



CFD Analysis on Rear and Front Wings of FSAE Vehicle

¹Patri Dhaneswar, ²Kankatala Sandeep Narayana, ³Panuganti Brahmateja, ⁴Kondapalli Sravanth Kumar, ⁵Bhukya Dheeraj

¹Mechanical engineer, ²Mechanical engineer, ³Mechanical engineer, ⁴Mechanical engineer, ⁵Mechanical engineer

¹Department of Mechanical Engineering,

¹Lovely Professional university, Punjab, India

Abstract : In this paper, study have been conducted on aerodynamic loads generated on FSAE car. As car moves in high speeds it encounter huge amount of force opposite to its moving direction which is called Drag. Wings are device used on the car for improving their aerodynamic performance. The main purpose of this paper is served by comparing different types of airfoils, analyzing various angle of attack of wings of the car, and integrating necessary changes to it to achieve the desired aerodynamic performance. The wings have designed based on the rule book of Formula Bharath 2022. The comparison between the three different airfoil NACA4412, NACA4415, S1210 geometries resulted to use NACA4415 for the design of wings. A 2D numerical analysis has been performed using ANSYS for the performance of the wings at various angles ranging from 0° to 20°. K-epsilon (k-ε) realizable turbulence model is used to simulate in numerical method with pressure-based solver. It indicated that the wings made it possible to generate favorable aerodynamic forces at angle of attack 12° of NACA4415 airfoil for the design of the wings. Simulation of the wings have been carried out with four different configurations to compare the aerodynamic forces. This study is mainly focused to find out optimized shape of car wings. Carried out with outputs of lift force, lift coefficient, drag force, drag coefficient.

Index Terms - Aerodynamics, Front and Rear wings, Angle of Attack, Drag, Lift.

I. INTRODUCTION

Aerodynamics is the study of how air interact with moving bodies. When cars move through air there are some aerodynamic forces generated by the relative motion between air and surfaces of the car. Airfoil is the shape of an object which help to generate the aerodynamic forces. Wings, spoilers, scoops, side deflectors are objects which are used on the cars for improving their aerodynamic performance by generating the significant aerodynamic forces. Aerodynamics should be considered during the design of cars to improve fuel efficiency, high speeds, more stability and handling at high speeds, noise level, aesthetic, and appearance of car. In this study wings are used on the car for improving their aerodynamic performance.

Aerodynamic Forces: Car moving through air distracts the original path of air and in such deflections lead to changes in the speed of the air. Bernoulli's Principle shows that the pressure exerted by the air on the wing is altered from that of the undisturbed stream. As a result of these processes, the car experiences a resultant aerodynamic force. There are mainly four Aerodynamic forces acting on a car they are thrust, drag, lift, and weight. Thrust is the forward force produced along the direction in which the car is moving. Thrust pushes or pulls the car forwards which helps the car to move forwards. Weight is the body force that acts vertically downwards along with the gravity. It is produced due to the mass of the object and acceleration due to gravity. In this paper it is mainly discussed about drag and lift.

Types of Aerodynamic Forces

1.1 Drag Force

Drag force is a force acting opposite to the direction in which the car is moving, or it can also define as the force opposite to thrust. It is generated by the difference in velocity between the surface of the car and the air. Drag force is important on the performance of the car as it affect directly on the fuel economy, top speed, and time to accelerate the car. To quantify the drag a dimensionless quantity drag coefficient is used. Lesser the drag coefficient less will be the drag. Drag force on any car is proportional to the density of the fluid and proportional to the square of speed between the car and the air. Drag coefficient also varies as a function of flow speed, shape of the object, object size, fluid density, flow direction, object position and fluid viscosity.

Drag is represented as the following equation:

$$F_D = \frac{1}{2} \rho V^2 C_D A$$

From the drag equation, it can be observed that the drag force is in proportion to the square of the speed between the car and the air. This indicates that the resistance due to air increases exponentially as the speed of the body increases. When a car is moving a low-pressure zone is generated at the back of the car. This low-pressure zone tries to pull the car backwards and

tries to reduce the speed and creates pressure drag which leads to high consumption of fuel. To reduce this under body diffuser can be used at the rear end of the car.

1.2 Lift Force

Lift Force is a force acting upwards on a car. It is the force that opposes the weight of the car. Lift is the component of this force that is perpendicular to the oncoming flow direction. Lift can be in any direction with respect to gravity as it is defined with respect to the direction of flow rather than to the direction of gravity. In cars inverted airfoil are used as they must remain contact with the ground rather than being in air like airplane. So here in this case the lift act as a downforce on the wings of the cars. Downforce is the downwards force created by the wings of the cars. The main reason for the downforce is to create more grip on the tires and let the car move faster by increasing the vertical force on the tires. Lift is affected by the size and shape of the wing, angle of attack, air density and same as that of drag coefficient, lift coefficient is a dimensionless quantity which quantify the lift force. Lift force on any car is proportional to the density of the fluid and proportional to the square of speed between the car and the air. Newtons third law of motion states that for every action of force there is an equal and opposite reaction. According to newtons third law of motion lift is the reaction force which acts upwards to the weight of the body as the airfoil pushes the air flow downwards.

Lift is represented as the following equation:

$$F_L = \frac{1}{2} \rho V^2 C_L A$$

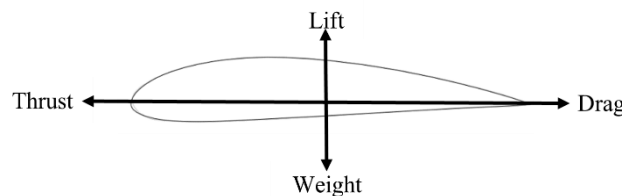


Figure I. Aerodynamics Forces

II. COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics (CFD) is an analysis of fluid flows which uses numerical solution methods and algorithms to solve and analyze problems. In this paper the CFD analysis is carried out using the Ansys 2021 R2 Fluid Flow (Fluent). It uses finite volume method to solve the differential equations with Reynolds Averaged Navier Stokes equations (RANS) in turbulence models for the simulations. The Reynolds Averaged Navier Stokes equations is used in turbulence model to find out the effect of turbulence fluctuation on the flow of the fluid. Reynolds Averaged Navier Stokes equations shows a lower computational cost than Large Eddy Simulations (LES), therefore Reynolds Averaged Navier Stokes equations is widely used in research. Large eddy simulation (LES) is a mathematical model for turbulence used in computational fluid dynamics which simulates large eddies and models small eddies contrast to RANS models which goes with all eddies.

Turbulence models in CFD are used to include the effect of turbulence in the simulation of fluid flows. Turbulent flow causes much more drag than laminar flow. K-epsilon (K-ε) model is one of the of turbulence models belongs to the RANS equations where all the effects of turbulence are modeled. All the simulations in this research were simulated using k-epsilon (K-ε) model. K-ε realizable model mainly provides improved predictions for the spreading rate of both planar and round jets with a superior ability to capture the mean flow of complex structures.

Pressure-based solver allows to solve flow problem in two ways they are segregated or coupled manner. There are four algorithm or a process or set of rules to be followed in calculations of problem-solving operations in pressure-based solver. The coupled approach offers some advantages over the other algorithm, it results a robust and efficient single-phase implementation for steady state flows with superior performance. It indicates that the methods scale generally very well and is necessary when the quality of the mesh is poor or if large time steps are used.

The gradient discretization Least Squares Cell Based or Green Gauss Node Based has similar accuracy between them and Least Squares Cell Based is less expensive to compute. Green Gauss Cell Based is the cheapest but it is least accurate. The discretization of the pressure, momentum, turbulent kinetic energy, and turbulent dissipation energy is best performed through second-order methods.

III. SELECTION OF AIRFOIL

Airfoil is a cross sectional shape used in many aerodynamic devices such as airplane wing, rotor blades, propeller blades. Generally, in airfoil the pressure above the wing is lower than the pressure below the wing. As the pressure below the airfoil is higher than the pressure above the airfoil, there is a net force acting upwards. The airfoil follows Bernoulli principle, it states that a velocity is inversely proportional to pressure. So, the higher the velocity of the flow the lower the pressure. Air flowing over an airfoil has more velocity which causes decrease in pressure. According to Bernoulli's principal aircraft can achieve upward force because of the shape of its wings. Airfoil are shaped in such a way that that air flows faster on the top of the airfoil and slower at bottom of the airfoil.

The airfoils sections used for wings in this study are S1210, NACA4412 and NACA4415. Research shows that S1210 and NACA4412 are one of the most used airfoils for spoilers to improve the aerodynamic efficiency of race cars. These three airfoils are compared to have a better knowledge about the aerodynamic performance. CFD is carried out in Ansys fluid flow fluent. The geometry is created in design modular by taking the reference of airfoil coordinate and an enclosure is created for the flow of air. Drag and lift forces of airfoils. To get an accurate result in simulation we required an efficient and fine mesh. To achieve that the mesh element size is given very small, and refinement has been applied. The finest mesh is obtained as follow:

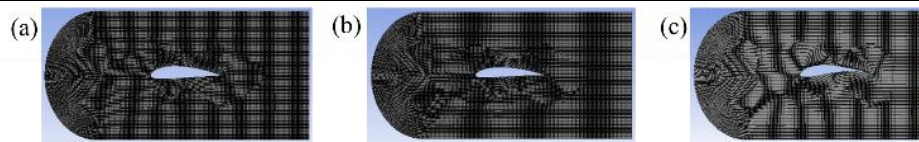


Figure II. Meshing (a) NACA4412 (b)NACA4415 (c)S1210

The inlet is considered as velocity inlet with air flow velocity of 35m/s (126KMPH) and the outlet is considered as pressure outlet. The aerodynamic characteristics of three airfoil NACA4412, NACA4415, S1210 is investigated at angle of attack 0° and to visualize the result contour is used and drag, lift forces output has been calculated [1].

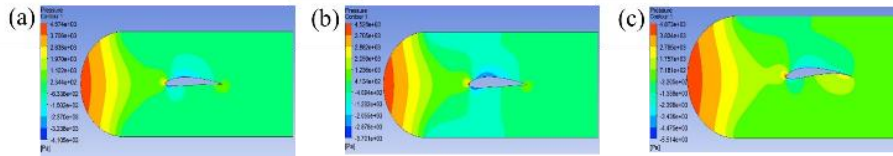


Figure III. Pressure Distribution (a)NACA4412 (b)NACA4415 (c)S1210

In the above figure it shows the pressure distribution of all the three airfoils. The red region shows high pressure region and blue region shows low pressure region. In the figure it can be observed that the pressure is less at the top surface than that of bottom surface which is due the shape of airfoil and Bernoulli's principle.

The most common quantity to measure the aerodynamic efficiency is lift to drag ratio. Airfoil efficiency is required to achieve an optimum aerodynamic performance. Lift to drag ratio can be determined by dividing the lift force to the drag force or by dividing lift coefficient to the drag coefficient. Airfoil with more lift to drag ratio are more efficient than that of less lift to drag ratio.

TABLE I. Drag and Lift of NACA4412, NACA4415, S1210

Air foil	Lift (N)	Lift Coefficient	Drag (N)	Drag Coefficient	Ratios	
					Lift/ Drag	Lift Coeff/ Drag Coeff
NACA 4412	791.34	1.05	96.52	0.12	8.19	8.19
NACA 4415	733.11	0.97	97.70	0.13	7.50	7.50
S1210	2279.75	3.03	274.3	0.36	8.31	8.31

From Table I it is observed that S1210 is more efficient airfoil, but it has very high lift forces which does not required for a FSAE vehicle. For optimization NACA4415 has been chosen for the research to continue as it has less but enough lift and drag forces compared to S1210. NACA4412 can also be chosen but it is the commonly used airfoil and there is not a much difference between the drag and lift forces values of NACA4415 and NACA4412. Therefore, by considering all the factors the most efficient airfoil to design the wings of the FSAE car is chosen as NACA4415 airfoil.

IV. ANGLE OF ATTACK

Airfoils are more effective lifting shapes. As compared to flat plate with same area, airfoil can generate more lift. Lift and drag are two major factors on which the airfoil efficiency is based on. Lift to drag ratio shows the amount of lift is generated by the car. Lift to drag ratio is the most common quantity to measure the aerodynamic efficiency. Airfoil efficiency is required to achieve an optimum aerodynamic performance. Lift to drag ratio can be determined by dividing the lift coefficient to the drag coefficient. Airfoil with more lift to drag ratio are more efficient than that of less lift to drag ratio.

Angle of attack is also one of the factors to get the optimum lift to drag ratio. For an airfoil the lift to drag ratio increases to maximum when the angle of attack is increased after certain value of angle of attack the lift to drag ratio starts decreasing until the stalling angle is reached. The angle of attack at which we obtain the best lift to drag ratio is called the most efficient angle of attack. When the airfoil is passing through the air, the angle at which the airfoil is passing is called angle of attack or it can also be defined as the angle between the longitudinal axis and the chord line of the airfoil.

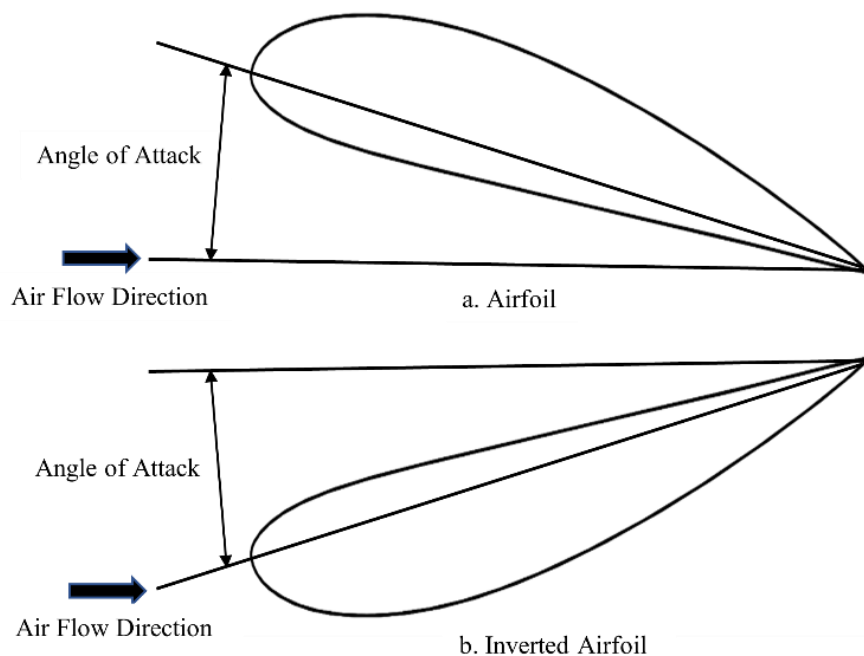
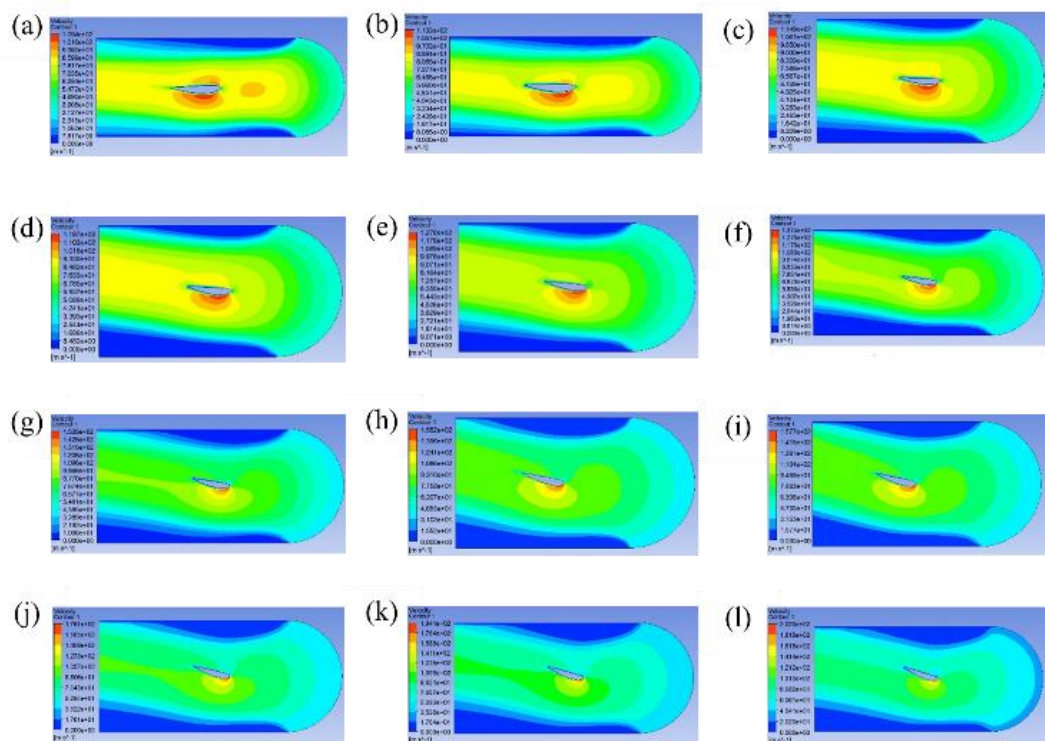


Figure IV. Angle of Attack

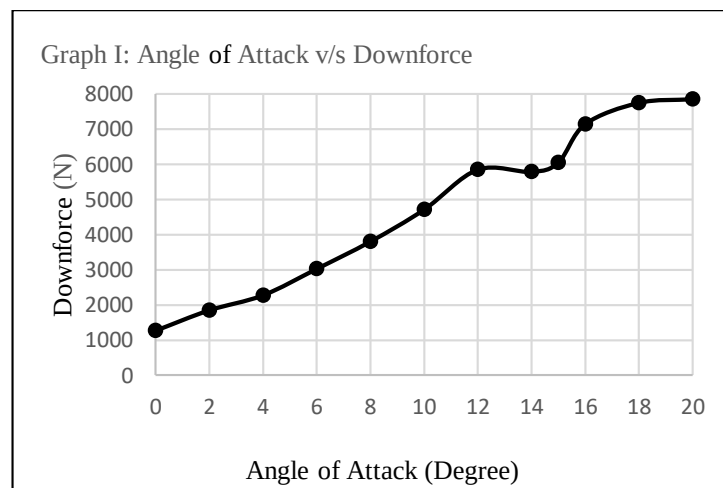
In this research simulation of the airfoil with a different angle of attack has been conducted to get an accurate angle of attack for the wing to design. The NACA4415 airfoil is analyzed with different angle of attack ranging from 0° to 20° . All the simulation is carried out in Ansys fluid flow fluent. The airfoil is an inverted airfoil or can be said as upside down airfoil. The main purpose of this simulation is to find out how the angle of attack of an airfoil impacts the aerodynamic properties of the wing as downforce, drag force.

Figure V. Velocity Distribution at Different angle of attack (a) 0° (b) 2° (c) 4° (d) 6° (e) 8° (f) 10° (g) 12° (h) 14° (i) 15° (j) 16° (k) 18° (l) 20°

The above figure indicates the velocity distribution of NACA4415 airfoil at different angle of attack. It shows the velocity visualization of the flow around the airfoil. The red region shows high velocity region and blue region shows low pressure region. It can be observed that the velocity is less at the top surface than that of bottom surface and the velocity decreases at the top of the surface as the angle of attack increases. It can be observed that top surface of the airfoil has less velocity than that of bottom surface which cause the airfoil to generate the negative lift which is called as downforce. As the velocity decreases, it causes in the change of downforce.

TABLE II. Drag and downforce of NACA4415 at different angle of attack

Angle of Attack (Degree)	Drag (N)	Drag Coefficient	Downforce (N)	Lift to Drag Ratio
0	39.549099	0.052710169	1270.0246	32.11260514
2	71.858065	0.095770849	1849.7852	25.74220722
4	216.93127	0.28912122	2275.294	10.48854782
6	282.89561	0.37703705	3024.9963	10.69297717
8	368.5093	0.4911411	3801.6136	10.31619446
10	438.22463	0.58405615	4709.0274	10.74569314
12	433.37733	0.57759578	5845.9696	13.48932949
14	610.77645	0.81402941	5782.203	9.466971099
15	773.83383	1.0313487	6048.1003	7.81576104
16	617.28284	0.822701	7137.8914	11.56340487
18	724.86422	0.96608309	7742.8689	10.8181969
20	816.38971	1.0880662	7846.9518	9.611772054



V. DESIGN

It is required to know all the rules and steps which are being involved in designing the FSAE car. To know about all the rules and regulation it is always required to go through rule books. In this research formula Bharat 2022 rule book is chosen to design the car. By keeping all the rules and constraints in mind the car is designed in SolidWorks. The dimension of the CAD model of the vehicle is shown in the figure VI.

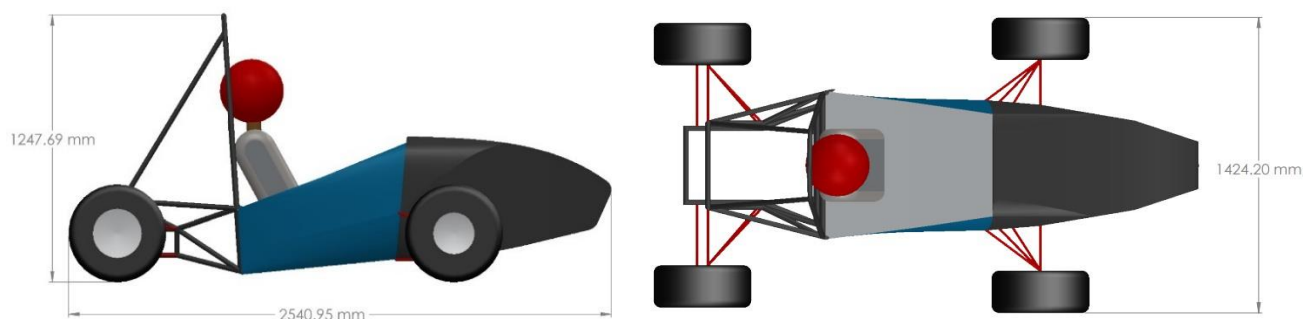


Figure VI. Dimension of FSAE car

Rear wing is an aerodynamic device which deflect the drag and turbulence movement which is occurred due to flow of air across the car. Rear wing is attached on the rear most part of Formula SAE car. Rear wing also generates downforce. It keeps the moving car stick to the ground, The track width of the car is 1424.2 mm. The wheelbase of the car is 1575. The ground clearance of the car without the wings is 38 mm or 1.5 inch. According to the rule book of Formula Bharat the minimum static ground clearance of any portion of the vehicle other than the tires including a driver must be 30 mm.

Front wing is an aerodynamic device which placed to the frontmost part of a race car. It consists of flaps, endplates, and negative attack wing. It generates most downforce in overall downforce of a car in the front part of the car with high aerodynamic efficiency. It reduces the drag generated in the front wheels and deflect the airflow to the rear components. It possesses a very high lift to drag ratio [4]. therefore, the grip between the rear tires and ground increases. It helps to have stability at high speeds, improves the cornering performance.

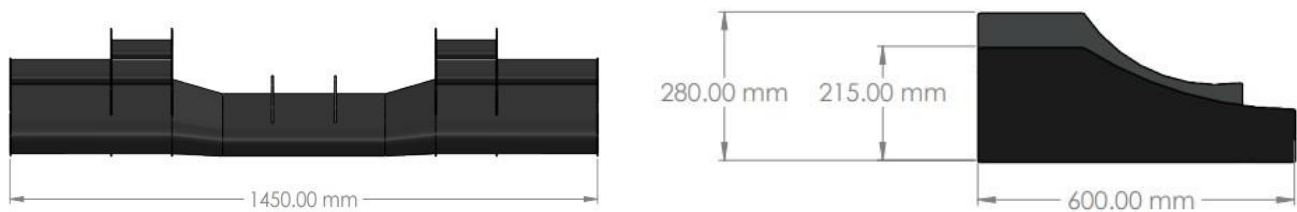


Figure VII. CAD model of Front Wing

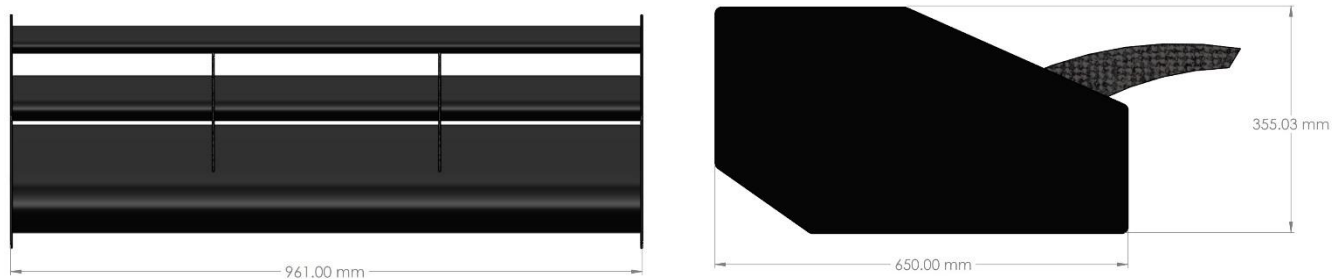


Figure VIII. CAD Model of Rear Wing

VI. ANALYSIS

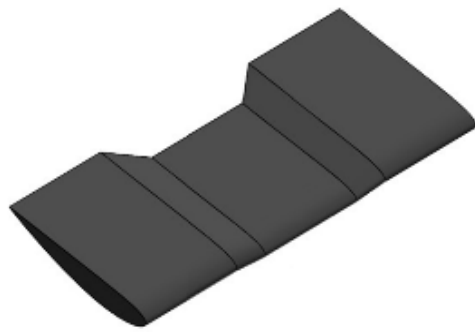
In this paper simulation of the front wing and rear wing is carried out to compare the drag and downforce to get efficient aerodynamic performance. The simulation in this research is performed with a hybrid mesh. Hybrid mesh is used as it is suitable and offers the highest accuracy cost ratio and default mesh sizes was assigned in mesh element size to control volume and number of elements. To do the simulation an enclosure is assigned to all the models in Ansys design modular. The enclosure acts as a tool used to enclose the bodies of a model so that the fluid enclosing the bodies can be assigned. The enclosure is created by sketching the required shape and extrude it with the required conditions. The enclosure should close all the side so that it allows the fluid to pass all over the car.

To simulate the wings, it is divided in to four different configurations. The first configuration is one airfoil without endplates, second is one airfoil element with endplates on both the sides, next comes with all the three airfoil elements and the last configuration is all the three elements with endplates. This configuration is done to compare the performance of each element present in the wing. The one element configuration will be considered as reference values for further simulations.

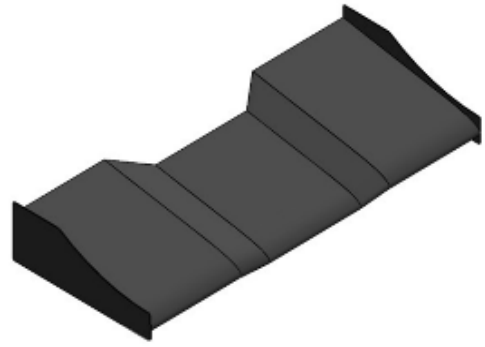
TABLE III. Boundary Conditions of CFD analysis

Boundary Conditions	Models	Inputs
Fluid Properties	Fluid Type	Air
	Density	1.225 kg/m ³
	Viscosity	1.7894 x 10 ⁻⁵ kg/(ms)
Inlet	Inlet type	Velocity Inlet
	Magnitude	35 m/s
	Direction	Normal to Boundary
	Turbulence Specification Method	Intensity and Viscosity ratio
	Turbulence Intensity	5%
Outlet	Turbulent Viscosity Ratio	10
	Type	Pressure Outlet
	Gauge pressure magnitude	0 Pascal
	Gauge pressure direction	Normal to boundary
	Turbulence specification method	Intensity and viscosity ratio
Walls	Back flow turbulence intensity	5%
	Back flow turbulence viscosity ratio	10
No Slip		

Table III shows the boundary conditions for the simulation of wings. Fluid is selected as air with the density 1.225 kg/m^3 and viscosity as $1.7894 \times 10^{-5} \text{ kg/(ms)}$. Inlet is assigned as velocity inlet of magnitude 35 m/s with direction normal to the boundary. Pressure is selected as pressure outlet with a zero-pressure gradient.



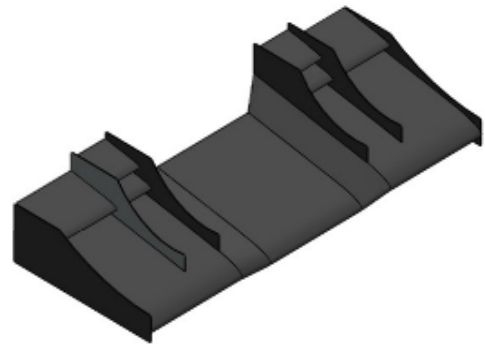
(a) One Element without Endplates



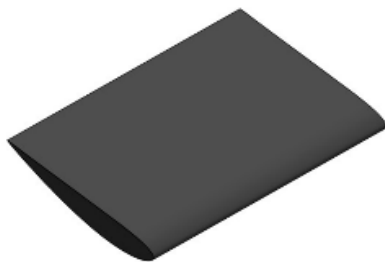
(b) One Element with Endplates



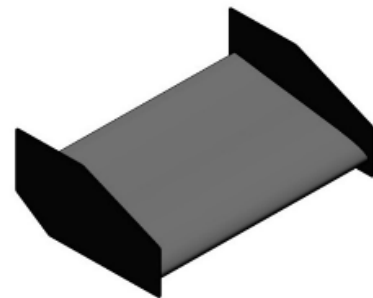
(c) Three Element without Endplates



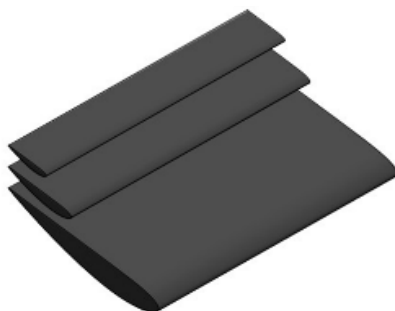
(d) Three Element with Endplates



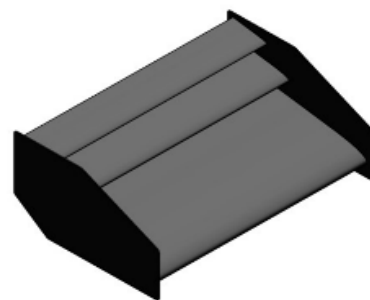
(a) One Element without Endplate



(b) One Element with Endplates



(c) Three Element without Endplate



(d) Three Element with Endplate

Figure IX. Configuration of Front and Rear Wings

TABLE IV. Drag and Downforce of wings with Different Configurations

Configuration	Front Wing			Rear Wing		
	Drag (N)	C _D	Downforce (N)	Drag (N)	C _D	Downforce (N)
a	153.68	0.20	1626.05	346.25	0.46	1710.24
b	161.71	0.21	1663.99	395.52	0.52	2068.35
c	205.91	0.27	1913.25	481.82	0.64	2166.23
d	218.09	0.29	1972.19	546.77	0.72	2489.20

From the TABLE IV it can be observed that in each configuration the drag and downforce both increases with compared to previous configurations. It states that the car can be more stable with the rear wing having three elements than that of wings with one element. According to the results obtained by CFD analysis of wings with different configuration states that 12° is the optimal angle of attack for the wing with three elements having the endplates gives more stability and aerodynamic performance.

VII. RESULTS AND DISCUSSION

To validate the computational results the numerical results for the NACA4412, NACA4415, S1210 air foils are validated. After validating the computational results NACA4415 has simulated with different angle of attacks (0, 2, 4, 6, 10, 12, 14, 15, 16, 18, 20) and presented angle of attack versus downforce to get the results for optimum angle of attack. The angle of attack is selected as 12° and the front wing and rear wing is designed according to the rule book of Formula Bharat 2022. Wings have used three air foil element each at 12° angle of attack with endplates on both sides. To simulate the wings, it has been divided in four different configurations. The computational results of the four configurations are compared in Table IV. It shows that the three elements with endplate have approximately 45% more downforce generated than that of one element without endplate for rear wing and 21% for the front wing.

VIII. CONCLUSION

A numerical analysis was performed to simulate the flow around three different airfoils NACA4412, NACA4415, and S1210. The numerical simulation was done using ANSYS Fluid Flow Fluent which uses Reynolds Averaged Navier Stokes equation. Computational results were validated by comparing the results with each other. Comparison showed that NACA4415 is efficient airfoil to use for the design of the wings. Different angle of attacks was simulated within the range of 0° to 20°. From the simulation of wings concluded that 12° angle of attack with three airfoil elements results high aerodynamic performance.

Aerodynamic performances are improved by a drag reduction. From this study we can conclude the following objectives:

1. The air foil S1210 has the highest value for the drag and lift forces.
2. Drag and lift are impacted by angle of attack of the airfoil.
3. The efficient lift to drag ratio is occurred at 12° angle of attack.
4. This study is mainly interested to get the efficient aerodynamic performance of multi element wing.
5. In this study it can be observed about the different types of the flow structure that can be created around a vehicle to get better aerodynamics performance.
6. The ways to get optimum drag and downforce for the car to get efficient aerodynamic forces.
7. To select an angle of attack to model the front and rear wing.
8. The main objective of this paper is to design an efficient front and rear wing for a Formula student design competition.

REFERENCES

- [1]. Eftekhari, H., Al-Obaidi, A. S. M., & Eftekhari, S. (2020). The Effect of Spoiler Shape and Setting Angle on Racing Cars Aerodynamic Performance. *Indonesian Journal of Science and Technology*, 5(1), 11-20.
- [2]. Cai, J., Kapoor, S., Sikder, T., & He, Y. (2017). Effects of active aerodynamic wings on handling performance of high-speed vehicles (No. 2017-01-1592). SAE Technical Paper.
- [3]. Panchal, K. (2015). Computational Assessment of Front and Rear Wing of Amity Racing Vehicle (No. 2015-26-0185). SAE Technical Paper.
- [4]. Azmi, A. R. S., Sapit, A., Mohammed, A. N., Razali, M. A., Sadikin, A., & Nordin, N. (2017, September). Study on airflow characteristics of rear wing of F1 car. In *IOP Conference Series: Materials Science and Engineering* (Vol. 243, No. 1, p. 012030). IOP Publishing.
- [5]. Howell, J. and Le Good, G. 1999. The influence of aerodynamic lift on high speed stability. SAE Paper No 1999-01-0651.
- [6]. Selig, M. S. (1996). UIUC airfoil data site.
- [7]. ANSYS Fluent Theory Guide; Release 19.2; ANSYS Inc.: Canonsburg, PA, USA, 2013.
- [8]. McKay, N. J., & Gopalarathnam, A. (2002). The effects of wing aerodynamics on race vehicle performance. *SAE Transactions*, 2254-2263.
- [9]. Chen, M. N., Tang, W. H., Yang, B., & Hu, X. J. (2012). Computer-Aided Front and Rear Wings Aerodynamic Design of a Formula SAE Racing Car. In *Applied Mechanics and Materials* (Vol. 120, pp. 20-25). Trans Tech Publications Ltd.
- [10]. Pehan, S., & Kegl, B. (2002). Aerodynamics Aspects of Formula S Racing Car. In *DS 30: Proceedings of DESIGN 2002, the 7th International Design Conference, Dubrovnik* (pp. 1109-1118).
- [11]. Chen, Y., Zhou, J., Su, G., & Wang, C. (2020, November). Design and Analysis of FSC Formula Racing Car Body. In *Journal of Physics: Conference Series* (Vol. 1635, No. 1, p. 012025). IOP Publishing.
- [12]. Hetawal, S., Gophane, M., Ajay, B. K., & Mukkamala, Y. (2014). Aerodynamic study of formula SAE car. *Procedia Engineering*, 97, 1198-1207.