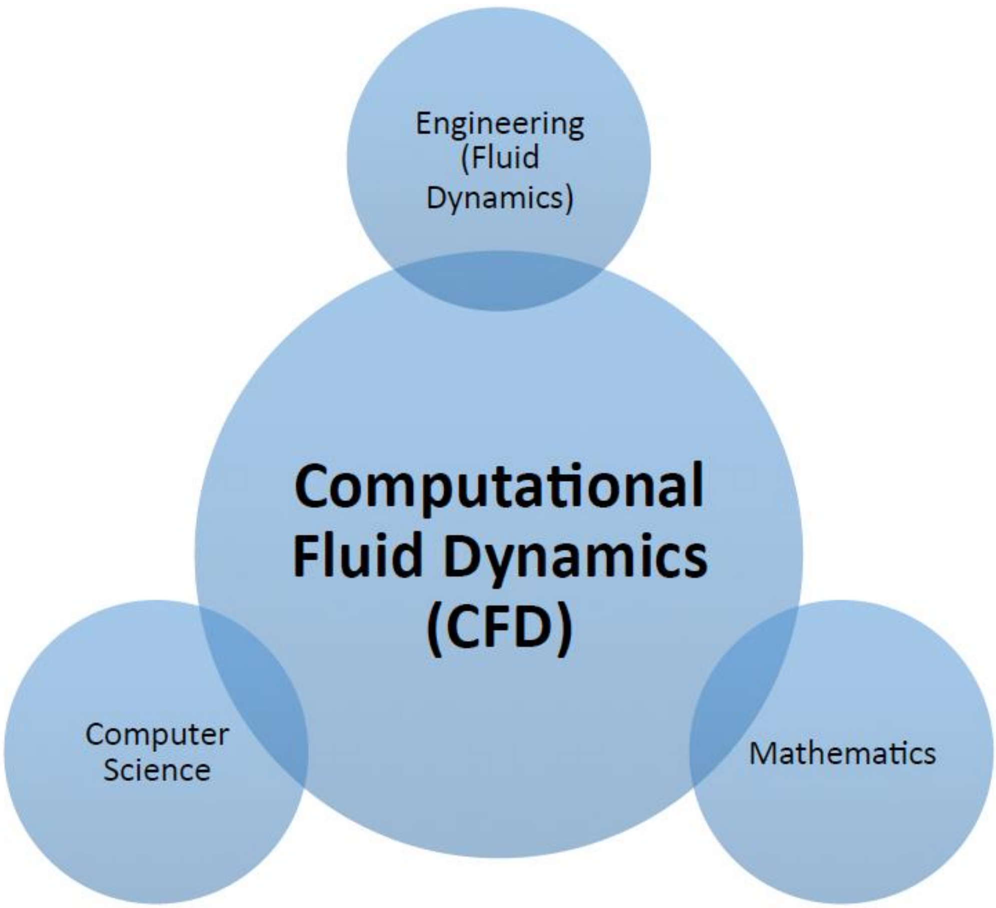

Fig. 4. The various disciplines in computational fluid dynamics [31]

According to Katz [33], wind tunnel methods were widely used for aerodynamic testing till the end of last century as it was the only logical testing available during the mid-60's. Testing a vehicle in a wind tunnel posed a lot of problems such as small clearance between the vehicle's underbody and the floor of the tunnel, then moving ground effect was introduced in which significant increase in drag and downforce was observed. Later over the years and with increase in technology, almost all the issues that were faced in the past were resolved. Most wind tunnels now a days can hold up to 50% model of the actual vehicle. Later, integration of computational fluid dynamics methods evolved into the most widely used tool in automobile industry due to its detailing like large body information of the solution, flow visualization etc. Wind tunnels can be used only for one certain configuration at a given time, whereas in CFD various configuration can be used, data can be reviewed, analysed over and over again. Wind tunnels are more expensive and resource consuming when compared to CFD techniques. Almost any configuration of the real-world problems can be simulated using CFD tools. It is a very useful preliminary design tool which is much cheaper than the wind tunnels and other conventional methods.

A brief comparison between computational fluid dynamics approach and the wind tunnel experimentation by Kumar et al [34], states that the experimentation time in wind tunnels is long whereas in computer simulation it is a short matter of seconds to hours. A full model can be simulated in CFD analysis but only up to 50% of the model can be used in wind tunnels. Wind tunnels are quite expensive and resource consuming in the other hand CFD simulations are cheap and can be done in a laptop with moderate system requirements. The main advantage of CFD simulation is that all related data about the analysis can be obtained whereas in wind tunnel experiment only data about the measured point could be obtained. CFD simulations can be repeated number of times with different configurations but only few experiments can be repeated in wind tunnels.

There are lot of advantages of using CFD, the theoretical development focuses on building and solution of governing equations and the study of different approximations to the governing equations. It mainly reduces the lead time, costs and man power in design and production. CFD gives both experimental and analytical approach at low cost means of real time simulation. CFD can simulate flows that cannot be reproduced using experimental methods. It provides a thorough, visualised and comprehensive data when compared to other methods [31]. CFD simulations allows the flow at a realistic scenario which will be the actual operating condition whereas in experimental methods the results need to be extrapolated always [32].

CFD also has disadvantages like everything. The accuracy is highly dependent on various factors such as; the quality and structure of mesh and its elements, the degree of equation that the flow is related to, understanding of the results, the precision of boundary conditions or rate of convergence of the solution. Numerical errors might occur during computation which will affect the results and reality. Mostly it comes to the user's ability, as each single flow problem will be different from each other hence a different modelling strategy must be used. It cannot entirely replace experimental testing even with all its advantages [35]. The main disadvantage is that the chaotic nature of turbulent flow which makes the mathematical prediction complex, the result may vary from reality so it cannot be relied alone [36]. Experimental data can however provide a useful reference point for the CFD model's validity [35].

1.4. Materials used for spoiler fabrication

Generally, spoilers are made out of lightweight polymers. Spoiler is an addon to a vehicle to improve the fuel economy by reducing the aerodynamic drag hence it must be made of lightweight materials so that it does not affect the performance of a vehicle. The various materials that are used to fabricate spoilers are as follows [37],

- ABS plastic
- Fiberglass
- Silicone
- Carbon fibre

1.4.1. ABS plastic

ABS plastic is the common name for Acrylonitrile butadiene styrene, $(C8H8)_x.(C4H6)_y.(C3H3N)_z$. It is amorphous in nature so it does not have any true melting point. Most manufactures fabricate spoilers by mixing ABS plastic with granular fillers in order to stiffen the material [37]. The acrylonitrile provides the material with thermal and chemical stability. Butadiene provides it with strength and toughness and styrene gives a glossy finish to the spoiler [38].

ABS have a very low melting point which makes it easier in injection moulding which is one of the popular spoiler manufacturing process. As stated earlier it can be moulded easily and shaped. And it is compactable with wide range of paints and glues allowing the spoiler to be dyed exact shades to meet specifications. Its high tensile strength, its resistant to chemical corrosion and physical impacts which makes the spoiler withstand heavy use and adverse environmental conditions [38]. The density of ABS plastic is 1.0 to 1.05 g/cm^3. It has a yield strength of 13 to 65 MPa. It has an ultimate tensile strength of about 22.1 to 49.0 MPa, modulus of elasticity about 1.00 to 2.65 GPa and Poisson's ratio of 0.35.

Its mechanical properties include high rigidity, good impact resistance even at low temperatures, good weldability, good dimensional stability and good abrasion and strain resistance [39]. It is very inexpensive almost less than 2 euros per kilogram. ABS plastic is also very harmless as it does not have any carcinogens and does not have any adverse health effects [40].

1.4.2. Fibreglass

Fibreglass is a type of fibre-reinforced plastic with glass fibre. They are used in automotive industry due to their low cost and also because of their durability. They have high ratio of surface area to weight. It also has an amorphous structure. It is used in fabrication of spoilers but not in large scale due to the fact that it takes more time to fabricate it [41]. Fibreglass are classified into five types, based on their raw materials used, and their proportions in manufacturing process and also based on their properties and applications.

Properties of fibreglass include high mechanical strength which is higher than steel itself, it is very good insulator of electricity even at low thickness, they are incombustible and do not support flame, fibreglass have good dimensional stability, they are non-rotting as they are unaffected by any insects and they have low thermal conductivity [42]. The E glass fibre has tensile strength of around 3445

MPa, compressive strength of 1080 MPa, elastic modulus of 73 GPa, density of 2.58 g/cm^3 and Poisson's ratio 0.21 to 0.23.

Their price ranges from 2 euros to 20 euros per kilogram depending on their classification. Due to their time-consuming fabrication process fibreglass are not preferred by automotive industries to fabricate spoilers despite their high strength which is of limited scope in this sector but finds applications in other parts of automotive industries and are widely used in aerospace and defence industries.

1.4.3. Silicone

Silicones are polymers made of large number of siloxane units which contain lot of silicon and oxygen atoms combined with other elements. Automotive industries use silicon organic polymers due to their phenomenal plasticity, high thermal characteristics and high durability [37]. Properties of silicone include low thermal conductivity, less reaction with chemicals, low toxicity, they have high thermal stability, they have very good electrical insulation properties. Mechanical properties of silicone include tensile strength of 165 to 180 MPa, compressive strength of 3200 to 3460 MPa, young's modulus about 140 – 180 GPa and Poisson's ratio of 0.265 – 0.275.

They are made out of rubber combined with other elements. Their price ranges are slightly over rubber prices around 2 to 14 euros per kilogram. They are widely used in brakes, engine components, gaskets, spark plug etc. but they aren't the first choice for spoilers.

1.4.4. Carbon fibres

Carbon fibres are lightweight and durable material but expensive. Because of its necessity of large manual labour, it is not widely used in automotive industries [37]. Carbon fibres have very high specific strength i.e., high strength to weight ratio. Properties of carbon fibres include high rigidity, 2.5 time stiffer than aluminium; they are corrosion resistive and chemically stable; high dimensional stability; carbon fibres are electrically conductive; they have good tensile strength and are non-flammable. Thermal conductivity of carbon fibres is low and they have low co-efficient of thermal expansion [43]. Mechanical properties of carbon fibres include average ultimate tensile strength of 1060 MPa, average yield strength of 1250 MPa, average density of 1.4 g/cm^3 and Poisson's ratio of 0.293 – 0.583.

The properties stated above are an average value of the actual property, the orthotropic property vary on two axes which are perpendicular to each other at 0° and 90°. However, the mechanical properties remain the same at both the axes, ultimate tensile strength along 0° 600 MPa and along 90° 600 MPa, young's modulus 85 MPa at both 0° and 90° and Poisson's ratio of 0.10 [44]. For this specific application use of carbon fibres may not be necessary as it will increase the cost of the product.

Carbon fibres have limited use in automotive industry due to its cost and it also contains lot of manual labour and manufacturing process which increase the cost of the spoiler. The average cost of light weight carbon fibre part is around 100 euros per kilogram out of which 20 euros is for the materials and 80 euros for manufacturing.

1.5. Summary of the literature review

From the previous researches, it could be seen that spoiler is the only aerodynamic component which reduces drag and also increases the downforce which is the most desired aerodynamic characteristic in a vehicle. But adding a wing type spoiler in most of the cases seem to have increased the downforce drastically at the expense of aerodynamic drag which is an undesired phenomenon. Hence, a front spoiler could also be introduced in order to get a better aerodynamic characteristic in terms of drag and as far as the downforce is concerned introducing of the rear spoiler will increase the downforce helping to achieve the most desired aerodynamic characteristics in a vehicle.

Based on the research papers mentioned earlier for investigating methods, it could be observed that CFD methodology is the most suited and economically efficient methodology than the wind tunnel experimentation. The cost and time play a major role between these two and also the measured data which can be obtained easily from CFD analysis whereas in wind tunnel experimentation lot of experiments has to been done in order to obtain different data and lot of measuring points and manpower has to be used. Hence, CFD analysis is the best option to continue further works on the thesis.

From the above-mentioned materials, ABS plastic and carbon fibre are the most suited for manufacturing spoilers. Due to its low cost and good tensile strength most of the automotive manufacturers fabricate spoilers from ABS plastic. It also has a very low melting point which makes it easier in injection moulding which is one of the popular spoiler manufacturing process. Since its compatibility with wide range of paints and glues it gives spoilers a sportive look. A comparative study to be done on the material of the spoiler in order to determine its strength on real life scenario.

2. Methodology

In this section, the processes followed for the investigations of the effect of spoilers on a vehicle are discussed. Starting with a comparative CFD study that has been done in SOLIDWORKS Flow Simulation software. The main reason to have chosen this software package is that it uses two main principles for the flow simulation approach; for the source of geometry the native CAD model is directly used and simpler engineering methods are used for a complete 3D CFD modelling. The flow simulation uses various technologies;

- CAD data management
- Mesh generation
- CFD solvers
- Engineering Modelling Technologies
- Results processing

The SOLIDWORKS Flow Simulation uses Cartesian-based meshes and it forms the main link for CAD embedded CFD. Mesh generation is carried out by dividing the computational domain into cuboids which are formed by intersection of planes parallel to the axes of coordinate system. Mesh refinements could be used to improve the mesh which resolves the features of CAD geometry such as surfaces with small curvatures, narrow channels etc. [45]

SOLIDWORKS Flow Simulation uses the Navier-Stokes equations for the fluid regions. Navier stokes equations formulations of mass, momentum and energy conservation laws [45];

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \tag{3}$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) + \frac{\partial P}{\partial x_i} = \frac{\partial}{\partial x_j}\left(\tau_{ij}\tau_{ij}^{R}\right) + S_i \tag{4}$$

$$\begin{gathered} \frac{\partial \rho H}{\partial t} + \frac{\partial \rho u_i H}{\partial x_i} = \frac{\partial}{\partial x_i}\left(u_j\left(\tau_{ij} + \tau_{ij}^{R}\right) + q_i\right) + \frac{\partial P}{\partial t} - \tau_{ij}^{R}\frac{\partial u_i}{\partial x_j} + \rho\varepsilon + S_i + Q_H \\ H = h + \frac{u^2}{2} \end{gathered} \tag{5}$$

When the Reynolds number exceeds its critical value the flow transits to turbulent. For turbulent flows, SOLIDWORKS flow simulation uses the Favre-averaged Navier-Stokes equation, additional information should be entered for this calculation, to close this system of equations k-ε turbulence model with damping functions is used. The modified k-ε turbulence model describes the laminar, turbulent and transitional flows of homogeneous fluids which consists the following turbulence conservation laws [45];

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho k u_i}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right) + \tau_{ij}^{R}\frac{\partial u_i}{\partial x_j} - \rho\varepsilon + \mu_t P_B \tag{6}$$

$$\frac{\partial \rho\varepsilon}{\partial t} + \frac{\partial \rho\varepsilon u_i}{\partial x_i} = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right) + C_{\varepsilon 1}\frac{\varepsilon}{k}\left(f_1\tau_{ij}^{R}\frac{\partial u_i}{\partial x_j} + C_B\mu_t P_B\right) - f_2 C_{\varepsilon 2}\frac{\rho\varepsilon^2}{k} \tag{7}$$

$$\tau_{ij} = \mu s_{ij}, \tau_{ij}^R = \mu_t s_{ij} - \frac{2}{3}\rho k \delta_{ij}, s_{ij} = \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k} \quad (8)$$

$$P_B = -\frac{g_i}{\sigma_B}\frac{1}{\rho}\frac{\partial \rho}{\partial x_i} \quad (9)$$

where, $C_\mu = 0.09$, $C_{\varepsilon 1} = 1.44$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 1$, $\sigma_\varepsilon = 1.3$, $\sigma_B = 0.9$, $C_B = 1$ *if* $P_B > 0$, $C_B = 0$ *if* $P_B < 0$.

The methodology exercised in this project are as follows;

- CFD analysis of various profiles of aerofoils and plate type with various angle of attack study and at different velocities. To develop spoilers with optimal shape and angle of attack based on the above results.
- CFD analysis of car without any spoiler (Case 1).
- CFD analysis of car with rear spoiler (Case 2).
- CFD analysis of car with front and rear spoiler (Case 3).
- Structural analysis of the final spoiler models with different materials using the forces derived from the flow analysis.

The first phase of the study is to carry out a CFD comparison between the various profiles of aerofoil and plate type in order to determine the shape and the angle of attack of the spoilers which is to be used in case 2 and case 3 of this project. The various aerofoils used for this phase are S9026 (configuration i), Clark Y (configuration ii), S1223 (configuration iii) and finally a plate type (configuration iv). The various angles of attack which were simulated in this project are; 0°, 5°, 10°, 15° and 20°. These configurations were simulated at different velocities such as 20 m/s (72 km/h), 30 m/s (108 km/h), 40 m/s (144 km/h) and 50 m/s (180 km/h). Based on the results obtained, spoilers were modelled with configuration iii and angle of attack 10° which will be discussed in the oncoming sections of this paper.

The second phase of the study is to do a CFD analysis on a passenger car and to determine its drag and lift co-efficient value. For this Audi A7, 2010 model was modelled in SOILDWORKS software and CFD study was carried out in SOLIWORKS flow simulation software at velocities such as 20 m/s, 30 m/s, 40 m/s and 50 m/s. The initial drag and lift coefficients of the car were determined from this study.

The third phase of the project is to perform a CFD analysis of the car with rear spoiler assembled on it. The rear spoiler was modelled based on the flow simulations performed on the shapes and the angle of attack of the aerofoil profiles, the shape of the aerofoil chosen was S1223 (configuration iii) with 10° angle of attack. This rear spoiler was assembled on the car and CFD simulation was carried out at the same velocities as mentioned in above case.

The fourth phase of this thesis is to perform a CFD analysis of the car with both front and rear spoilers assembled on it. Even the front spoiler was modelled using the configuration iii and at 10° angle of attack based on the results obtained from the first phase of the thesis. With both front and rear spoilers assembled on the car, the flow simulation was carried out at the same velocities as mentioned in the previous cases.

Fifth and the final phase of this paper is to perform a static structural analysis of the spoilers on SOLIDWORKS simulation software in order to evaluate the strength of the spoilers in real life applications. The forces used on the spoilers for static structural analysis were derived from the flow analysis of the aerofoils performed earlier in this paper. For the static structural analysis two different materials were taken in order to compare and propose the best economically feasible material for the spoilers. The two materials chosen were ABS plastic and carbon fibre. The properties and the results will be discussed in upcoming sections of this paper.

2.1. Modelling of spoilers

The various aerofoils used for this phase are S9026 (configuration i), Clark Y (configuration ii), S1223 (configuration iii) and finally a plate type (configuration iv). All these spoilers have a chord length of 0.2 m, wing span of 1.5 m and aspect ratio of 7.5. The spoilers were modelled in SOLIDWORKS software. The angle of attack configurations are as follows; 0° – a, 5° – b, 10° – c, 15° – d and 20° – e.

2.1.1. Configuration i – S9026 aerofoil

S9026 aerofoil was modelled in SOLIDWORKS software based on the aerofoil profile and the points available on the aerofoil database [46]. The chord length of the aerofoil is 0.2 m and its wing span is 1.5 m. The aspect ratio of this aerofoil is 7.5.

Fig. 5. S9026 aerofoil (configuration i)

2.1.2. Configuration ii – Clark Y aerofoil

Inverted Clark Y aerofoil was modelled in SOLIDWORKS software based on the aerofoil profile and the points available on the aerofoil database [47]. Inverting the aerofoil from Clark Y is because of the aerodynamics required for the application. The chord length of the aerofoil is 0.25 m and its wing span is 1.5 m. The aspect ratio of this aerofoil is 6.

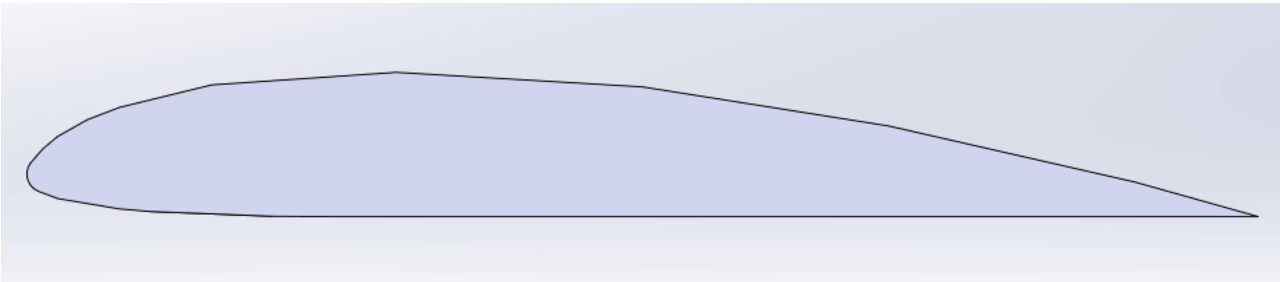

Fig. 6. Clark Y aerofoil (configuration ii)

2.1.3. Configuration iii – S1223 aerofoil

S1223 aerofoil was modelled in SOLIDWORKS software based on the aerofoil profile and the points available on the aerofoil database [48]. The chord length of the aerofoil is 0.2 m and its wing span is 1.5 m. The aspect ratio of this aerofoil is 7.5.

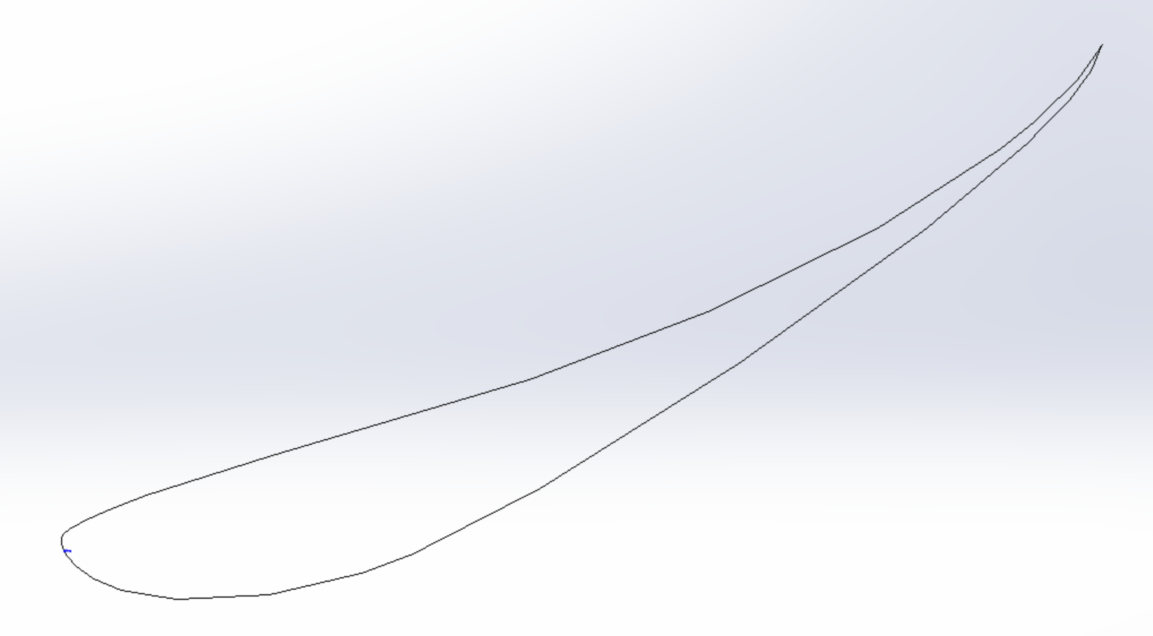

Fig. 7. S1223 aerofoil (configuration iii)

2.1.4. Configuration iv – Plate type

Plate type spoiler was modelled in SOLIDWORKS software in accordance with the previous aerofoils modelled previously for comparison. The plate type has a length of 0.2 m, span of 1.5 m and width of 0.05 m.

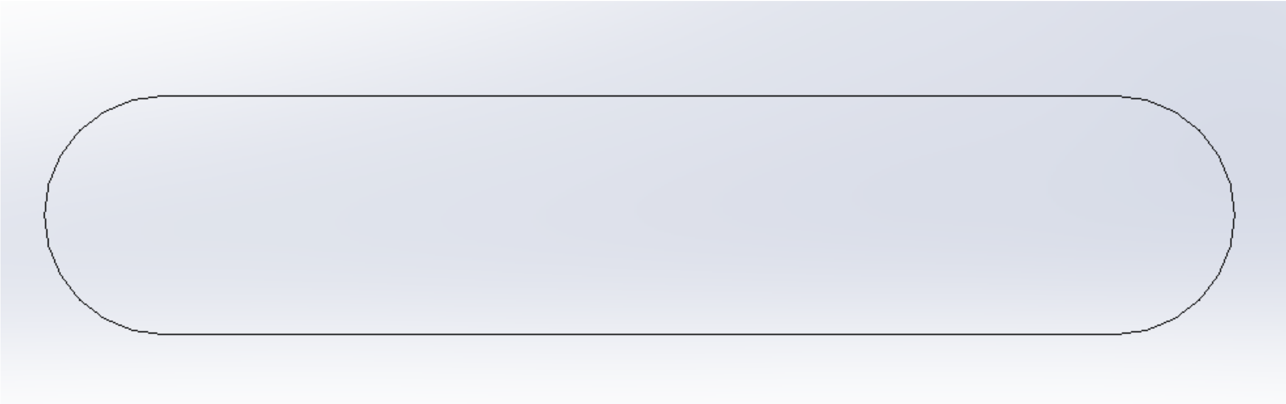

Fig. 8. Plate type (configuration iv)

The various configurations used for the aerofoil testing are given below Fig. 9. From left to right are the various profile configurations of the aerofoils; S9026, Clark Y, S1223 and plate type. From top to bottom are the angle of attack configurations of the corresponding aerofoils; 0°, 5°, 10°, 15° and 20°.

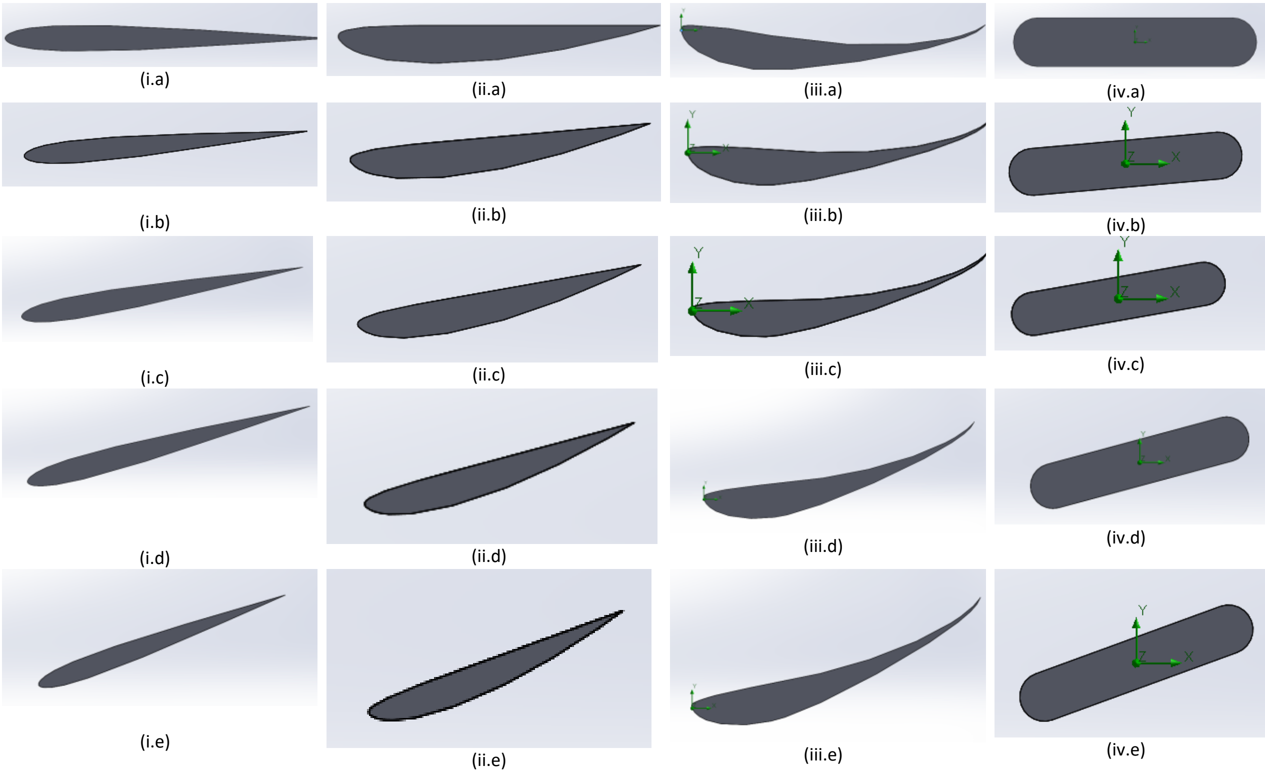

Fig. 9. Profile configurations for aerofoil trials; S9026 (i), Clark Y (ii), S1223 (iii) and plate type (iv) (from left to right). Angle of attack configurations; 0° (a), 5° (b), 10° (c), 15° (d) and 20° (e) (from top to bottom)

Each configuration was simulated at four flow velocities, 20 m/s, 30 m/s, 40 m/s and 50 m/s. The computational domain used for these simulations were based on the wing span and chord length of the profiles, i.e., three times the length chord length in the front, top and bottom, five times the chord length at the rear and three times the wing span at the sides. The initial conditions such as pressure and temperature were maintained according to computational testing standards, 101325 Pa and 293.2 K respectively. The flow was assumed to be incompressible as the Mach number is low.

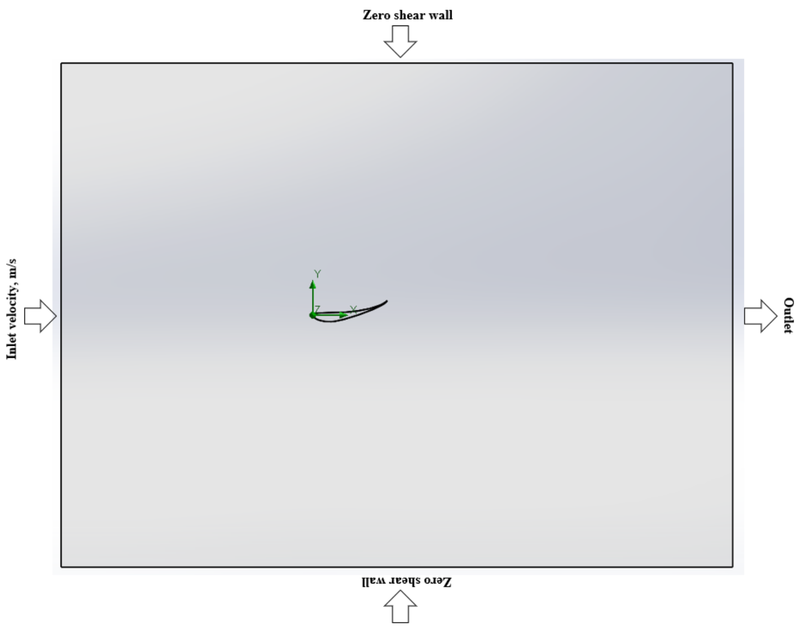

Fig. 10. S1223 aerofoil (at 10° angle of attack) inside the computational domain

The mesh set up was maintained constant for all the configurations. The level of initial mesh was set to 6 and the refinement parameter of the global domain under the calculation control option was set to 5. The mesh set up was set as per the above-mentioned parameters in order to get a fine mesh along the computational domain and around the profile of the aerofoil.

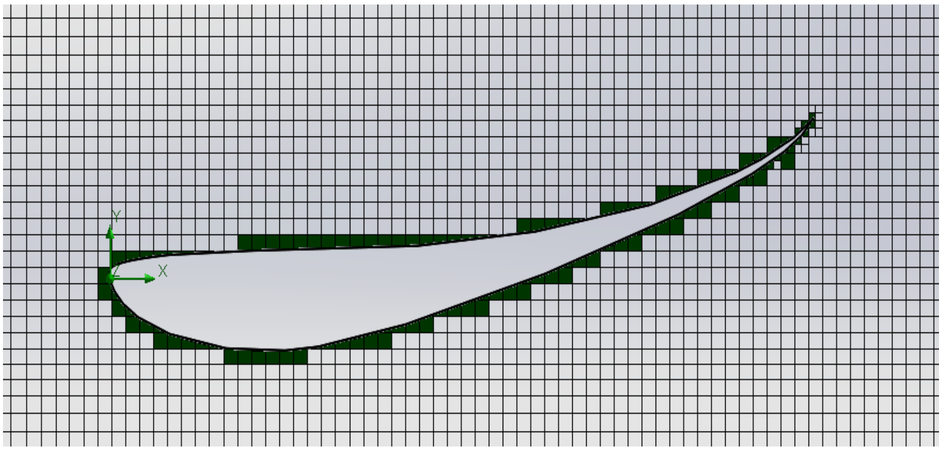

Fig. 11. Mesh set up for selected S1223 aerofoil at 10° angle of attack

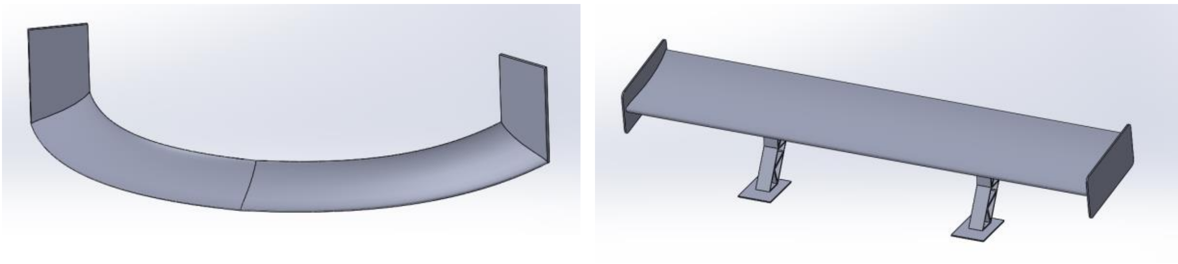

Fig. 12. Front spoiler and rear spoiler (from left to right) modelled with S1223 aerofoil at 10° angle of attack

2.2. Modelling of car

For the analysis of this project, 3-D model of Audi A7 was taken and modelled in SOLDWORKS software. The drag coefficient of the car is 0.27 according to the manufacturer's specification. The stock car does not have any spoiler. The purpose of adding spoiler is to reduce the drag coefficient further and to increase the downforce which is the most desired phenomenon in the automobile sector. Aerodynamic drag is the square of the vehicle speed or flow velocity in this case; hence velocity is a critical factor to be considered for CFD simulations.

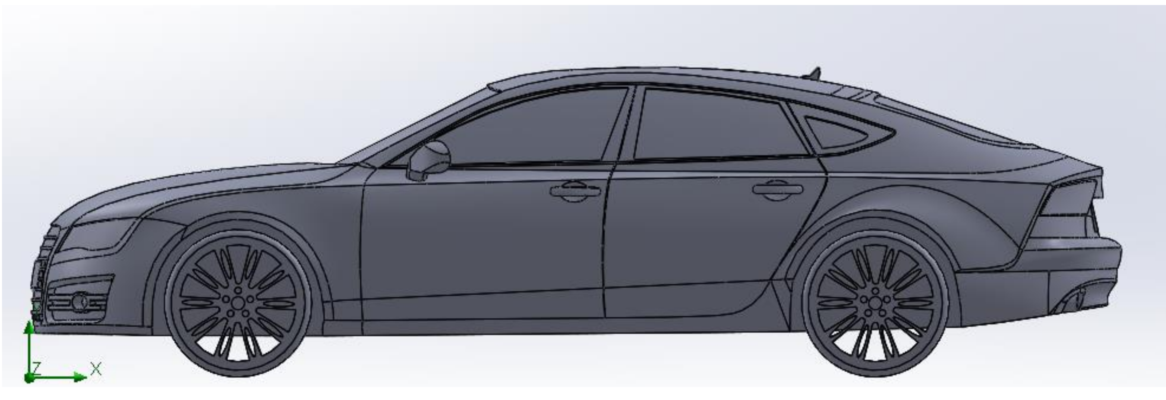

Fig. 13. 3-D model of Audi A7 (without any spoilers)

For cases 2 and 3, as mentioned earlier the spoilers were assembled on the car. Spoilers were modelled with S1223 aerofoil profile at 10° angle of attack based the results of comparison study of aerofoil profiles (which will be discussed in the further sections of this thesis). For case 2, the rear spoiler is assembled on the car as shown in Fig. 14., and for case 3 both front and rear spoilers are assembled on the car as shown in Fig. 15.

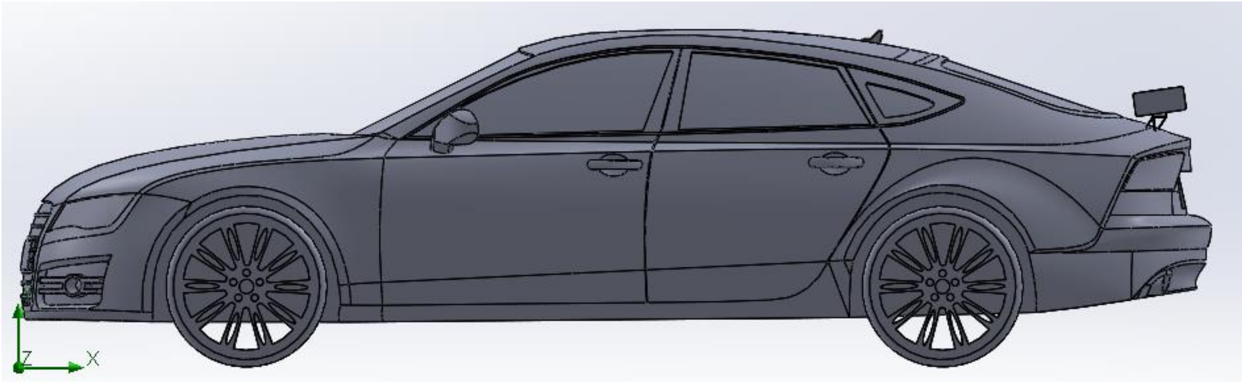

Fig. 14. Audi A7 assembled with rear spoiler

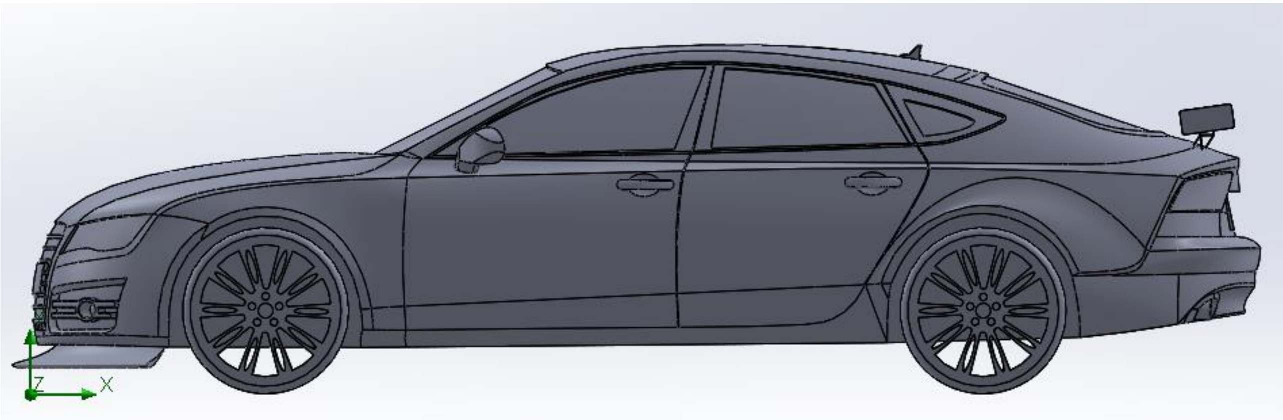

Fig. 15. Audi A7 assembled with both front and rear spoilers

The computational domain for the flow simulation of the car was set as three car length in front and at the sides of the car, five car length at the rear and a movable wall at the bottom tangent to the tires in order to simulate road set up. The road velocity is the same as the flow velocity of each case. The computational domain is the same for all the cases simulated in this section. The computational domain set up is shown in Fig. 16. The flow velocities for each case are 20 m/s, 30 m/s, 40 m/s and 50 m/s.

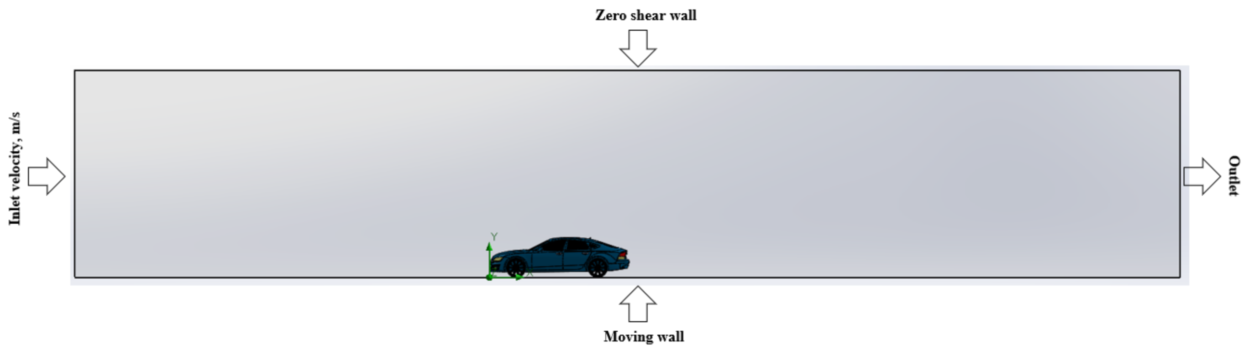

Fig. 16. Car inside the computational domain

The level of initial mesh was set to 7 and the refinement parameter of the global domain under the calculation control option was set to 7. The mesh set up was maintained same for all the cases. The mesh set up for all the three cases are shown in Fig. 17, 18 and 19.

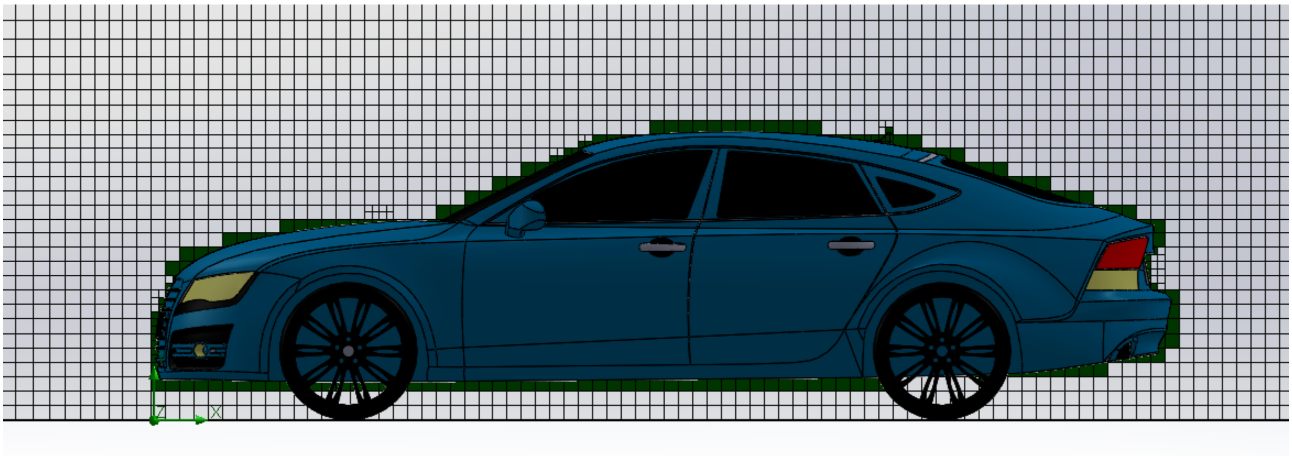

Fig. 17. Mesh generated for case 1 (car without spoilers)

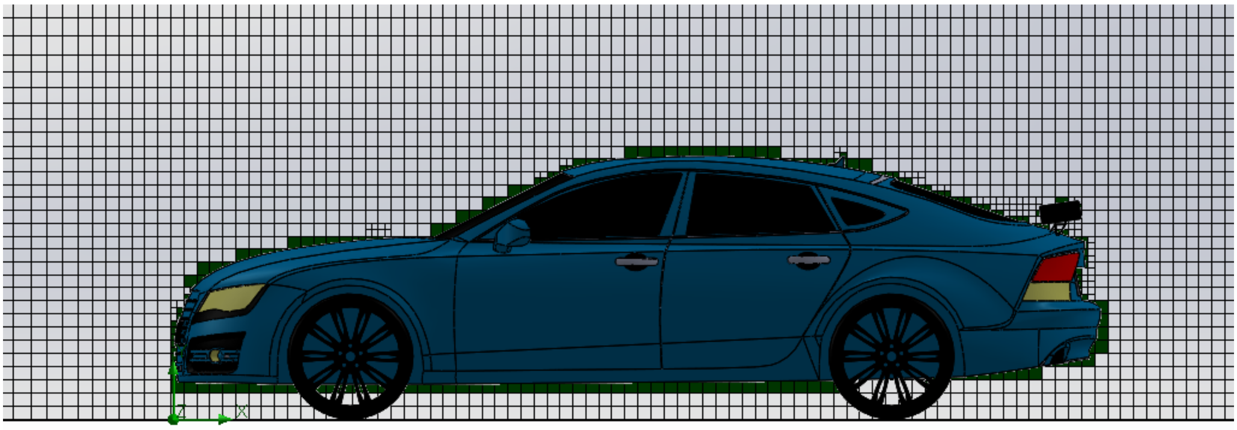

Fig. 18. Mesh generated for case 2 (car with rear spoiler)

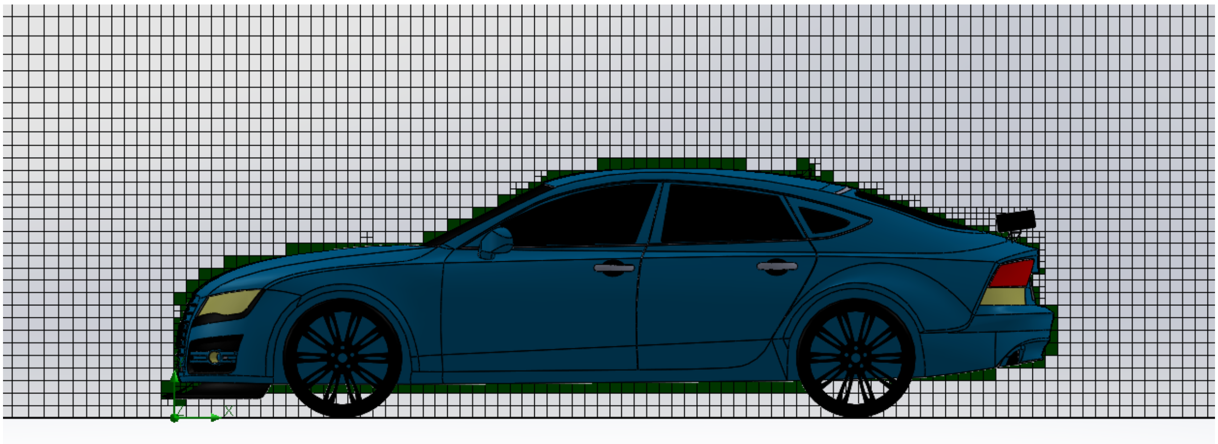

Fig. 19. Mesh generated for case 3 (car with both front and rear spoilers)

2.3. Modelling of static structural analysis

The purpose of doing static structural testing to determine the strength of the spoiler and to test the spoilers in real life scenarios. The forces acting on the spoilers were taken from the flow analysis of the aerofoils performed earlier. S1223 aerofoil with 10° angle of attack configuration was taken and the spoilers were modelled accordingly. Two materials were taken in order to do static analysis. The two materials chosen were ABS plastic and carbon fibre. The principle purpose of the static testing is to find out the stress, displacements and the factor of safety for both the spoilers with two materials mentioned earlier. The properties of the materials used for the testing are [44];

Table 1. Material properties

Properties	**ABS plastic**	**Carbon fibre**
Elastic modulus, N/mm^2	2000	7000
Poisson's ratio	0.394	0.45
Shear modulus, N/mm^2	318.9	7000
Mass density, kg/m^3	1020	1800
Tensile strength, N/mm^2	30	600
Compressive strength, N/mm^2	55.2	570

The carbon fibre properties used in this paper is carbon reinforced polymer (CFRP). It contains high strength and stiffness due to the high material damping of carbon fibre epoxy composites which has the ability to dissipate any vibration that is induced in them [49].

All the four structural testing configurations were meshed with curvature-based mesh with 1.6 element size growth ratio and mesh density set to fine. The mesh generated for front and rear spoilers are shown in Fig. 20 and 21.

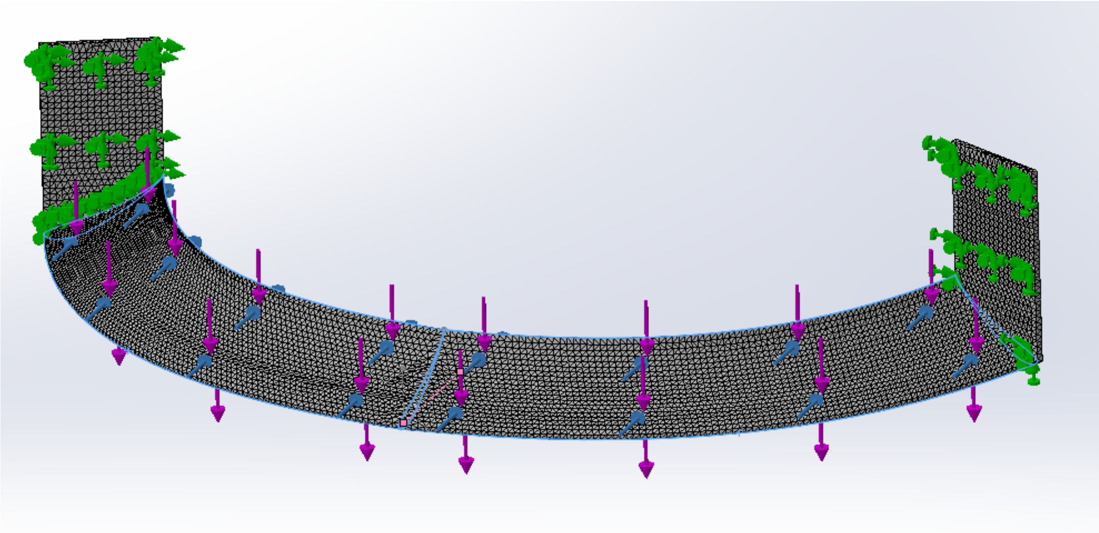

Fig. 20. Mesh generated with boundary conditions for front spoiler

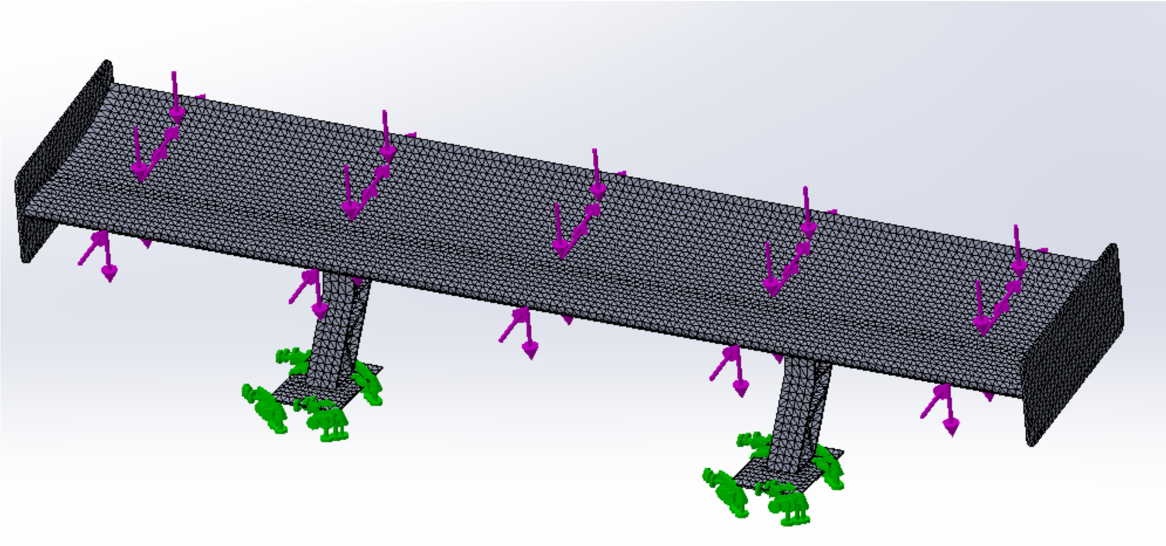

Fig. 21. Mesh generated with boundary conditions for rear spoiler

The spoilers were fixed using fixed support. The front spoiler was fixed on faces which mates with the inner face of vehicle body and the rear spoiler was fixed at the base face where it is assembled on the vehicle. The forces acquired from the flow analysis of S1223 aerofoil with 10° angle of attack at flow velocity 50 m/s (180 kmph) were applied, the force on x-direction was 79.334 N and the force on y-direction was 397.891 N. A separate flow was also done at 70 m/s to obtain the forces of drag and lift in order to do structural analysis of the spoilers at worst case scenario, the force on x-direction was 114.348 N and y-direction was 568.243 N (negative lift or downforce). No additional forces were taken into account.

3. Results of CFD simulations and structural analysis

This chapter is divided into two parts, first the approach and results of CFD analysis are presented and then the approach and results of structural analysis are presented. For CFD analysis as discussed in the previous chapter SOLIDWORKS flow simulation software was used. For structural analysis SOLIDWORKS simulation software was used. The results of the simulations are shown in the upcoming subsections, all the results will be discussed in the next chapter.

3.1. Results of CFD simulations

CFD simulations of aerofoils and the plate type profile were done first in order to determine the best profile and the angle of attack configuration suited for the final spoiler to be assembled on the car. After choosing the optimal profile and the angle of attack, the spoilers were modelled and assembled on the car. Three cases were then simulated with the car and spoiler setups in SOLIDWORKS flow simulation software as discussed in earlier sections.

Drag force and lift force or down force values were acquired in newtons from the simulations, from these values the drag coefficients and lift coefficients were calculated based on the formula mentioned in equation 1 and 2.

3.1.1. Results of CFD simulations of aerofoils and plate type profiles

The configurations for all the profiles and angles of attack were shown in Fig. 9. Each configuration was simulated at 20 m/s, 30 m/s, 40 m/s and 50 m/s flow velocity.

Table 2. CFD simulations results of S9026 aerofoil based on the different angle of attack configurations

Angle of attack	Flow velocity, m/s	Drag Force, N	Lift force, N	Drag co-efficient	Lift co-efficient
0°	20	0.611	0.189	0.008	0.003
	30	1.26	0.305	0.008	0.002
	40	2.119	-0.198	0.007	-0.001
	50	3.354	0.65	0.007	0.001
5°	20	1.75	-8.675	0.024	-0.118
	30	3.836	-19.8	0.023	-0.120
	40	6.516	-34.916	0.022	-0.119
	50	10.764	-60.82	0.023	-0.132
10°	20	5.822	-24.86	0.079	-0.338
	30	12.949	-56.1	0.078	-0.339
	40	22.959	-100.232	0.078	-0.341
	50	36.048	-157.831	0.078	-0.344
15°	20	11.931	-36.011	0.162	-0.490
	30	26.784	-81.736	0.162	-0.494
	40	47.745	-146.172	0.162	-0.497

Angle of attack	Flow velocity, m/s	Drag Force, N	Lift force, N	Drag co-efficient	Lift co-efficient
15°	50	74.509	-227.771	0.162	-0.496
20°	20	20.447	-42.825	0.278	-0.583
	30	45.727	-96.138	0.277	-0.581
	40	81.207	-171.507	0.276	-0.583
	50	126.959	-267.262	0.276	-0.582

Table 3. CFD simulations results of inverted Clark Y aerofoil based on the different angle of attack configurations

Angle of attack	Flow velocity, m/s	Drag Force, N	Lift force, N	Drag co-efficient	Lift co-efficient
0°	20	2.665	-11.751	0.029	-0.128
	30	5.877	-27.301	0.028	-0.132
	40	11.763	-63.587	0.032	-0.173
	50	20.061	-113.476	0.035	-0.198
5°	20	4.716	-29.746	0.051	-0.324
	30	10.669	-68.754	0.052	-0.333
	40	20.512	-134.658	0.056	-0.366
	50	33.096	-223.014	0.058	-0.388
10°	20	9.247	-49.211	0.101	-0.536
	30	20.932	-112.012	0.101	-0.542
	40	37.189	-199.581	0.101	-0.543
	50	59.599	-321.114	0.104	-0.559
15°	20	20.498	-62.737	0.223	-0.683
	30	46.015	-141.563	0.223	-0.685
	40	81.721	-251.992	0.222	-0.686
	50	127.456	-395.772	0.222	-0.689
20°	20	26.181	-75.305	0.285	-0.820
	30	58.953	-169.669	0.285	-0.821
	40	104.788	-301.342	0.285	-0.820
	50	163.871	-470.107	0.285	-0.819

Table 4. CFD simulations results of S1223 aerofoil based on the different angle of attack configurations

Angle of attack	Flow velocity, m/s	Drag Force, N	Lift force, N	Drag co-efficient	Lift co-efficient
0°	20	4.375	-18.862	0.060	-0.257
	30	9.591	-42.387	0.058	-0.256

Angle of attack	Flow velocity, m/s	Drag Force, N	Lift force, N	Drag co-efficient	Lift co-efficient
0°	40	22.787	-106.983	0.078	-0.364
	50	35.452	-167.968	0.077	-0.366
5°	20	6.814	-41.57	0.093	-0.566
	30	16.671	-100.103	0.101	-0.605
	40	33.404	-197.654	0.114	-0.672
	50	54.988	-319.94	0.120	-0.696
10°	20	12.38	-62.076	0.168	-0.845
	30	27.968	-140.43	0.169	-0.849
	40	50.116	-251.282	0.170	-0.855
	50	79.334	-397.891	0.173	-0.866
15°	20	17.64	-79.967	0.240	-1.088
	30	39.28	-176.321	0.238	-1.066
	40	70.153	-317.196	0.239	-1.079
	50	109.977	-496.563	0.239	-1.081
20°	20	25.952	-73.649	0.353	-1.002
	30	57.993	-164.657	0.351	-0.996
	40	104.11	-296.759	0.354	-1.009
	50	162.133	-461.888	0.353	-1.005

Table 5. CFD simulations results of plate type profile based on the different angle of attack configurations

Angle of attack	Flow velocity, m/s	Drag Force, N	Lift force, N	Drag co-efficient	Lift co-efficient
0°	20	5.459	0.224	0.297	0.012
	30	11.51	0.867	0.278	0.021
	40	20.86	0.333	0.284	0.005
	50	30.649	1.236	0.267	0.011
5°	20	8.555	-28.339	0.466	-1.542
	30	19.577	-66.198	0.474	-1.601
	40	35.3	-116.546	0.480	-1.586
	50	55.535	-180.812	0.484	-1.574
10°	20	14.096	-30.481	0.767	-1.659
	30	31.835	-68.55	0.770	-1.658
	40	56.801	-119.154	0.773	-1.621
	50	88.751	-183.791	0.773	-1.600
15°	20	21.186	-39.764	1.153	-2.164

Angle of attack	Flow velocity, m/s	Drag Force, N	Lift force, N	Drag co-efficient	Lift co-efficient
15°	30	47.558	-88.95	1.150	-2.151
	40	84.713	-159.577	1.153	-2.171
	50	131.915	-252.598	1.149	-2.199
20°	20	27.661	-43.647	1.505	-2.375
	30	62.195	-98.49	1.504	-2.382
	40	110.907	-173.494	1.509	-2.360
	50	174.03	-269.599	1.515	-2.348

3.1.2. Results of CFD simulations of car

The computational domain set up is shown in Fig. 15 and the configuration of all the setups were shown in Fig. 16, 17 and 18. The flow velocities used are 20 m/s, 30 m/s, 40 m/s and 50 m/s.

Table 6. CFD simulations results of car without spoilers (case 1)

Flow velocity, m/s	Drag Force, N	Downforce, N	Drag co-efficient	Lift co-efficient
20	168.304	-137.024	0.300	-0.244
30	383.38	-308.353	0.304	-0.244
40	681.294	-548.204	0.304	-0.244
50	1056.63	-857.533	0.301	-0.245

Table 7. CFD simulations results of car with rear spoiler (case 2)

Flow velocity, m/s	Drag Force, N	Downforce, N	Drag co-efficient	Lift co-efficient
20	178.663	-149.625	0.318	-0.267
30	404.756	-338.21	0.321	-0.268
40	721.498	-599.734	0.321	-0.267
50	1124.585	-932.847	0.321	-0.266

Table 8. CFD simulations results of car with both front and rear spoilers (case 3)

Flow velocity, m/s	Drag Force, N	Downforce, N	Drag co-efficient	Lift co-efficient
20	166.926	-153.753	0.298	-0.274
30	375.74	-346.742	0.298	-0.275
40	669.072	-616.443	0.298	-0.275
50	1044.481	-967.24	0.298	-0.276

Fig. 22. Pressure distribution for case 1 at different velocities; 20 m/s, 30 m/s, 40 m/s and 50 m/s (from top to bottom)

In the above pressure contour diagram, the pressure around the vehicle and the pressure due to flow on different parts of the vehicle can be seen. By adding spoilers, the flow of air around the vehicle gets disturbed. The main objective of adding a spoiler to a vehicle is to increase the downforce at the same to decrease the drag co-efficient. The spoiler tends to push the vehicle downwards which disturbs the air flow at the rear and thereby increasing the downforce acting on the vehicle [3].

Fig. 23. Pressure distribution for case 2 at different velocities; 20 m/s, 30 m/s, 40 m/s and 50 m/s (from top to bottom)

After adding the spoiler, the pressure at the rear of the vehicle was disturbed which could be seen from Fig. 23. After adding the front spoiler, the air at the front gets dispersed to the sides thus reducing the drag and increasing the downforce helping the car to have better traction [3].

Fig. 24. Pressure distribution for case 3 at different velocities; 20 m/s, 30 m/s, 40 m/s and 50 m/s (from top to bottom)

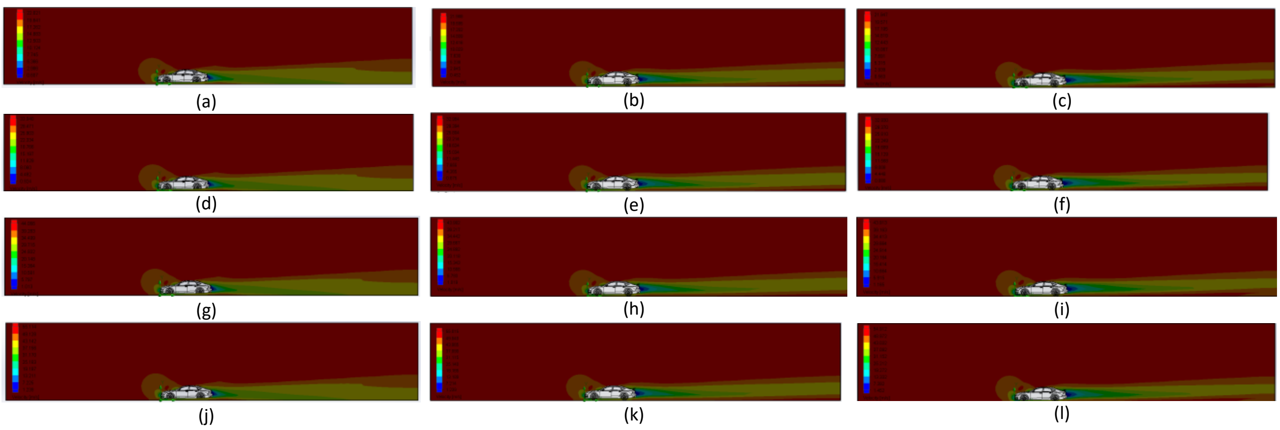

Fig. 25. Velocity contours for all three cases at different velocities; 20 m/s (a, b, c), 30 m/s (d, e, f), 40 m/s (g, h, i) and 50 m/s (j, k, l). Case 1 (a, d, g, j), case 2 (b, e, h, k) and case 3(c, f, i, l)

3.2. Results of static structural analysis

The analysis was linear static analysis since the main goal was to find of the strength of the materials used and to study the material behaviour within its elastic range. Fixed support was used to fix the spoilers [38]. Fixed support was used on the faces of front spoiler which mates with the inner face of vehicle body and base of the rear spoiler which mates with the trunk of the vehicle. The forces acquired from the S1223 aerofoil flow analysis (at 10° angle of attack) at flow velocity 50 m/s (180 kmph) were utilised, the force on x-direction was 79.334 N and the force on Y-direction was 397.891 N.

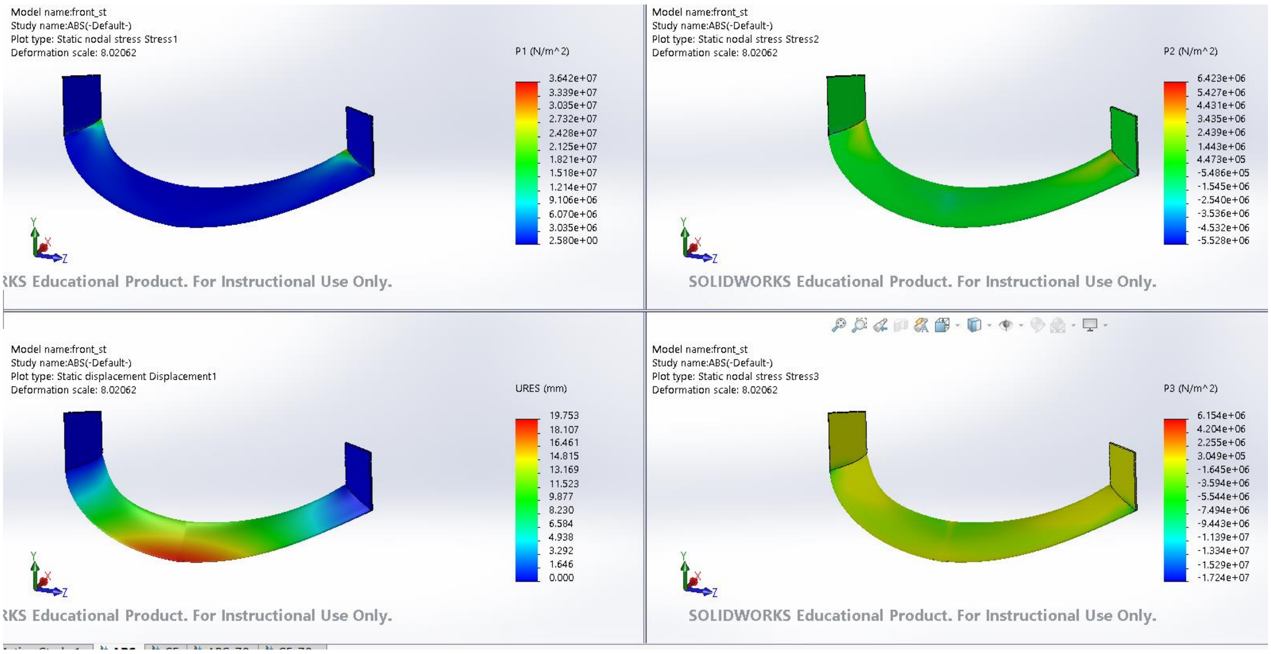

Fig. 26. Results of structural analysis of front spoiler with ABS plastic with forces obtained from 50 m/s flow velocity (Clockwise from top left; Principle stress P1, P2, P3 and displacement)

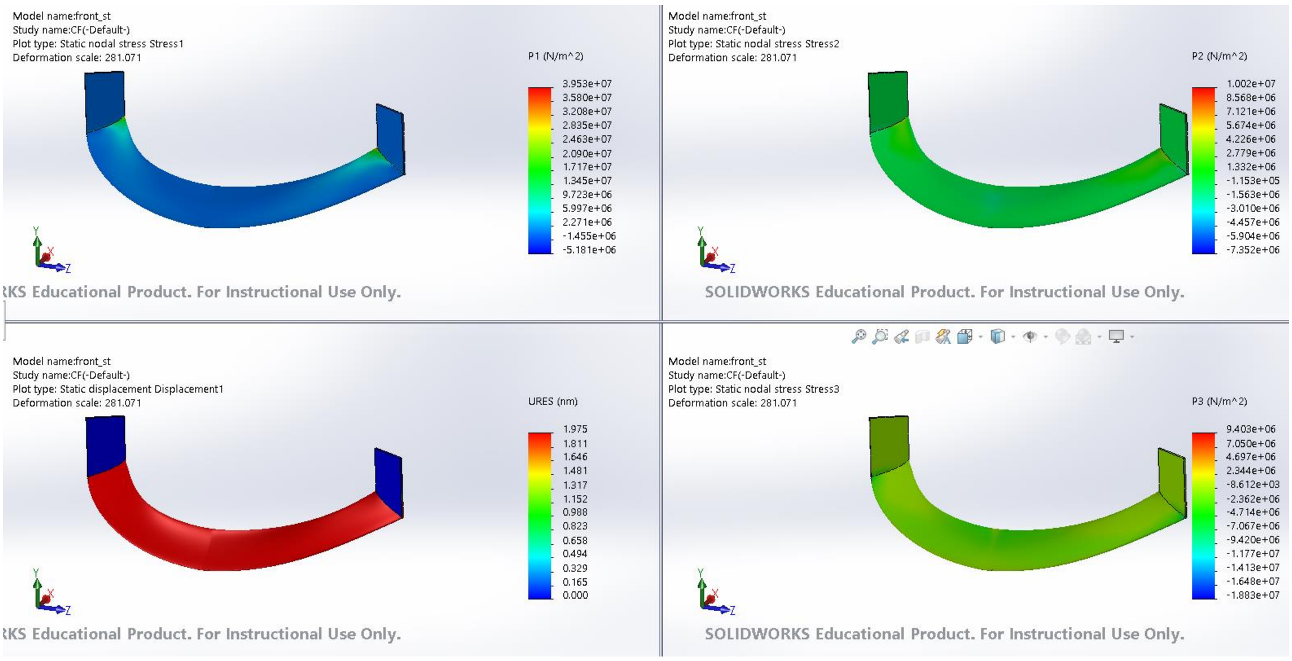

Fig. 27. Results of structural analysis of front spoiler with carbon fibre with forces obtained from 50 m/s flow velocity (Clockwise from top left; Principle stress P1, P2, P3 and displacement)

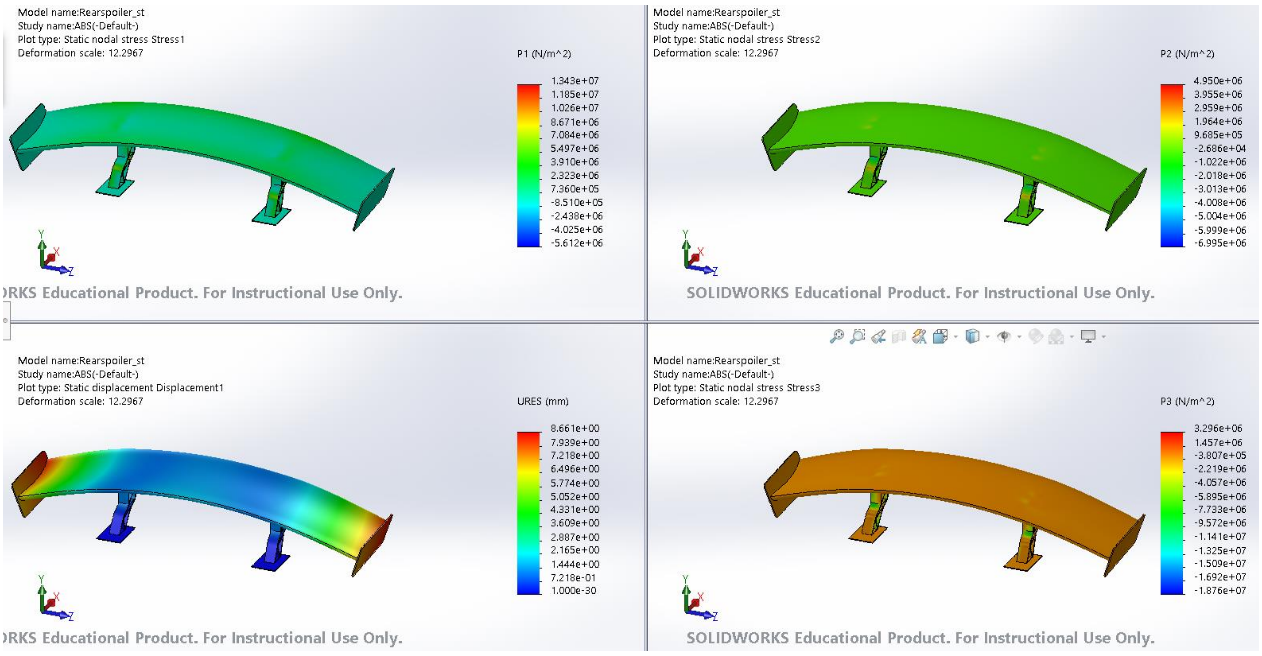

Fig. 28. Results of structural analysis of rear spoiler with ABS plastic with forces obtained from 50 m/s flow velocity (Clockwise from top left; Principle stress P1, P2, P3 and displacement)

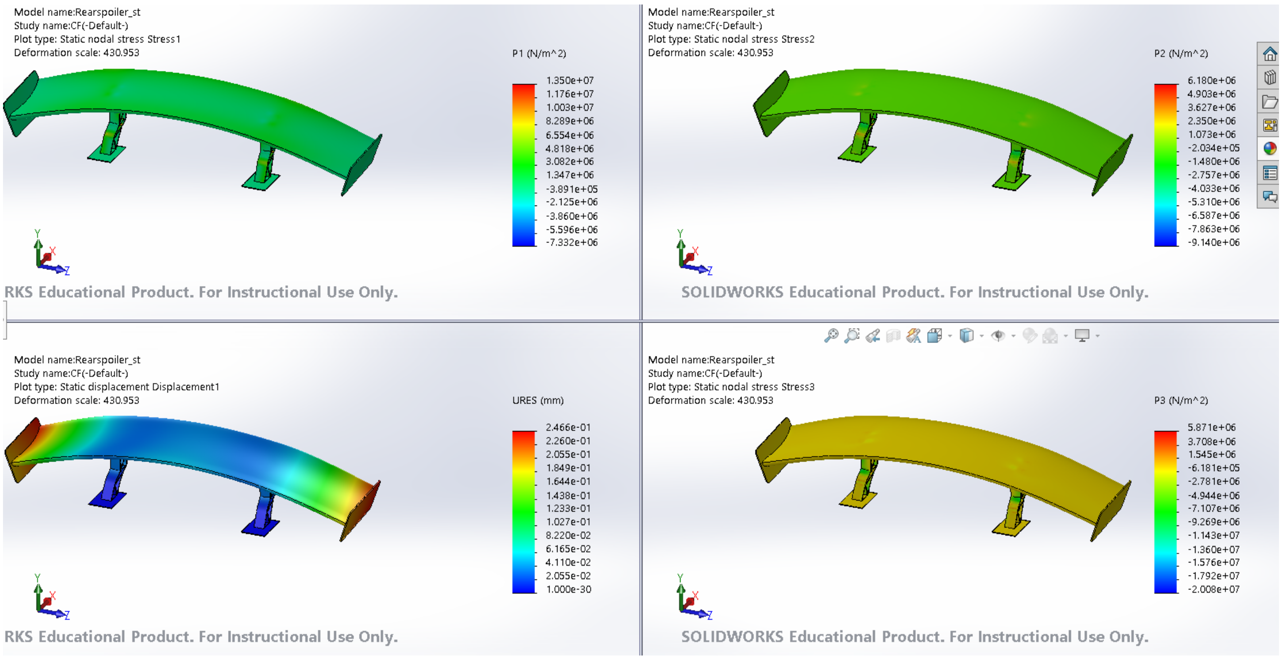

Fig. 29. Results of structural analysis of rear spoiler with carbon fibre with forces obtained from 50 m/s flow velocity (Clockwise from top left; Principle stress P1, P2, P3 and displacement)

Table 9. Results of structural analysis of spoilers with ABS plastic with forces obtained from 50 m/s flow velocity

Description (Maximum values)	**Principle stress, P1, N/mm^2**	**Principle stress, P2, N/mm^2**	**Principle stress, P3, N/mm^2**	**Displacement, mm**
Front spoilers	36.42	6.423	6.154	19.753
Rear spoiler	13.43	4.95	3.296	8.661

Table 10. Results of structural analysis of spoilers with carbon fibre with forces obtained from 50 m/s flow velocity

Description (Maximum values)	**Principle stress, P1, N/mm²**	**Principle stress, P2, N/mm²**	**Principle stress, P3, N/mm²**	**Displacement, mm**
Front spoilers	39.53	10.02	9.403	1.975
Rear spoiler	13.50	6.18	5.871	0.247

To do structural analysis at worst case scenario, a flow analysis was carried out at 70 m/s and forces of drag and lift were acquired. The lift force was 114.348 N (force on x-direction) and downforce was 568.243 N (force on y-direction). These forces were used to do structural testing at worst case scenario and the results obtained are shown below.

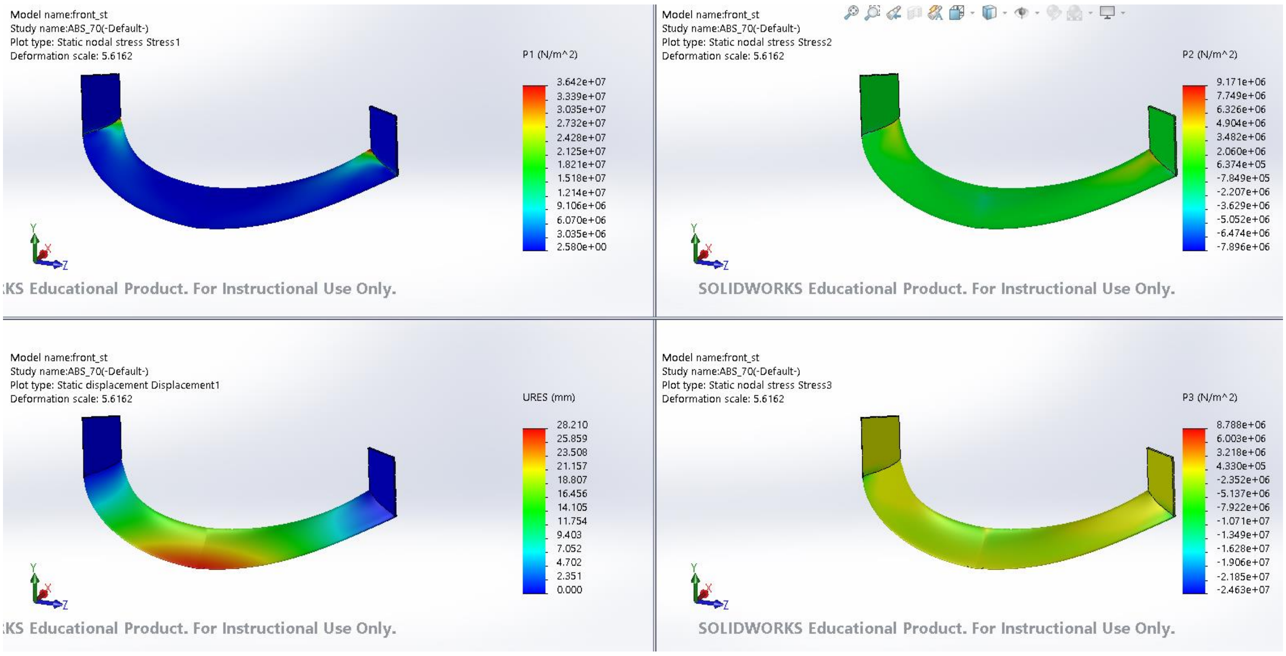

Fig. 30. Results of structural analysis of front spoiler with ABS plastic with forces obtained from 70 m/s flow velocity (Clockwise from top left; Principle stress P1, P2, P3 and displacement)

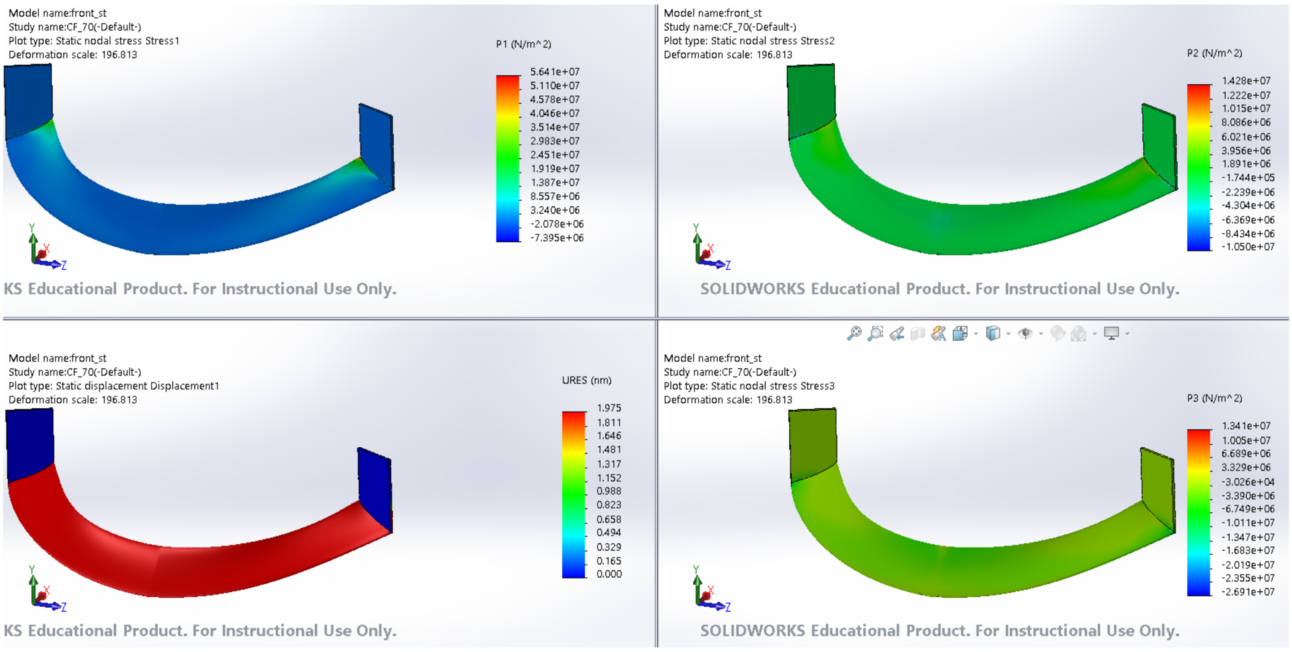

Fig. 31. Results of structural analysis of front spoiler with carbon fibre with forces obtained from 70 m/s flow velocity (Clockwise from top left; Principle stress P1, P2, P3 and displacement)

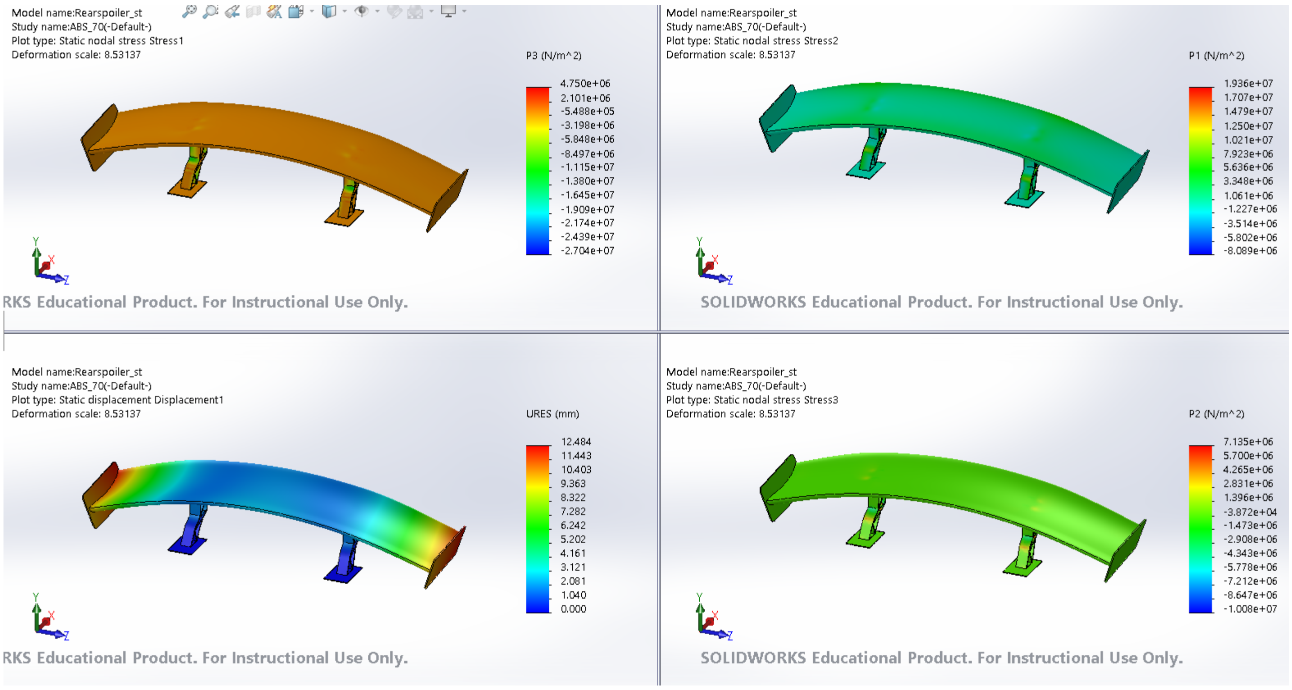

Fig. 32. Results of structural analysis of rear spoiler with ABS plastic with forces obtained from 70 m/s flow velocity (Clockwise from top left; Principle stress P3, P1, P2 and displacement)

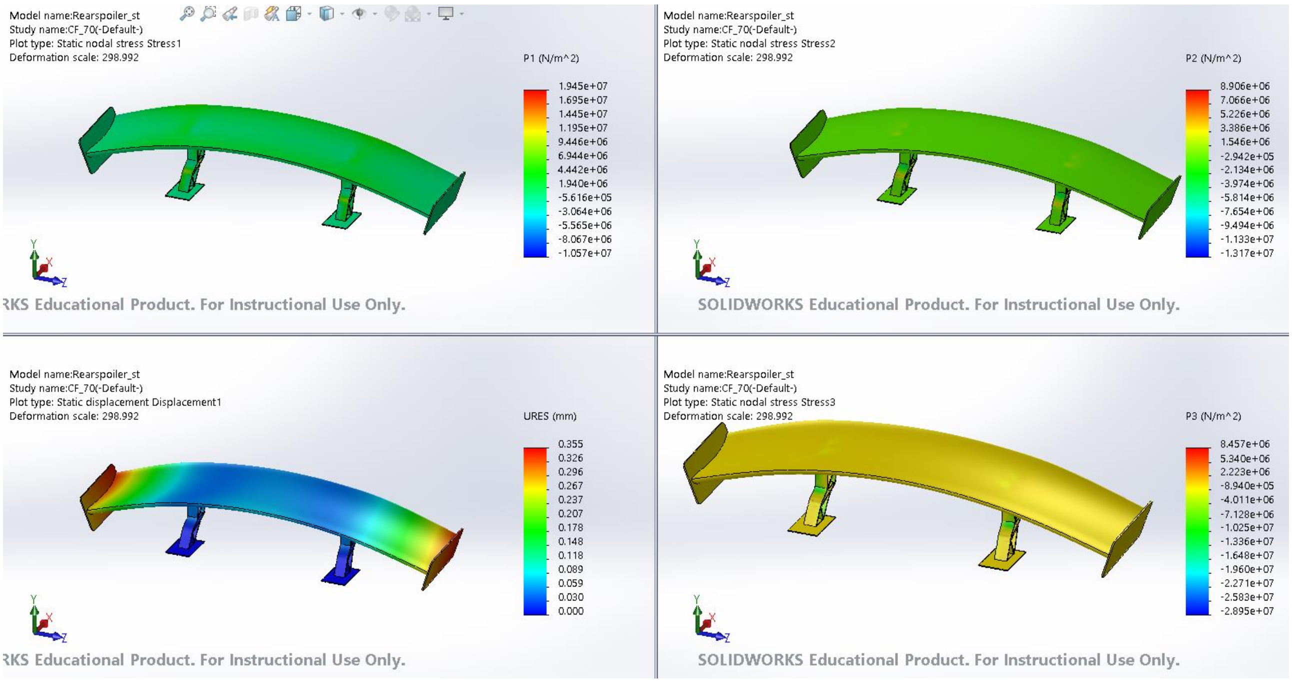

Fig. 33. Results of structural analysis of rear spoiler with carbon fibre with forces obtained from 70 m/s flow velocity (Clockwise from top left; Principle stress P1, P2, P3 and displacement)

Table 11. Results of structural analysis of spoilers with ABS plastic with forces obtained from 70 m/s flow velocity

Description (Maximum values)	**Principle stress, P1, N/mm^2**	**Principle stress, P2, N/mm^2**	**Principle stress, P3, N/mm^2**	**Displacement, mm**
Front spoilers	36.42	9.171	8.788	28.21
Rear spoiler	19.36	7.135	4.75	12.484

Table 12. Results of structural analysis of spoilers with carbon fibre with forces obtained from 70 m/s flow velocity

Description (Maximum values)	**Principle stress, P1, N/mm^2**	**Principle stress, P2, N/mm^2**	**Principle stress, P3, N/mm^2**	**Displacement, mm**
Front spoilers	56.41	14.28	13.41	1.975
Rear spoiler	19.45	8.906	8.457	0.355

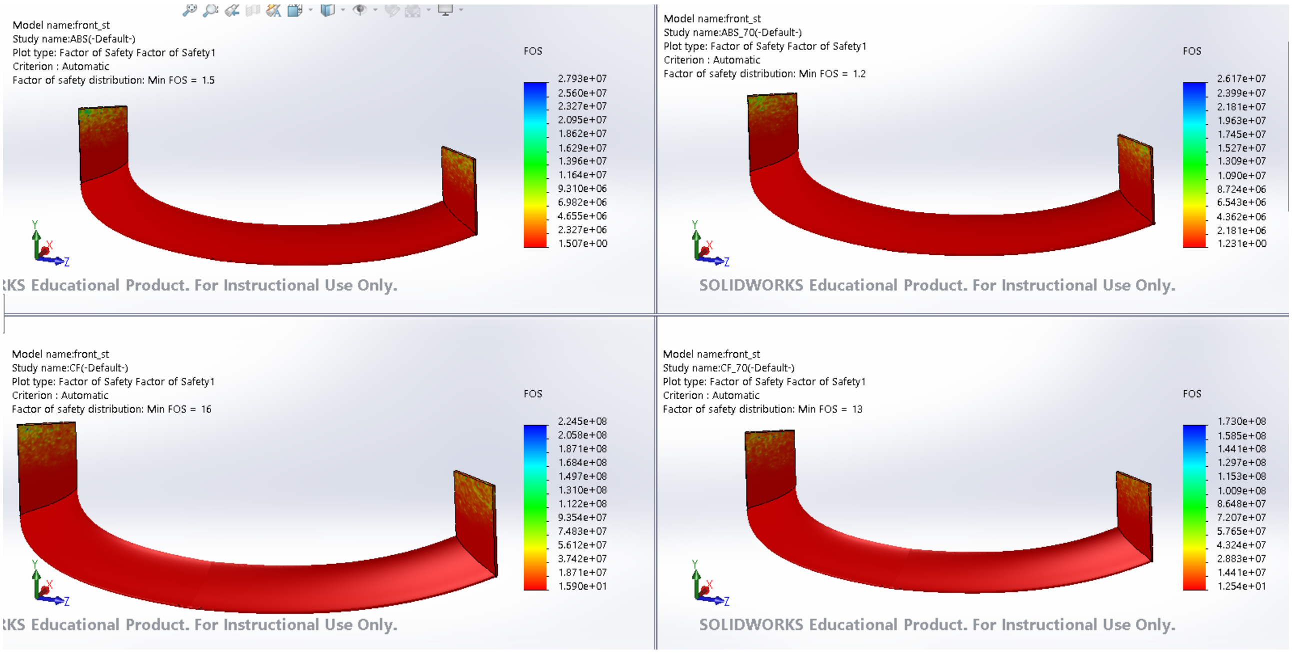

Fig. 34. Factor of safety of front spoiler with ABS plastic (top) and carbon fibre (bottom) with forces obtained from 50 m/s (left) and 70 m/s (right) flow velocities

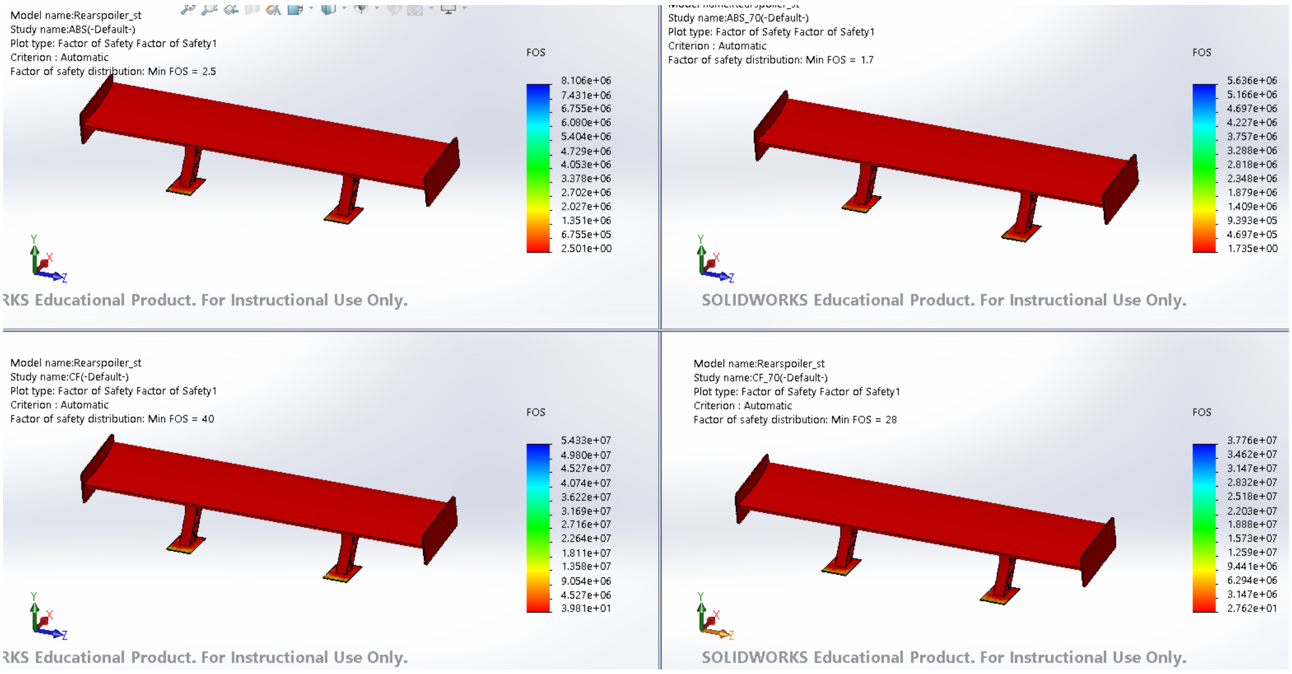

Fig. 35. Factor of safety of rear spoiler with ABS plastic (top) and carbon fibre (bottom) with forces obtained from 50 m/s (left) and 70 m/s (right) flow velocities

4. Discussions

In this chapter all the results that were shown in the previous chapters are discussed with the help of graphical tools. From the data (Table. 2) of the flow simulation trials done on S9026 aerofoil, it could be seen that the drag increases as the angle of attack increases from 0°, downforce also increase along with it. In fig. 36, drag force increases drastically after 10° angle of attack, it means that there might be a "stall" condition after 10° [18]. There is also a drastic change in the co-efficient of drag (from 0.007 to 0.078) and lift values (from 0.001 to -0.344) from 0° to 10° angle of attack.

It could be stated that the optimum angle of attack for this aerofoil configuration is 10°, the reason to have arrived at this conclusion is from the plot of co-efficient of drag and lift plot where this configuration has the high co-efficient of lift values with low co-efficient of drag values compared to the other angle of attack configuration trials performed on this aerofoil.

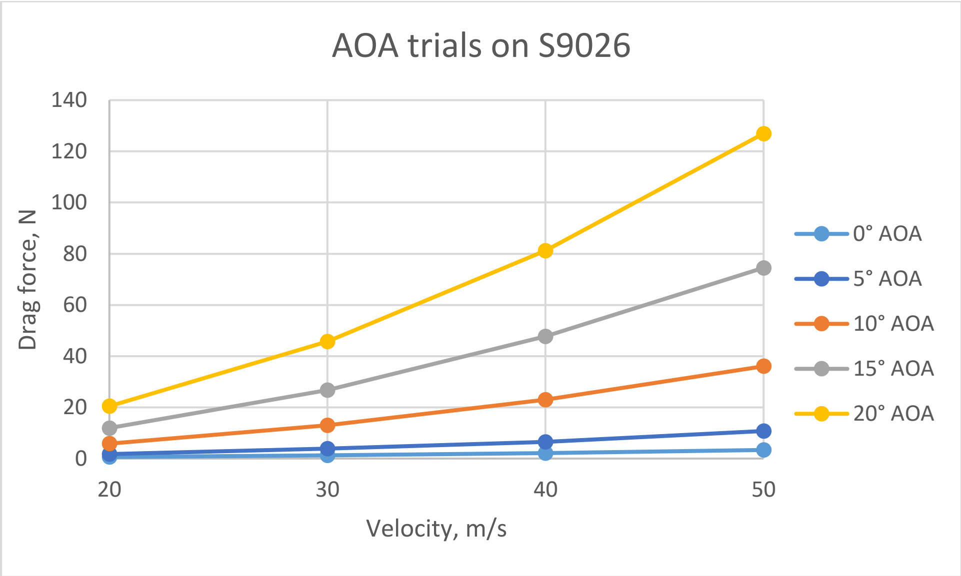

Fig. 36. Drag force for the angle of attack trials on S9026 aerofoil

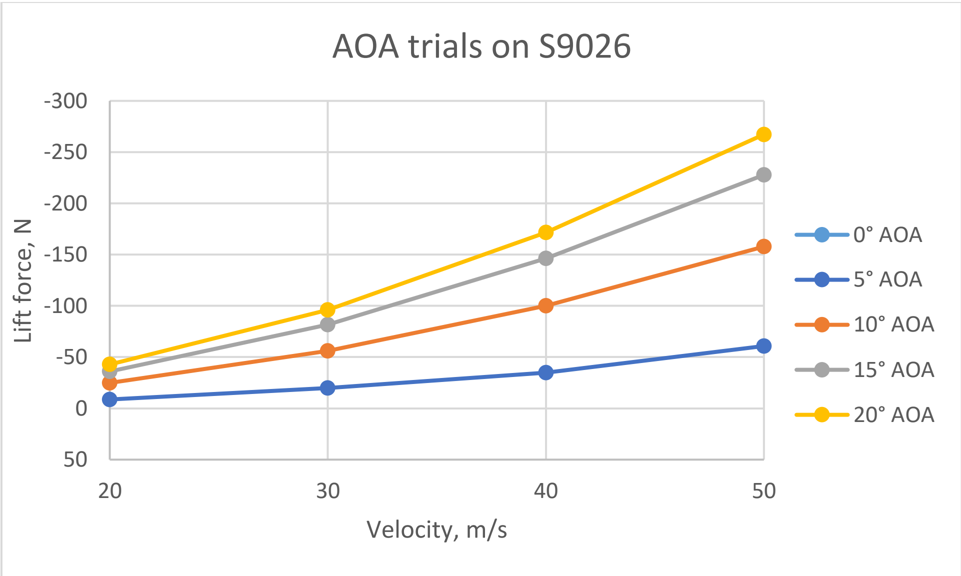

Fig. 37. Lift force for the angle of attack trials on S9026 aerofoil

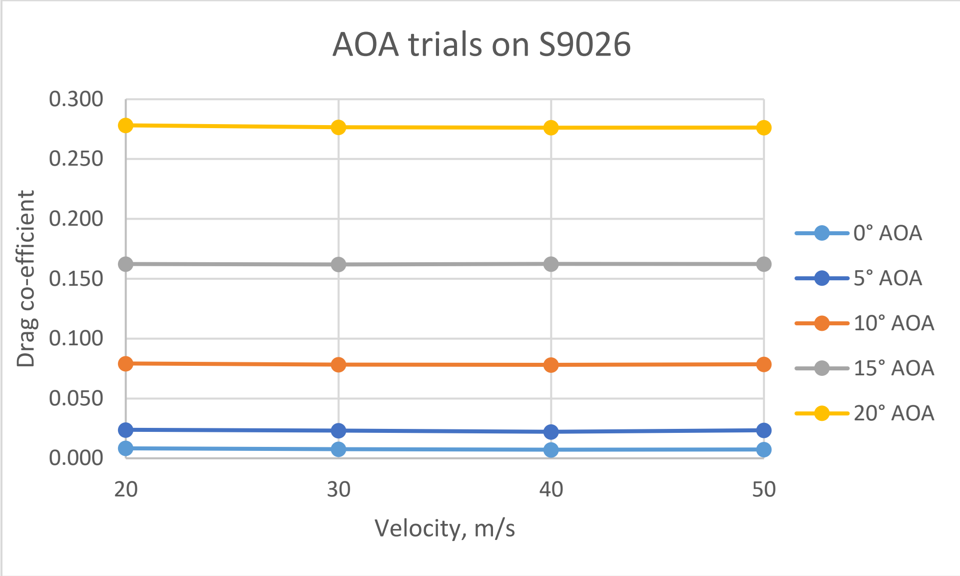

Fig. 38. Drag co-efficient for the angle of attack trials S9026 aerofoil

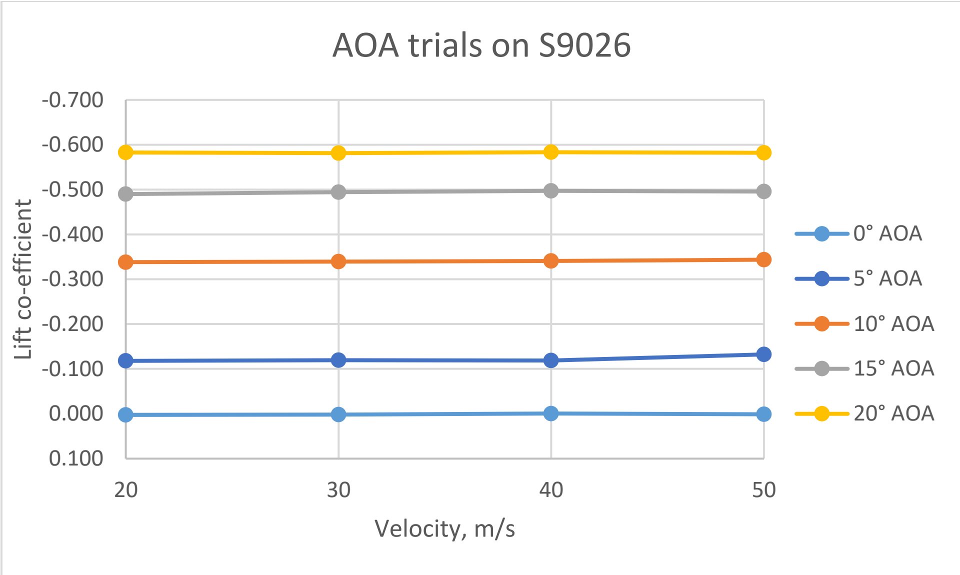

Fig. 39. Lift co-efficient for the angle of attack trials on S9026 aerofoil

From the results of the flow simulation trials done on the inverted Clark Y aerofoil (Table. 3), it could be seen that the lift co-efficient is much better than the results obtained in S9026 aerofoil. The same condition which was earlier observed in S9026 aerofoil could also be seen in Clark Y aerofoil, after 10° angle of attack the drag co-efficient increased from 0.104 to 0.222 for 15° angle of attack.

At the same 10° angle of attack the lift co-efficient value is -0.559 which is good compare to the lift co-efficient value obtained from S9026 configurations. From this it could be stated that the best angle of attack configuration for inverted Clark Y aerofoil is with 10° angle of attack, it could also be stated that on comparing with S9026; inverted Clark Y with 10° angle of attack configuration is the optimal configuration so far.

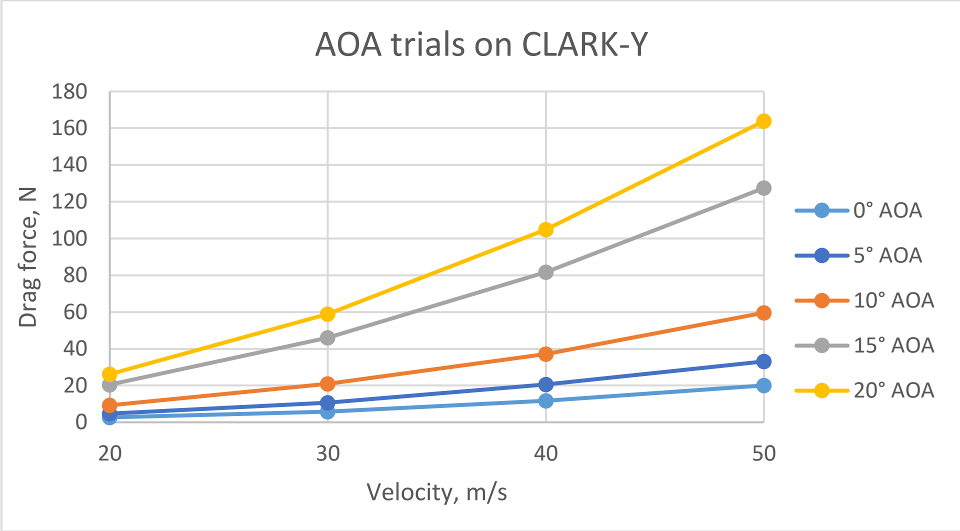

Fig. 40. Drag force for angle of attack trials on Clark Y aerofoil

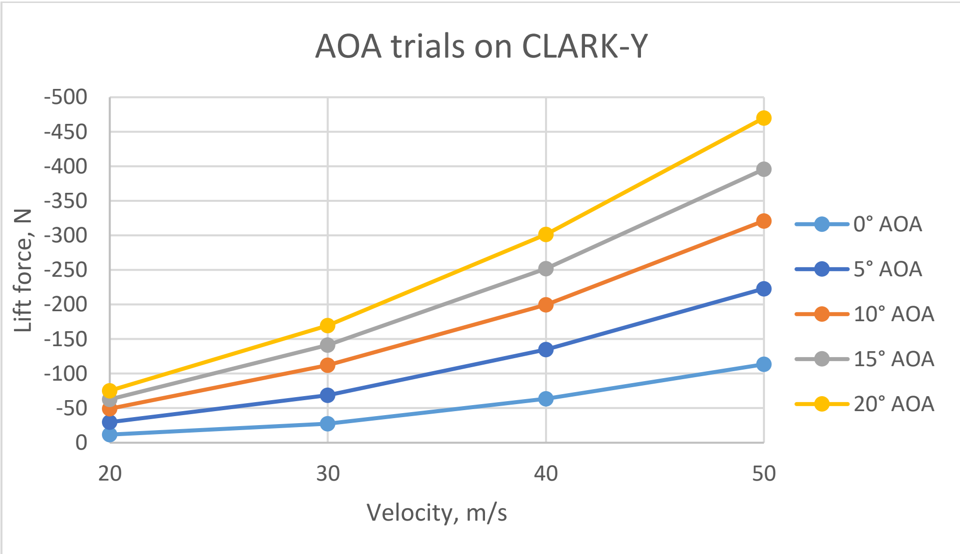

Fig. 41. Lift force for angle of attack trials on Clark Y aerofoil

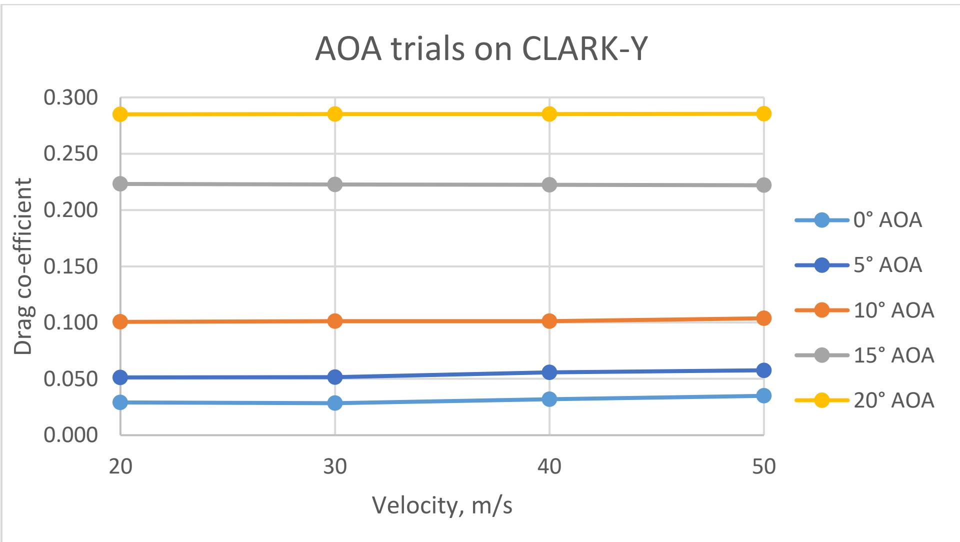

Fig. 42. Drag co-efficient for angle of attack trials on Clark Y aerofoil

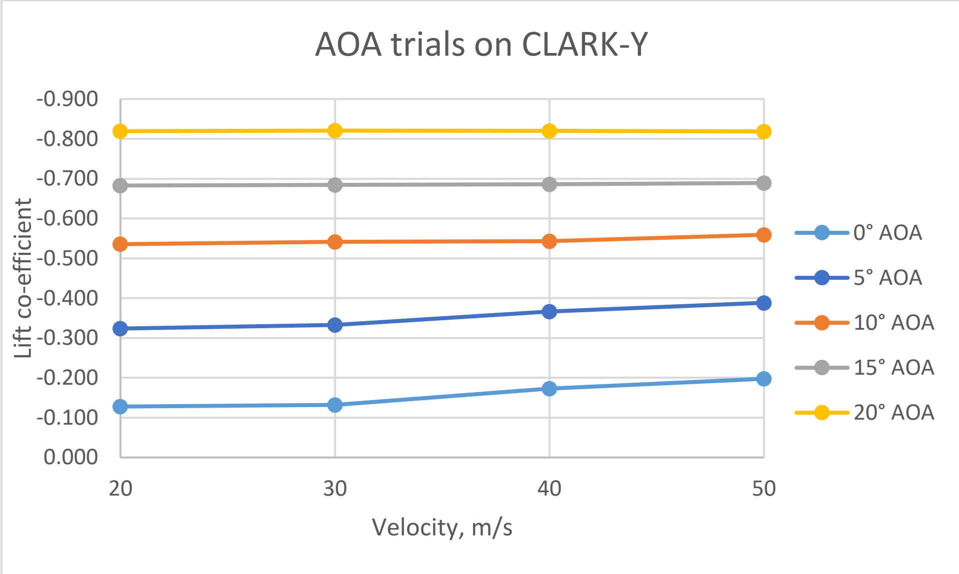

Fig. 43. Lift co-efficient for angle of attack trials on Clark Y aerofoil

Table. 4 shows the results obtained from the angle of attack trials performed on S1223 aerofoil. In S1223 aerofoil, it could be seen that the "stall" condition occurs at 15° angle of attack where the drag co-efficient value is 0.239 and rises to 0.353 at 20° angle of attack. On comparing the drag and lift co-efficient values between 10° and 15° angle of attack, it could be noted that the drag value is less by 39% to a 25% reduced lift co-efficient value. Reducing the drag with a good downforce is the main objective of this paper hence the configuration with less drag co-efficient could be selected as the best configuration, hence 10° angle of attack could be chosen as the optimal configuration for S1223 aerofoil.

Comparing the drag co-efficient and lift co-efficient values of S1223 and Clark Y (which is previously selected as the best configuration), it could be noted that the drag co-efficient is higher by 66% at the same time the lift co-efficient is higher by 55%, in this scenario the priority could be given to lift co-efficient because the maximum downforce created in Clark Y configuration very less than that of the downforce created in S1223 aerofoil with less co-efficient of drag. It could be decided with the previous explanation that S1223 aerofoil with 10° angle of attack configuration is the optimal configuration so far in this aerofoil trials.

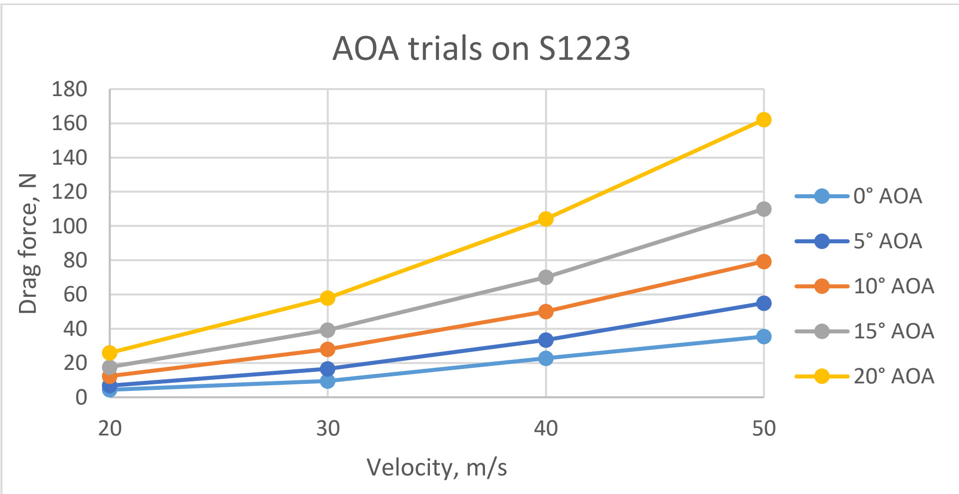

Fig. 44. Drag force for angle of attack trials on S1223 aerofoil

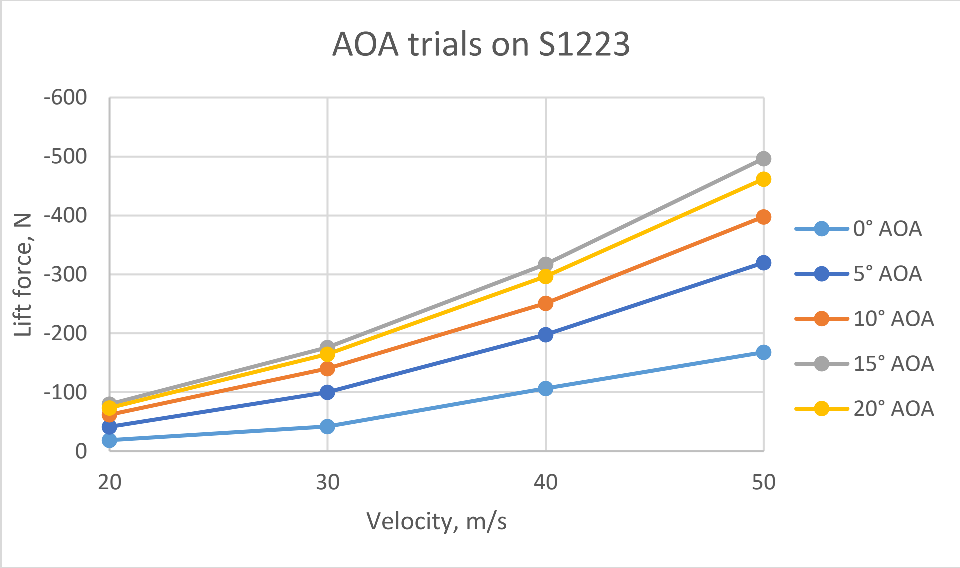

Fig. 45. Lift force for angle of attack trials on S1223 aerofoil

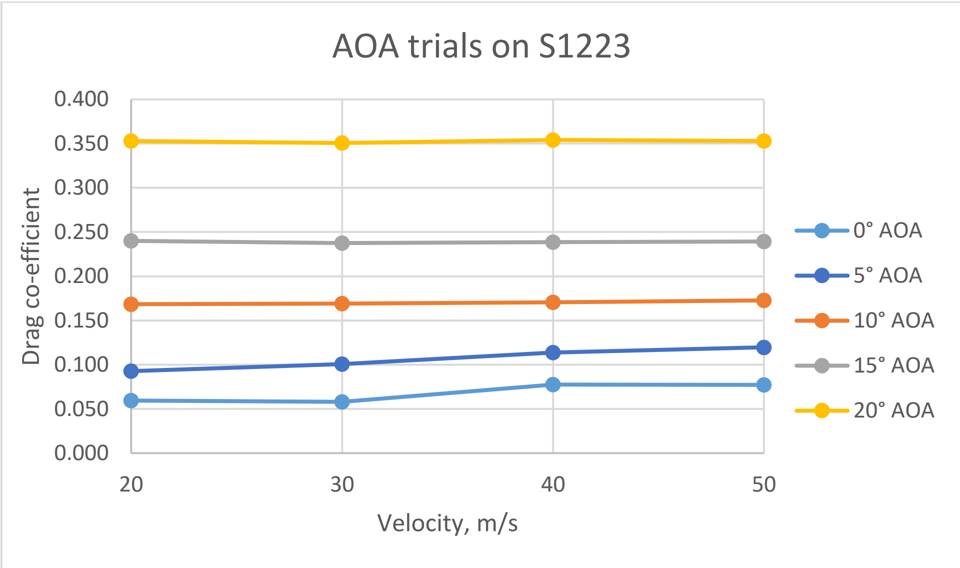

Fig. 46. Drag co-efficient for angle of attack trials on S1223 aerofoil

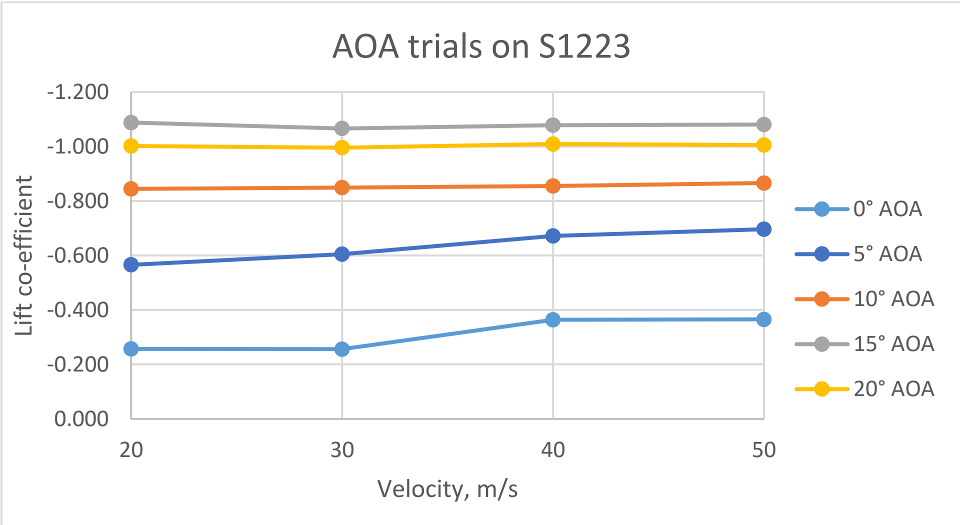

Fig. 47. Lift co-efficient for angle of attack trials on S1223 aerofoil

Table. 5 deals with results obtained from the flow simulation trials done on plate type profile for spoilers. From the data recorded it could be observed that the profile is not effective at 0° angle of attack as it produces positive lift co-efficient which means there is upward force acting on the profile which will produce undesired phenomenon on the vehicle. At 5° and 10° angle of attack the lift forces produced are very low and from 15° the drag forces and lift forces are relatively high when compared to 10° angle of attack. So, 10° angle of attack can be taken as better configuration for this plate type profile with 0.773 co-efficient of drag and -1.6 co-efficient of lift.

Although, comparing these values with the previously obtained values from S1223 configuration, the drag co-efficient is higher by 78% and lift co-efficient is higher by 85%. This plate type profile is ineffective when compared to the other aerofoil profiles tested earlier in this paper. Hence, it could be concluded that the best and the optimum configuration of spoiler for further trials on car is S1223 aerofoil with 10° angle of attack configuration.

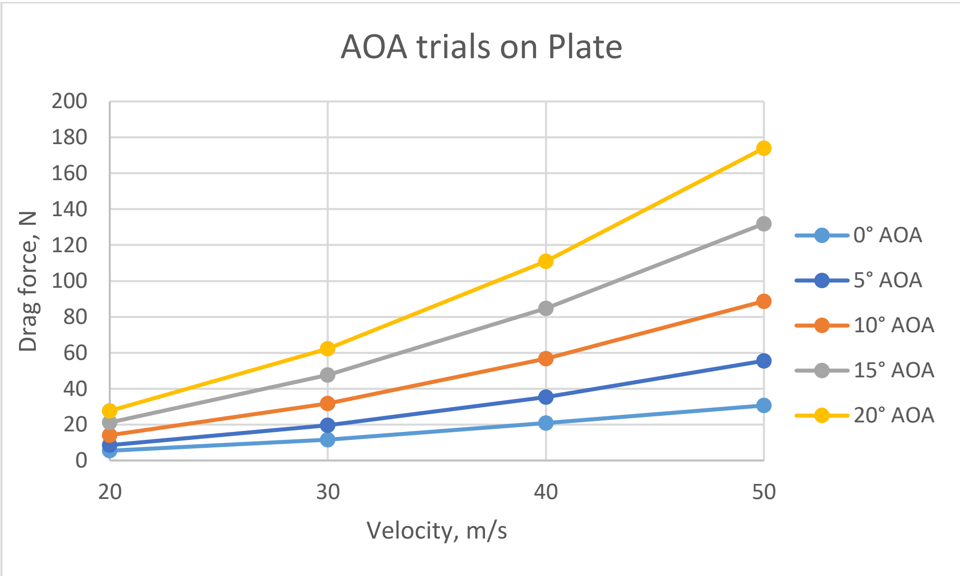

Fig. 48. Drag force for angle of attack trials on plate type profile

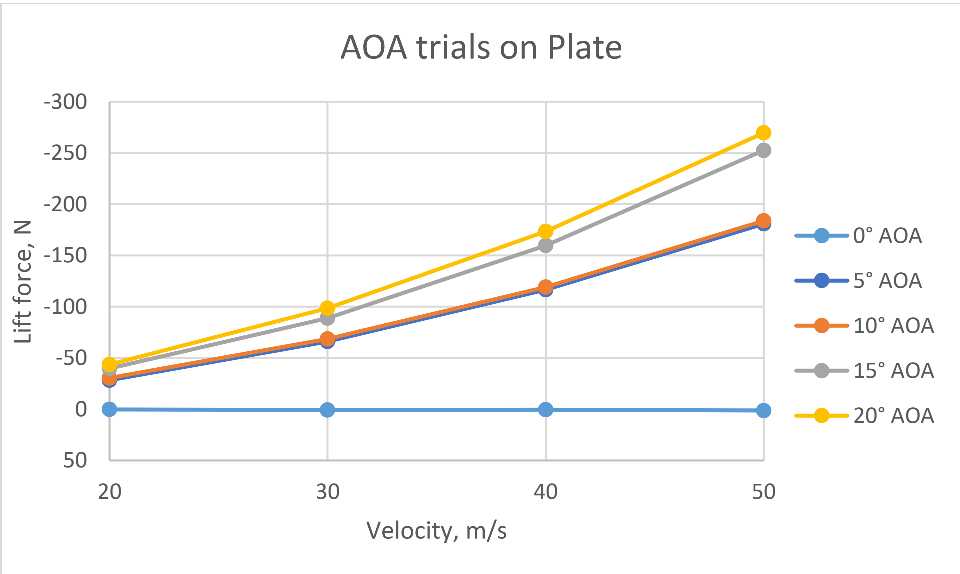

Fig. 49. Lift force for angle of attack trials on plate type profile

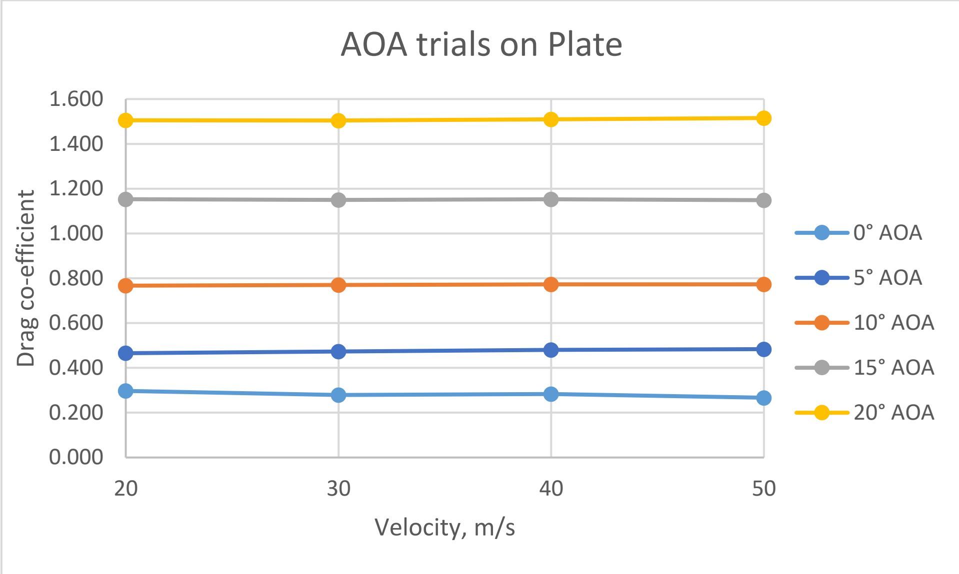

Fig. 50. Drag co-efficient for angle of attack trials on plate type profile

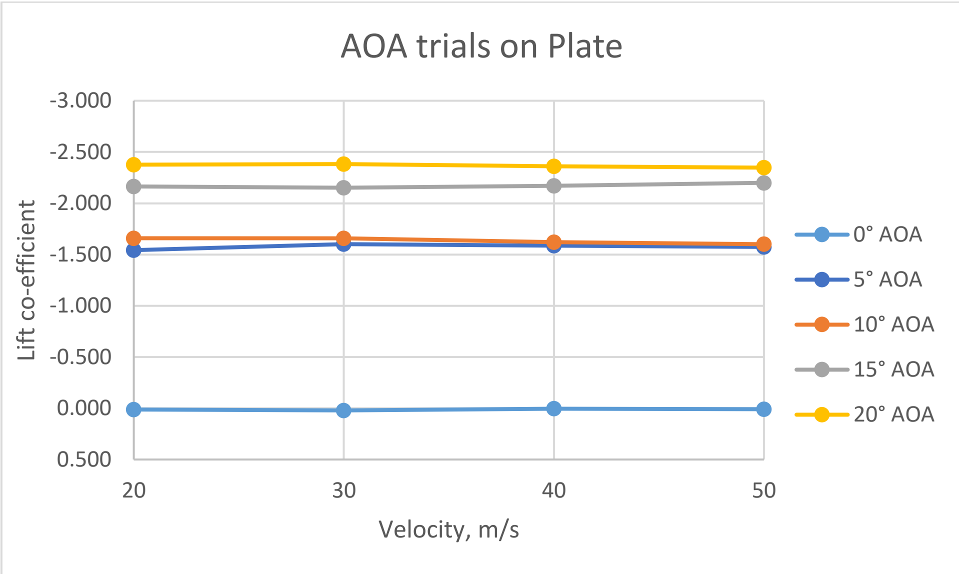

Fig. 51. Drag co-efficient for angle of attack trials on plate type profile

In order to investigate the aerodynamic effect of spoilers on the aerodynamics of the car, flow simulations were done on the car without any spoilers on different flow velocities starting from 20m/s to 50 m/s at every 10 m/s, this configuration is considered as case 1. The flow simulation results recorded for case 1 are tabulated (Table. 6). The maximum downforce acting on the car is -857.53 N and the maximum drag force acting on the car is 1056.63 N at 50 m/s flow velocity, for which the corresponding lift and drag coefficients are -0.245 and 0.301. This is to be considered as the base data from which the addition of spoilers will be compared with.

For case 2, rear spoiler was assembled onto the car and flow simulations were done at all the previously mentioned velocities. The rear spoiler was modelled based on the angle of attack trials done and discussed earlier. The best configuration was chosen from all the configurations which was tested, the chosen configuration is S1223 aerofoil with 10° angle of attack. This spoiler is assembled and flow simulation trials were carried out. Table.7 shows the results obtained from these simulations. It could be noted that the downforce increased after adding the spoiler at the same time the drag force also increased. This is an expected phenomenon after the literature review. The maximum drag force acting on the vehicle is 1124.59 N and the maximum downforce acting on the vehicle is 932.85 N at 50 m/s flow velocity. The drag and lift coefficients for the case 2 are 0.321 and -0.266 respectively. An increase of 6.4% in drag co-efficient and 8.8% in lift co-efficient can be observed on comparing case 2 with case 1. This is not the desired phenomenon but it is the expected phenomenon.

For case 3, front spoiler was assembled on the vehicle along with the rear spoiler in order to decrease the drag and lift coefficients. The front spoiler was also modelled with the same configuration as the rear spoiler. Flow simulation trials were conducted for case 3 at all the velocities mentioned in the earlier cases and the data are tabulated (Table. 8). From the data, the decrease in the drag force acting on the vehicle could be noted and at the same time the downforce acting on the vehicle has increased which is the desired phenomenon and the objective of this paper. The drag force achieved at 50 m/s flow velocity is 1044.48 N and the corresponding lift force is -967.24 N. This is highest downforce achieved in all the three cases. The corresponding drag and lift coefficients are 0.298 and -0.276.

On comparing the results of case 3 with case 1, there is 1.1% reduction in the drag co-efficient value with 12.8% increase in the lift co-efficient value, with this it could be stated that adding both front and rear spoilers to the vehicle will reduce the drag force acting on the vehicle and the increases the downforce which will help in the good traction and cornering ability of the vehicle. The added benefit of reduced drag force is the reduction in fuel consumption.

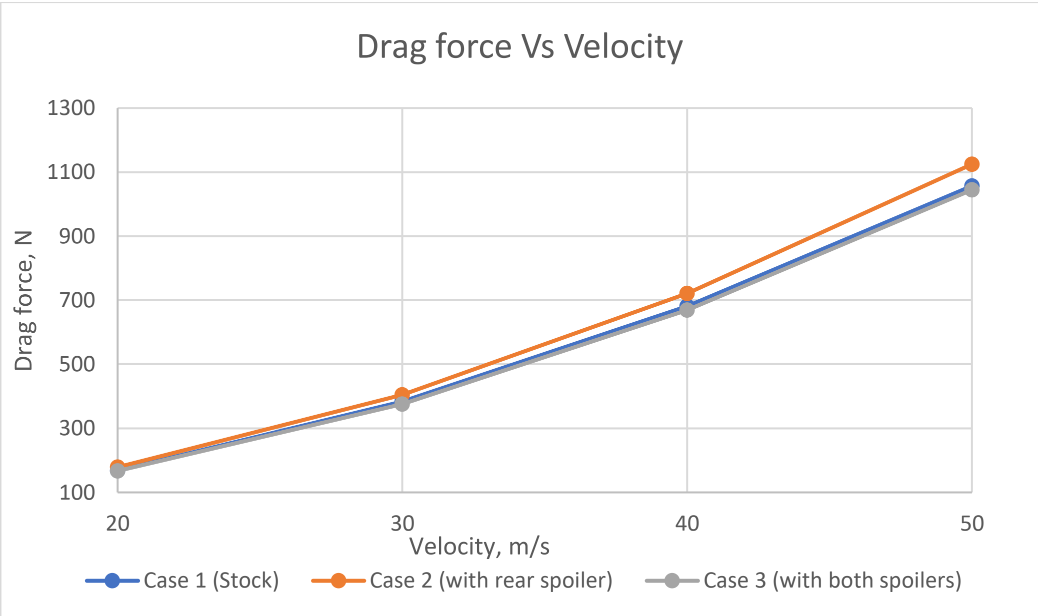

Fig. 52. Drag force for trials on car

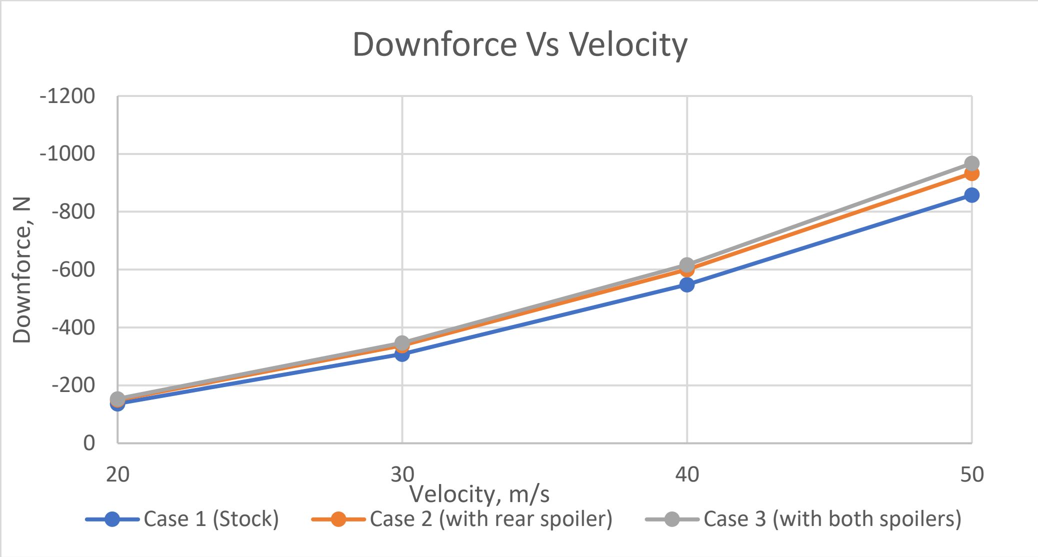

Fig. 53. Downforce for trials on car

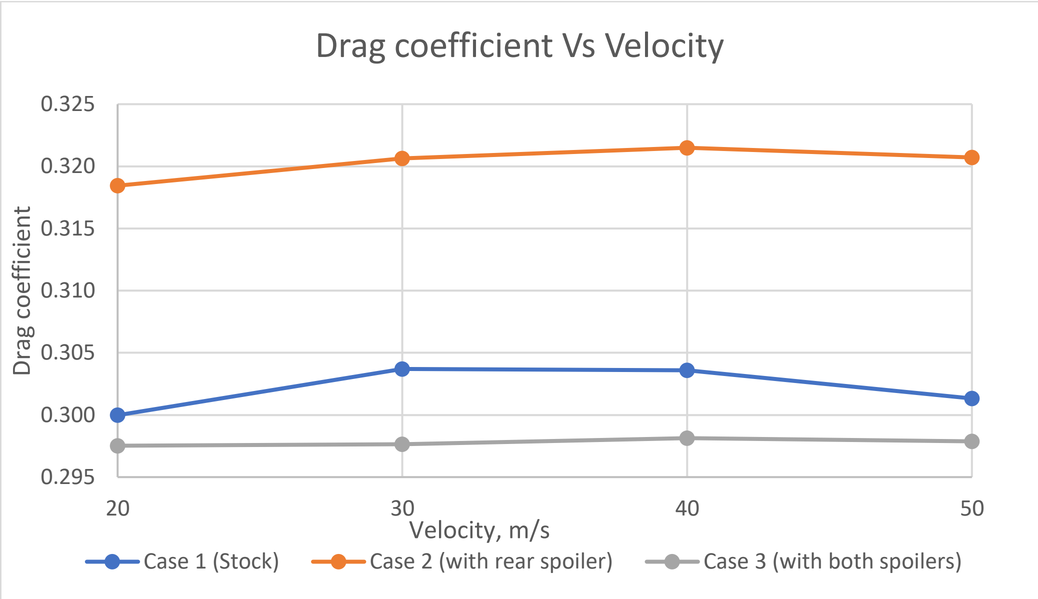

Fig. 54. Drag co-efficient for trials on car

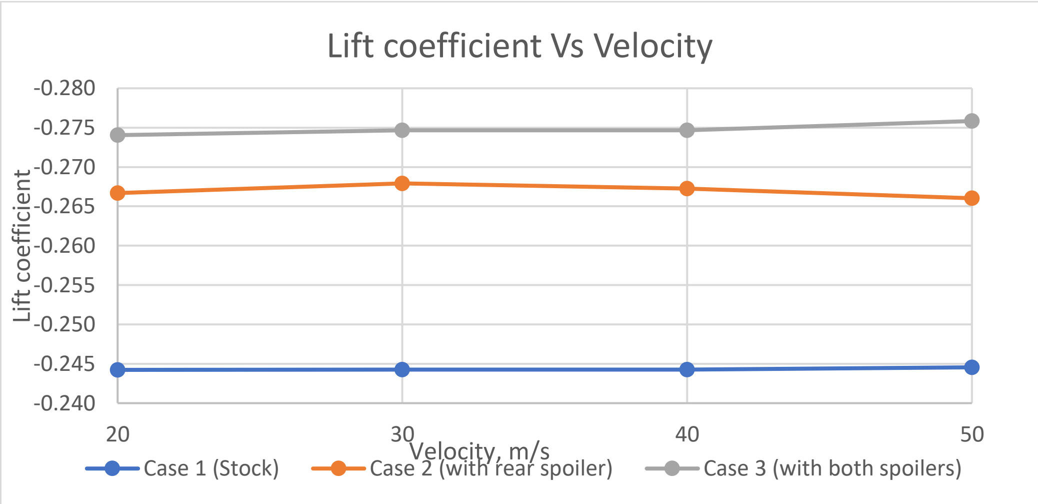

Fig. 55. Lift co-efficient for trials on car

The structural analysis for the spoilers were done with two materials; ABS plastic and carbon fibre, the data acquired from the testing with forces from 50m/s have been tabulated (Table. 9 and 10) in the previous chapter. The principle stresses (P1, P2 and P3) for front spoiler obtained from simulation were 36.42 N/mm^2, 6.423 N/mm^2 and 6.154 N/mm^2 with ABS plastic and 39.53 N/mm^2, 10.02 N/mm^2 and 9.403 N/mm^2 with carbon fibre respectively. The principle stress (P1, P2 and P3) induced in rear spoiler obtained acquired from the simulation were 13.43 N/mm^2, 4.95 N/mm^2 and 3.296 N/mm^2 with ABS plastic and 13.5 N/mm^2, 6.18 N/mm^2 and 5.871 N/mm^2 with carbon fibre respectively. It could be observed that the stresses induced in front spoilers were similar and did not exceed the allowable stress. Even in the case of rear spoiler the stress induced with ABS slightly higher than that of carbon fibre but the stresses did not exceed the allowable stress in both the cases. The displacement values got from the structural testing could be seen in Table. 9 and 10 for both the spoilers with ABS plastic and carbon fibre respectively. The maximum displacement of front spoiler with ABS plastic was 19.8 mm and with carbon fibre was 1.98 mm. The maximum displacement of rear spoiler with ABS plastic was 8.7 mm and with carbon fibre was 0.25 mm. It could be observed that the displacement values are very less with carbon fibre on comparing with ABS plastic in both the spoilers. The difference in the displacement with ABS plastic and carbon fibre is high but, the displacement value with ABS plastic is only 19.8 mm with the front which has the maximum displacement of the two spoilers.

The spoilers were then tested for worst case scenario with the forces obtained from the flow analysis at flow velocity of 70 m/s. The data could be seen in Table. 11 and 12. It could be observed that the principle stress values (P1, P2 and P3) acquired for front spoilers are 36.42 N/mm^2, 9.17 N/mm^2 and 8.79 N/mm^2 with ABS plastic and 56.41 N/mm^2, 14.28 N/mm^2 and 13.41 N/mm^2 with carbon fibre respectively. In the case of rear spoiler, the principle stress values obtained were 56.41 N/mm^2, 14.28 N/mm^2 and 13.41 N/mm^2. As it could be seen from the data that the stresses have not reached the allowable stress in both the cases. The displacement values for the front spoiler was 28.21 mm with ABS plastic and 1.98 mm with carbon fibre. For rear spoilers the displacement values 12.48 mm with ABS plastic and 0.36 mm with carbon fibre. The difference between the displacement values of ABS plastic and carbon fibre is high but the displacement value of ABS plastic is also very not high and the material has not reached its elastic limit.

The spoilers were then tested for the factor of safety and the data were data could be seen in Fig. 33 and 34. The minimum factor of safety for front spoiler at worst case scenario was 1.2 with ABS plastic and 13 with carbon fibre. The minimum factor of safety for rear spoiler was 1.7 with ABS plastic and 28 with carbon fibre. It could be stated that in both the cases the factor safety is over 1.2 which is suitable for automotive applications [50]. While considering the cost and the fabrication process involved with the carbon fibres, ABS plastic still remains the favourable material in this thesis and it could be stated that the best suited material for manufacturing of these spoilers is ABS plastic.

Conclusions

1. From the previous researches it could be seen that, spoiler is the only aerodynamic component which reduces drag and also increases the downforce which is the most desired aerodynamic characteristic in a vehicle. But adding a wing type spoiler in most of the cases seem to have increased the downforce drastically at the expense of aerodynamic drag which is an undesired phenomenon. Hence, a front spoiler could also be introduced in order to get a better aerodynamic characteristic in terms of drag and as far as the downforce is concerned introducing of the rear spoiler will increase the downforce helping to achieve the most desired aerodynamic characteristics in a vehicle. ABS plastic and carbon fibre are the most suited for manufacturing spoilers. In this project literature survey about the aerodynamics of the vehicle and the materials used to fabricate spoilers has been taken and completed. Various papers have been reviewed and necessary data have been taken which was very useful in understanding the concept of aerodynamics and gave a clear idea about the further proceedings of the project.
2. The optimum configuration for the spoiler models in order to provide good downforce and reduced drag was determined from the flow simulation trials which was conducted with various aerofoils and plate type profile at different angle of attacks. S1223 aerofoil at 10° angel of attack was chosen to be the optimum configuration. It has a drag co-efficient of 0.173 and lift co-efficient of -0.866 which on comparing with other aerofoils is better. Both front and rear spoilers were modelled with this configuration.
3. Flow simulation trials were first done on the car (Audi A7) without any spoilers to determine the base aerodynamic parameters from which it could be improved by adding the spoilers. With rear spoiler assembled on the car, the downforce increased by 8.8% but the drag also increased by 6.4% which is expected and it was also reviewed in the literature survey of this thesis. Hence, a front spoiler was also added and flow simulation trials were carried on with this setup. After adding both the spoilers the downforce increased further by 12.8% from the base value and the drag also reduced 1.1% which is the desired aerodynamic phenomenon. With this it could be concluded that the aim of this project is achieved.
4. With the forces derived from the flow analysis of the aerofoil, the static structural testing of the spoilers was carried out. Both ABS plastic and carbon fibre has not reached their allowable stress for both front and rear spoilers at worst case scenario. From the displacement results, it could be noted that carbon fibre has lesser displacement values than ABS plastic but even the displacement value of ABS plastic is very less; 28.21 mm and 12.48 mm for front and rear spoiler respectively. Also, the spoilers have minimum factor of safety value of 1.2 and 1.7 for front and rear spoiler respectively. The direct and indirect cost of the carbon fibre is very high compared to ABS plastic, the same was discussed earlier in the literature survey of this thesis. Considering all the parameters for the final spoiler’s strength and cost, it could be concluded that ABS plastic is the best suited material for the spoilers discussed in this thesis.

List of references

[1] S. Diamond, “Heavy vehicle systems optimization,” annual progress report for heavy vehicle systems optimization, Washington, DC, 2004.

[2] S.-Y. Cheng, K.-Y. Chin, S. Mansor and A. B. A. Rahman, “Experimental study of yaw angle effect on the aerodynamic characteristics,” Journal of Wind Engineering & Industrial Aerodynamics, vol. 184, pp. 305-312, 2019.

[3] “What Is The Function & Importance Of Rear Spoiler?,” 12 April 2019. [Online]. Available: https://carbiketech.com/rear-spoiler-design-function-importance/. [Accessed 10 May 2019].

[4] A. E. Minal Pandurang Marathe, “CFD Analysis of Rear Spoiler of Passenger Car,” vol. 5, no. 3, 2017.

[5] W.-H. Hucho and G. Sovran, “AERODYNAMICS OF ROAD VEHICLES,” vol. 25, no. 1, pp. 485-537, 1993.

[6] “Race car aerodynamics basics and design,” [Online]. Available: https://www.buildyourownracecar.com/race-car-aerodynamics-basics-and-design/2/. [Accessed 13 May 2019].

[7] “What is Drag?,” 5 May 2015. [Online]. Available: https://www.grc.nasa.gov/www/k-12/airplane/drag1.html. [Accessed 5 May 2019].

[8] Thiagarajan, “CAR AERODYNAMICS,” 25 July 2015. [Online]. Available: http://itsmyblogvbe.blogspot.com/2015/07/car-aerodynamics.html. [Accessed 5 May 2019].

[9] W.-H. Hucho, Aerodynamics of Road Vehicles, Butterworth-Heinemann, 1987.

[10] “What is Lift?,” 5 May 2015. [Online]. Available: https://www.grc.nasa.gov/www/k-12/airplane/lift1.html. [Accessed 5 May 2019].

[11] S. R. Ahmed, G. Ramm and G. Faltin, “Some Salient Features of the Time - Averaged Ground Vehicle Wake,” vol. 93, pp. 473-503, 1984.

[12] W. Hanfeng, Z. Yu, Z. Chao and H. Xuhui, “Aerodynamic drag reduction of an Ahmed body based on deflectors,” vol. 148, pp. 34-44, 2016.

[13] G. Fourrié, L. Keirsbulck, L. Labraga and P. Gilliéron, “Bluff-body drag reduction using a deflector,” vol. 50, no. 2, pp. 385-395, 2011.

[14] H. Chowdhury, B. Loganathan, I. Mustary, H. Moria and F. Alam, “Effect of various deflectors on drag reduction for trucks,” vol. 110, pp. 561-566, 2017.

[15] J.-L. Aider, J.-F. Beaudoin and J. E. Wesfreid, “Drag and lift reduction of a 3D bluff-body using active vortex generators,” vol. 48, no. 5, pp. 771-789, 2010.

[16] G. Pujals, S. Depardon and C. Cossu, “Drag reduction of a 3D bluff body using coherent streamwise streaks,” vol. 49, no. 5, pp. 1085-1094, 2010.

[17] C.-H. Tsai, L.-M. Fu, C.-H. Tai, Y.-L. Huang and J.-C. Leong, “Computational aero-acoustic analysis of a passenger car with a rear spoiler,” Applied Mathematical Modelling, vol. 33, pp. 3661-3673, 2009.

[18] R. Bansal and R. B. Sharma, “Drag Reduction of Passenger Car Using Add-On Devices,” Journal of Aerodynamics, vol. 2014, 2014.

[19] A. R. Paul, A. Jain and a. F. Alam, “DRAG REDUCTION OF A PASSENGER CAR USING FLOW CONTROL TECHNIQUES,” International Journal of Automotive Technology, vol. 20, no. 2, pp. 397-410, 2019.

[20] X.-x. Hu and E. T. Wong, “A Numerical Study On Rear-spoiler Of Passenger Vehicle,” International Journal of Mechanical and Mechatronics Engineering, vol. 57, no. 9, 2011.

[21] M. Bambhania, D. Jani and S. Patel, “CFD analysis of Roof Spoiler used in Light Duty Commercial Vehicle,” Surat, Gujarat, May 28, 2016.

[22] H. Sadettin Hamut, R. Salah El-Emam, M. Aydin and I. Dincer, “Effects of rear spoilers on ground vehicle aerodynamic drag,” vol. 24, no. 3, pp. 624-642, 2014.

[23] K. Thummar, N. Shah, V. Vaghrecha and S. Sarkar, “Variation in Aerodynamic Drag and Lift by addition of a Rear Spoiler in a Passenger Car,” Journal of Emerging Technologies and Innovative Research (JETIR), vol. 6, no. 4, 2019.

[24] R. C. Das and M. Riyad, “CFD Analysis of Passenger Vehicle at Various Angle of Rear End,” vol. 194, pp. 160-165, 2017.

[25] S. M. R. Tousi, P. Bayat and P. Bayat, “Evaluating the Importance of Rear Spoiler on Energy Efficiency of Electric Vehicles,” International Journal of Automotive Engineering, vol. 5, no. 4, 2015.

[26] P. Auto, “Aerodynamic 101- How airflow will affect your car,” 26 May 2015. [Online]. Available: http://www.pu-autotech.com/aerodynamic-101/. [Accessed 14 May 2019].

[27] Engineering_Explained, “Engineering Explained: 10 Aerodynamic Features Of Race Cars,” 2015. [Online]. Available: https://www.carthrottle.com/post/engineering-explained-10-aerodynamic-features-of-race-cars/. [Accessed 14 May 2019].

[28] “Benefits of having a car spoiler,” 19 September 2016. [Online]. Available: http://blog.rcmart.com/rc-benefits-of-having-a-car-spoiler-2/. [Accessed 14 May 2019].

[29] S. Ghulaxe, “Benefits of rear spoiler in cars,” 12 October 2014. [Online]. Available: https://physics.stackexchange.com/q/140983. [Accessed 19 May 2019].

[30] O. Zikanov, Essential Computational Fluid Dynamics, Hoboken, New Jersey: John Wiley & Sons, INC., 2010.

[31] M. Cakir, “CFD study on aerodynamic effects of a rear wing/ spoiler on a passenger vehicle,” no. 1, 2012.

[32] A. Sayma, Computational Fluid Dynamics, Ventus Publishing ApS, 2009.

[33] J. Katz, “Aerodynamics of race cars,” Annu. Rev. Fluid Mech, vol. 38, pp. 27-63.

[34] V. N. Kumar, K. L. Narayan and L. N. V. N. R. a. Y. S. Ram, “Investigation of Drag and Lift Forces over the Profile of Car with Rear spoiler using CFD,” International Journal of Advances in Scientific Research, vol. 1, no. 8, pp. 331-339, 2015.

[35] J. Leary, “Computational Fluid Dynamics Analysis of a Low Cost Wind Turbine,” 2010.

[36] T. Hughes, “CFD Study of Flow over a Simplified Car (Ahmed Body) Using Different Turbulence Models,” 2018.

[37] V. Johnston, “4 Essential Things to Know About Your Car’s Spoiler,” 15 December 2015. [Online]. Available: https://www.yourmechanic.com/article/4-essential-things-to-know-about-your-car-s-spoiler. [Accessed 15 May 2019].

[38] ADMIN, “ABS plastic properties,” 20 December 2018. [Online]. Available: https://www.adrecoplastics.co.uk/abs-plastic-properties-and-application/. [Accessed 9 June 2019].

[39] “Acrylonitrile Butadiene Styrene (ABS) and its Features,” [Online]. Available: https://omnexus.specialchem.com/selection-guide/acrylonitrile-butadiene-styrene-abs-plastic. [Accessed 9 June 2019].

[40] T. Rogers, “Everything You Need to Know About ABS Plastic,” 13 July 2015. [Online]. Available: https://www.creativemechanisms.com/blog/everything-you-need-to-know-about-abs-plastic. [Accessed 6 June 2019].

[41] E. Ghassemieh, “Materials in Automotive Application, State of the Art and Prospects,” in New Trends and Developments in Automotive Industry, IntechOpen, 2011.

[42] “Fiberglass – Types, Properties, and Applications Across Industries,” 7 April 2019. [Online]. Available: https://www.phelpsgaskets.com/blog/fiberglass--types-properties-and-applications-across-industries. [Accessed 6 June 2019].

[43] P. Bhatt and A. Goel, “Carbon Fibres: Production, Properties and Potential Use,” Material Science Research India, vol. 14, no. 1, pp. 52-57, 2017.

[44] “Mechanical Properties of Carbon Fibre Composite Materials, Fibre / Epoxy resin (120°C Cure),” Performance Composites Limited, [Online]. Available: http://www.performance-composites.com/carbonfibre/carbonfibre.asp. [Accessed 06 20 2019].

[45] A. Sobachkin, G. Sobachkin and A. Dumnov, “Numerical Basis of CAD-Embedded CFD,” in NAFEMS World Congress 2013, Salzburg, 2015.

[46] “S9026 (9.5%) (s9026-il),” Airfoil Tools, 2020. [Online]. Available: http://airfoiltools.com/airfoil/details?airfoil=s9026-il. [Accessed March 2020].

[47] “CLARK Y AIRFOIL (clarky-il),” Airfoil Tools, 2020. [Online]. Available: http://airfoiltools.com/airfoil/details?airfoil=clarky-il. [Accessed March 2020].

[48] “S1223 (s1223-il),” 2020. [Online]. Available: http://airfoiltools.com/airfoil/details?airfoil=s1223-il. [Accessed March 2020].

[49] J. Fekete and H. J.N, “Design of auto body: Materials perspective,” in Automotive Steels: Design, Metallurgy, Processing and Applications, Woodhead Publishing, 2017, pp. 1-18.

[50] "Factor of Safety in Design and Engineering: The Basics," 11 October 2016. [Online]. Available: https://www.creativemechanisms.com/blog/factor-of-safety-in-design-and-engineering-the-basics. [Accessed 15 May 2020].