International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

Optimization Analysis of Race Car Design

Alwin Thomas, Vivek Wagh,

Dr. M. Gunasekaran

(Assistant Professor)

Bachelor of Technology

Department of Mechanical Engineering

College of Engineering and Technology

SRM Institute of Science and Technology

Kattankulathur – 603 203

ABSTRACT

Modern sports vehicles got enhanced in terms of performance by using better tools to design and analyze. The study of aerodynamics has always been a very intense, challenging as well as exciting task for professionals working on the research and development. This work presents the numerical analysis of our modeled race car aerodynamics and the simulation on it. We have studied different airfoils and used them in the rear wing. We studied the air flow over the body of the car at different velocities and its effect on the various aerodynamic parameters of the car, i.e., downforce/negative lift, stress, pressure and drag. We collected and tried to study what the numbers meant and analyzed the data which we got from the simulation solver. This was a tricky and time-consuming part as simulation has much to study and understand. Our aim was to get meaningful results from our simulation study. Our simulation study was without aero kit and with aero kit. In the following

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

sections we will further discuss and explain more in detail. Final conclusion of this project will be; to find out the best fit airfoil and most efficient placement (angle of attack & height) of it in the rear of our modeled car. In this project we have designed a car, studied air flow over the body of the car at different velocities and its effect on the various aerodynamic parameters of the car i.e., downforce/negative lift, stress, pressure and drag.

**From now on we will be discussing specifically about race cars and motorsports*.

INTRODUCTION

1.1 What is Aerodynamics?

Aerodynamics is the science of air movement and its impact on solid bodies placed in the flow space as a barrier. As a subdivision, many equations from fluid dynamics also apply to aerodynamics and, including all governing figures, chaos, boundary layer theory, and appropriate gas speculation. It is the study of how gases interact with moving bodies. Because the gas we are most exposed to is air, aerodynamics is primarily concerned with gravity and gravity, caused by the wind passing through or around solid bodies. Engineers apply the principles of aerodynamics to the construction of many different objects, including buildings, bridges, and even soccer balls; however, the biggest concern is the movement of planes and cars. Aerodynamics plays a vital role in aeronautical research and aeronautics science known as aeronautics. Aerodynamics engineers use aerodynamics foundations to design the aircraft in Earth's atmosphere. Supercar and motorsports companies are also doing a lot of research and development in liquid mechanics. They also spend a lot of time and money to get good results from their cars.

1.2 History

Air is already widely used by humans as a tool (windmills, sailing ships), but scientific research into aerodynamics dates back to the 17th century. The dream of flying with "light than air" equipment has been around for a long time. Early attempts to explain the size of the flow are evident in the works of

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

Aristotle and Archimedes. However, the scientific field "aerodynamics" does not appear on the note.

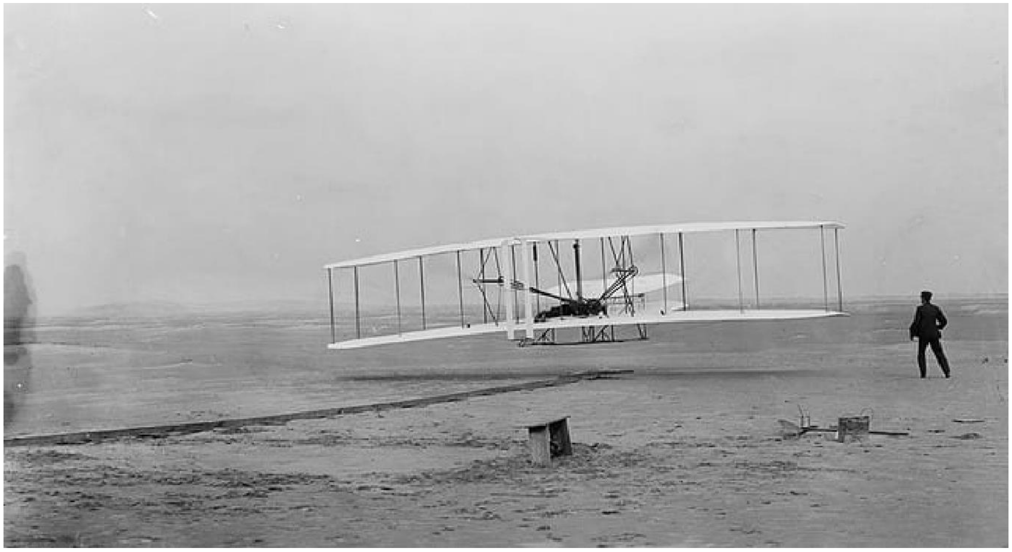


Figure 1.1: A double-decker airplane in the early days of aircraft.

The first aerodynamicist was Sir Isaac Newton, who invented and described the theory of flow resistance, also known as drag. Other well-known scientists, such as Bernoulli, Euler, Navier, and Stokes, have provided more accurate and accurate interpretations of gas mechanical calculations. The Navier-Stokes-dominated liquid machine equations were created in the 1800s, but they are also the most difficult models to solve. The desire to fly has always been a major factor in the development of the field of aerodynamics engineering. In the early 1800's, engineers and scientists began to study the power of flying. Words like dragging, lifting, and throwing seem to describe the relationship between you. The first air tunnel was built in 1871. Opposition theory was developed by engineers such as Kirchhoff and Rayleigh. Otto Lilienthal made the first successful flight, and after all these studies, in 1903 the Wright brothers flew me on my first commercial airplane.

LITERATURE SURVEY

Most race cars and sports cars have spoilers attached to them at the back. This is so because these cars are meant to be driven fast on the tracks for races. They are light in weight and so at higher speeds the velocity of wind gushing through the car aerodynamics may cause instability and loss of control over the car leading to serious accidents. So, to increase handling and stability, the spoilers play a major role in aerodynamics of a car. The spoilers also help the car keep low to the ground and close to ground so as it remains in good contact with the ground.

- **Problem statement.**

The first step for motorsport teams is to go through the rulebook provided to them. They have to read the rulebook carefully. The teams have very intelligent people who try to understand and make the base design of their cars. They continuously revise the exterior design and shape of the body while complying to the design rules and regulations.

- **Design.**

Aerodynamics is the trickiest part here because everyone wants more grip and high speed. More grip means the driver needs a crazy amount of downforce on the car. So, to get that amount of downforce the car needs aero kits or small aero components which help increase downforce. But there are limitations to this, the team engineers work continuously to reduce the drag created by the car and the aero components. If we do not reduce the drag, the driver will feel too much force in the negative direction of his travel which in turn will make him slower than the other drivers and that excessive amount of downforce will not be very helpful. So, the aerodynamics will always remain the most crucial domain of the car to work on.

- **Modeling Simulation.**

Teams use various CFD simulation software to run and solve 3d simulations. Software like ANSYS, Open FOAM, SIEMENS Star-CCM + commonly used to obtain real world simulations. One, integrated area covers everything from CAD, automated meshing, Multiphysics computational fluid dynamics, complex background processing, and design studies. This allows engineers to successfully navigate the entire designarea in order to make better design decisions faster.

- **Material of construction (MOC).**

Race Cars such as DTM, Formula 1 cars and NASCAR are made of materials such as carbon fiber/glass fiber, epoxy resins blended with plastics or Kevlar. These composites are used in order to make the car parts strong and lightweight as well as aerodynamically efficient. The aero kits on a car help the car gain more downforce and also reduce drag in some cases (such as DRS-Drag reduction system). There are many ways in which the aero kits can be installed on a racecar. But everything comes with proper limitations. We cannot just install the aero kits anywhere on a race car's body. If we do so without understanding the results, there might be serious consequences. The study of CFD simulation steps in here and it is a very detailed study and sometimes complex. Performing simulations and giving input might be easy but, understanding the results and then reworking on them to get more efficient and better results is a challenging task. There are multiple software's in the market which we can use to perform simulations. We will discuss more about our ideology and process of simulation study.

- **Manufacturing.**

After the simulation process, the team runs CAD simulations and creates a small model to compare the results. The compact model will help identify problems that may arise in the real vehicle, and they will help change a few parameters (if necessary). Instead of making a full car and performing on it. It will be very expensive. So, when all simulations and fixes are done, it goes into production. The F1 uses a number of additional gears aimed at reducing lift and drag to shorten the 's lap times. Lift and drag are inversely proportional to each other. The accessory used in F1 vehicles can change the center of pressure (CP), an important factor in aerodynamics. The rear wing has two sets of aero wings connected to each other by wind end plates. The upper aerodynamic wing typically consists of

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

three elements, providing the greatest downforce. The lower airfoil provides a low-pressure zone just below the wing. As the L/D ratio decreases, the lift coefficient increases and the number of airfoils increases. The rear end plate helps reduce the effectsof updrafts from the wheels.

2.1 Bernoulli's Theorem

The spoilers are of inverted airfoil shape. There are different types of airfoil shape geometries. The have specific purpose and applications at different places. The airplane wings are designed to give lift to the airplane whereas, in race cars the wings are designed to give maximum downforce to the car. The airfoil shape works on Bernoulli's Theorem. It can be explained by of the Figure 2.1.

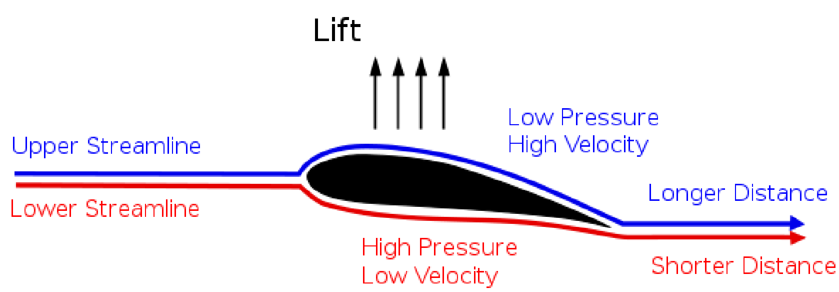


Figure 2.1. The air flow separation over the airfoil Bernoulli's theorem.

You can observe the upper streamline and lower streamline or air flow. This is the section view of airfoil. The air coming from the front side slides into two ways over the airfoil. The distance covered by the upper streamline of air is more than the lower streamline of air. This is because of the slight curve and bulgy shape of the airfoil on the upper side. Whereas the speed of air on the below side of the wing is less. The speeds vary because when the air distributes into two at the front of the wing, it has different distance to travel but within same time. So, to reach at the end of the wing, the speed of air on the upper streamline is more. This difference in speed causes pressure difference. The down part of the wing becomes high or more pressure area whereas, the upper or top part of the wing is less pressure area. This causes lift to the airplane wing. When this type of wing is used inverted on a car, it creates the negative lift or downforce to the car. When a car goes through corners, the forces acting on it are different. We cannot simulate perfectly the downforce through corners as we all are aware that the incoming air has different angle of attack to different body parts of the car. The study experiment and numerical and mathematical analysis of inverted T026 wing in a curved path of same radius showed that the flow simulation was very much disturbed and asymmetric. This led to study air flow simulation of car on a straight line and the simulation results were near to accurate. The asymmetric air flow through a corner causes change in air pressure acting on the wing surface area, thus not providing us proper results. In open wheel cars, such as F1, the front wings are most important as they will influence the air flow streamline throughout the aerodynamic body. The front wing is the first part to experience the high-speed winds attacking from different angles. These front wings help to put more downforce and direct pressure on the front tires contact patch with the ground. This gives more grip for the car to steer and maneuver through corners and straights. This also helps to balance the roll of the car and minimize it. The velocity gradient is the most important factor. Velocity gradient is the difference in speed and pressure of fluid layers flowing over each other. This velocity gradient is responsible for creating high pressure and low-pressure area points on a car surface. This also causes drag which is not efficient for the car aerodynamics and reduces top speed reach of cars. Another important thing which we will be studying is: Velocity of wind attacking on different parts, radius, yaw angle. On a constant radius curve, we can consider the car into three different parts: inner middle and outer.

$$R_{inner} < R_{middle} < R_{outer} \quad V_{inner} < V_{middle} < V_{outer} \quad \psi_{outer} < \psi_{middle} < \psi_{inner}$$

where;

R is radius of curvature;
V is velocity region of the wing and ψ is yaw angle.

2.2 Add-on devices

Aerodynamics plays a very important role in Formula 1 racing. But other than that engineers are trying to bring efficient aerodynamics in commercial vehicles too, so that they can improve ride handling quality, fuel efficiency etc. At present the Formula 1 norms are very strict and the teams have many restrictions for various domains such as emission norms, length of car, height above the ground etc. They (FIA) have made the rule book very difficult to make any changes or add extra add-on to the car. The engineers have to find loopholes while abiding the rules and then implement them in the car. Steady state CFD simulations on a F1 car using add-ons. Fluent Solver was used initially to study free air flow over the body. Then the add-on was added. The diffusers, roof wing, front wing etc. are examples of add-ons. When the add-ons are attached to the car, they bring much drag to the car which is not good as it will slow down the car's top speed. Less drag means good lap times and thereby leading the race. In order to maintain good handling of car, the aerodynamic center of pressure is the most important. More angle of wing causes more downforce but also more drag, reducing the speed by significant digits.

1. **Modified front wing**

In an F1 car 30 to 40 percent of downforce comes from front wing. In original front wing there are no end plates and deflectors. After adding these we could get a streamlined flow of wind without making more drag force on tires.

2. **Bargeboard**

The vehicle's barge is located directly behind the suspension arm. These five circulate through the body with the flow of wind, reducing resistance. The main function of the bargeboard is to organize this flow around the body of the vehicle, returning it through the rear wing, which is important to create downforce.

3. **Nose wing**

This provides a reverse negative lift airfoil shape that balances aerodynamic moments at the front and rear of the vehicle and optimizes airflow under the and double wishbone upper suspension arms.

4. **Roof spoiler**

It's above the driver's seat. Its main purpose is to provide downforce and optimize flow. The midsection should also have sufficient downforce to balance the vehicle's overall aerodynamic moments. The basic idea is to keep the center of pressure as close as possible to the vehicle's center of gravity (CG) toensure maximum stability in operation.

5. **Rear Wheel Scallops**

Grab the front of the rear wheel. This reduces drag due to the rear tires and helps direct air from the bead and front fenders to the rear fenders.

6. **Rear Wing**

The rear wing creates downforce towards the rear of the vehicle. The is designed to direct air to the rear fenders to generate the most downforce in the back. Reduced drag and drag coefficient. Downforce and lift factor have been doubled on models with all attachments. Downforce and lift coefficient increased in modified models with added devices.

EXPERIMENTAL INVESTIGATION

1. DESIGN OF VEHICLE

Car design is a creative process used to define the appearance of a car. Design thinking gives you theopportunity to look at problems from a completely different perspective.

Steps used in designing this car:

1. Sketch using pen and paper
2. Sketch using Wacom tablet in Photoshop
3. 3d model using class a surfacing tool Autodesk Alias software
4. Visually realistic renders using Keyshot

After completing the design process, we moved to CFD simulation part. We have taken design idea from 'White Tiger' in Fig. 3.1.

Figure 3.1. The highlighted parts are referred and adopted to make car surfaces.

You can see this image which shows how we extracted design lines from the body. This is the first and most important step in ideation. Then we sketched 30 to 70 different car designs from these lines and finalized theshape and design as shown below.

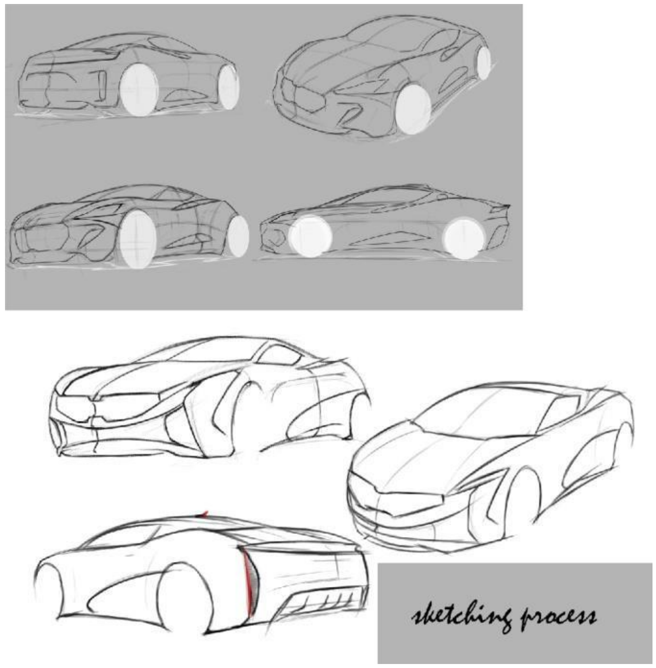


Figure 3.2: Line sketch of our car.

We drew 50+ car sketches and selected 1 design and started sketching digitally to know more about the design. We used photoshop for this process. We had to check the dimensions to know whether it stays withinthe Indian standards.

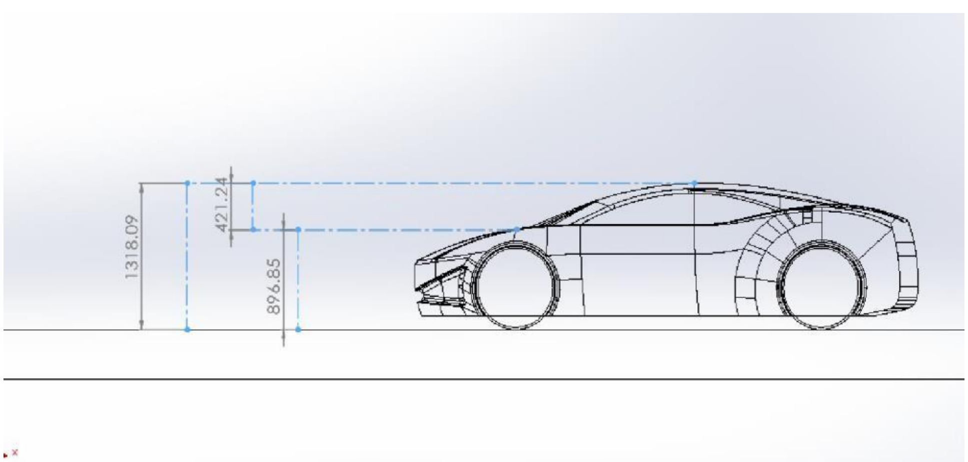


Figure 3.3: Dimensions of our car. (Side View). All dimensions in mm.

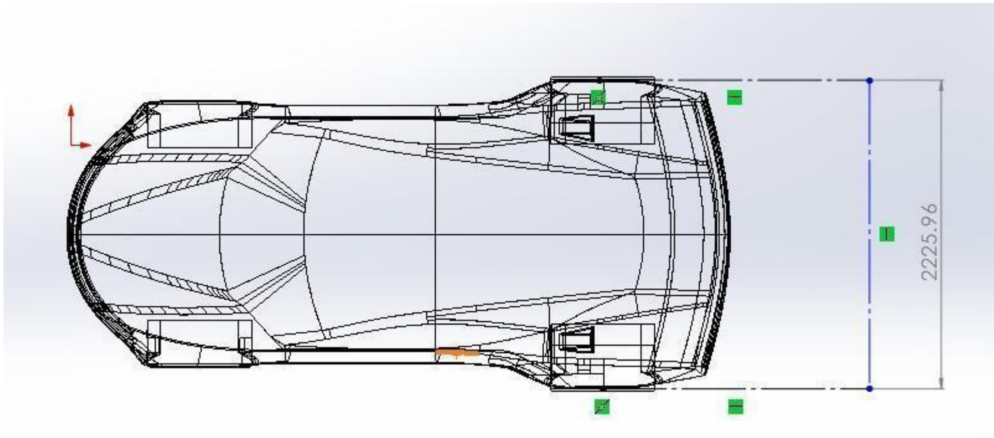


Figure 3.4: Dimensions of our car (Top View). All dimensions in mm.

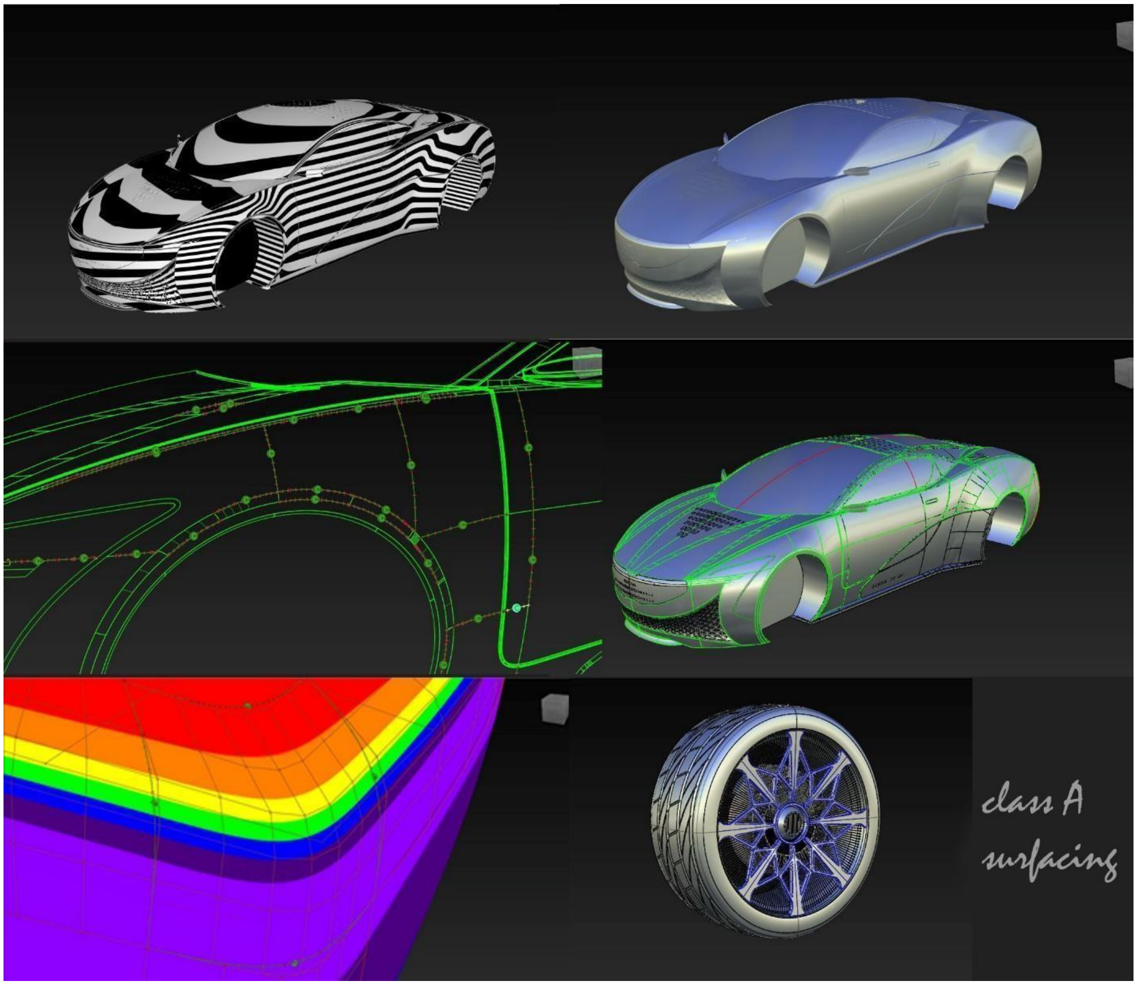


Figure 3.5: Class A Surfacing of our model in Autodesk Alias

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

Figure. 3.5. shows the Class A surfacing process, which was challenging and time consuming. Each and every part is designed in Autodesk Alias. In automotive design, a class A surface is any of a set of free form surfaces of high efficiency and quality. The minimum math requirement for Class A Surface is that the transition across the edge is a continuous curvature. Another way to explain it is when you don't know where one point ends at and another starts when the reflection hits the surface. Grade A surfacing requires the perfect combination of sharp visualization skills, artistic insight, engineering skills, and experience and knowledge.

Figure 3.6: Rendered parametric design

Figure. 3.6. shows the parametric design. It has been gone under rendering process
to get more aesthetic and pleasing look. ***Parametric*** term used to describe the ability of magnitude to change the shape of a geometric model when the value of magnitude changes. Parametric modeling uses computers to design objects or systems that reflect the structures of parts with real-world behavior. The element base is a term used to describe the various parts of a model. For example, a section may have different types of elements, such as holes, grooves, ties, and chamfers. The "element" is the basic unit of a solid parameter model. One of the most important features of parametric modeling is that linked attributes automatically change their characteristics. In other

words, parametric modeling allows the designer to define all shape classes, not just specific situations. Organizations often turn to parameters when creating product families that incorporate minor variations into the main design, because the designer will need to create a design purpose between sizes, components and assemblies. This supports designs that need to be changed or repeated. It also creates models with unique features that can be modified or modified that are photographed on a 'model tree.' Keyshot is able to deliver additional Class A requirements for visual and artistic components, including Automotive Exteriors. We then moved on to the digital painting process using photoshop itself to get a clearer view of the painting. Then the next part was time consuming and the most challenging part of our design process, which is 3d modeling from drawing. It took about one month to complete the entire process.

Initially we chose Siemens STAR-CCM+ for our core simulation process. We were completely new to this software and neither of us had used this previously. We started with the basics of STAR-CCM beginner tutorials but soon we realized that the process of learning and understanding was a little brief journey and so we decided to not directly get involved in complex calculations and processes. We also had used open source CFD Simulation which was only available online on SIMSCALE website. For us it was time consuming due to connectivity and limited network data accessibility. But for beginners this SIMSCALE is very helpful to learn and start. Finally, we decided to use SOLIDWORKS software for the CFD Simulation. We had to close the surfaces of car to make that a solid model, which we had to change design of the car where the aerodynamics won't affect much. After these processes we started calculating CFD using Solidworks flow simulation which helped us to get accurate results. We started designing the car by keeping in mind the aerodynamics of car and the Indian car exterior design rules and regulations. We have also taken proper careof dimensions of the car.

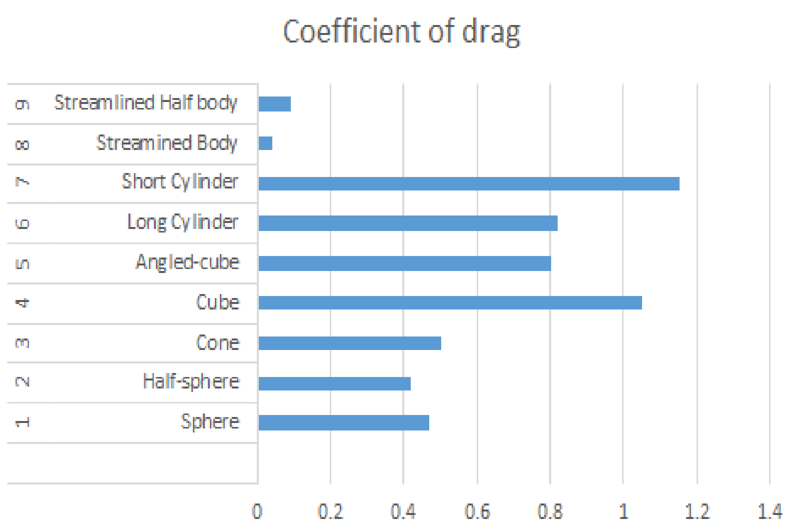
- **Ideas Layout**

    - Designing concept idea
    - Software selection
    - Developing design
    - Making aero components
    - Troubleshooting design flaws

- Proceeding to basic simulation
- Software selection for simulation
- Begin with Simulation
- Take more reference and journal paper study into account
- Make more efficient design according to simulation results

2. AERO-PARTS DESIGN

Before starting anything in designing Aero parts of our car, we figured out any flaws and defects in our car design. There arise many flaws and defects in surfacing, meaning, the surfacing has to be done perfectly and symmetrically because dissymmetry causes wrong air flow simulation over the car body unless and until we do it intentionally. So, checking the exterior closed design of the surfacing is very important in order to study air flow over the car surface. We studied about various shapes and their drag coefficients as you can see the chart 4.1. The different shapes and their drag co-efficient help us to understand and choose the design or shape for a perfect aero shape. It means the lesser the drag co-efficient the much better it is and more efficient or also, in other words less loss of energy or lesser drag. Basic Aero parts on F1 car are shown here and their small subparts and components are



mentioned in detail.

Table 3.1: Drag coefficients of different shapes.

1. Simulation Ideology

- Start with basic simulation study (types and processes involved)
- Basic requirements for the simulation
- Study of various factors affecting simulation
- Factors in simulations i.e., the physics involved in simulation
- Basic Formulas and equations used for simulation in solver
- What does the various simulation line color indicate?
- Types of input to be given in simulation
- Reading and understanding the output
- Graphical representation of the simulation data
- Plotting respective graphs

Inputs:

We followed these steps:

1. We gave the fluid type which was air and many other boundary conditions and assigning them their purpose.
2. Added velocity and started simulation.
3. After adding these details we have to add the bounding box in which themodel sits fits.
4. Then we give the boundary conditions like road friction etc.
5. Then we check the boxes for what all results we require.
6. Meshing the entire model.
7. Run simulation.

After processing these steps, we get results which we need in a box which shows graph and everything we require. We do these process 4 times with and without body kit to get the result.

These results are noted down in a tabular method. The boundary conditions which we give are; Ideal wall which is road, Velocity - 90,110,130,150 km/ph., Gravity in negative direction, and forces are on negative Y-axis which means downforce and positive means lift.

Simulation Results which we chose to study:

1. Force(Y)
2. Velocity points
3. Temperature points
4. Pressure points
5. Drag over the body surface

2. Detailed study on the Airfoil

A. Rear wing analysis:

Incompressible Turbulent Airflow around a Spoiler:

Here we explain what and how we did this simulation part.

- Run an incompressible turbulent flow simulation.
- Give boundary conditions, Color, material, and tunnel conditions to the simulation.
- Mesh the geometry

The typical SimScale workflow will be followed:

- Prepare the 3d CAD model for the flow simulation.
- work in the simulation.
- Make the mesh.
- Run the flow simulation and find the results.

❖ **Preparing the CAD model and selecting the analysis choice:**

We modelled the s1223-il air foil with the following specifications in Solidworks CAD software. Air foil plotter (s1223-il) S1223 - Selig S1223 high lift low Reynolds number air foil

Reason for choosing this air foil:

Details:

(s1223-il) S1223

Selig S1223 high lift low Reynolds number air foilMax thickness 12.1% at 19.8% chord.

Max camber 8.1% at 49% chordAir foil plots:

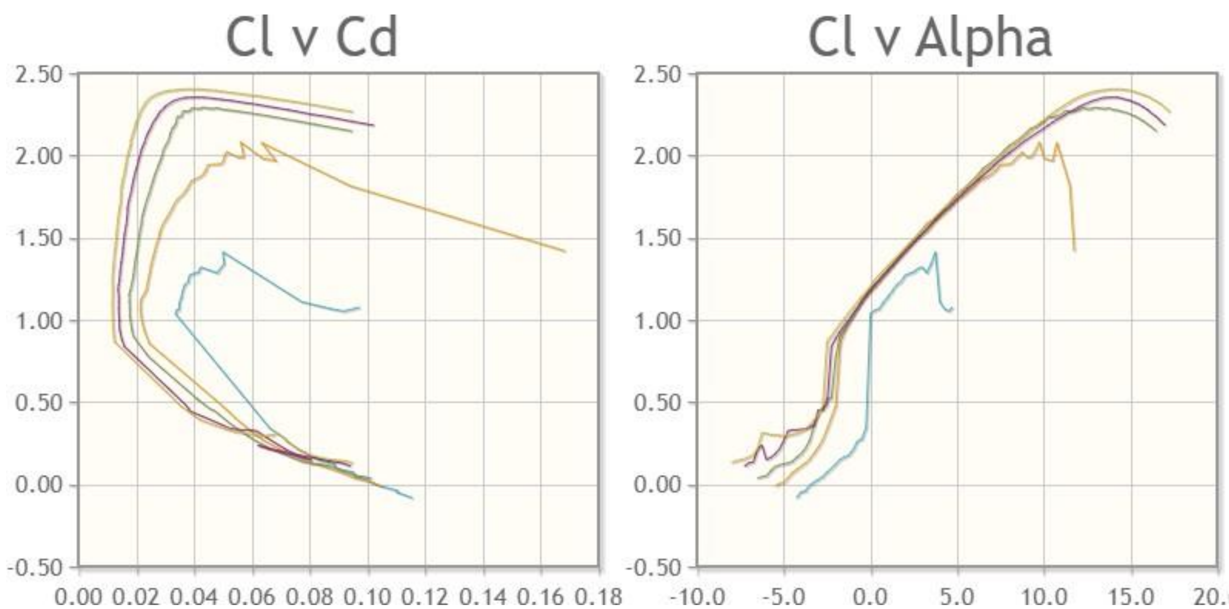
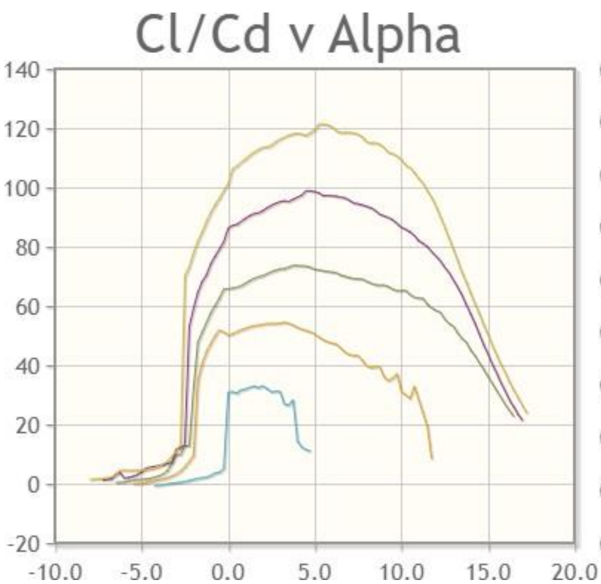


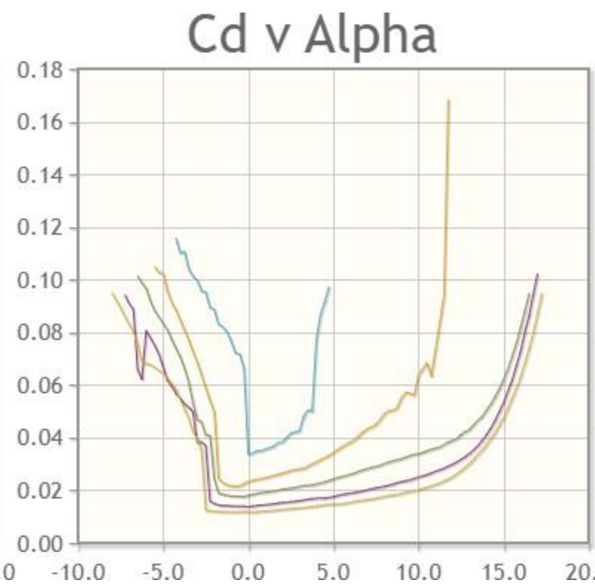
Figure 3.7: Airfoil plots with different relations.

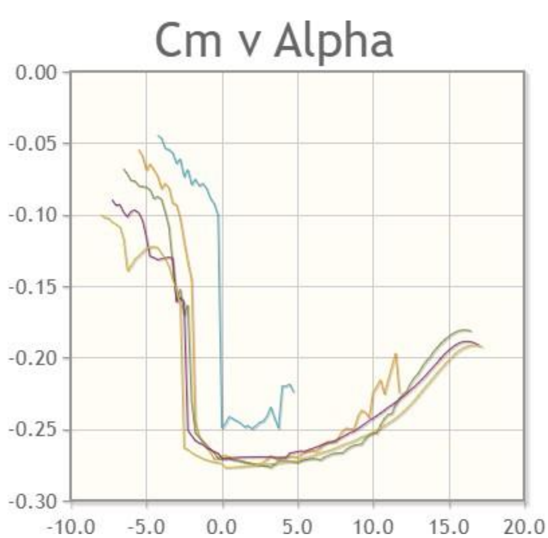
1. C_L v/s C_D - Coefficient of the lift to Coefficient of the drag.
2. C_L v/s α - Coefficient of the lift to Coefficient of the drag.
3. Coefficient of lift to drag ratio v/s α.
4. C_D v/s α.

α – Angle of attack.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930







Airfoil	s1223-il - S1223	Choose from database list or add you own airfoils here
Chord (mm)	200	Chord width in millimetres. (1 inch = 25.40mm)
Radius (mm)	0	Radius of camber in millimetres, Zero for no curve
Thickness (%)	100	Thickness adjustment 100% is normal thickness. 50% is half. 200% is double
Origin (%)	0	Adjust the position of the origin e.g. 50% is mid chord
Pitch (degrees)	0	Pitch or angle of attack. 180 flips the plot
Halo (mm)	0	Line parallel to airfoil for wing covering or jig. Negative values are external, positive internal.
Halo (mm)	0	Second line parallel to airfoil as above
Colour	Colour	Colour palette or black & white
Line thickness (%)	100	Scale the line thickness (10% to 500%)
Reverse	☑	Plot a mirror image
Data box	☑	Print the airfoil data on the image
Camber line	☑	Show camber line on image
X grid (mm)	10	X grid size in millimetres
Y grid (mm)	10	Y grid size in millimetres
Paper width (mm)	280	Used for printing plan. A4 landscape approx 280mm
Paper height (mm)	180	Used for printing plan. A4 landscape approx 180mm

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

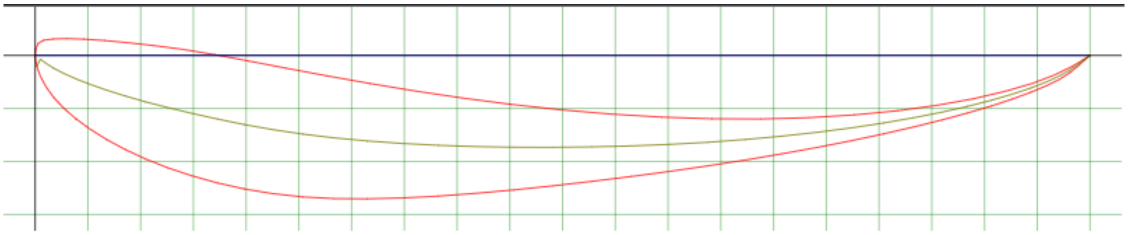


Figure 3.8: Inverted s1223-il airfoil with Chord length 200mm.

3D CAD model:

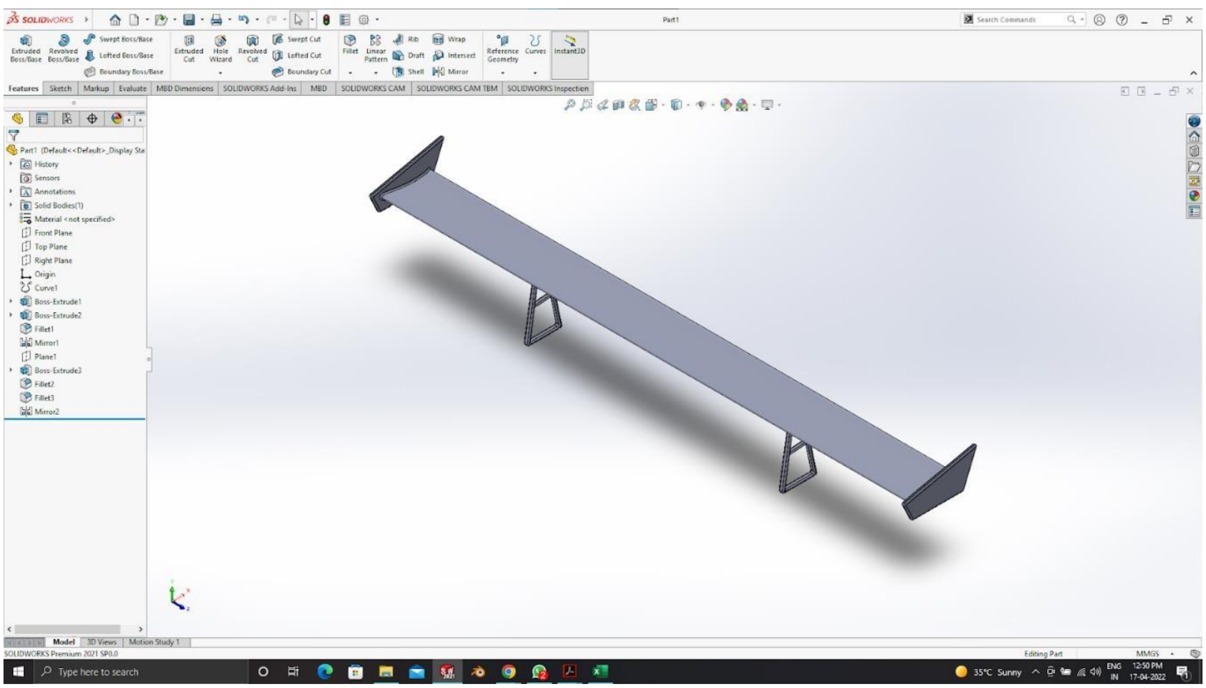


Figure 3.9: Inverted s1223-il airfoil with Chord length 200mm designed in Solidworks and is going to be installedon our car for further simulations

The above The figure shows the geometry used in the simulation. It corresponds to a typical car spoilerfitted to increase the downforce of the rear axle:

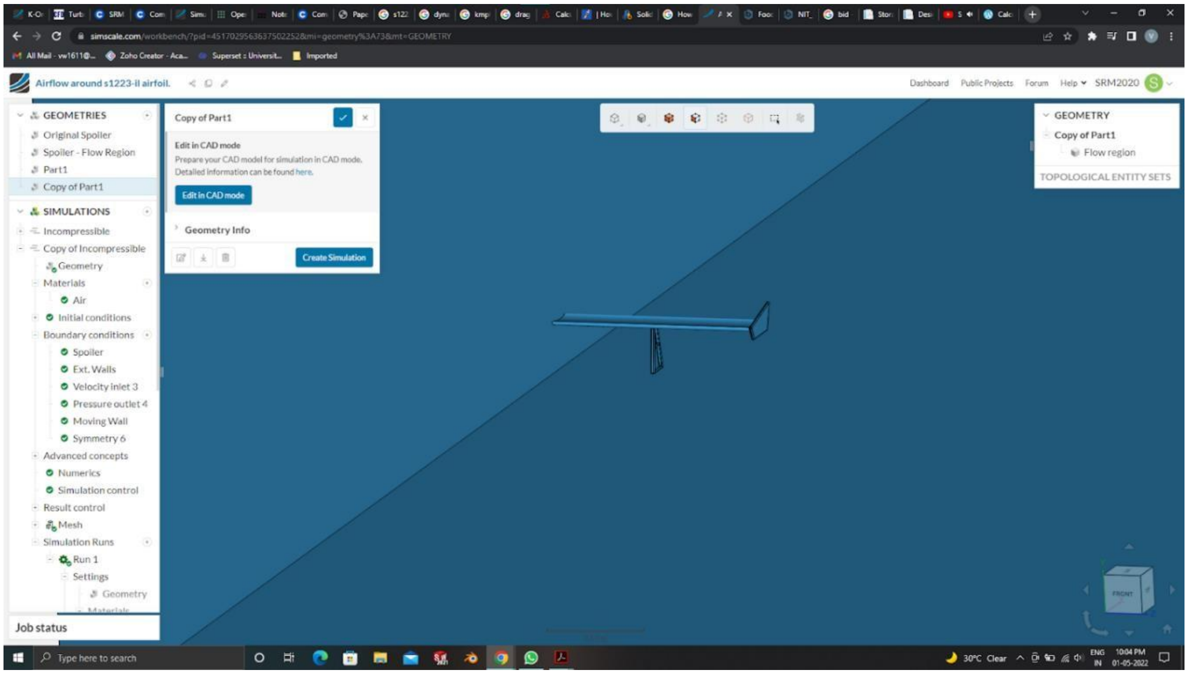


Figure 3.10: This is the spoiler-flow region.

You can see that the imported project has two geometries. The first is called the original spoiler and the second is called the spoiler - flow area. The original model shows the geometry of the analyzed solid part.

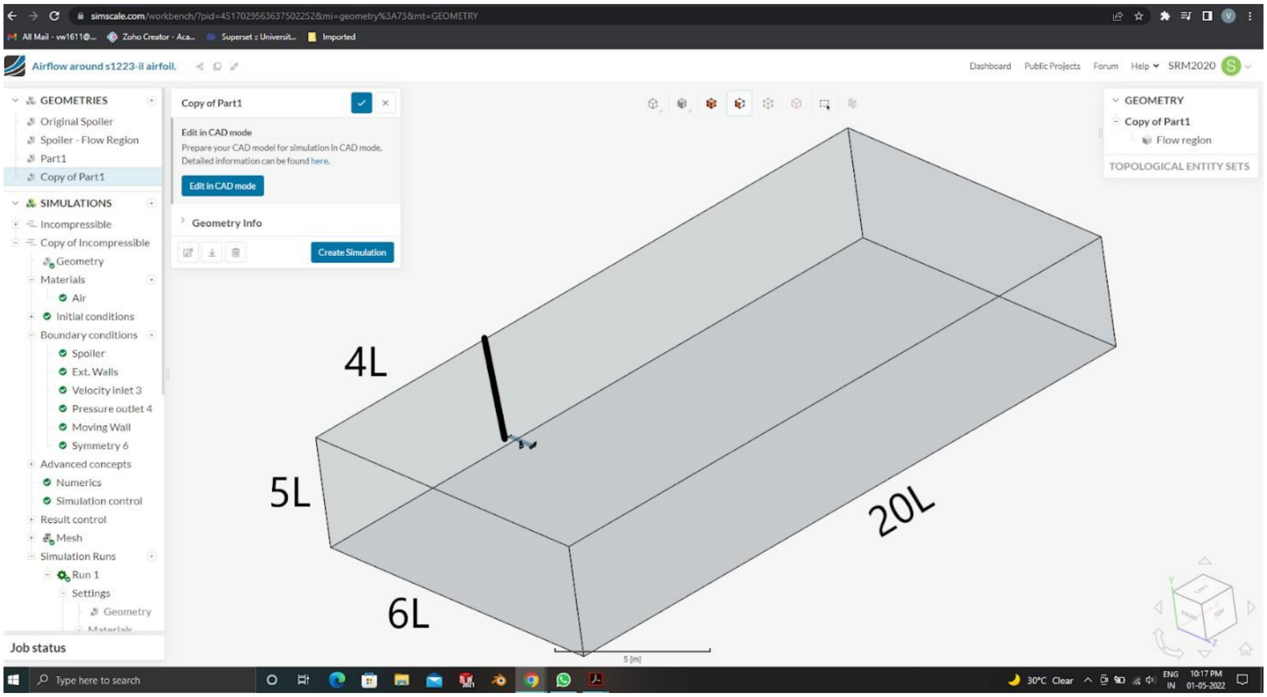


Figure 3.11: The above image shows the wind tunnel around the wing for air flow study.

Spoiler - The flow area part contains a virtual wind tunnel representing the amount of air around the spoiler. Since the spoiler is symmetrical, we only model half the geometry to optimize our modeling resources. The image below shows how the dimensions of the bounding box were chosen for the length of the spoiler cord,L = 0.20 m. Two operations were performed in CAD mode.

- External or outer flow volume, to create the flow region (virtual wind tunnel).
- Delete bodies, to remove or replace the spoiler part from the model.

As this is an incompressible analysis, only the flow volume is necessary to run the simulation. Therefore, thespoiler volume is deleted with the operation below:

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

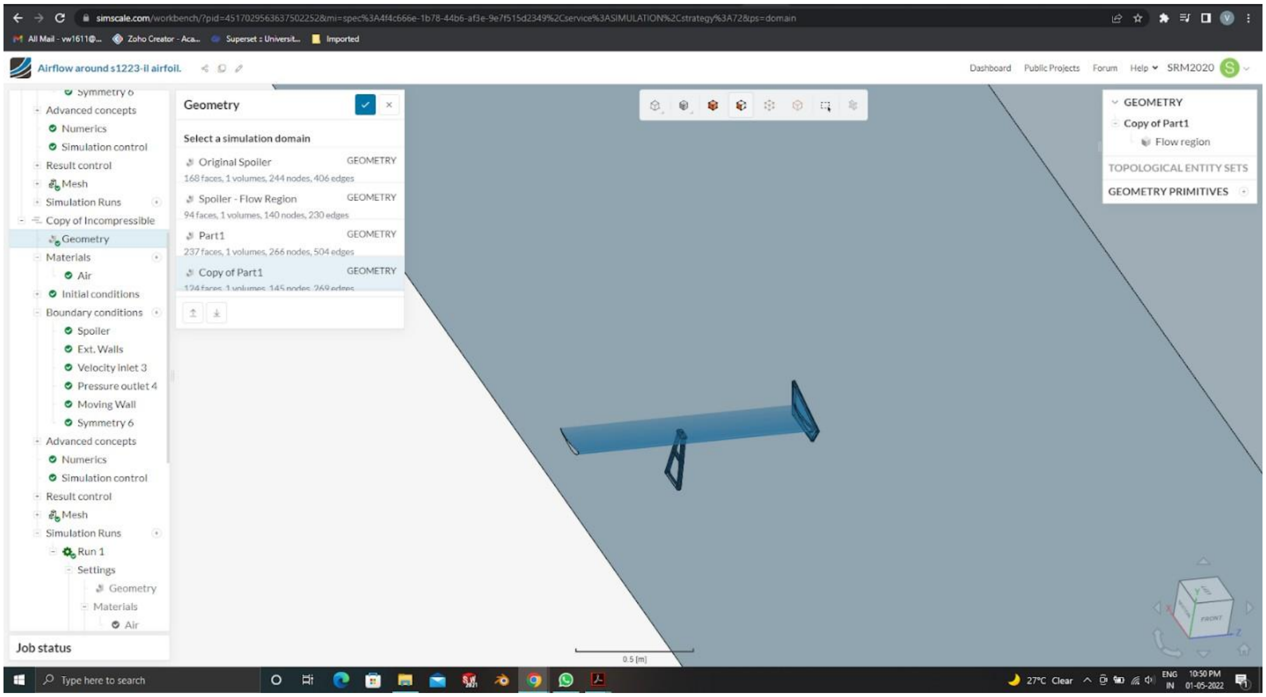


Figure 3.12: The above image shows the half wing profile in the wind tunnel or boundary box created.

❖ Create Topological Entity Sets

In this we will give the different sides and purpose for the walls of the wind tunnel. Each wall has a purposein the wind tunnel and any small parameter mistake will disturb our calculations.

A layout feature set is a group of faces used for boundary conditions or other purposes. With their help, thesimulation setup can be completed faster.

For the simulation setup, we will use the Spoiler – Flow Region geometry. After selecting the geometry, youwill find pre-defined topological entity sets on the right-hand side panel:

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

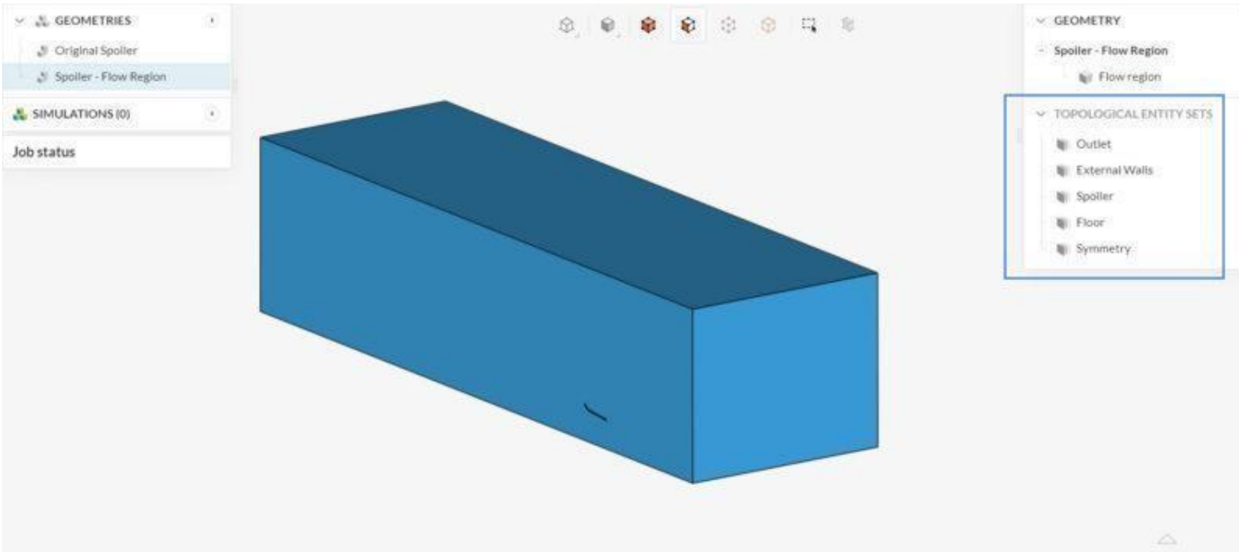


Figure 3.13: Five sides of the boundary condition are given the purpose.

The five required sets were already provided in the template design, but the flow inlet face set was stillmissing. The picture below shows how to add it.

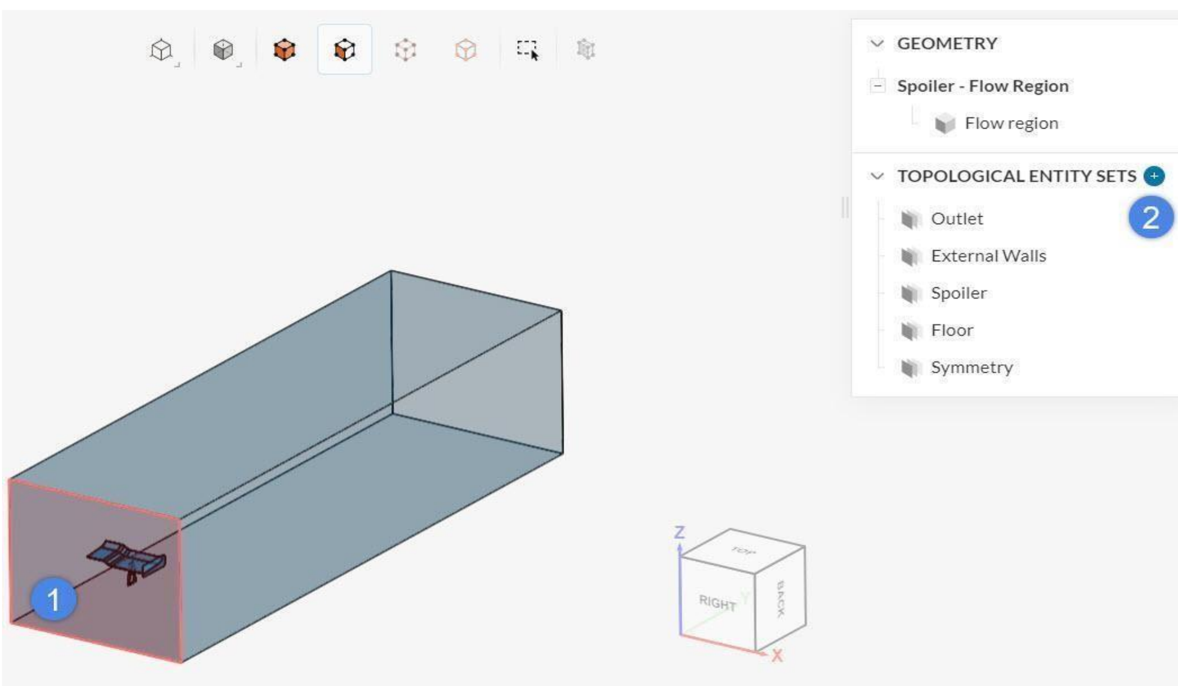


Figure 3.14: The above image shows method of adding the wall.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

❖ **Create the simulation:**

Now we can start by setting up our simulation. Click the "Generate Simulation" button in the lower right corner of the panel. The Simulation Library window appears, allowing you to select the appropriate simulation type.

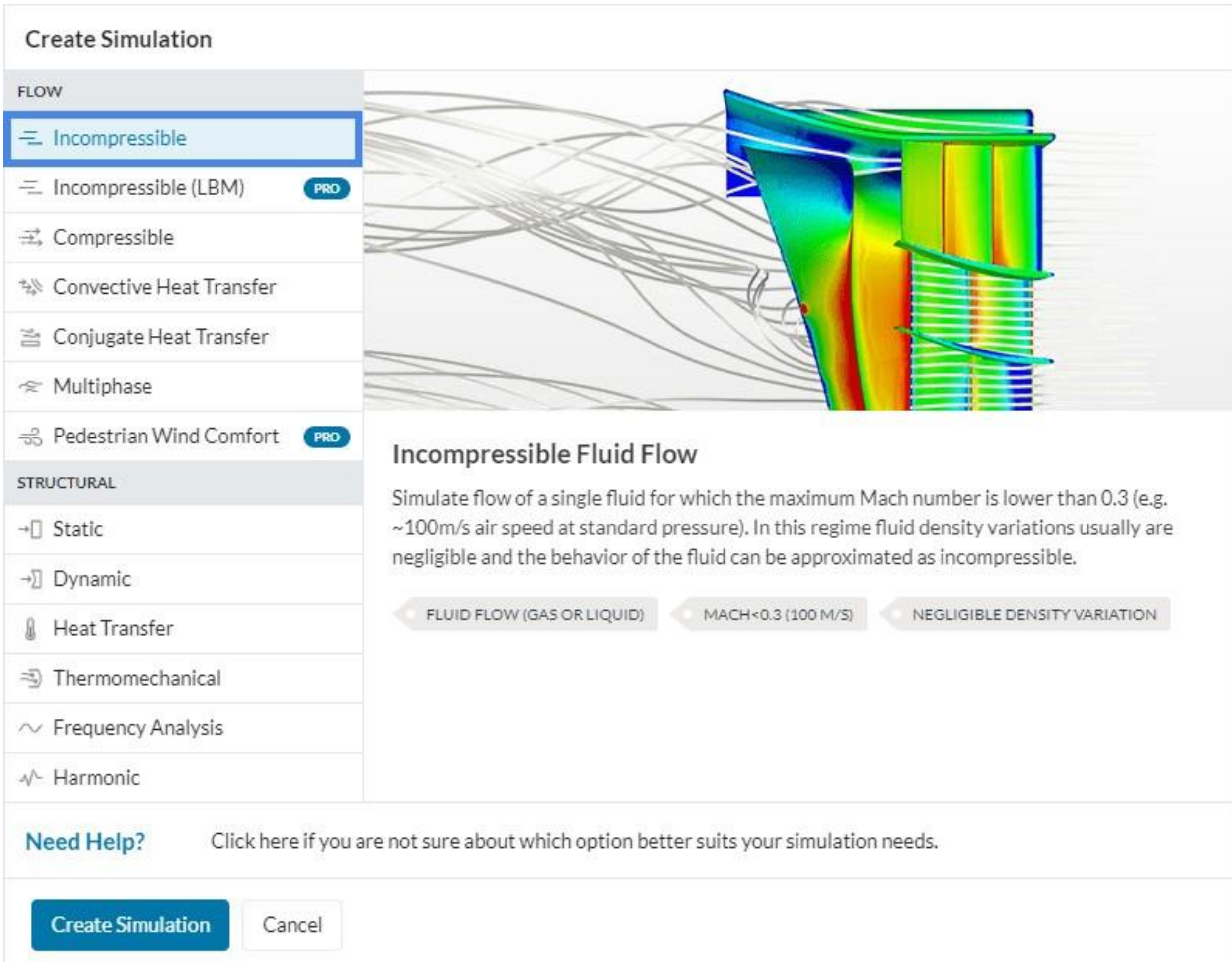


Figure 3.15: Setting up and creating the simulation.

Select ‘Incompressible’ and press on ‘Create Simulation’.

❖ **Set up the simulation:**

A new simulation tree will appear on the left hand of side of the Workbench. The first entry of the simulation tree is the global settings of the simulation. For this tutorial, please keep the default values:

We use ‘k-omega SST’.

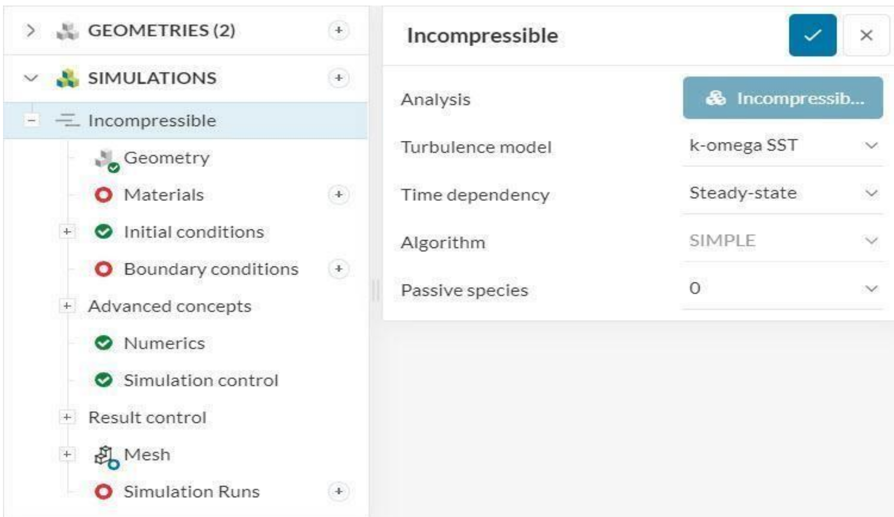


Figure 3.16: Choosing analysis type setting up simulation.

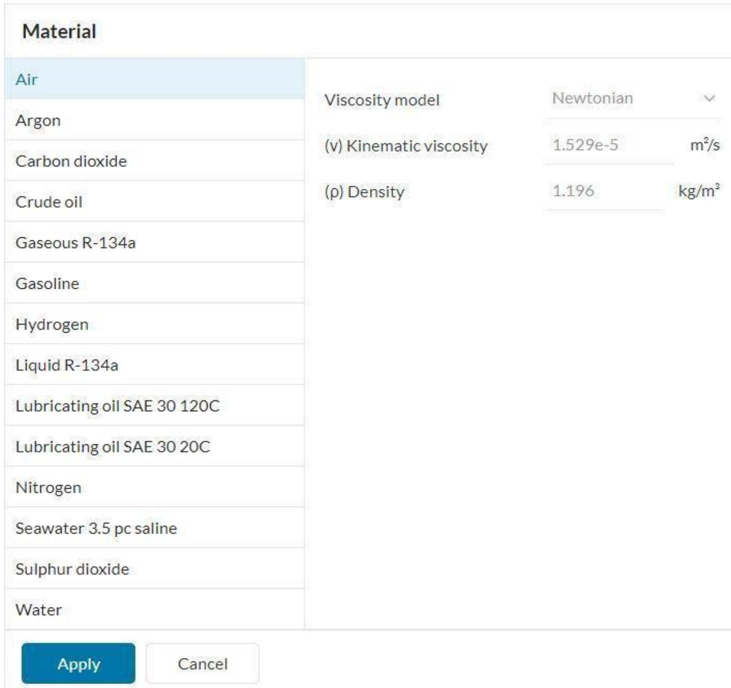


Figure 3.17: Adding material and conditions.

The Material Properties window appears. The Flow Region volume is automatically selected because it isthe only volume in the model. So, click the checkmark to accept your selection.

- **Boundary Conditions:**

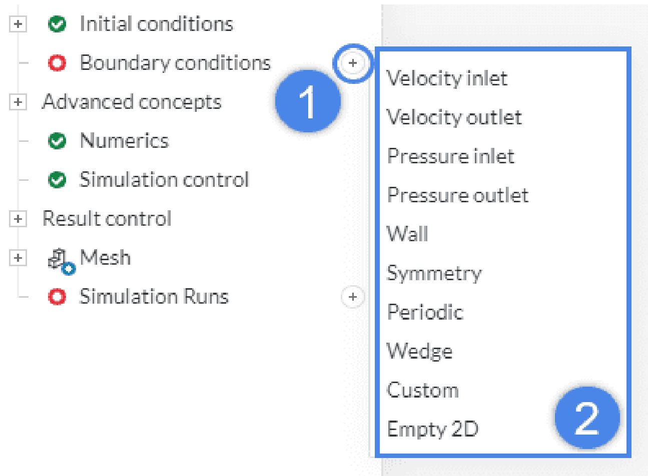
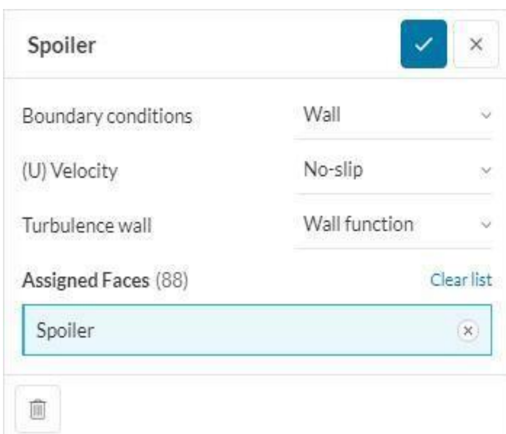


Figure 3.18: Initial conditions and boundary condition.



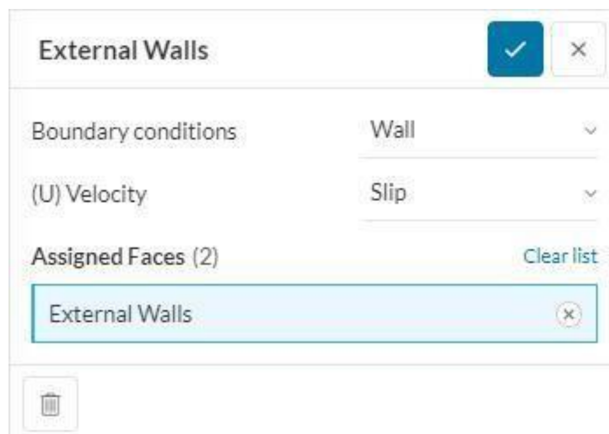
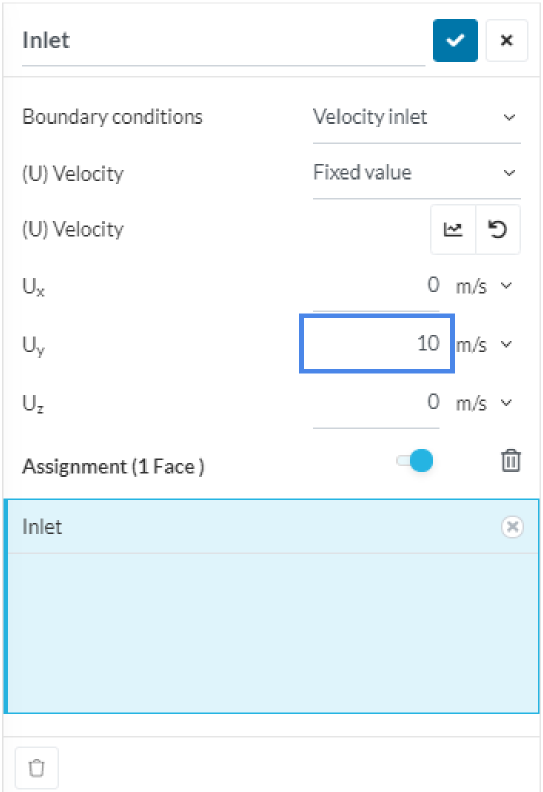


Figure 3.19: Spoiler conditions. Figure 3.20: External wall conditions.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930



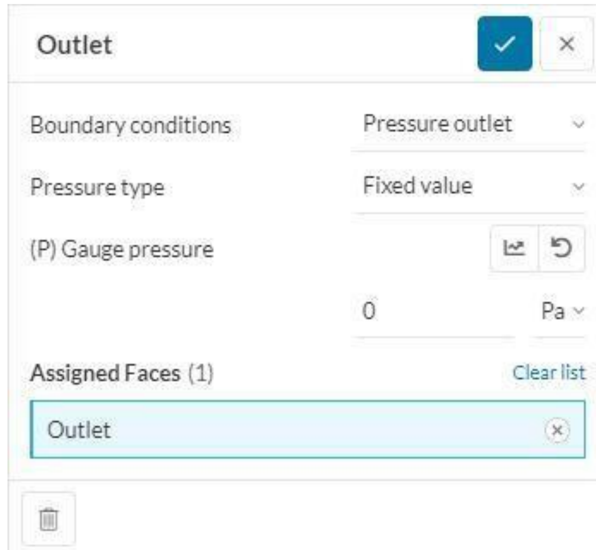
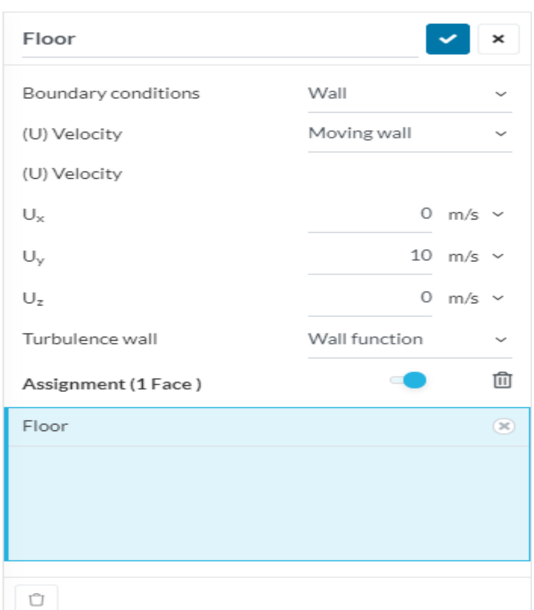
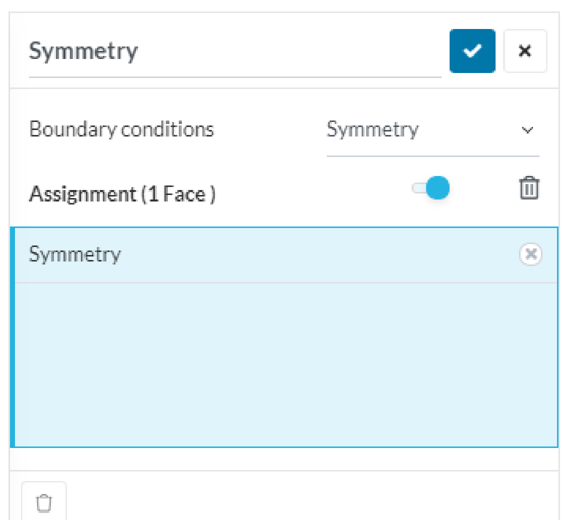


Figure 3.21: Inlet wall conditions. *Figure 3.22: Outlet wall conditions.*



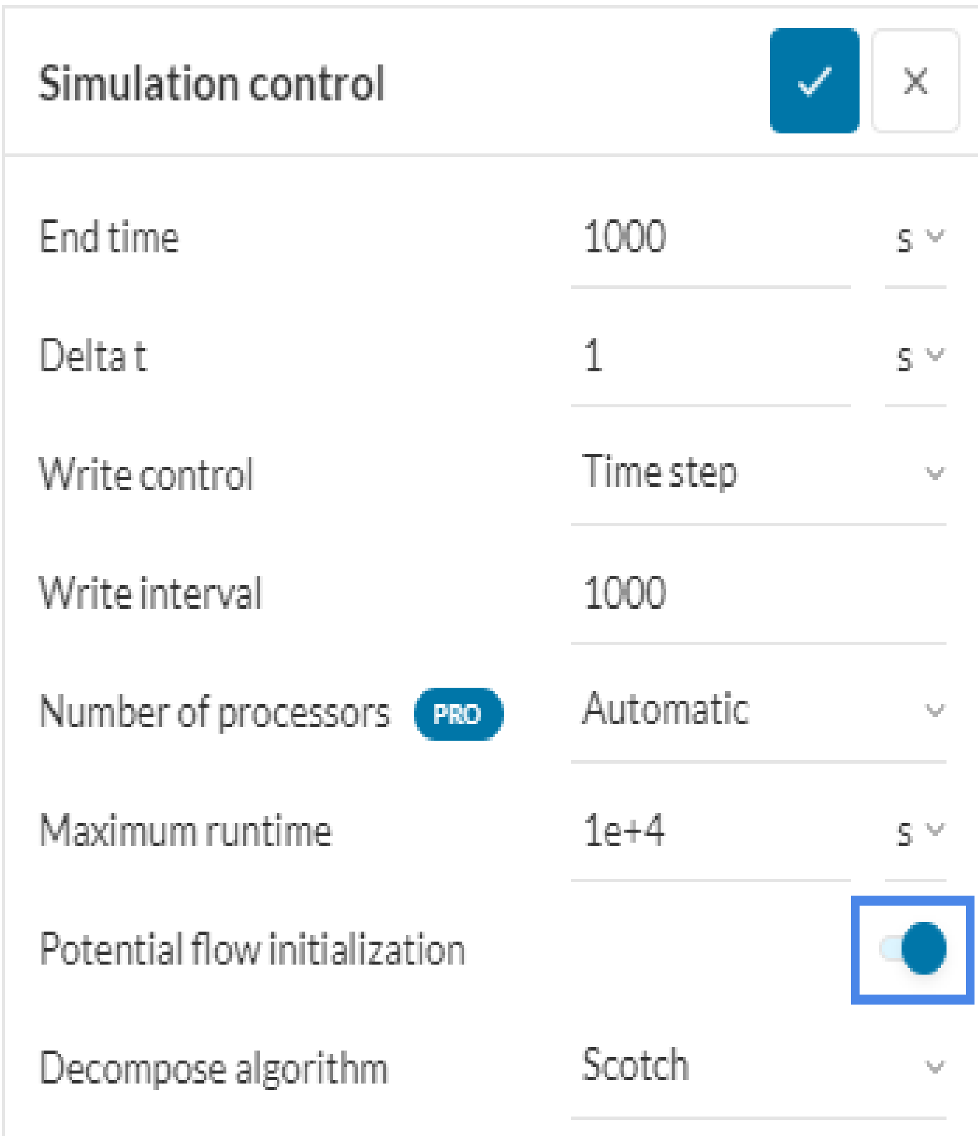


International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

Figure 3.23: Floor conditions. *Figure 3.24: Symmetry conditions.*

❖ **Simulation Control:**

One of the important settings to help the simulation get good results and converge faster is the initializationof potential streams. Activate it in the Simulation Control tab as shown.



International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

4. Mesh:

Figure 3.25: Simulation control.

We use Hexdominant Automatic Mesher. Click Mesh in the modeling tree and set the global settings asfollows. You can rename it to Hexahedral Mesh.

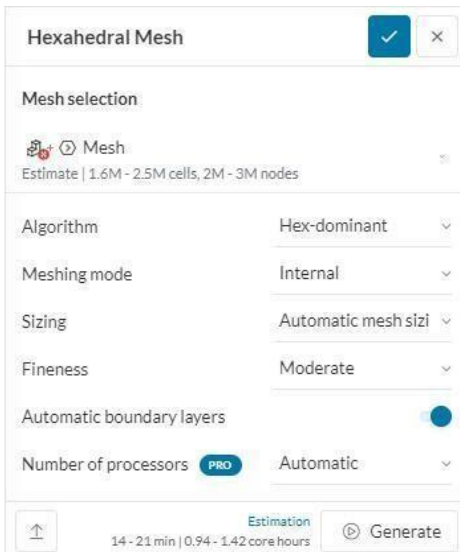
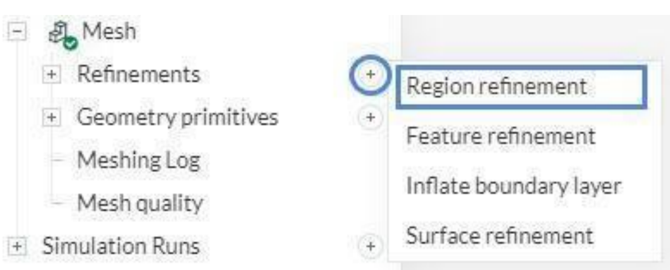


Figure 3.26: Meshing conditions.

Don't generate the mesh just yet –press on the blue check mark and save.

Before generating the mesh, we want to set a mesh refinement to capture the wake more accurately.Therefore, we will create a region refinement:



International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

Figure 3.27: Giving course refinement according to the best combination.

Click the + button next to Refinements and select Refine Area. The settings panel opens.

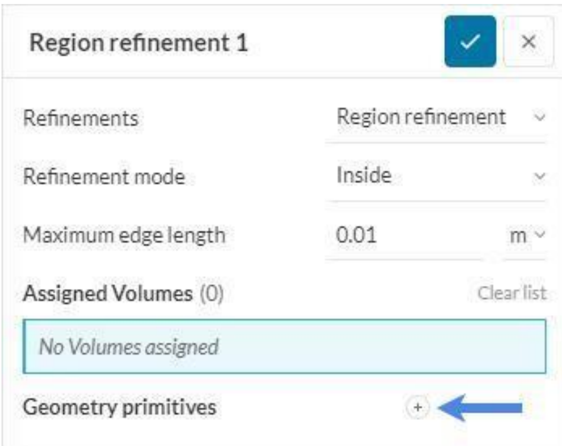


Figure 3.28: Setting up region for region refinement.

Set the maximum edge length to "0.01 m" and click the "+" button to create an "orthogonal box" geometriccircle defining the volume.

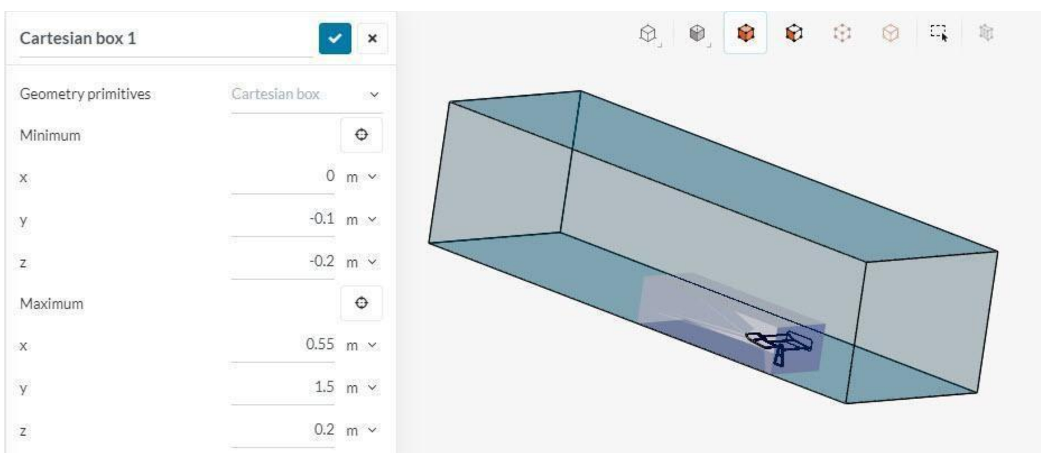


Figure 3.29: Cartesian box set-up

Determine the following dimensions of the Cartesian box in meters, as shown in the picture above.

- Minimum x: '0' m
- Minimum y: '-0.1' m
- Minimum z: '-0.2' m
- Maximum x: '0.55' m
- Maximum y: '1.5' m
- Maximum z: '0.2' m

Saving the Cartesian box settings will redirect you to the area subdivision settings. Assign the Cartesian field we just created. This will give you a maximum edge length of 0.01m for all cells inside this rectangle. Save the settings and return to the Mesh tab of the modeling tree and click New to create the mesh.

After around 30 to 40 minutes, you will see this mesh:

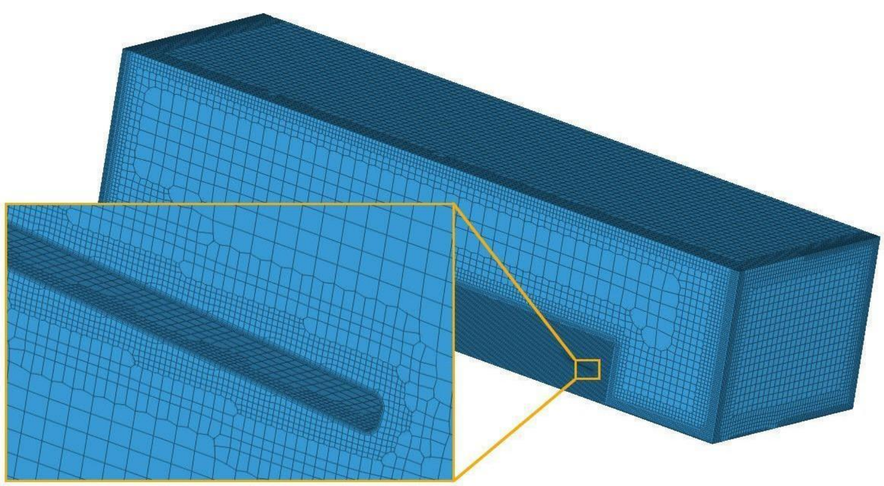


Figure 3.30: This is the mesh created.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

❖ **Start the simulation:**

Now that your simulation setup is complete, you can create a new simulation run. To do this, click the "+" button next to Run Simulation in the Simulation Tree. In the pop-up window, give the run a valid name andclick "Start".

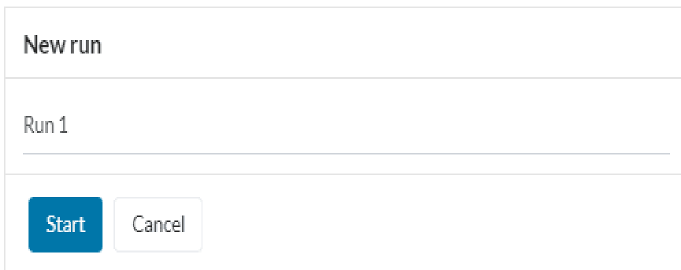


Figure 3.31: Give 'New' and 'Run 1' as simulation name.

RESULTS AND DISCUSSION

The following image shows the velocity magnitude and the colour around the wing profile shows the velocity values.

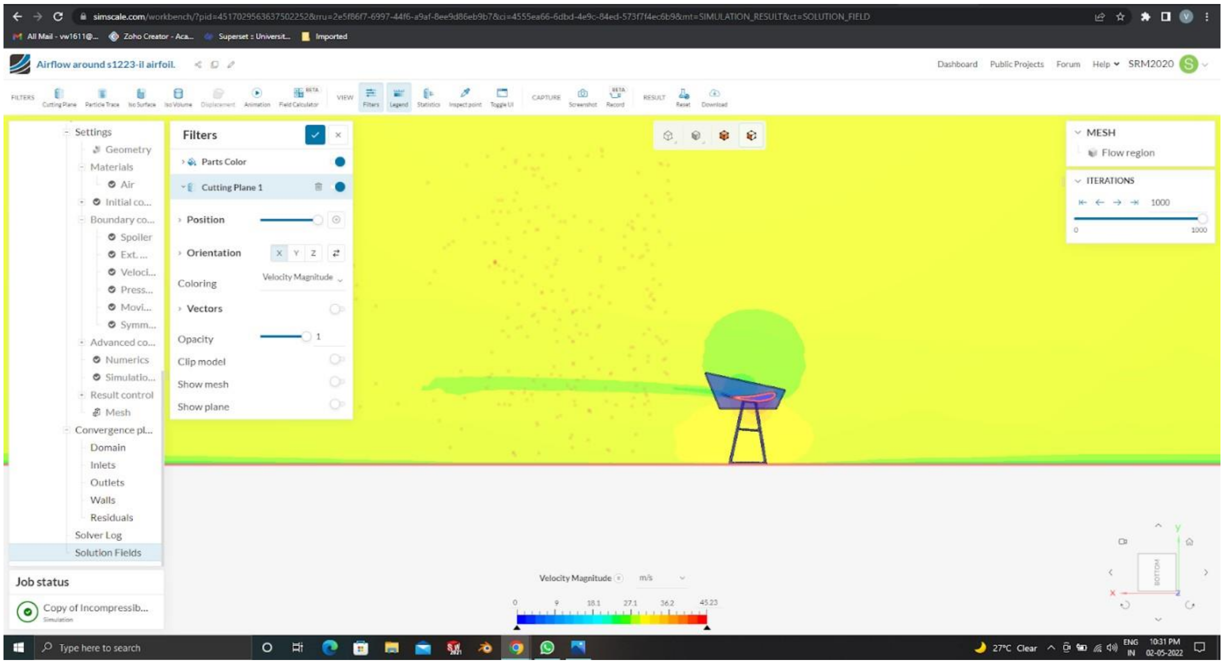


Figure 4.1: Velocity magnitude color graph results.

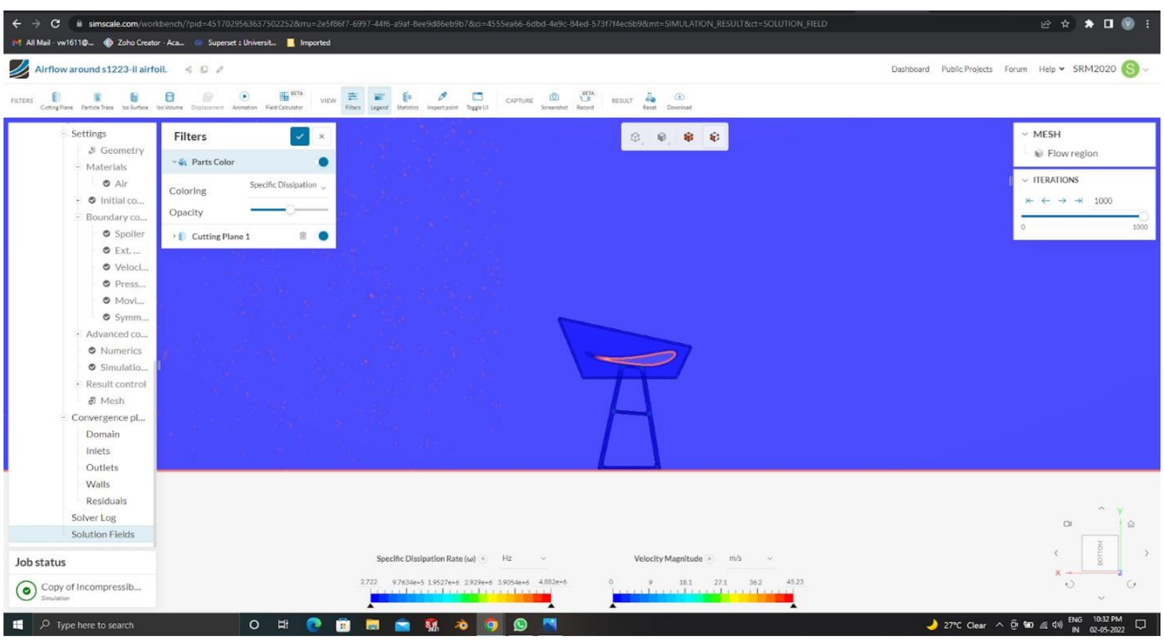


Figure 4.2: Velocity magnitude color graph and Specific Dissipation rate color graph.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

The above image shows the specific dissipation rate and the color shows the values as shown on the scale.

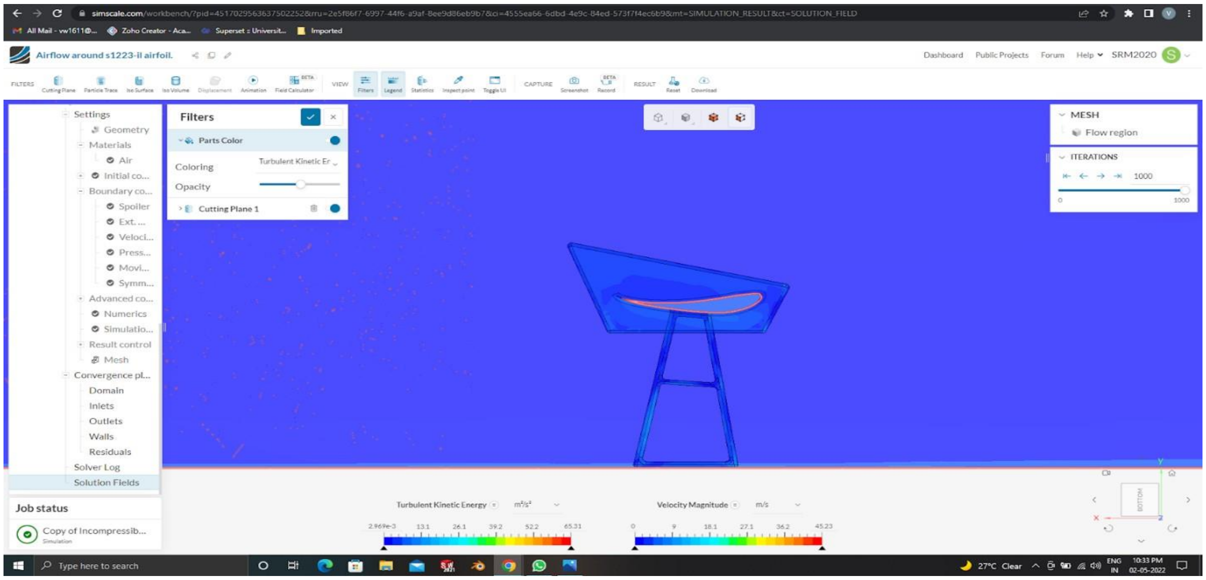


Figure 4.3: Velocity magnitude color plot and Turbulent Kinetic Energy color plot.

The below image shows the turbulent kinematic viscosity and the color shows the values as shown on the scale.

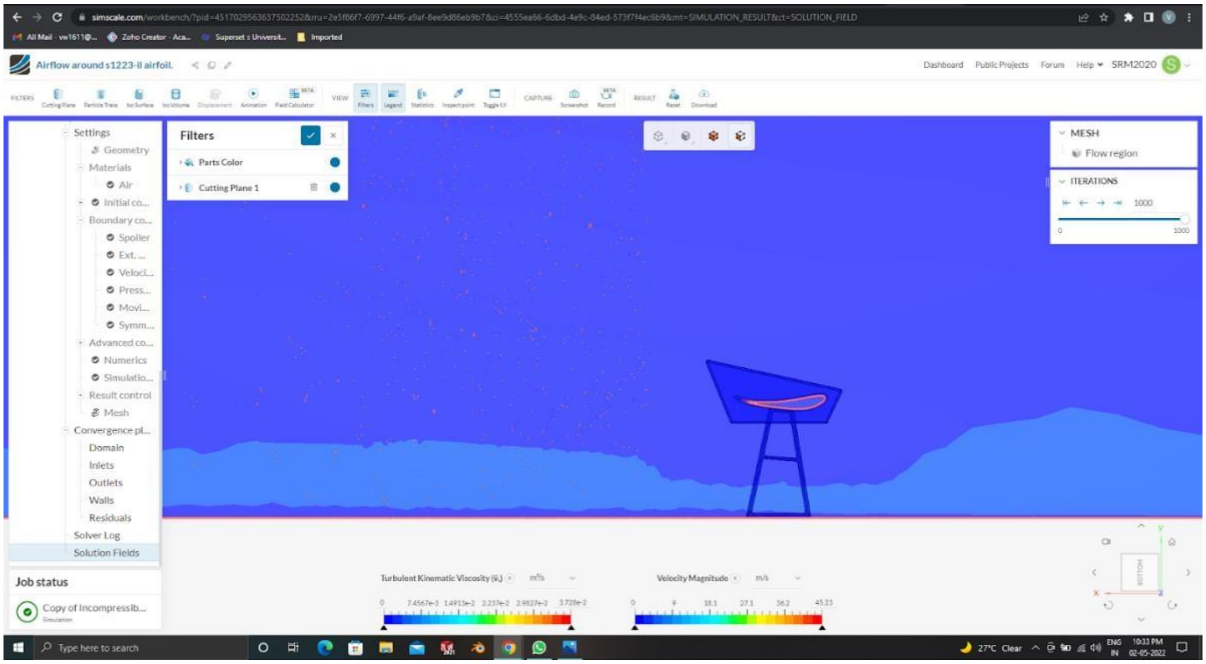


Figure 4.4: Velocity magnitude color plot and Turbulent Kinematic viscosity color plot.

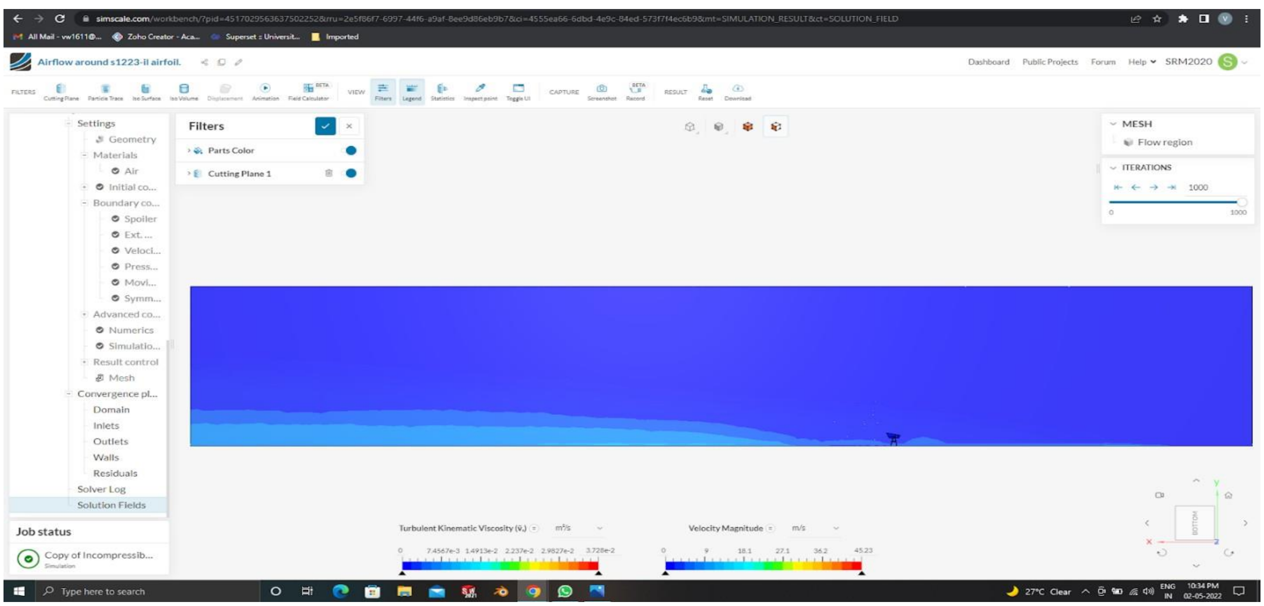


Figure 4.5: Velocity magnitude color plot and Turbulent Kinematic viscosity color plot.

The below image shows the Pressure and the color shows the values as shown on the scale.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

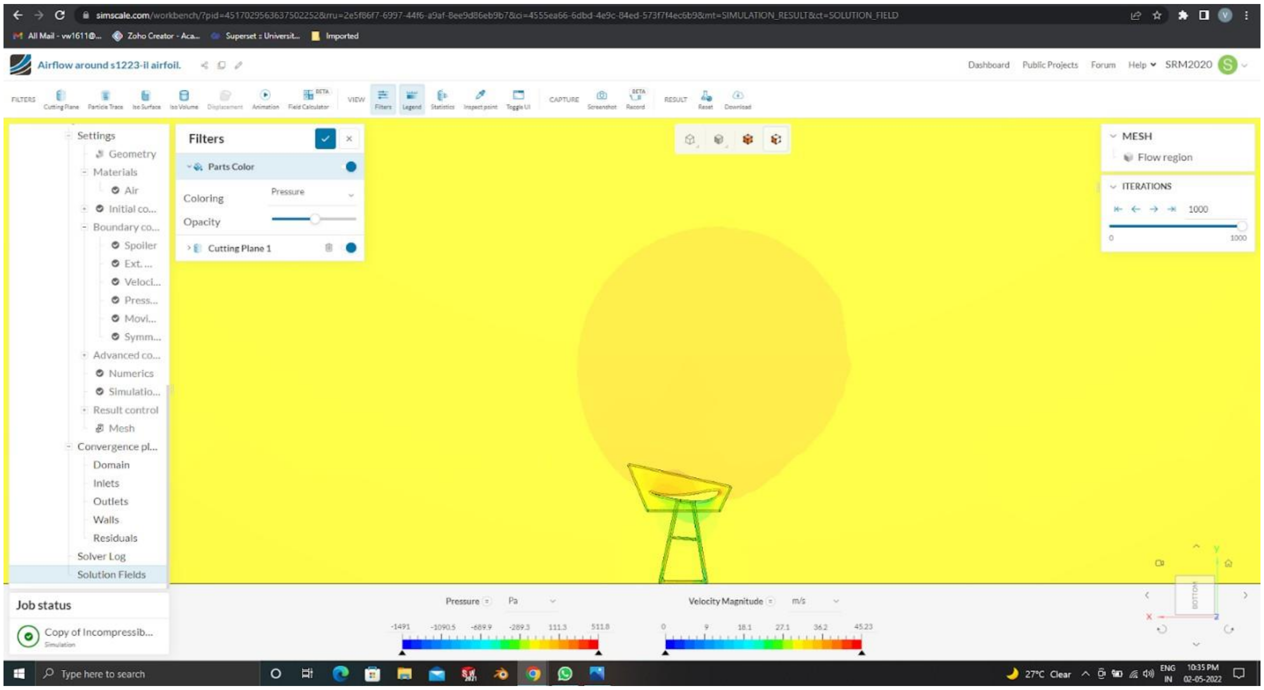


*Figure 4.6: Velocity magnitude color plot and Pressure color plot.*The above image shows the specific dissipation rate and the color shows the values as shown on the scale.

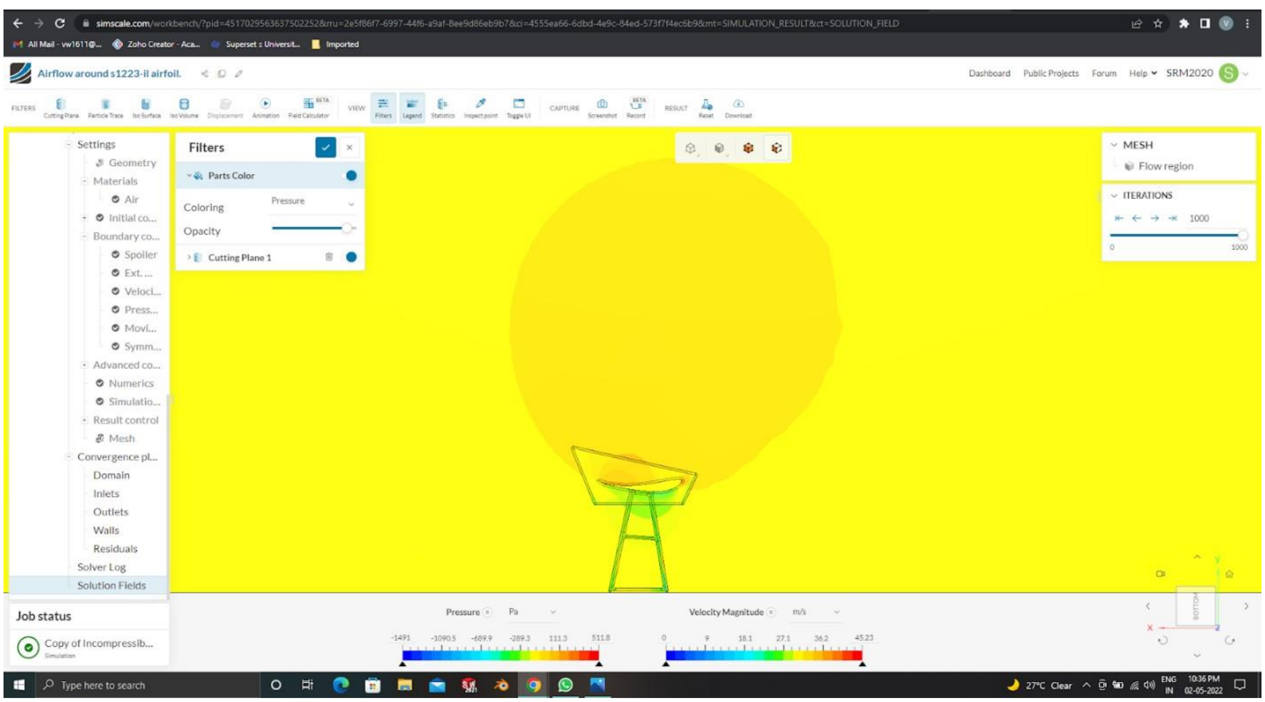
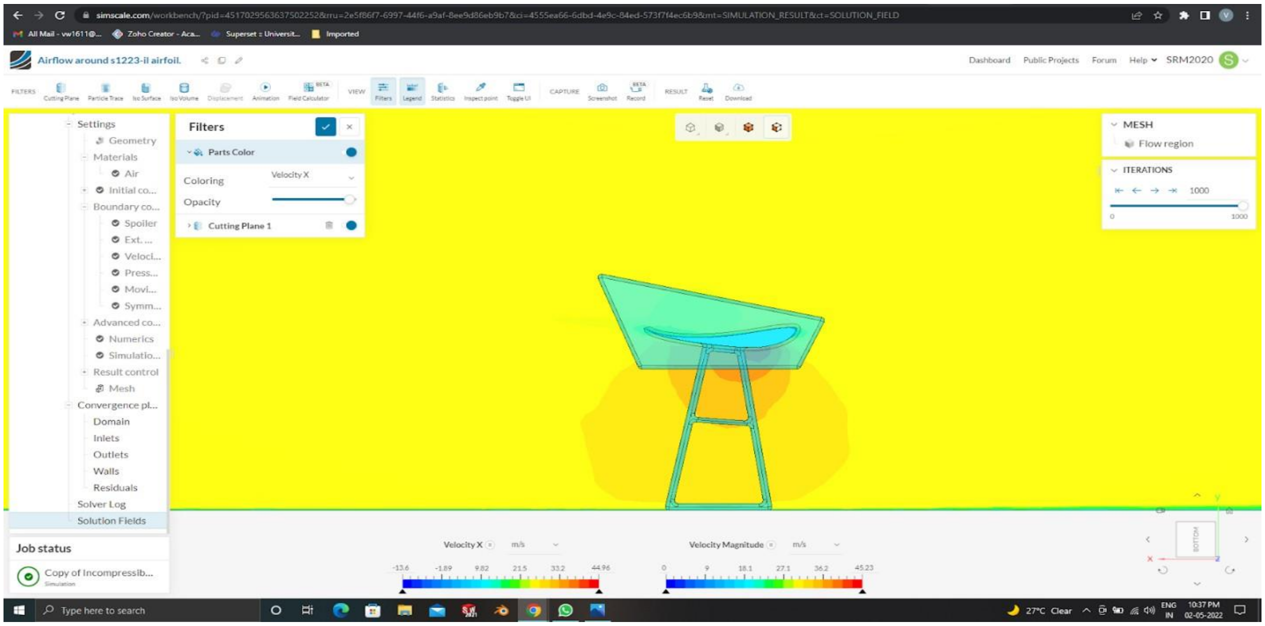


Figure 4.7: Velocity magnitude color plot and Pressure color plot.



The above image shows the Pressure and the color shows the values as shown on the scale.

Figure 4.8: Velocity magnitude color plot and Velocity at X color plot.

We can calculate downforce using this formula:Downforce = 1/2p * A * Cl * V^2.

Where;

- 1/2p - constant or same value of .00119.
- A - This the area of the wing in square feet.
- Cl - This is the Coefficient of Lift.
- V^2 - Velocity or speed in feet per second, squared.

Dynamic pressure (q) = (rho*u^2)/2Where;

- q = dynamic pressure in pascals
- rho = fluid mass density
- u = flow speed in m/sFrontal area of our wing:

0° - 66,138 mm^2

5° - 86,724mm^2

10° - 1,13,880 mm^2

15° - 1,44,540 mm^2

20° - 1,73,010 mm^2

Validation of Results:

Bernoulli's Formula for Relation Between Pressure and Velocity:

The first formula that describes the relationship between pressure and speed is the Bernoulli Principle. The formula states that in thermodynamics or hydrodynamics, an increase in incomparable speed or a non- inviscous liquid is the result of a decrease in the constant pressure applied to the liquid. The formula given to Bernoulli in terms of this principle to explain the relationship between pressure and speed is:

$P + (\rho v2 / 2) + \rho gh = K$

In the formula above, P shows the pressure of a compressed, invisible liquid measured by N / m2 ho means the density of a compressed liquid, measured using Kg / m^2 v means pressure, invisible, measured. liquid. using m / s g represents acceleration due to gravity, measured in m / s2 h means the height from the reference level at which the liquid is contained. Measured in meters (m). In simple terms, Bernoulli's formula describes the relationship between pressure and speed in relation. It means that when the pressure increases, the speed decreases, which keeps the algebraic value of potential energy, kinetic energy, and pressure unchanged. In the same way, when the speed increases, the pressure decreases. Bernoulli's principle of the relationship between pressure and speed can be applied to many types of fluid flows. But it has to be the other way around. Bernoulli's simple calculation method only works in the flow of undisturbed and invisible liquid.

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

❖ Software Simulation results:

We calculated the downforce on the wing at 5 different angles of attack (α). The angles are 0, 5, 10, 15, 20 deg.

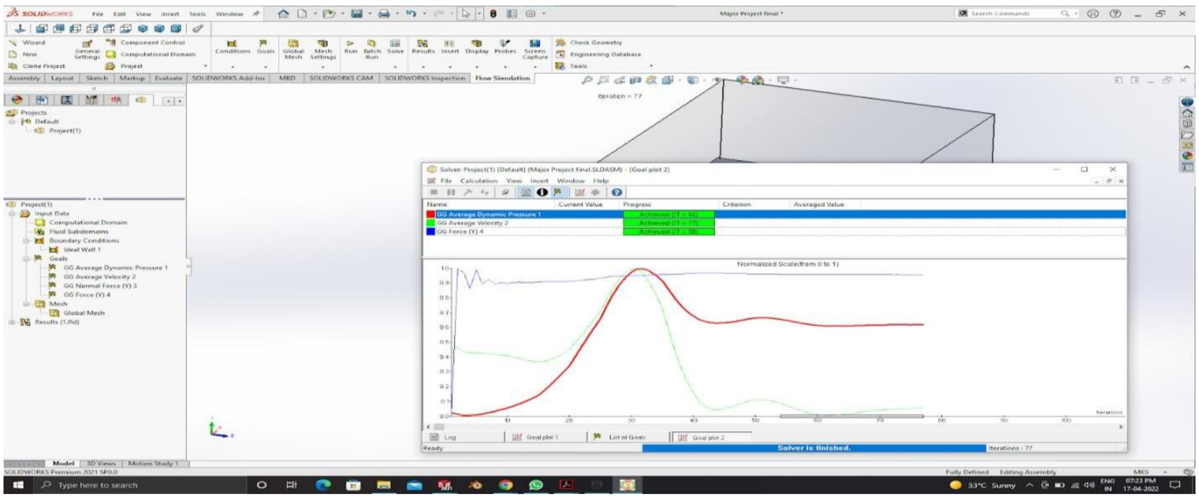


Figure 4.9: Result at 0

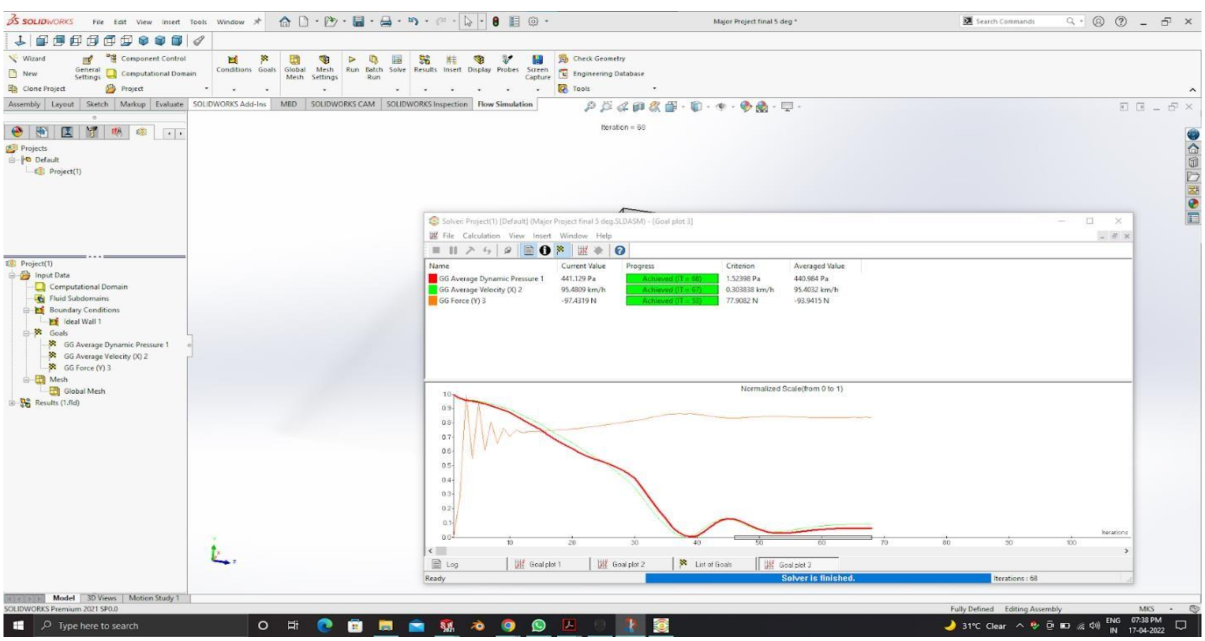


Figure 4.10: Result at 5°

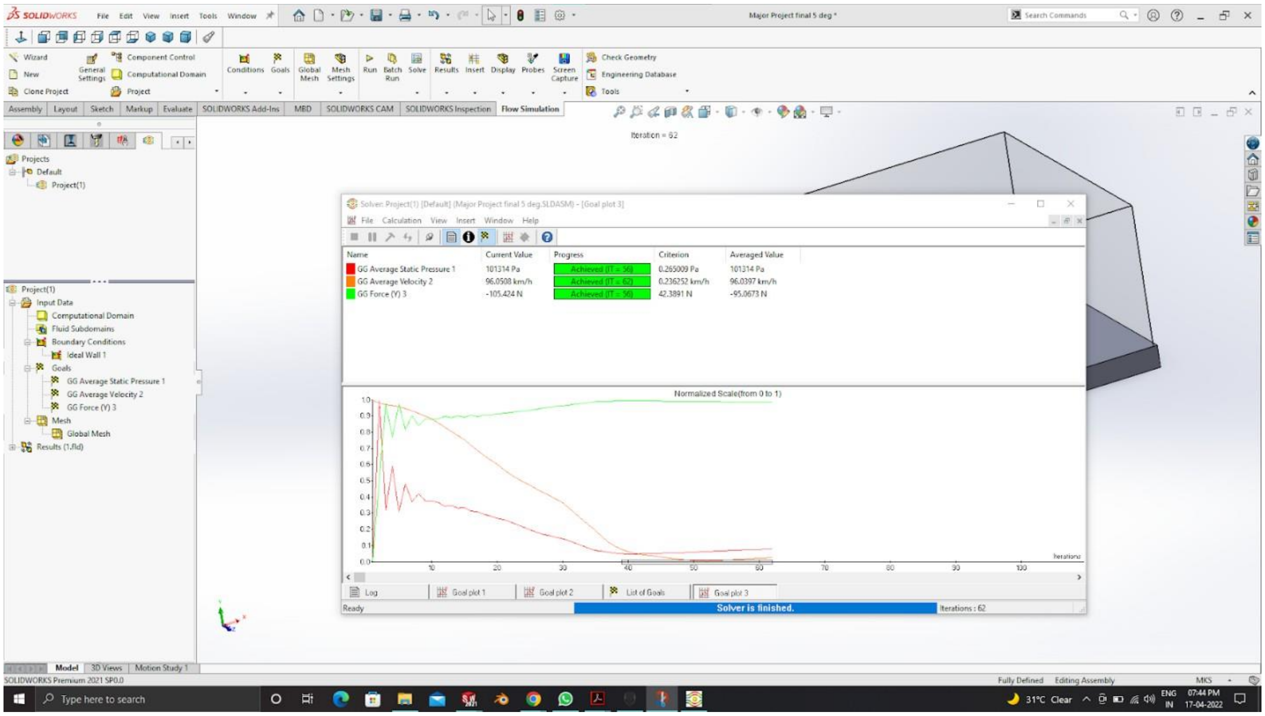
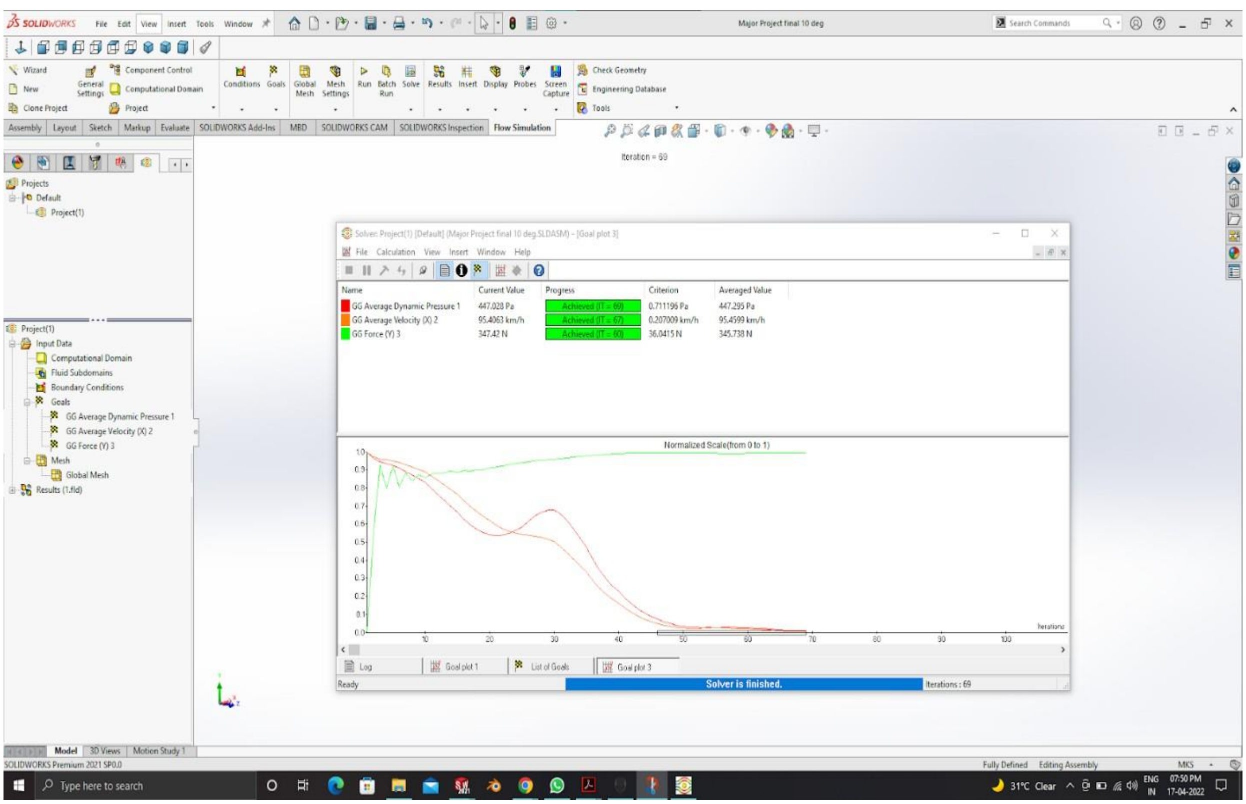


Figure 4.11: Result at 10



International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 01 | January - 2023 Impact Factor: 7.185 ISSN: 2582-3930

Figure 4.12: Result at 15°

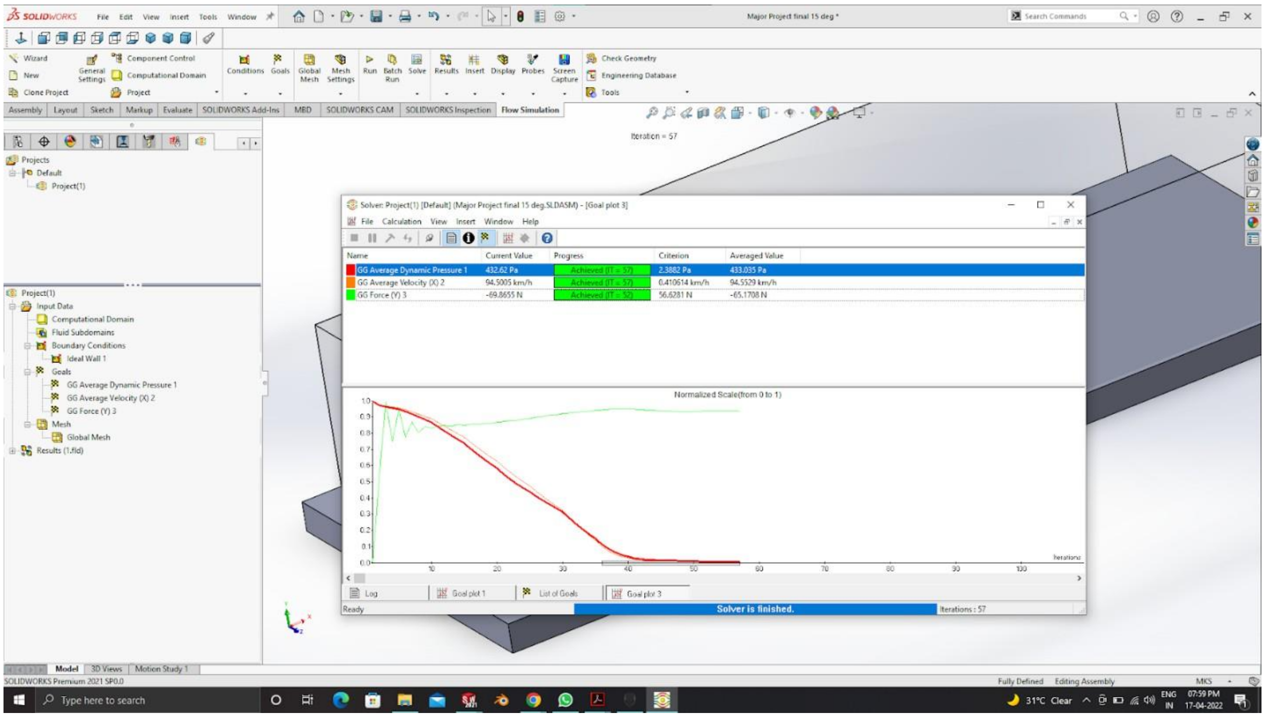


Figure 4.13: Result at 20°

CONCLUSIONS

After calculating with and without aero kits we came to know that we have got more efficient result. The lift has reduced which means that the down force has increased which helps to give better grip while cornering and on straight path. We have also got the simulation result of velocity flow, pressure and temperature. These helped us to know about the temperature and pressure are connected to each other, in each part of the vehicle. The tabular columns and graph show the result at each iteration and we take the value when the graph become stable. We got the result that the drag force has decreased and down force have increased. We got efficient result because we spent more time to model the aero kits which helped us to get more downforce without effecting other drag and weight to the vehicle. The simulation study and efficiency in simulation is a very vast topic and this project can be further enhanced and studied to increase the effective aerodynamics.

SCOPE OF WORK

Simulation studies are very broad and vast to perform because there are 'n' types of combination one can perform in this by giving different simulation calculation entities. This project can be further expanded on giving the car cornering wind tunnels and turbulent straights and corners and they can study further more ahead on this. This always remains open to perform more simulation studies with 'n' types of conditions.

REFERENCES

[1] Prabhu, L., Krishnamoorthi, S., Gokul, P., Sushan, N., Harshed, P., S., H., and Jose, A. (2020) "Aerodynamics analysis of the car using Solidworks flow simulation with rear spoiler using CFD", ICMECE2020, IOP Publishing, IOP Conf. Series: Materials Science and Engineering 993 - 012002

[2] Keogh, K., Doig, G., Tracie, J., Barber, Diasinos, S.(2014) "The Aerodynamics of a Cornering InvertedWing in Ground Effect", Applied Mechanics and Materials Vol. 553 (2014) pp 205-210

[3] Unmukt, B., R. (2014) "Formulae 1 racecar performance improvement by optimization of the aerodynamic relationship between the front and rear wings"

[4] Agathangelou, B. and Gascoyne, M. (1998) "Aerodynamic Design Considerations of a Formula 1 RacingCar", SAE Technical Paper-980399

[5] Zhang, X., Toet , W., and Zerihan, J.(2006) "Ground effect aerodynamics of race cars", AutoTechnologyJournal, Volume 6, pages 40–43

[6] Toet, W.(2013) "Aerodynamics and aerodynamic research in Formula 1", The Aeronautical Journal 117pp. 1-26

[7] Dominy, R.(1992) "Aerodynamics of Grand Prix Cars", Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, pg 267-274.

[8] F1 Nutter, "The Aerodynamic Development of Formula One Car"

[9] Katz, J.(2006) "Race Car Aerodynamics", Annu. Rev. Fluid Mech 38:27–6337

[10] Muzzupappa, M.; Pagnotta, L.(2002) "Optimum Design of a Formula 1 RearWing using a

GeneticAlgorithm", International Design Conference - Design (pp. 1217- 1222). Dubrovnik.

[11] Katz, J.(1995) "High Lift Wing Design for Race-Car Applications," SAE Technical Paper-951976

[12] Katz, J. and Plotkin, A.(1991) "Low-Speed Aerodynamics From Wing Theory to Panel Methods",
McGraw-Hill, Inc., 90-38041

[13] Sriram, P., Gopalarathnam, A., and Misenheimer, A.(2012) "High-Downforce Airfoil Design for Motorsports," SAE Int. J. Mater. Manf. 5(2):478-489

[14] William, G. , Bastedo , Jr. , and Thomas, J. , Mueller (1986) "Spanwise Variation of Laminar Separation Bubbles on Wings at Low Reynolds Numbers", Journal of Aircraft, Vol. 23, No. 9, pp 687 – 694

[15] Abbott , Ira, H., and Doenhoff, V., Albert, E. (1959) "Theory of wing Sections",Library 01 Congress Catalog Card Number: 60-1601

[16] Bangash, Z. A., Sanchez, R. P. and Ahmed, A.(2006) "Aerodynamics of Formation Flight", Journal ofAircraft, 43(4), 907-912.

[17] Devaiah, B., N., S., Umesh (2013) "Enhancement of aerodynamic performance of a F1 car using add- on devices", SASTECH Journal, Volume 12, Issue 1

[18] CD-Adapco , STAR-CCM+ User guide , Version 9.06, 2015.

[19] Versteeg , H. , K. , and Malalasekera , W.(2007) "An introduction to computational fluid dynamics : the finite volume method", Prentice Hall, Harlow, 2nd edition GBA649081 bnb. Uk Includes bibliographical references (p. [472]-494) and index.

[20] Krzyszt of Kurec , Remer, M., Broniszewski, J., Bibik, P., Tudruj, S., and Piechna, J. (2019)

"Advanced Modeling and Simulation of Vehicle Active Aerodynamic Safety", Journal of Advanced Transportation Volume , Article ID 7308590

[21] Genta, G. (1997) "Motor vehicle dynamics", River Edge, NJ : World Scientific, (DLC) 9605102038

[22] Diasinos, S., Barber, T., J., and Doig, G. (2013) "Influence of wing span on the aerodynamics of wings in ground effect", Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering 227 (3) pp. 569-573

[23] Jean-Francois, Beaudoin, Jean-Luc Aider (2008), "Springer-Verlag 2008 Drag and lift reduction of a 3D bluff body using flaps", Springer-Verlag, Exp Fluids 44:491–501

[24] S., P., S. , M., H., Kabir, and M. M. M. Talukder (2015) "Proceedings of the International Conference on Mechanical Engineering and Renewable Energy", (ICMERE2015) Aerodynamic analysis of a concept car model

[25] Kumar, A., Vashist, P.(2020) "Experimental Study of Race Car Aerofoil", September 2020 IJSDR | Volume 5 Issue 939

Format - I

SRM INSTITUTE OF SCIENCE AND TECHNOLOGY

(Deemed to be University u/s 3 of UGC Act, 1956)

Office of Controller of Examinations

REPORT FOR PLAGIARISM CHECK ON THE DISSERTATION/PROJECT REPORTS FOR UG/PG PROGRAMMES

(To be attached in the dissertation/ project report)

1	Name of the Candidate **(IN BLOCK LETTERS)**	VIVEK WAGH. ALWIN THOMAS.
2	Address of the Candidate	DEPARTMENT OF MECHANICAL ENGG., SRMIST, KATTANKULATHUR **Mobile Number :** 9999799999
3	Registration Number	RA1811002010213, RA1811002010208
4	Date of Birth	04/03/2000, 22/11/2000
5	Department	MECHANICAL ENGINEERING
6	Faculty	FACULTY OF ENGINEERING & TECHNOLOGY
7	Title of the Dissertation/Project	EXPERIMENTAL & OPTIMIZATION ANALYSIS OF RACECAR DESIGN.
8	Whether the above project/dissertation is done by	~~Individual~~ or group : (Strike whichever is not applicable) a) If the project/ dissertation is done in group, then how many students together completed the project : 2 b) Mention the Name & Register number of other candidates : RA1811002010208, RA1811002010213
9	Name and address of the Supervisor / Guide	Dr. M. Gunasekaran, ASSISTANT PROFESSOR DEPARTMENT OF MECHANICAL ENGG. gunasekm@srmist.edu.in **Mail ID : Mobile Number :**
10	Name and address of the Co-Supervisor / Co- Guide (if any)	N.A. **Mail ID : Mobile Number :**

11	Software Used	TURN IT IN		
12	Date of Verification	11 / 05 / 2000		
13	**Plagiarism Details: (to attach the final report from the software)**			
Chapter	**Title of the Chapter**	**Percentage of similarity index (including self citation)**	**Percentage of similarity index (Excluding self citation)**	**% of plagiarism after excluding Quotes, Bibliography, etc.,**
1	INTRODUCTION	<2%	—	<2%
2	LITERATURE SURVEY	<2%	—	<2%
3	EXPERIMENTAL INVESTIGATION	<2%	—	<2%
4	RESULTS & DISCUSSIONS	<1%	—	<1%
5	CONCLUSION	<1%	—	<1%
6				
7				
8				
9				
10				
	Appendices			

I / We declare that the above information have been verified and found true to the best of my / our knowledge.

Signature of the Candidate	11/5/22 **Name & Signature of the Staff (Who uses the plagiarism check software)**
Name & Signature of the Supervisor/Guide	**Name & Signature of the Co-Supervisor/Co-Guide**

Name & Signature of the HOD

11/5/2022

Dept. of Mechanical Engineering
Kattankulathur 603 203
SRM IST