AERODYNAMICS OF FASE CAR – CFD PROJECT

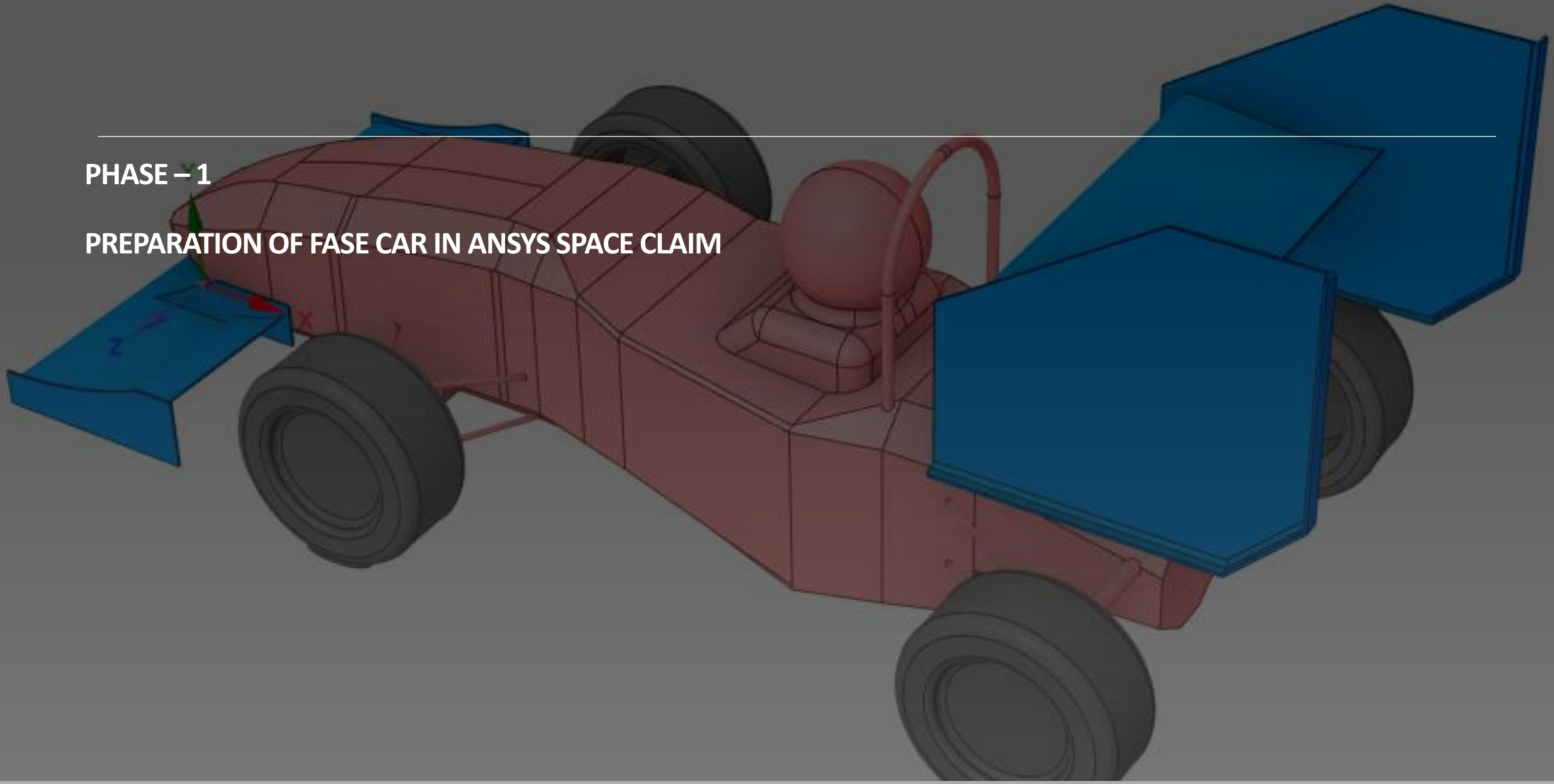
SINGLE WING

GUIDED BY: PROF .DR. MEHMET SARIMURAT

-JAYAPRAKASH CHANDRAN
522242685

PHASE – 1

PREPARATION OF FASE CAR IN ANSYS SPACE CLAIM



In this module, we will compare the previous model with the modified model featuring a single wing. Through this comparison, we aim to understand and analyze the airflow patterns and behaviors associated with these two designs. By examining the flow characteristics, we can gain insights into how the modifications introduced by the single wing affect the overall performance of the car.

Enclosure: Create an enclosure around the car that is typically three times larger than the car itself. The larger enclosure enables accurate testing in wind tunnels or virtual CFD simulations.

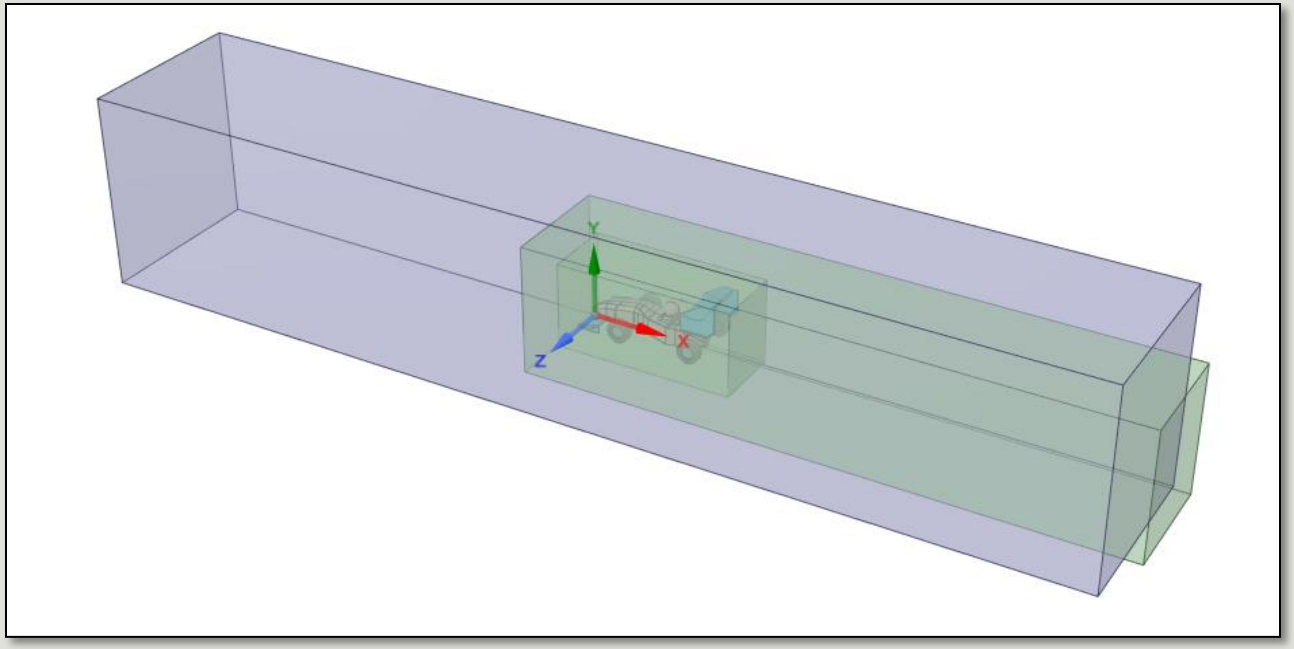


FIG. A. Enclosure and BOI

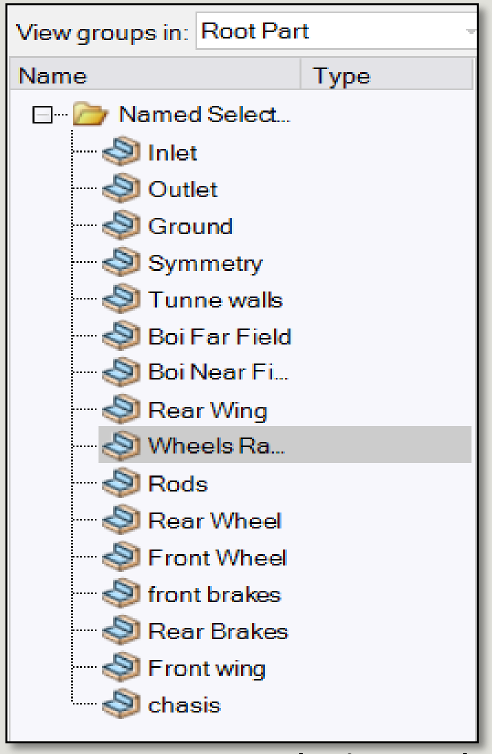


FIG. A2. Name selection panel

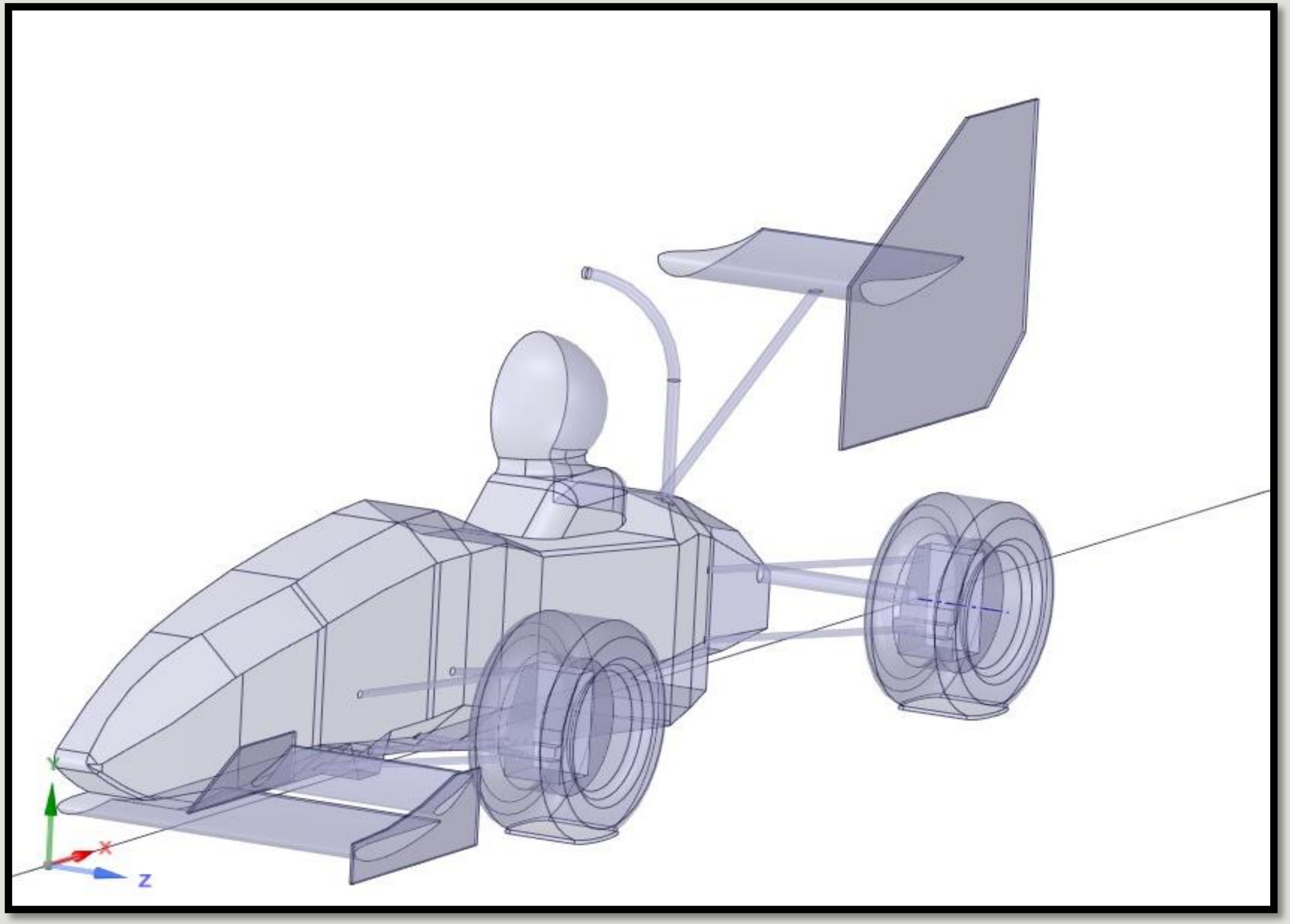


FIG. A3. Geometry

Body of Influence: Identify the portion of the car model that has the most significant impact on aerodynamics, known as the body of influence. Refining this specific region by controlling the mesh size allows for more precise results near the car. This refinement captures important flow characteristics and aerodynamic forces acting on the car's body.

Boundary Conditions and Meshing: Define the boundary conditions before generating the mesh. Boundary conditions involve specifying parameters such as airflow velocity, temperature, pressure, and any constraints or interactions with the environment surrounding the car. Once the boundary conditions are set, the car model is divided into small elements or cells, forming a mesh that facilitates fluid dynamics calculations.

PHASE – 2

TO CREATE THE COMPUTATIONAL MESH

LOCAL SIZING - CONDITIONS

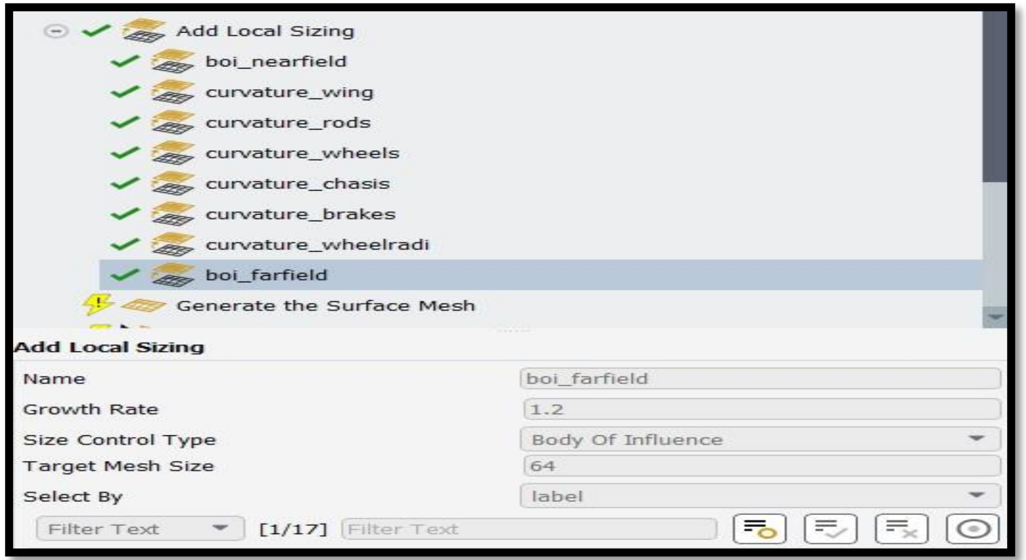


FIG. B1 – Bio far-field

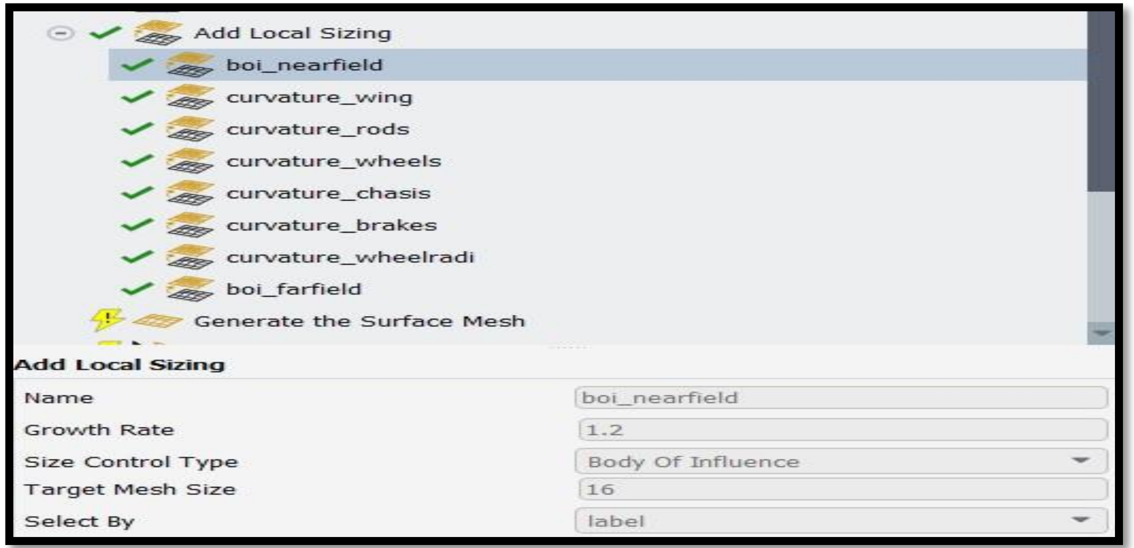


FIG. B2 – Bio Near-field

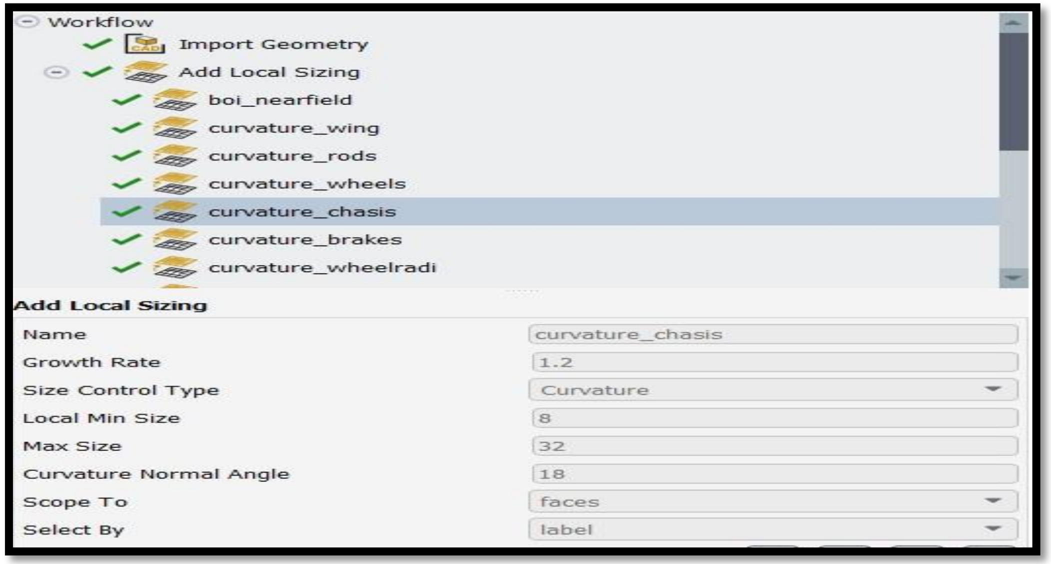


Fig.B3 – Curvature Chassis

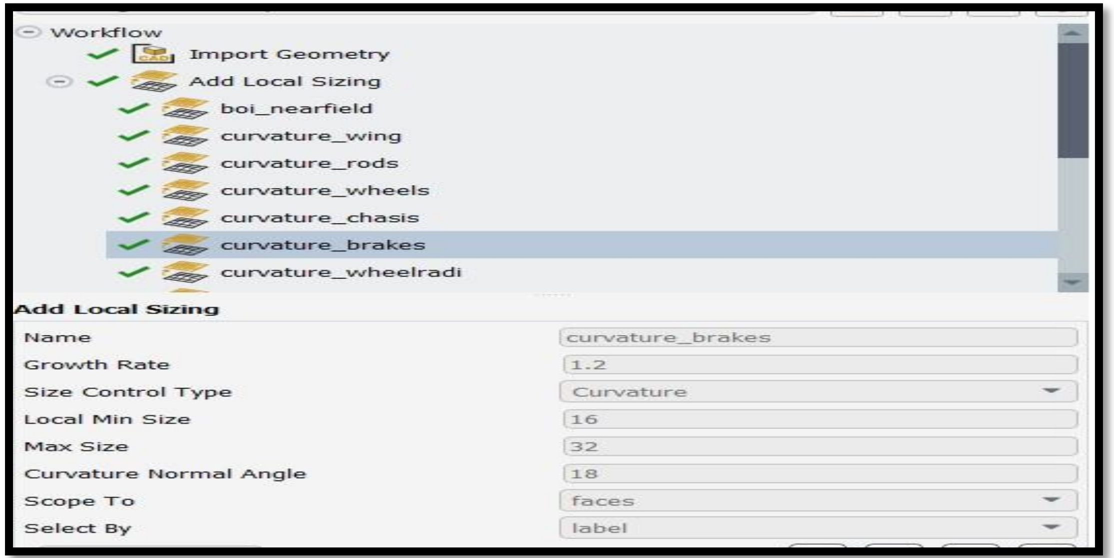


Fig.B4 – curvature brakes

LOCAL SIZING – CONDITIONS (Cont.)

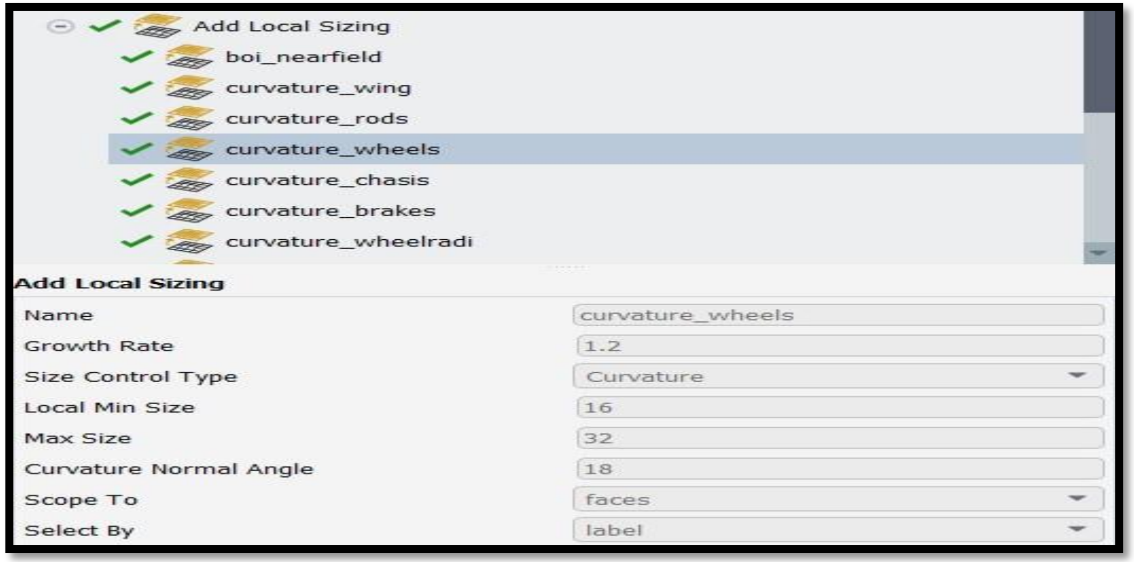


FIG. B5 – Curvature wheels

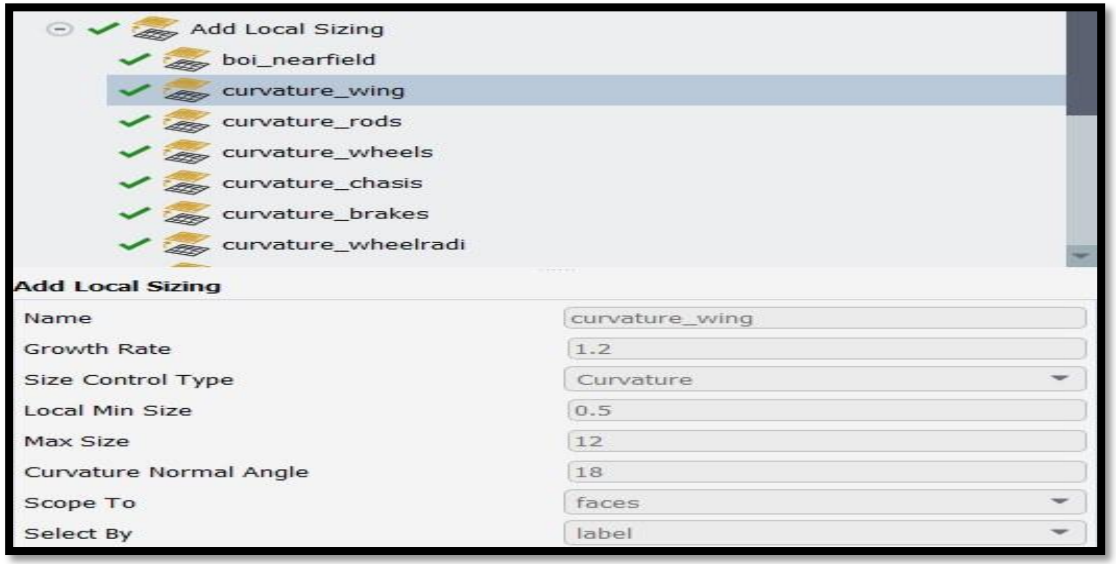


FIG. B6 – Curvature wings

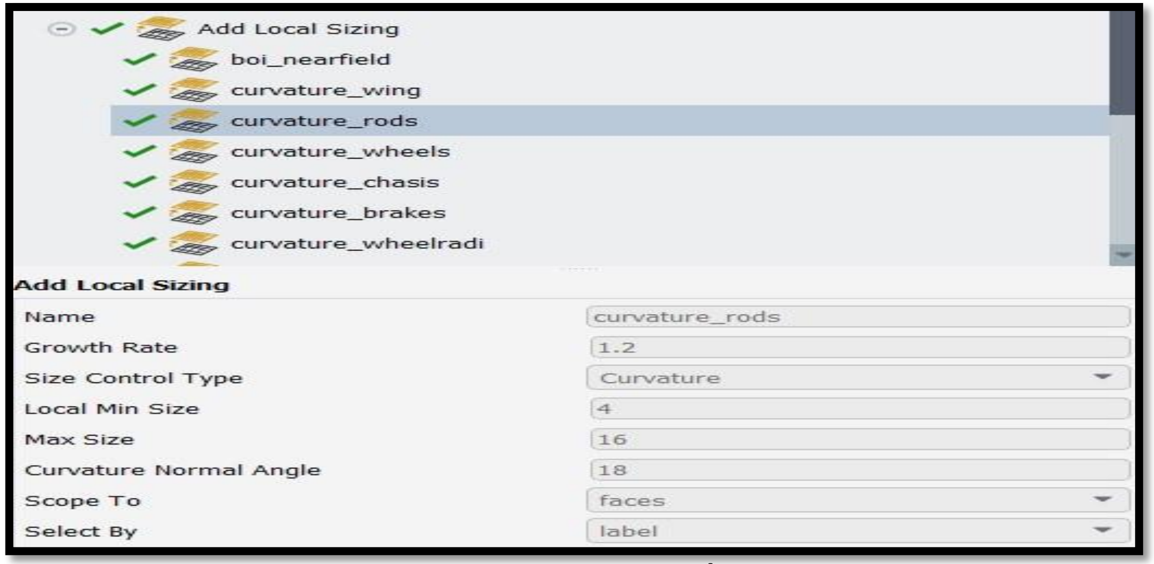


FIG. B7 – curvature rods

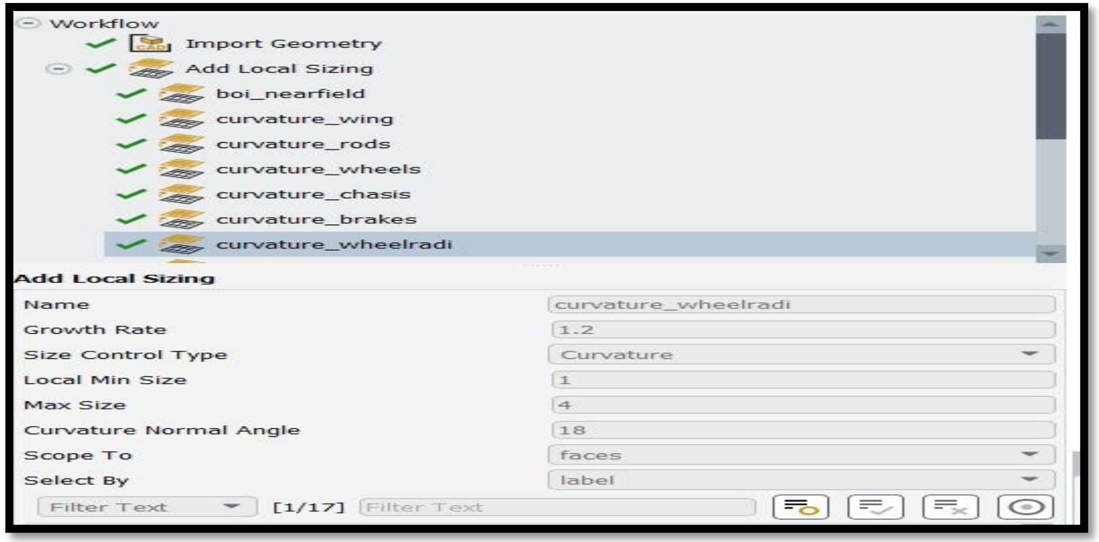
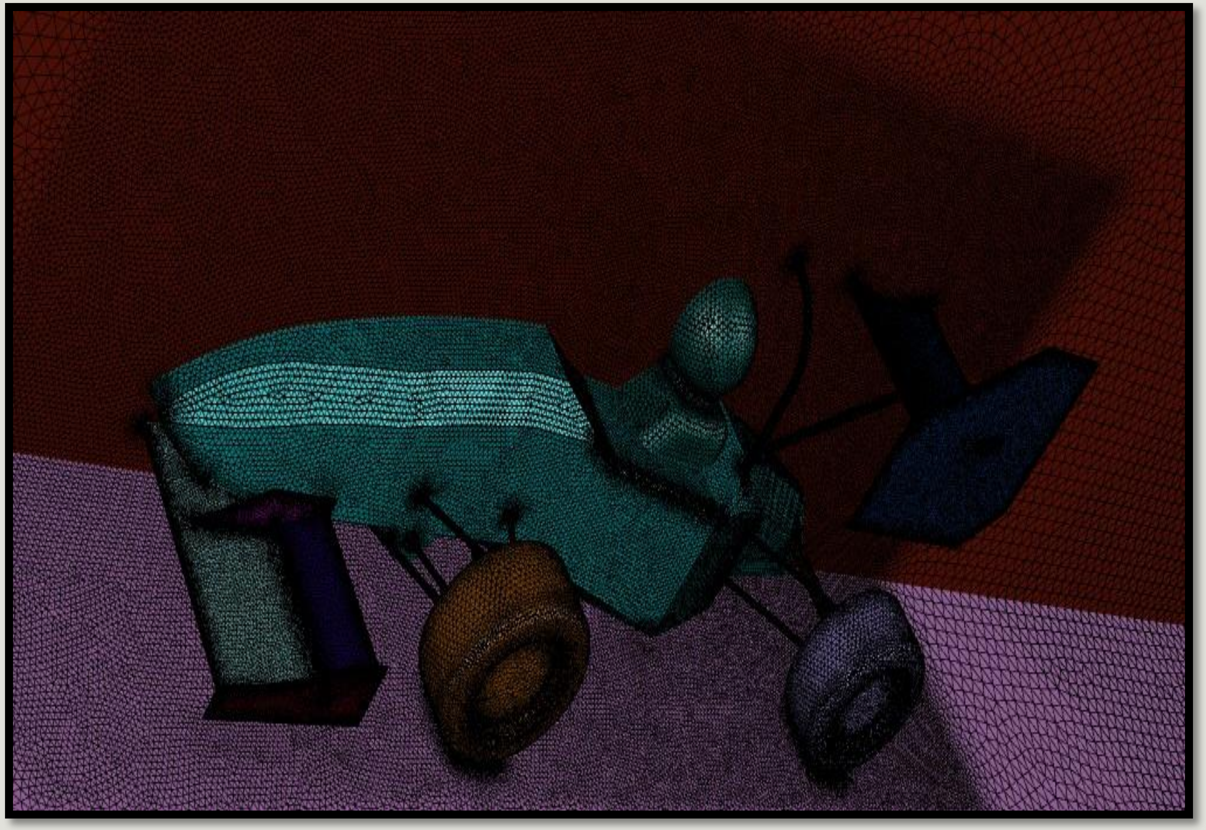


FIG. B8 – Curvature wheel radii

SURFACE MESH

- In order to analyze the flow of the FSAE Car, we utilize Ansys Fluent Meshing to create specific properties for mesh generation. These properties include the surface mesh, boundary layer mesh, and poly hexacore volume mesh.

- To ensure accurate meshing, we apply local sizing to the body of influence and curvature. This allows us to focus on analyzing the flow in the wake region and capture detailed information in these areas. Additionally, we maintain desired settings for the surface mesh to ensure its quality and resolution.



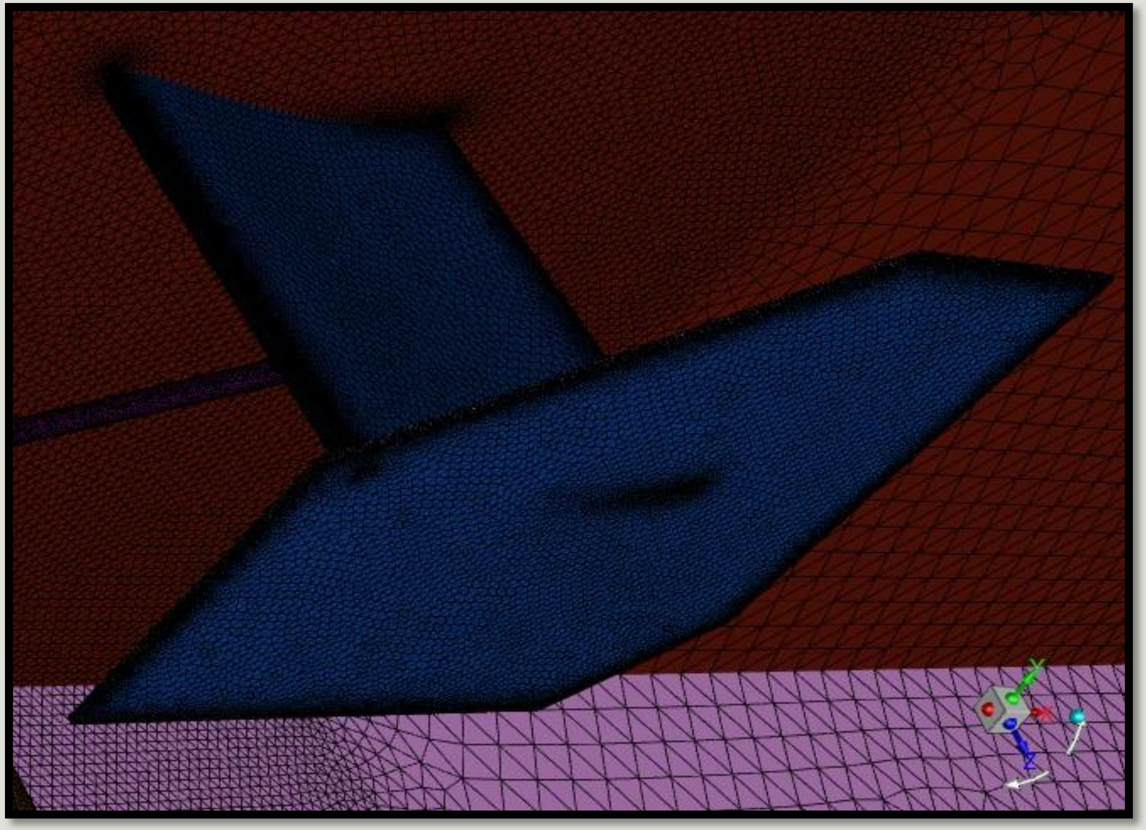
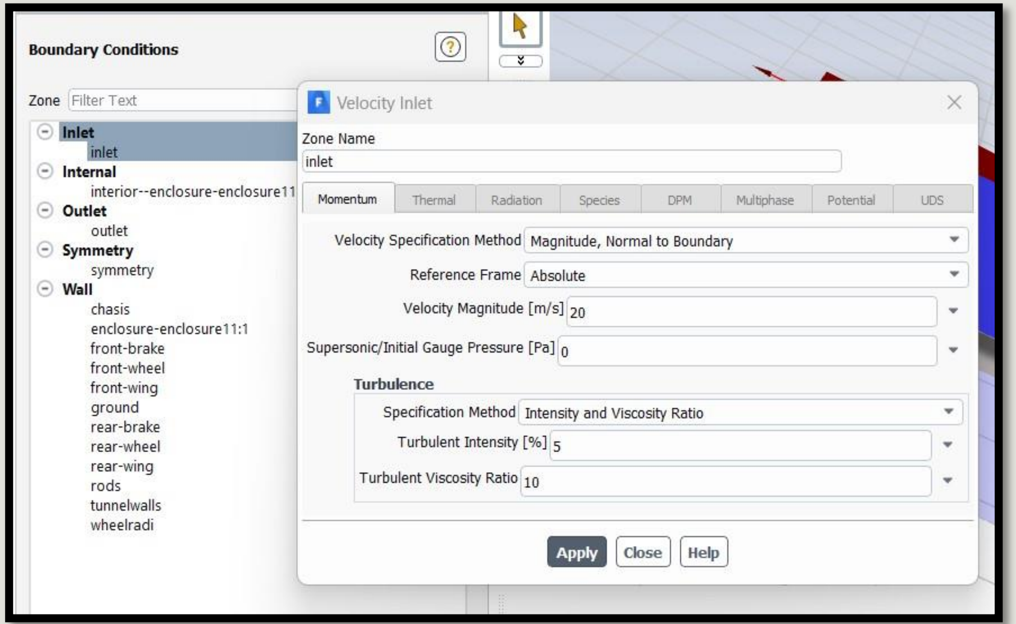
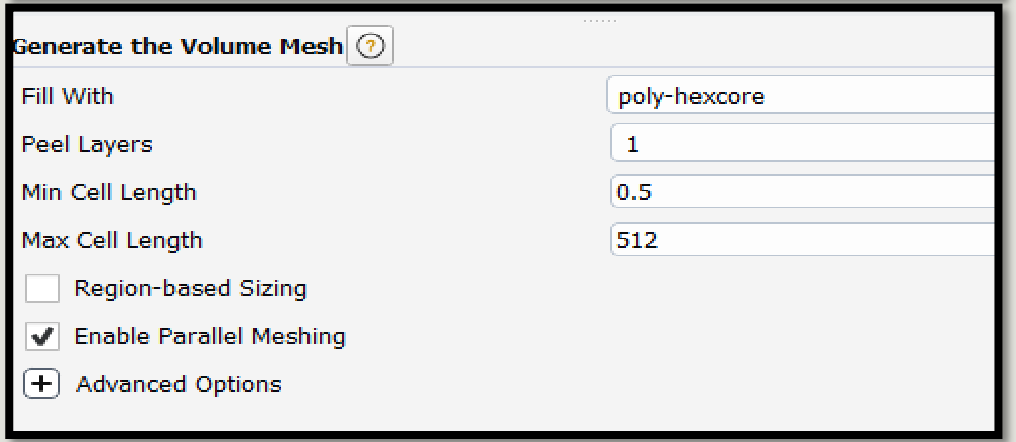


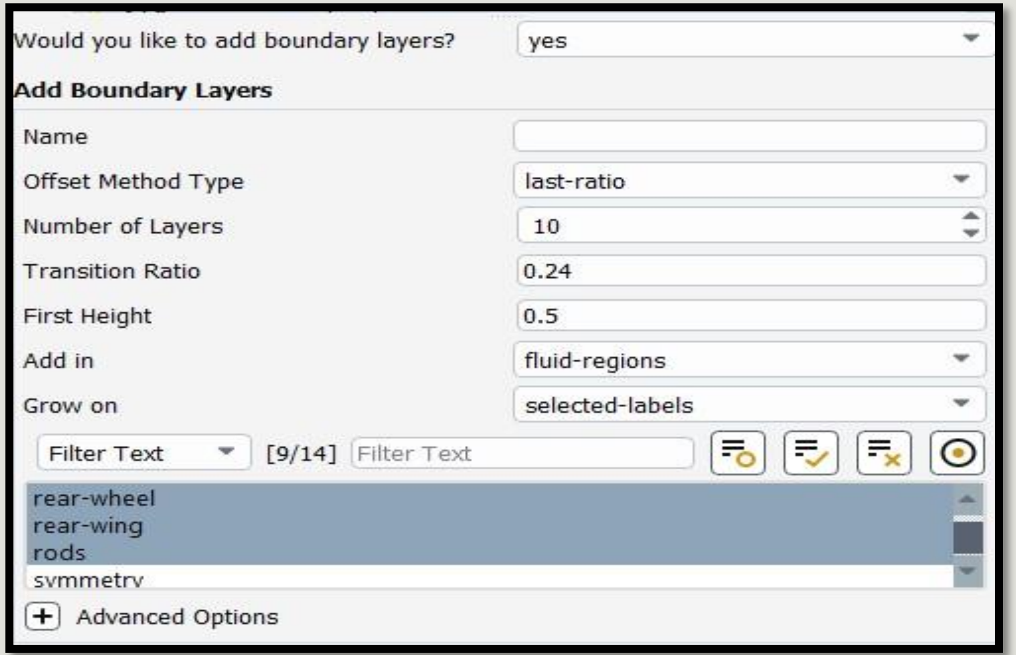
FIG. B9 – SURFACE MESH – SINGLE WING

BOUNDARY LAYER

- Boundary layer conditions play a crucial role in the analysis of the flow around the FSAE car. These conditions are typically defined to capture the characteristics of the boundary layer, which is the thin layer of fluid adjacent to the car's surface.







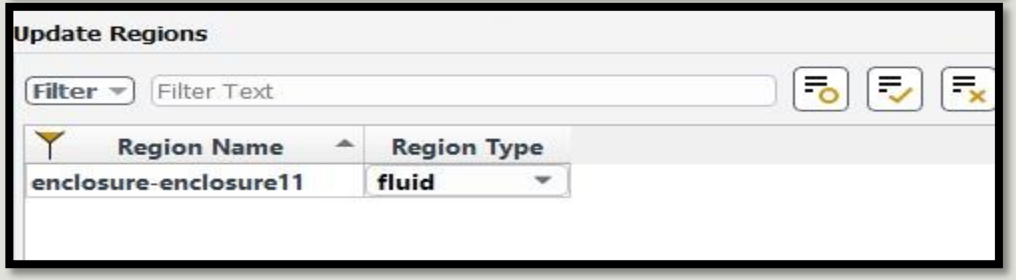


FIG. B10 – BOUNDARY CONDITIONS

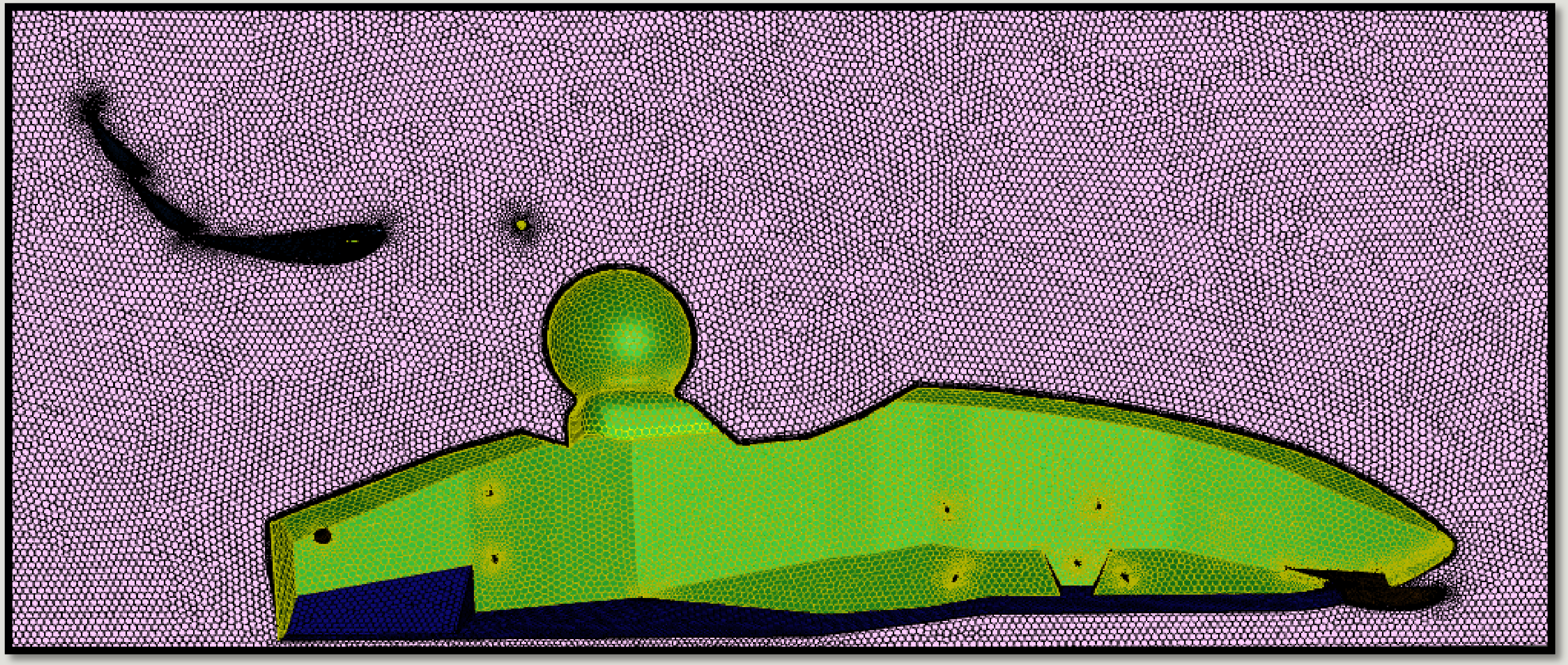


FIG. B11. volume mesh for full wing

```
Console
[Quality Measure : Inverse Orthogonal Quality]

---------------- 4361142 cells were created in :  4.17 minutes

---------------- The mesh has a minimum Orthogonal Quality of:  0.07

---------------- The volume meshing of enclosure-enclosure11 is complete.

zone id: 114, name: symmetry, type: symmetry, count: 38452
zone id: 124, name: chasis, type: wall, count: 23168
```

FIG. B12. Orthogonal Quality

```
Mesh Quality:

Minimum Orthogonal Quality =  1.01262e-01
(To improve Orthogonal quality , use "Inverse Orthogonal Quality"
 where Inverse Orthogonal Quality = 1 - Orthogonal Quality)

Maximum Aspect Ratio =  8.11377e+01
```

FIG. B13. Mesh Quality

Since our meshing value is greater than 1.0, our mesh quality is well defined under the volume mesh conditions.

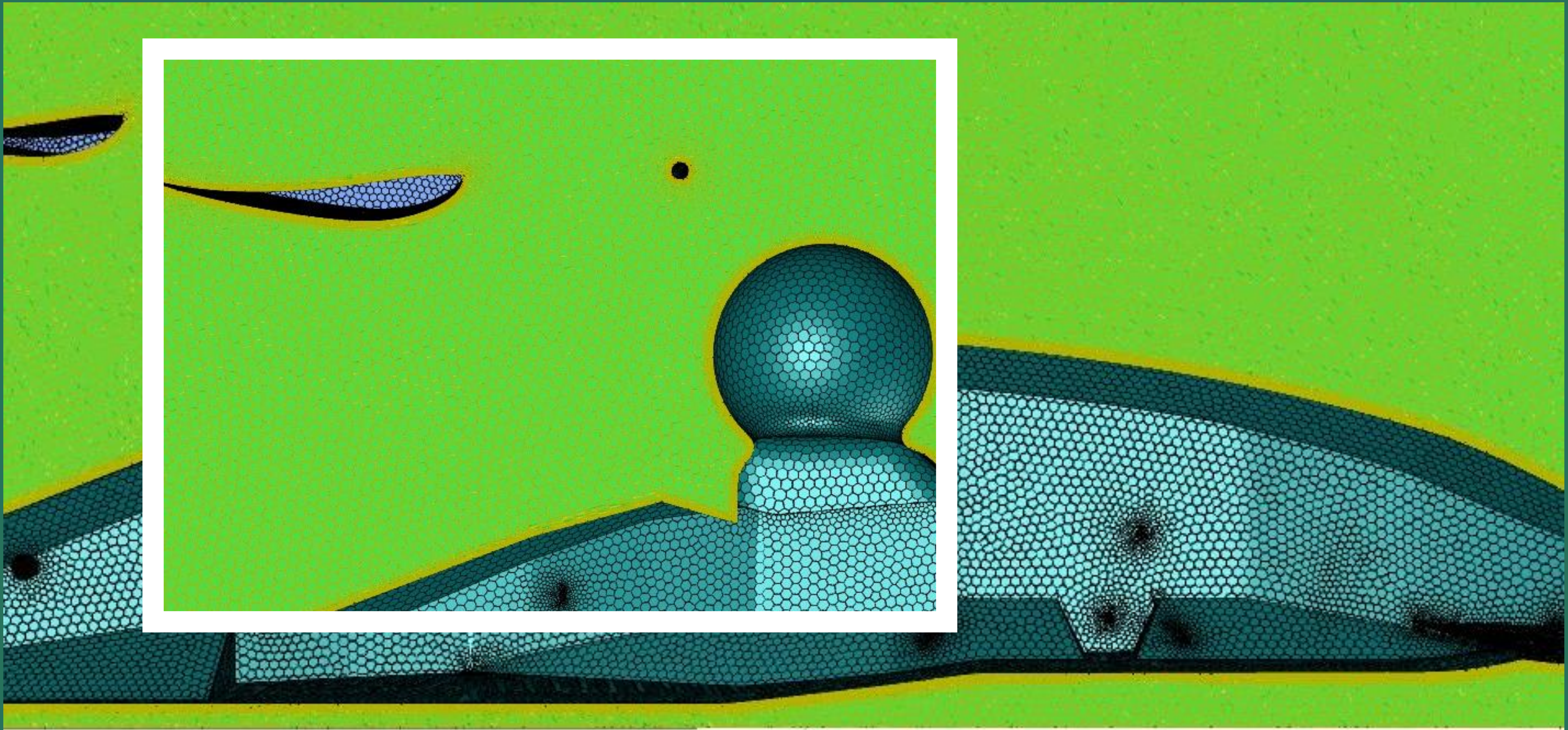
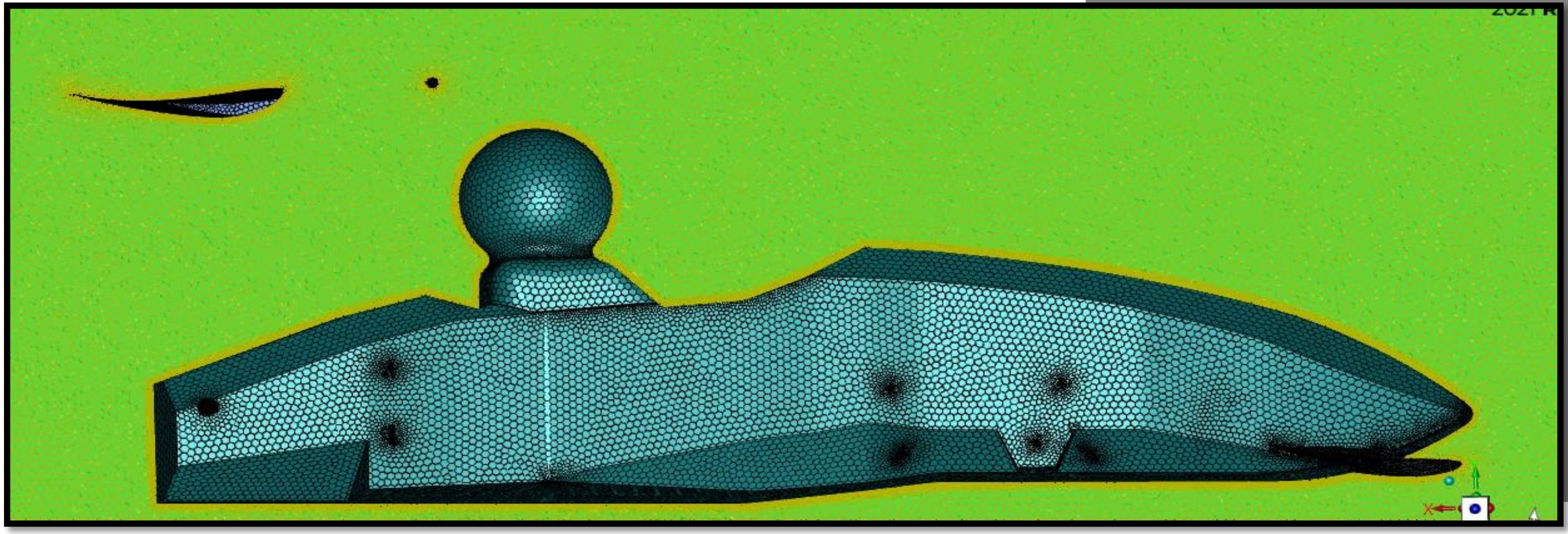


FIG. B11. volume mesh for single wing wing

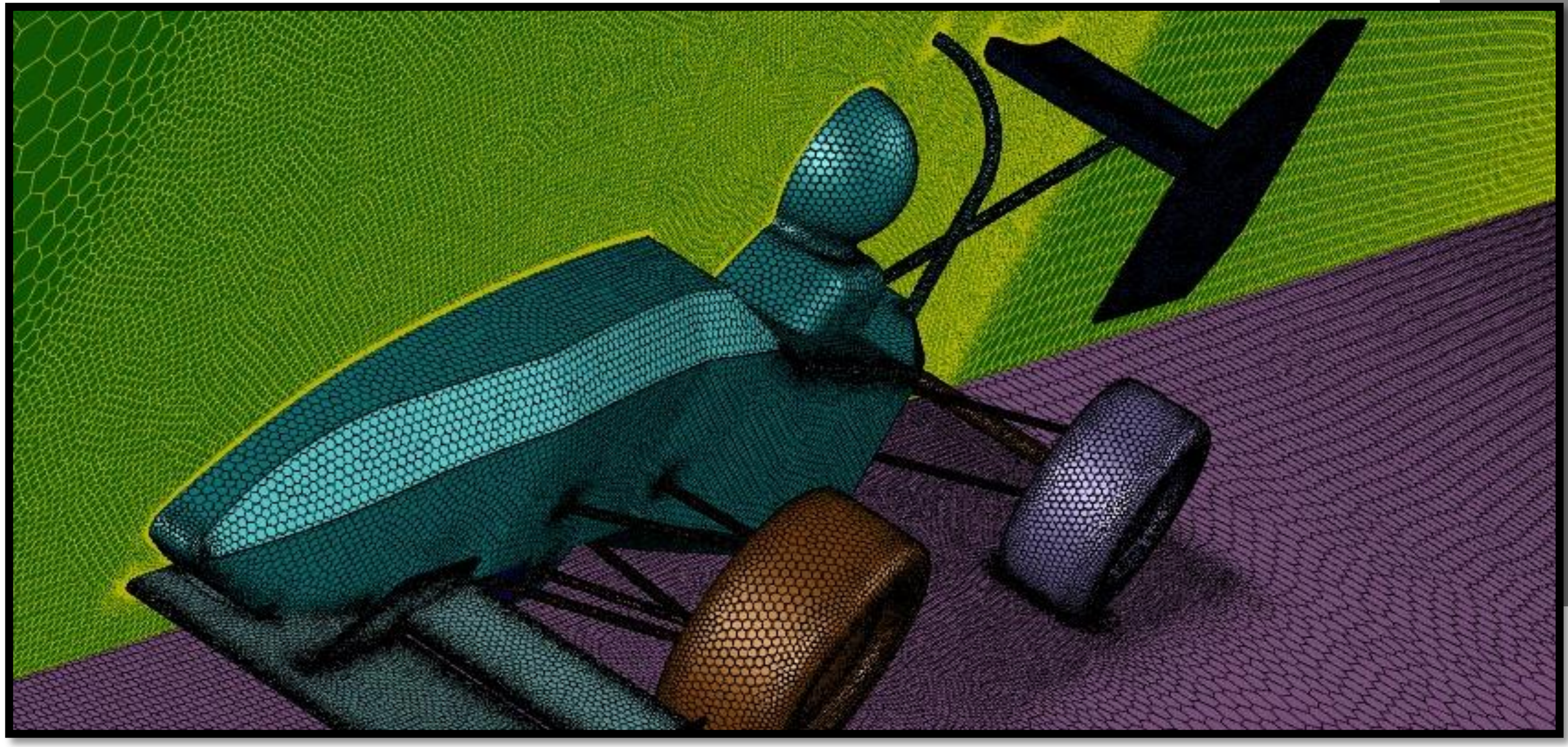


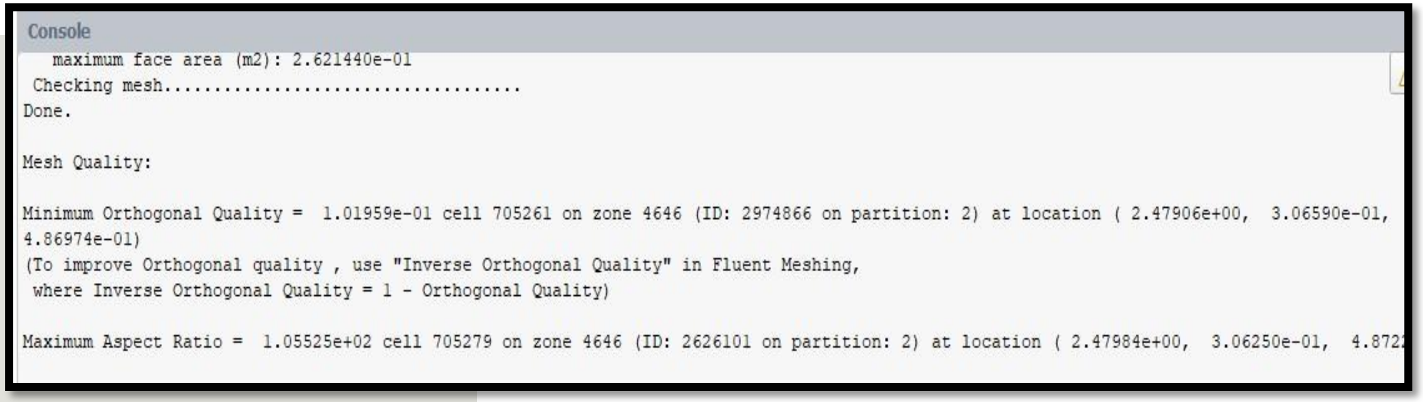
```
---------- The mesh has a minimum Orthogonal Quality of:  0.06

---------- The volume meshing of enclosure-enclosure11 is complete.
```

FIG. B12. Orthogonal quality

- By combining the two setups, we are able to achieve a mesh and orthogonal quality that meets our requirements, specifically with a value greater than 1.0.

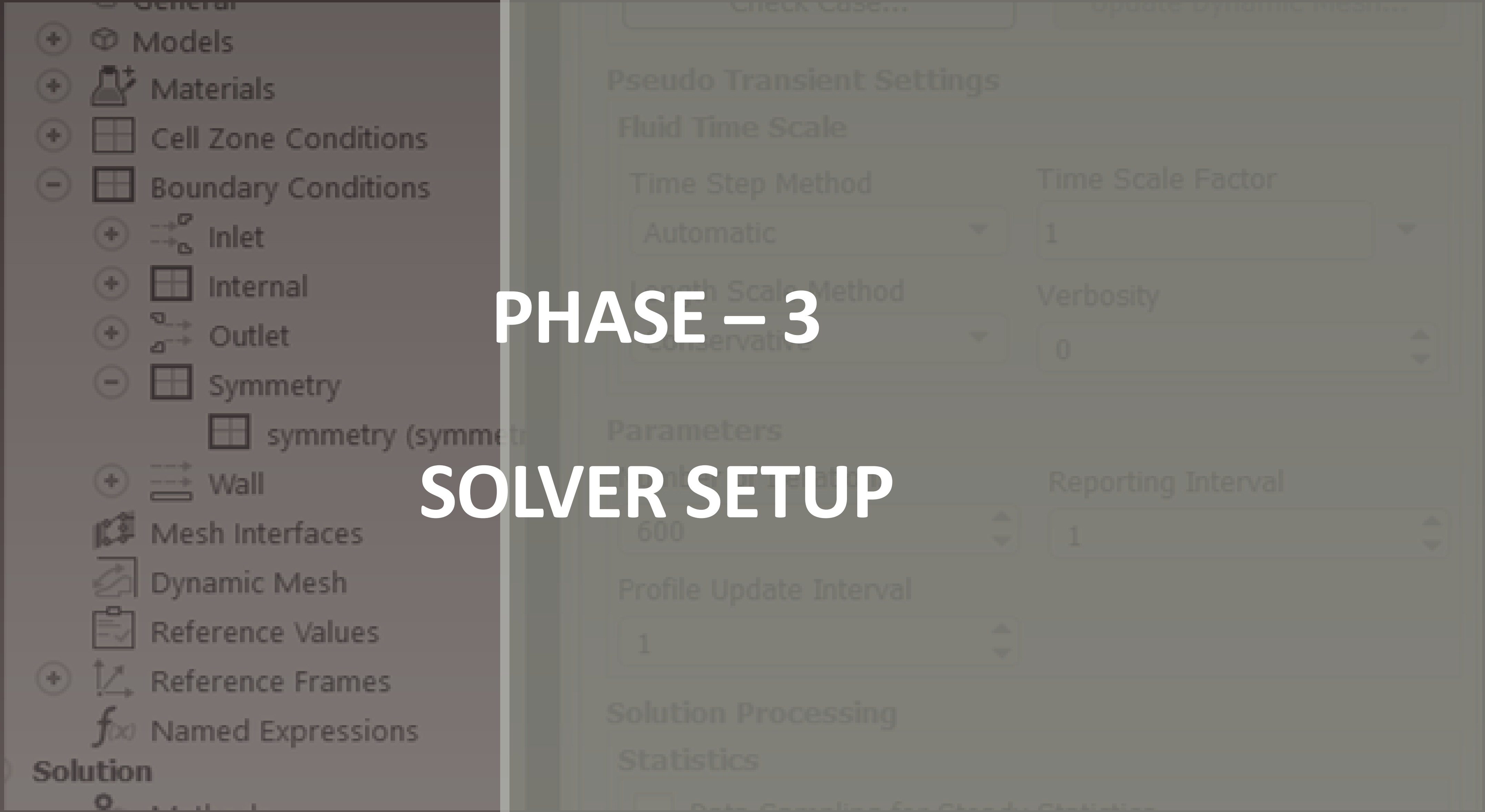
- The process involves carefully considering and optimizing the meshing parameters and settings to ensure the resulting mesh meets the desired criteria. One crucial aspect is achieving an orthogonal quality greater than 1.0. Orthogonal quality refers to the measure of the angle between the edges of adjacent mesh cells and is an important factor in maintaining accuracy and stability in the simulation.





```
Console
  maximum face area (m2): 2.621440e-01
 Checking mesh..................................
Done.

Mesh Quality:

Minimum Orthogonal Quality =  1.01959e-01 cell 705261 on zone 4646 (ID: 2974866 on partition: 2) at location ( 2.47906e+00,  3.06590e-01,
4.86974e-01)
(To improve Orthogonal quality , use "Inverse Orthogonal Quality" in Fluent Meshing,
 where Inverse Orthogonal Quality = 1 - Orthogonal Quality)

Maximum Aspect Ratio =  1.05525e+02 cell 705279 on zone 4646 (ID: 2626101 on partition: 2) at location ( 2.47984e+00,  3.06250e-01,  4.872
```

FIG. B13. Mesh Report



PHASE – 3

SOLVER SETUP

In this module, we will conduct a comparison between the Full wing and rear wing configurations to examine the differences in their respective conditions and values. Once we have ensured the quality of the mesh, we proceed to establish the physics conditions for the simulation.

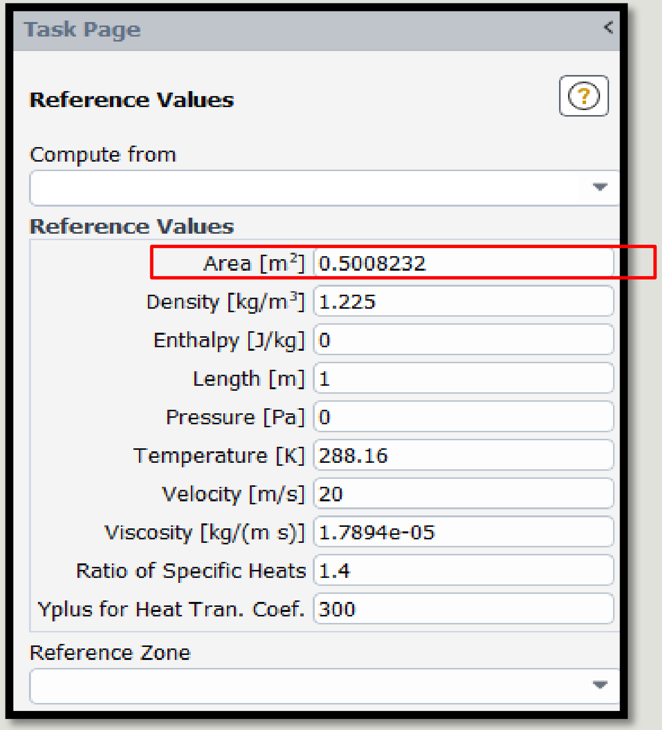


FIG. C1. Reference - FW

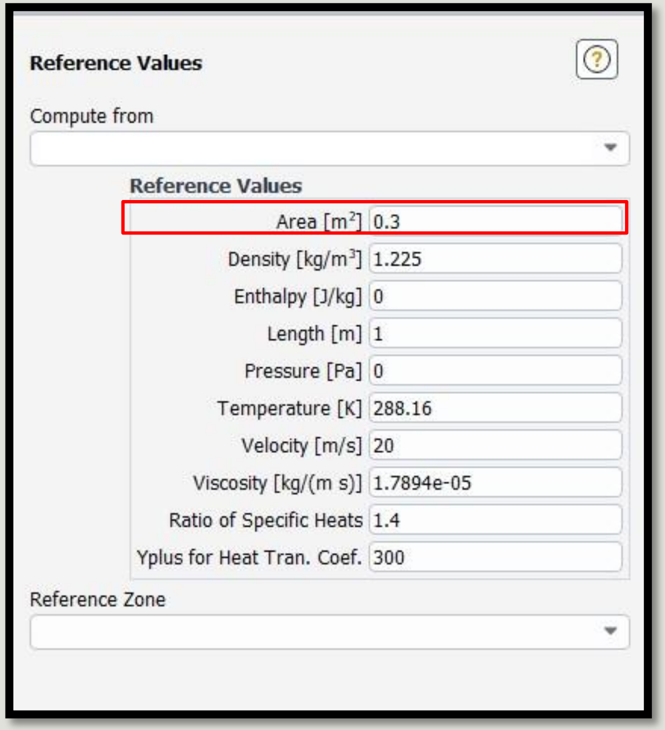


FIG. C2. Reference - SW

By comparing the original wing configuration to the modified version, we can observe the differences in the respective areas. These differences in the area can have significant implications for the aerodynamic performance of the wing and, consequently, the overall performance of the FSAE car.

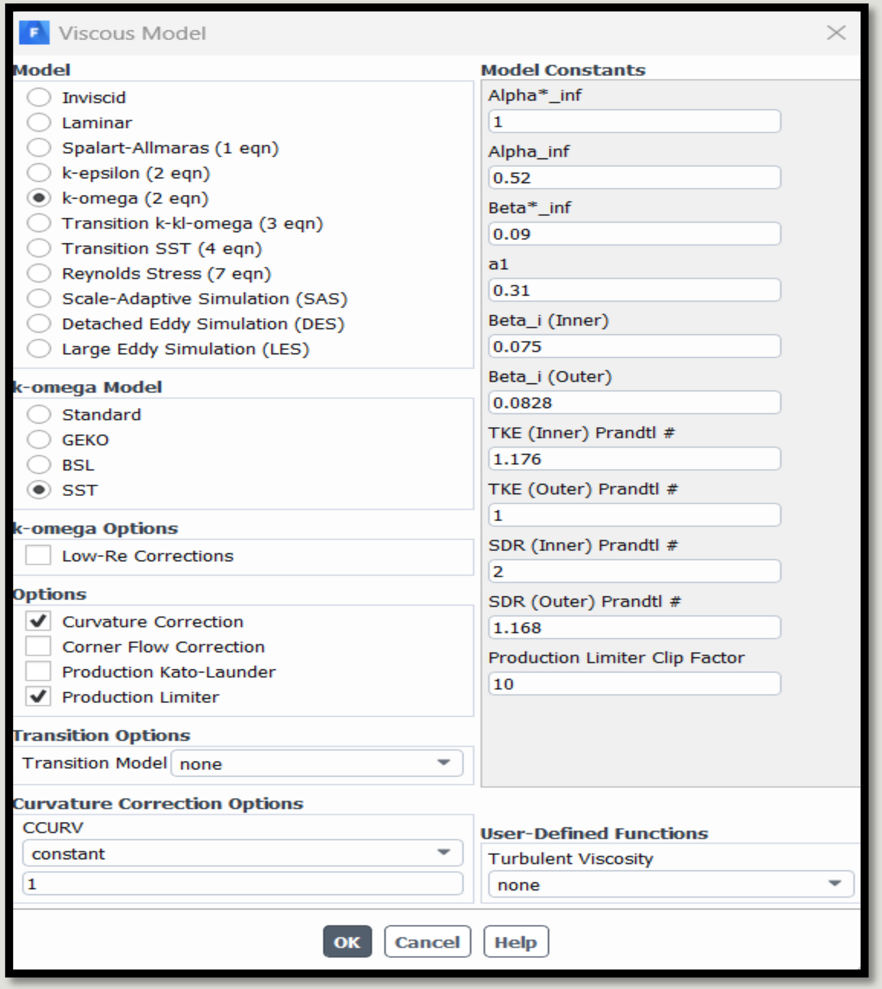


FIG. C3. Viscous Model conditions for both wings

Given that the flow in this simulation is anticipated to be incompressible, we opt to use the default settings for the both the wing analysis. The default settings are suitable for capturing the behavior of incompressible flows accurately, ensuring reliable results.

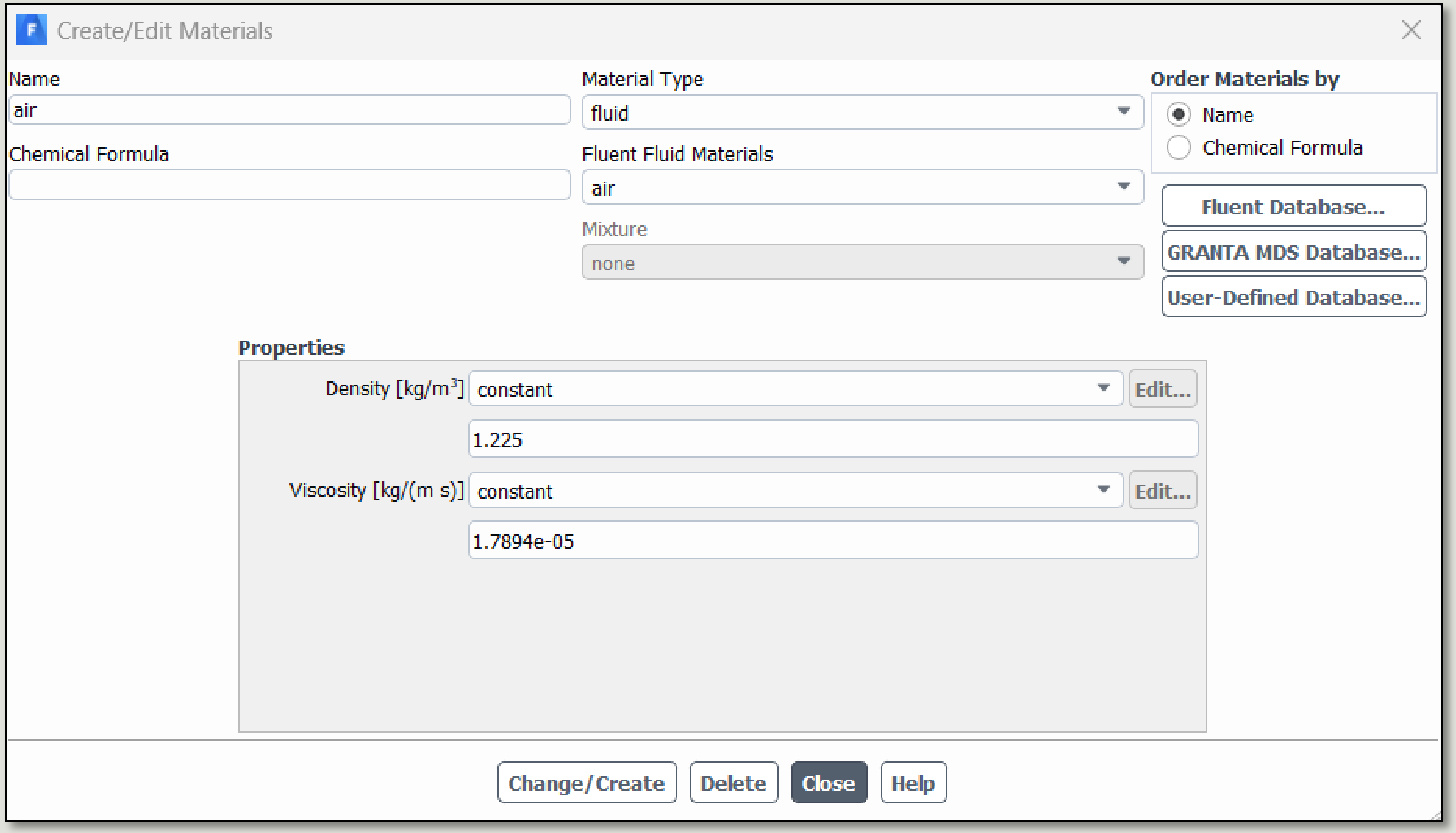


FIG. C4. Material Conditions

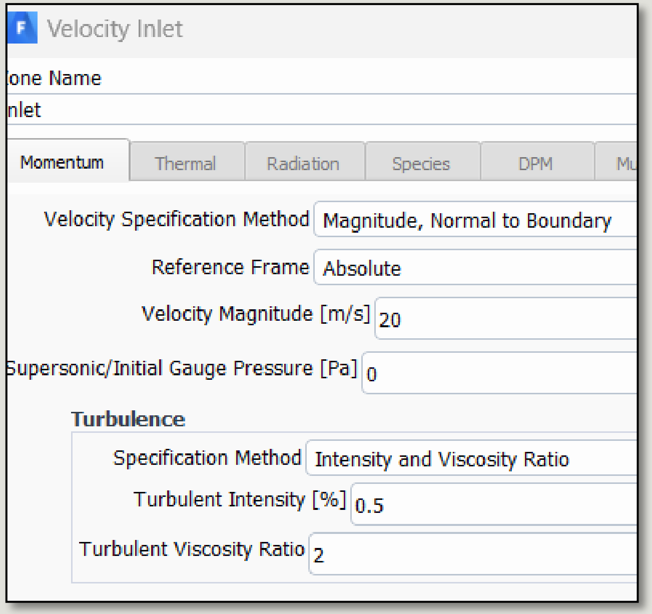


FIG. C5. Velocity Inlet

Conditions for the pressure outlet and conditions for the zone wall as set as Moving as well as rotational. As an example, I have shown for the rear wheel.

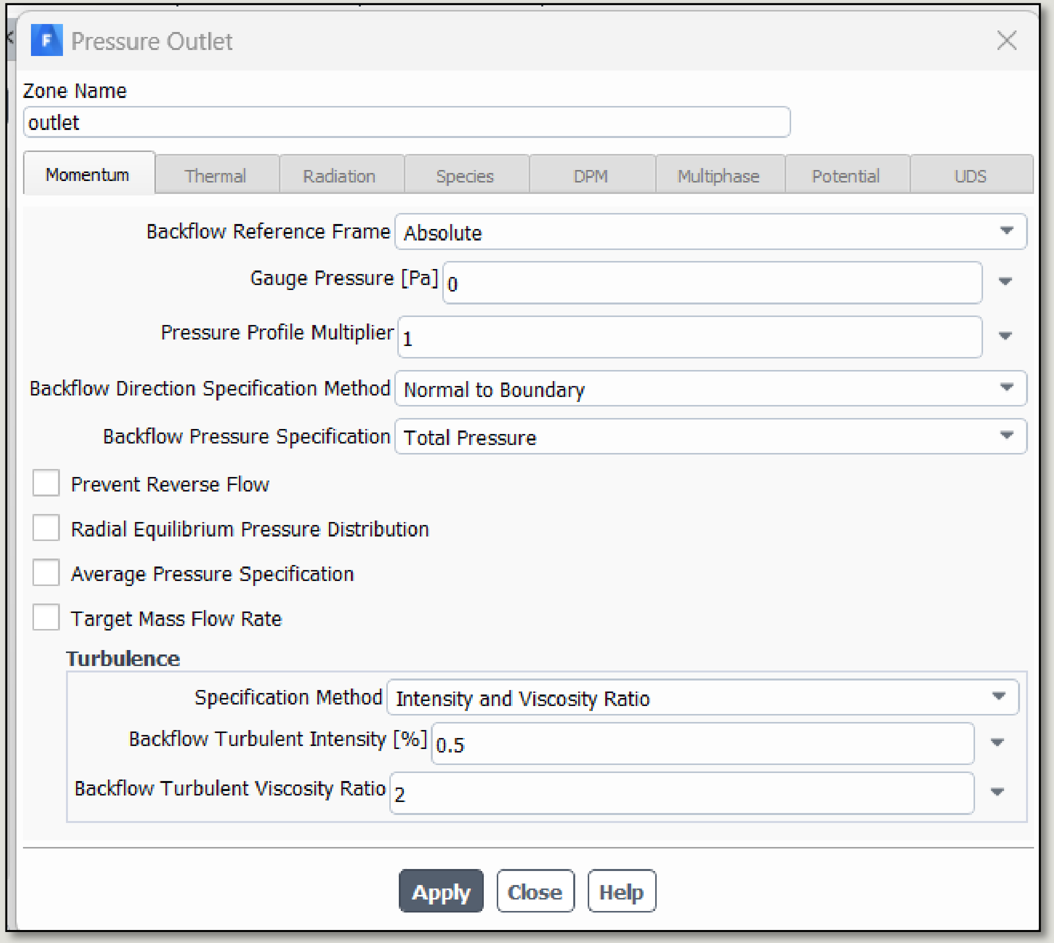


FIG. C6. Pressure outlet

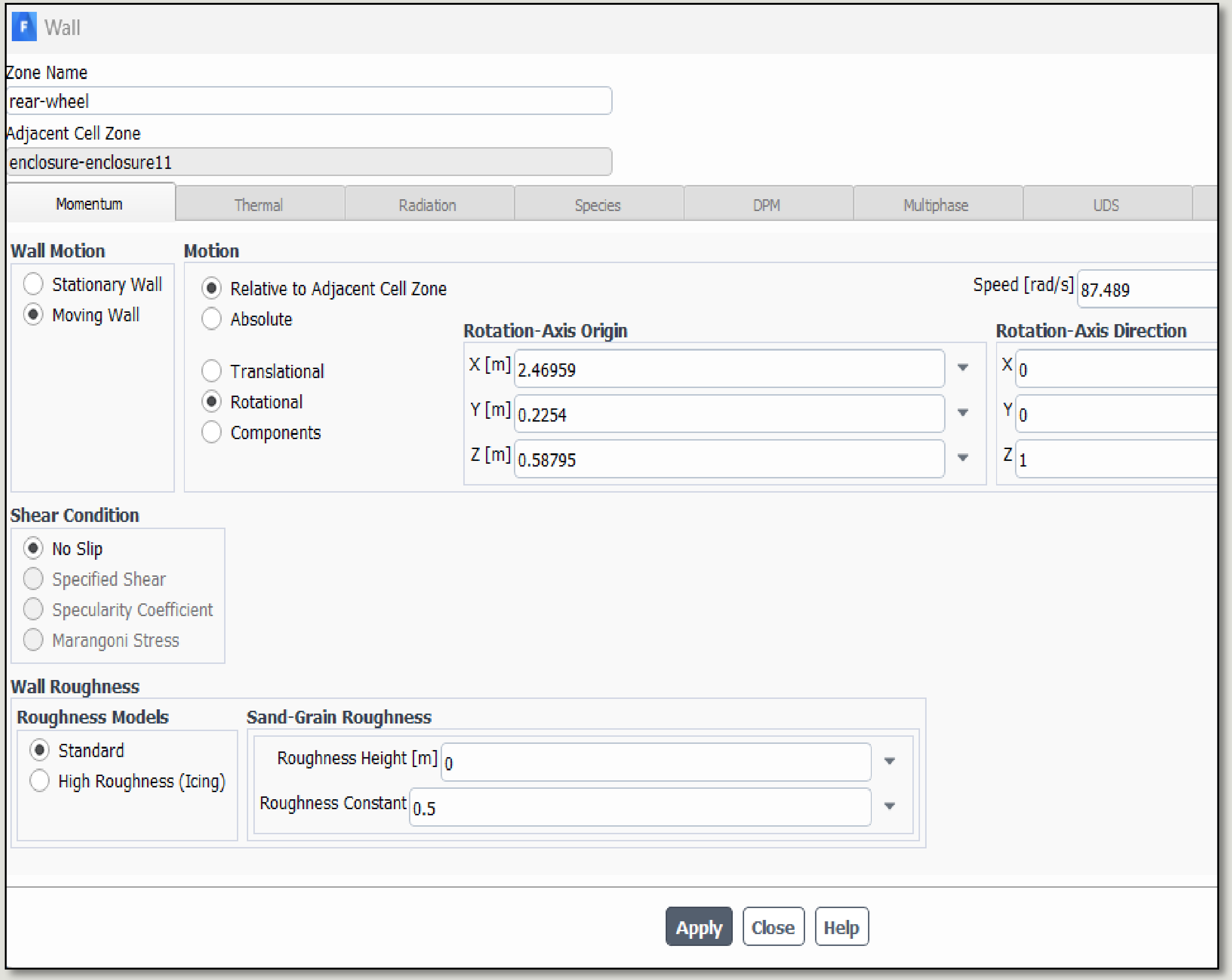


FIG. C7. wall Conditions

To initiate the solver iteration in Fluent, specific conditions must be configured along with FMG (Full Multigrid) initialization. In this process, we conducted mesh checks to ensure its quality and analyzed the assigned material properties. Additionally, we carefully set up the appropriate boundary conditions to accurately represent the physical system being simulated.

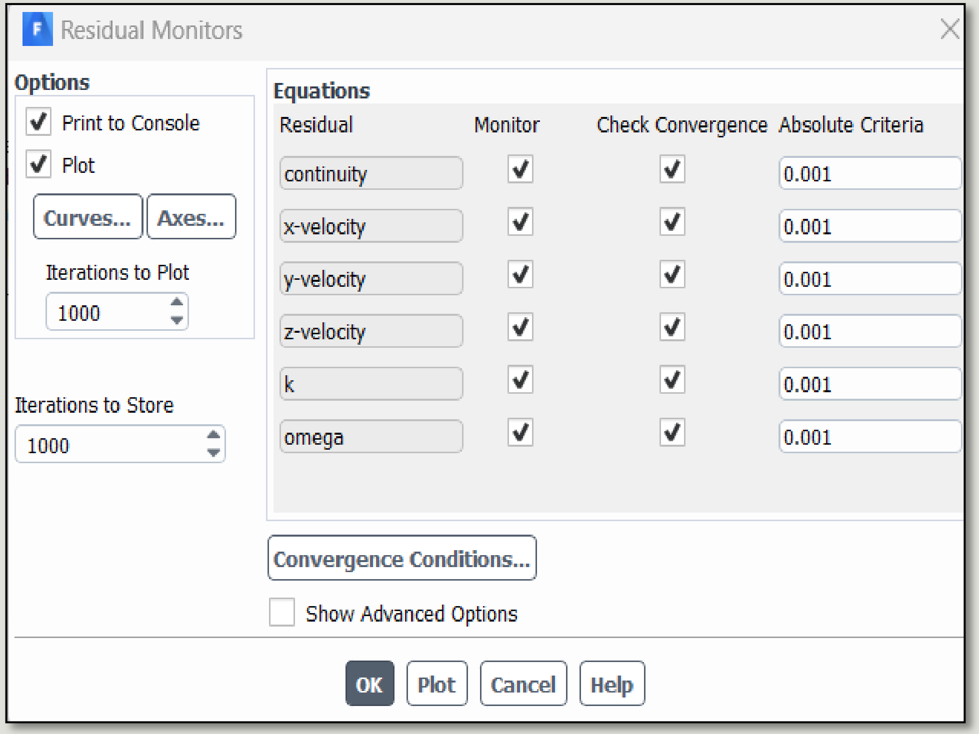


FIG. C8. Residual Monitors

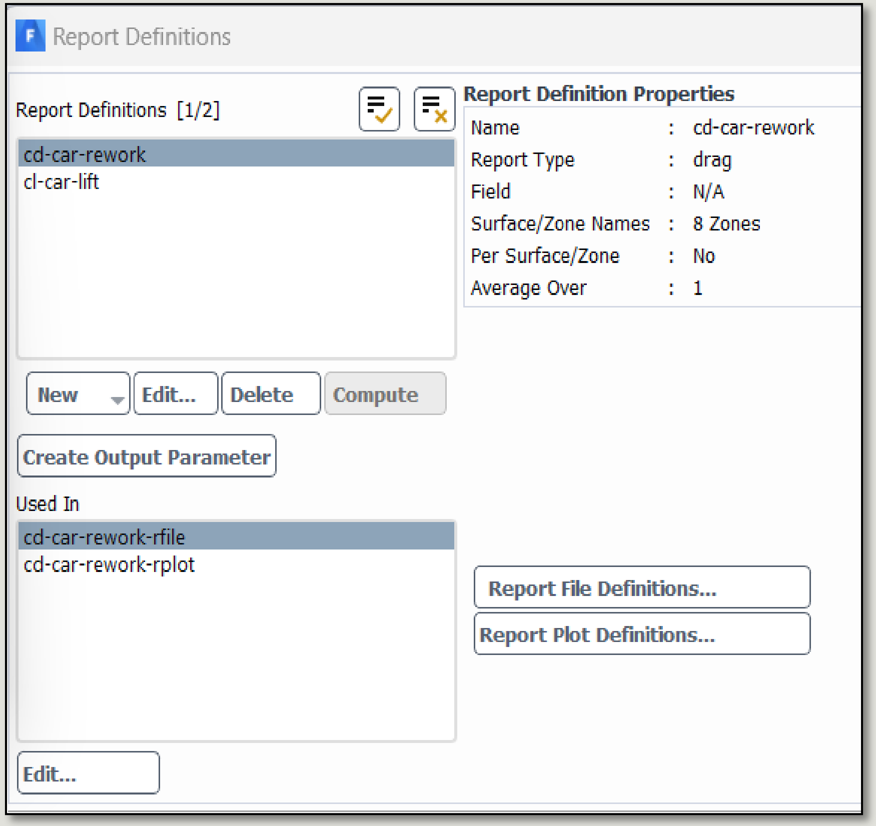


FIG. C9. Definition Reports

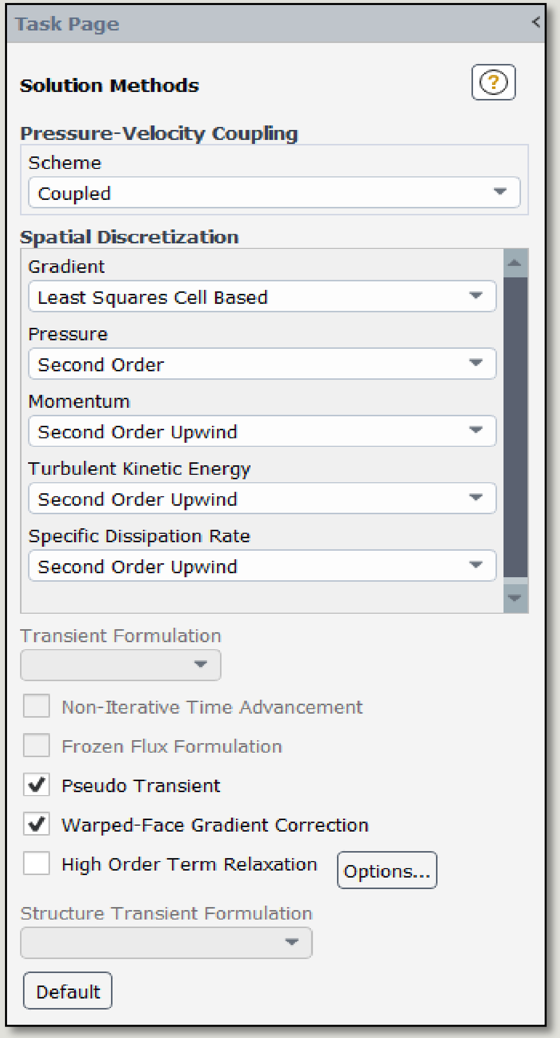


FIG. C10. Solution Methods

Single-wing Iterations

Since the problem at hand is unsteady, it is expected to observe inherent fluctuations in the plotted data. When examining the plots of Cd (drag coefficient) and CL (lift coefficient), it can be observed that these values stabilize after approximately 300 iterations and exhibit oscillations around certain values. Based on this behavior, we can conclude that the simulation has reached a steady state analysis.

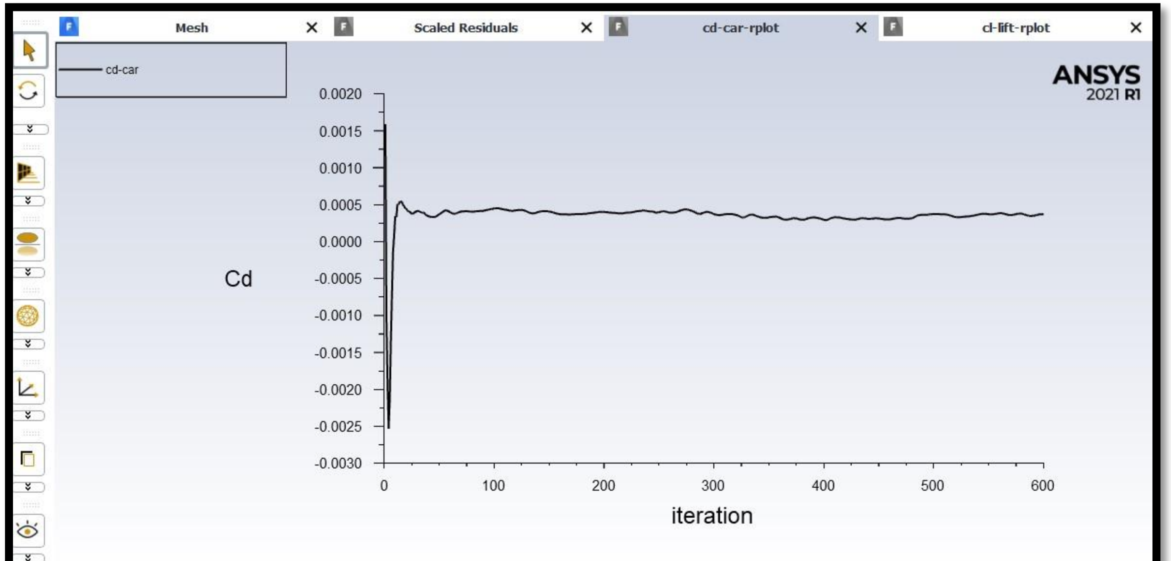


FIG. C11. Cd car plot graph

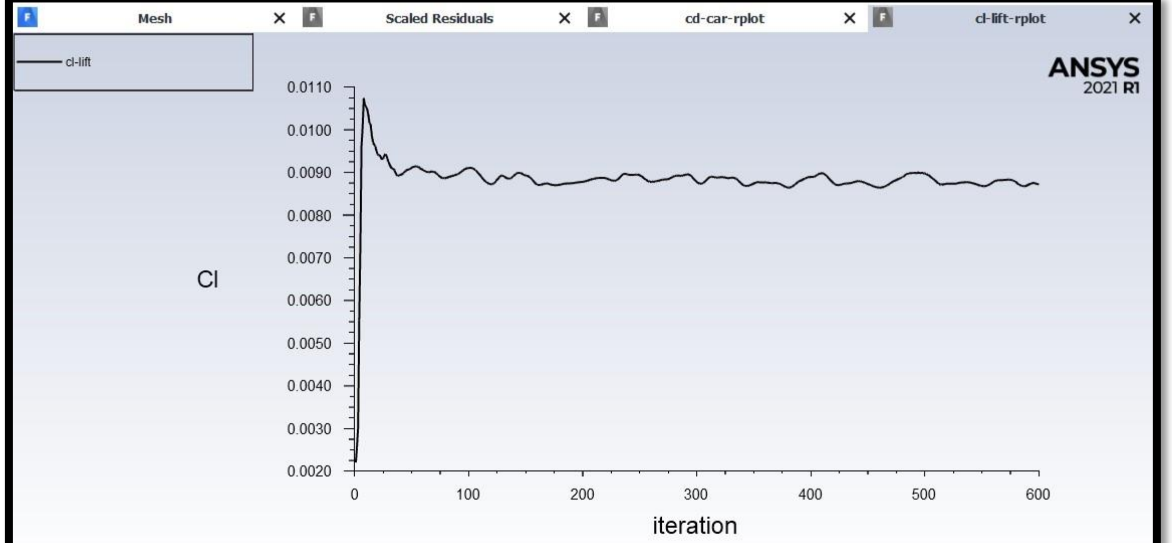


FIG. C12. Cl car plot graph

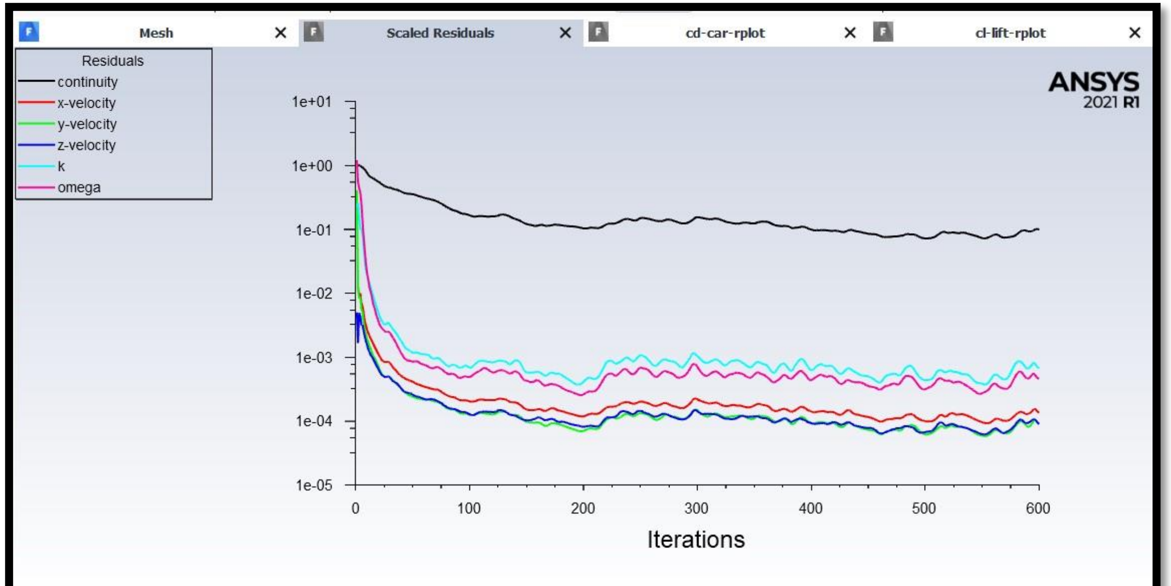


FIG. C13. Scaled residuals

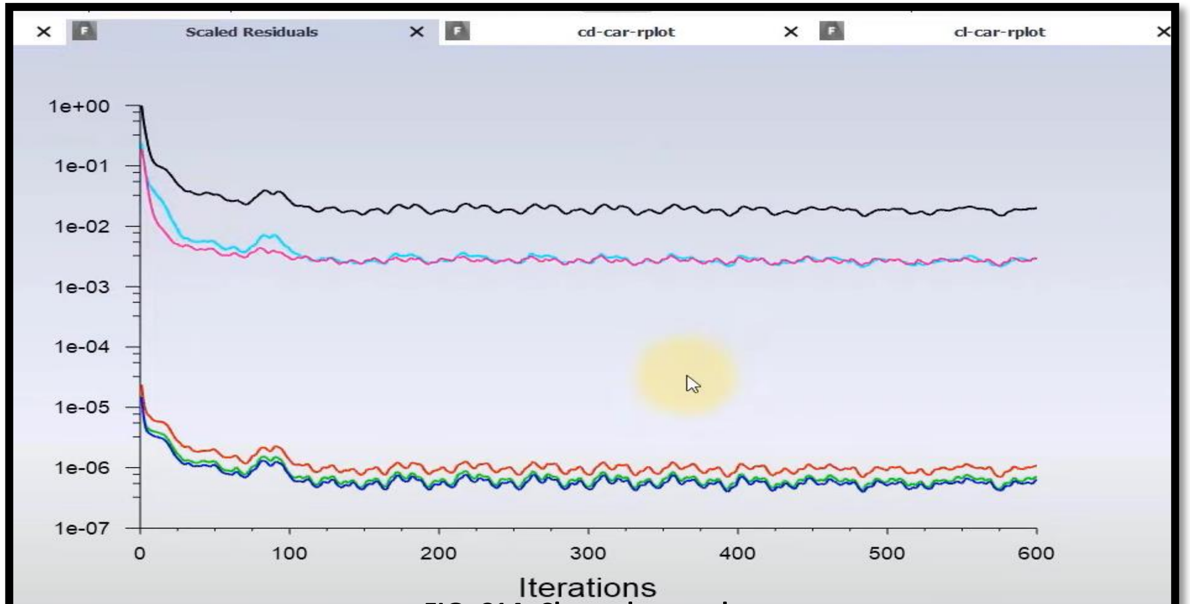


FIG. C14. Cl car plot graph

FIG. C15. Cl car plot graph

Full-wing Iterations

Upon comparing both graphs, it is evident that the flow exhibits oscillations at certain stages. These oscillations can be observed in the data plotted, indicating fluctuations in the analyzed parameters.

By closely examining the graphs, we can identify distinct periods where the flow behavior demonstrates these oscillatory patterns. These fluctuations signify variations in the flow dynamics, potentially caused by factors such as vortex shedding, flow separation, or other aerodynamic phenomena.

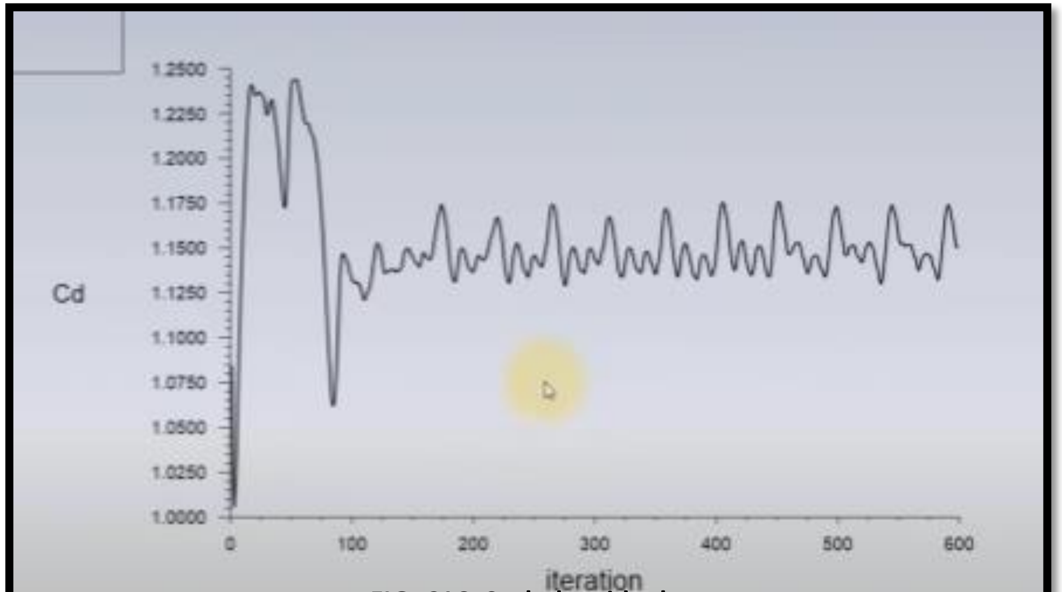
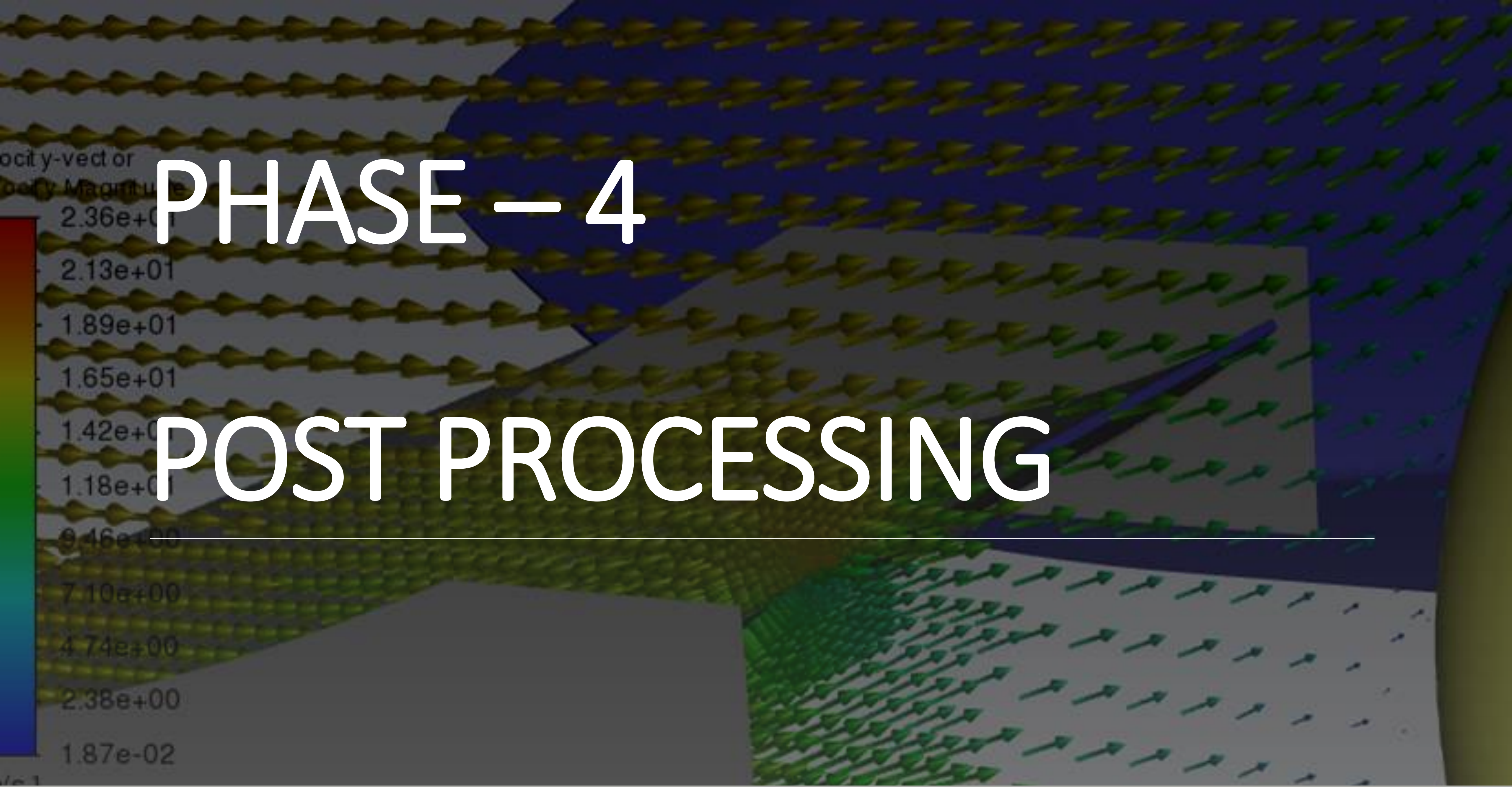


FIG. C16. Scaled residuals

PHASE – 4

POST PROCESSING



In this phase, we will analyze forces and then create and interpret contours of velocity magnitude and static pressure. Also, we will create a velocity vector plot to help us understand the fluid flow behavior.

Forces - Direction Vector (1 0 0)

Zone	Forces [N] Pressure	Viscous	Total	Coefficients Pressure	Viscous	Total
chasis	35.962715	-2.1707741	33.791941	0.29309064	-0.017691477	0.27539916
front-brakes	-0.34442477	-0.016504246	-0.36092902	-0.0028070093	-0.00013450709	-0.0029415164
front-wheel	19.660764	-0.46832583	19.192438	0.16023222	-0.0038167841	0.15641544
front-wing	9.2948045	-1.0131151	8.2816894	0.075751238	-0.0082567334	0.067494505
rear-brakes	-2.570721	-0.010036409	-2.5807574	-0.020950984	-8.1795202e-05	-0.021032779
rear-wheel	15.855069	-0.49108806	15.363981	0.12921639	-0.0040022927	0.1252141
rear-wing	51.220593	-1.5583998	49.662193	0.41744001	-0.012700721	0.40473929
rods	6.9556174	-0.09630149	6.859316	0.056687221	-0.00078484244	0.055902378
wheels-radii	0.56878552	0	0.56878552	0.0046355152	0	0.0046355152
Net	136.6032	-5.824545	130.77866	1.1132953	-0.047469153	1.0658261

FIG. D1. Forces direction – X axis (FULL WING)

In the above console, we come to know that the REAR WING & CHASIS plays a major contribution in the force – Direction vector (x-axis) based on the average value of 600 iterations.

Forces - Direction Vector (1 0 0)

Zone	Forces [N] Pressure	Viscous	Total	Coefficients Pressure	Viscous	Total
chasis	-0.013049665	-0.0029984726	-0.016048137	-0.00017754645	-4.0795544e-05	-0.000218342
front-brake	-0.0025259241	-8.8809568e-05	-0.0026147337	-3.4366313e-05	-1.2082934e-06	-3.5574607e-05
front-wheel	0.0083670533	0.003754694	0.012121747	0.00011383745	5.108427e-05	0.00016492173
front-wing	0.0045873722	-0.0012948596	0.0032925126	6.2413225e-05	-1.7617137e-05	4.4796088e-05
rear-brake	-0.0025556941	-6.2498804e-05	-0.0026181929	-3.4771347e-05	-8.5032383e-07	-3.5621671e-05
rear-wheel	0.037173074	0.0035558096	0.040728884	0.00050575609	4.837836e-05	0.00055413445
rear-wing	7.5974618e-05	-0.001771592	-0.0016956174	1.0336682e-06	-2.4103292e-05	-2.3069624e-05
rods	-0.0053203303	-0.00066056787	-0.0059808982	-7.2385443e-05	-8.9873176e-06	-8.1372761e-05
wheelradi	0.17744892	0	0.17744892	0.0024142709	0	0.0024142709
Net	0.20420078	0.00043370315	0.20463449	0.0027782418	5.9007229e-06	0.0027841426

FIG. D2. Forces direction – X axis (SINGLE WING)

Forces - Direction Vector (0 1 0)

	Forces [N]			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
chasis	19.007141	-0.1788727	18.828269	0.1549053	-0.0014577852	0.15344752
front-brakes	-0.59176272	-0.0039339551	-0.59569668	-0.0048227759	-3.2061133e-05	-0.004854837
front-wheel	9.4062241	-0.15308461	9.2531395	0.076659291	-0.0012476162	0.075411675
front-wing	-48.650647	-0.088890956	-48.739538	-0.39649535	-0.00072444773	-0.39721979
rear-brakes	-1.0931785	-0.00094878956	-1.0941273	-0.008909238	-7.7324901e-06	-0.0089169705
rear-wheel	7.33138	-0.002568397	7.3288116	0.059749629	-2.0932043e-05	0.059728696
rear-wing	-115.15758	-0.22178924	-115.37937	-0.93851672	-0.0018075484	-0.94032427
rods	0.96419702	-0.011099468	0.95309755	0.0078580586	-9.045897e-05	0.0077675996
wheels-radii	0.31010663	0	0.31010663	0.0025273217	0	0.0025273217
Net	-128.47412	-0.66118811	-129.13531	-1.0470445	-0.0053885822	-1.0524331

FIG. D3. Forces direction – Y axis Full wing

On the Y-axis console, we observe that the total forces for the full wing configuration are in the negative direction. This indicates that the forces exerted by the full wing are directed toward the ground, ensuring compliance and stability during operation. These negative forces allow the car to effectively generate downforce, enhancing traction and grip.

Forces - Direction Vector (0 1 0)

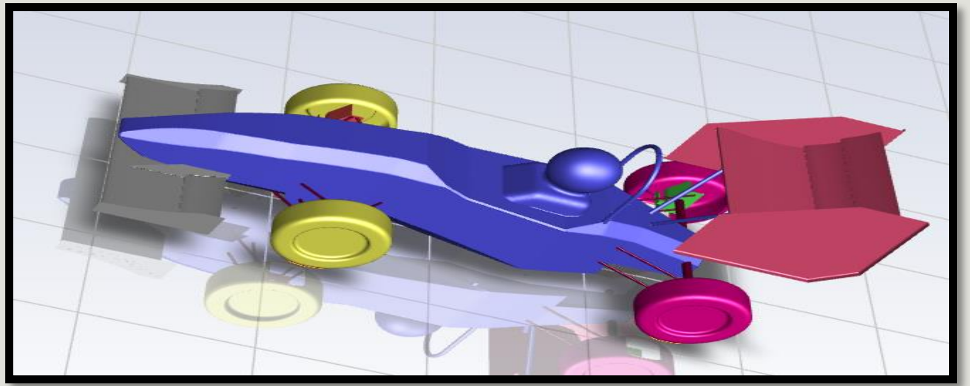
	Forces [N]			Coefficients		
Zone	Pressure	Viscous	Total	Pressure	Viscous	Total
chasis	0.17651731	0.0020972826	0.17861459	0.0024015959	2.8534456e-05	0.0024301304
front-brake	0.00086202997	5.0503225e-05	0.00091253319	1.1728298e-05	6.8711869e-07	1.2415417e-05
front-wheel	0.23149646	0.055705087	0.28720155	0.0031496116	0.00075789231	0.0039075039
front-wing	-0.025085955	-0.0001983263	-0.025284281	-0.00034130549	-2.6983169e-06	-0.00034400381
rear-brake	-0.0029624121	5.0884834e-05	-0.0029115273	-4.0304925e-05	6.9231064e-07	-3.9612614e-05
rear-wheel	0.21640534	-0.0074557003	0.20894964	0.0029442903	-0.00010143809	0.0028428522
rear-wing	-0.0088695366	0.0011850166	-0.00768452	-0.00012067396	1.6122674e-05	-0.00010455129
rods	0.00092976458	0.00017018214	0.0010999467	1.2649858e-05	2.3154032e-06	1.4965261e-05
wheelradi	-0.037800119	0	-0.037800119	-0.00051428731	0	-0.00051428731
Net	0.55149289	0.05160493	0.60309782	0.0075033043	0.00070210786	0.0082054121

FIG. D4. Forces direction Y axis single wing

In contrast, for the single wing configuration, the forces are not in the negative direction. This implies that the design fails to generate downward forces and lacks compliance with the ground. Consequently, it may not sustain stable performance when subjected to airflow or hits the ground.

The absence of negative forces with the single-wing design suggests a potential deficiency in generating sufficient downforce, which is crucial for maximizing grip and optimizing the car's performance. Addressing this issue becomes imperative to ensure stability and maneuverability during operation.

Now, we create contour plots to understand the general flow field of the car.



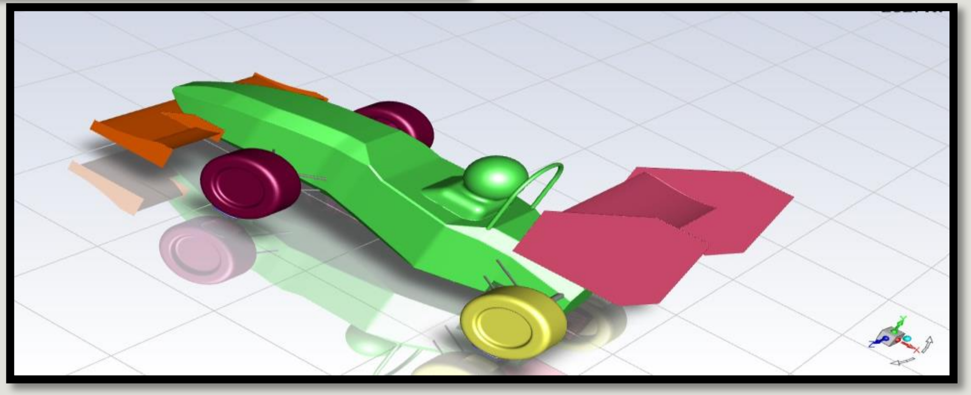


FIG. D5. Graphical representation of the full car

Here we see the velocity contour of the car:

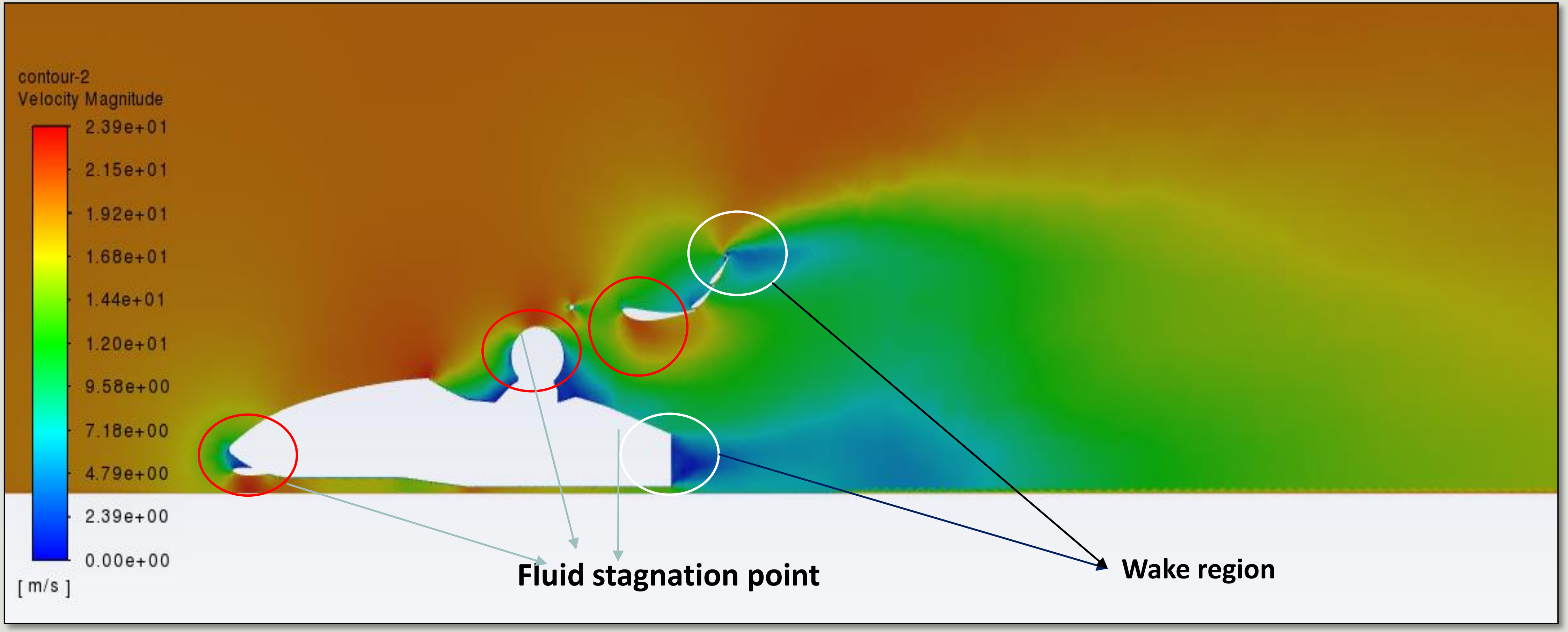


FIG. D6. Velocity contour of the FW car

The Single wing car configuration exhibits a larger wake region in the velocity contour compared to the Full wing configuration. This is due to the absence of negative forces directing the airflow toward the ground. The extended wake region indicates reduced aerodynamic efficiency, potentially impacting the car's stability, handling, and overall performance. Optimizing the design or making modifications to minimize the wake region is crucial for improving the aerodynamic performance of the Single wing car.

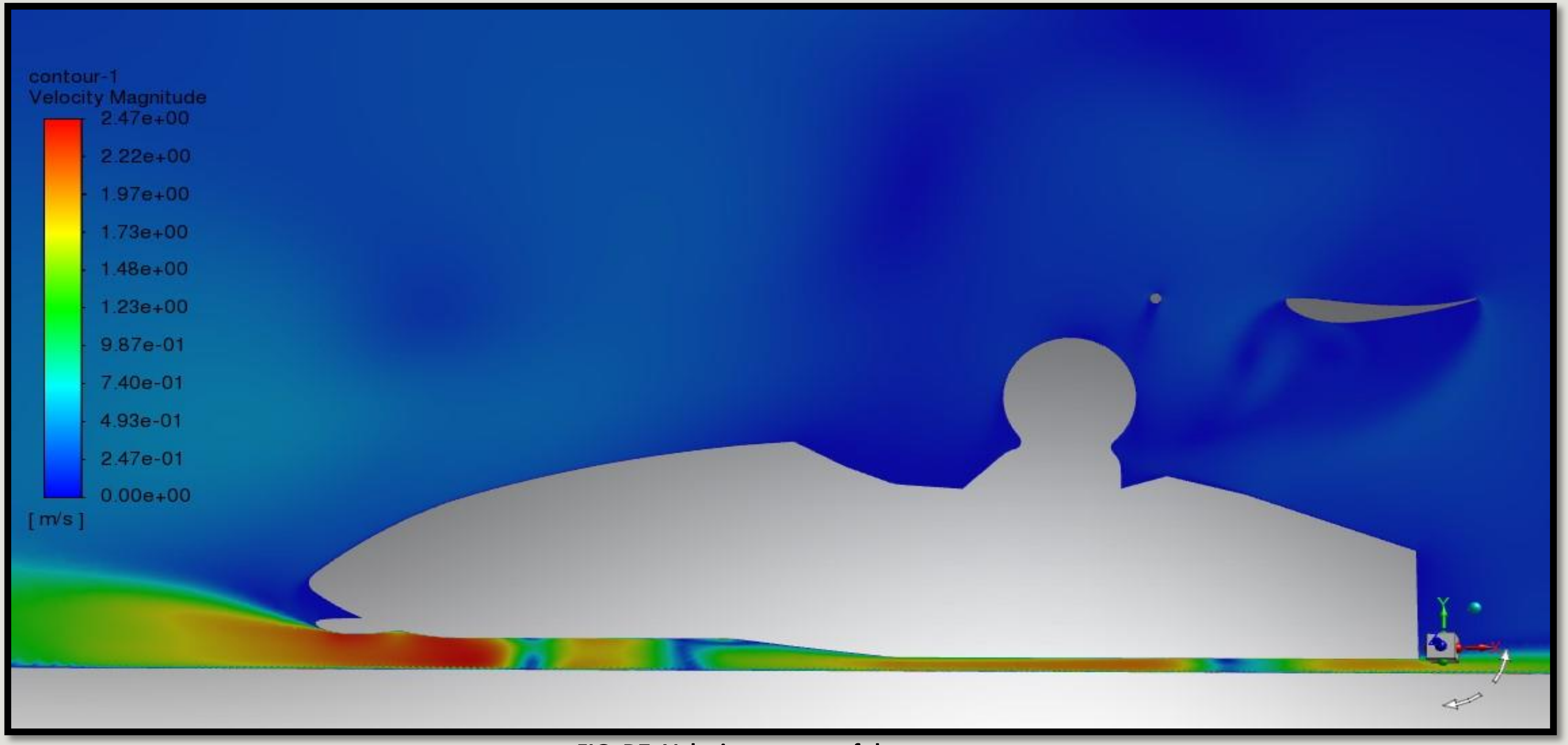


FIG. D7. Velocity contour of the car - sw

Here we see the pressure contour of the car:

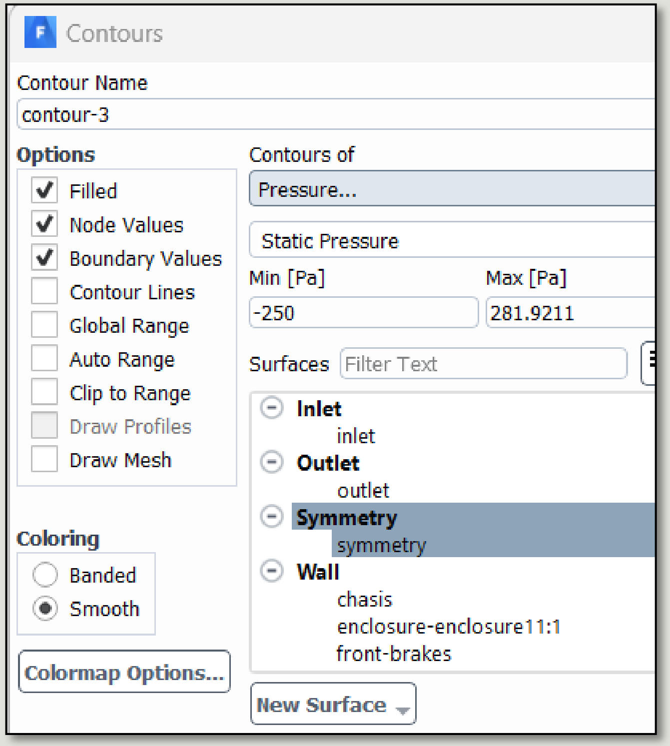


FIG. D8. Pressure contour of the car

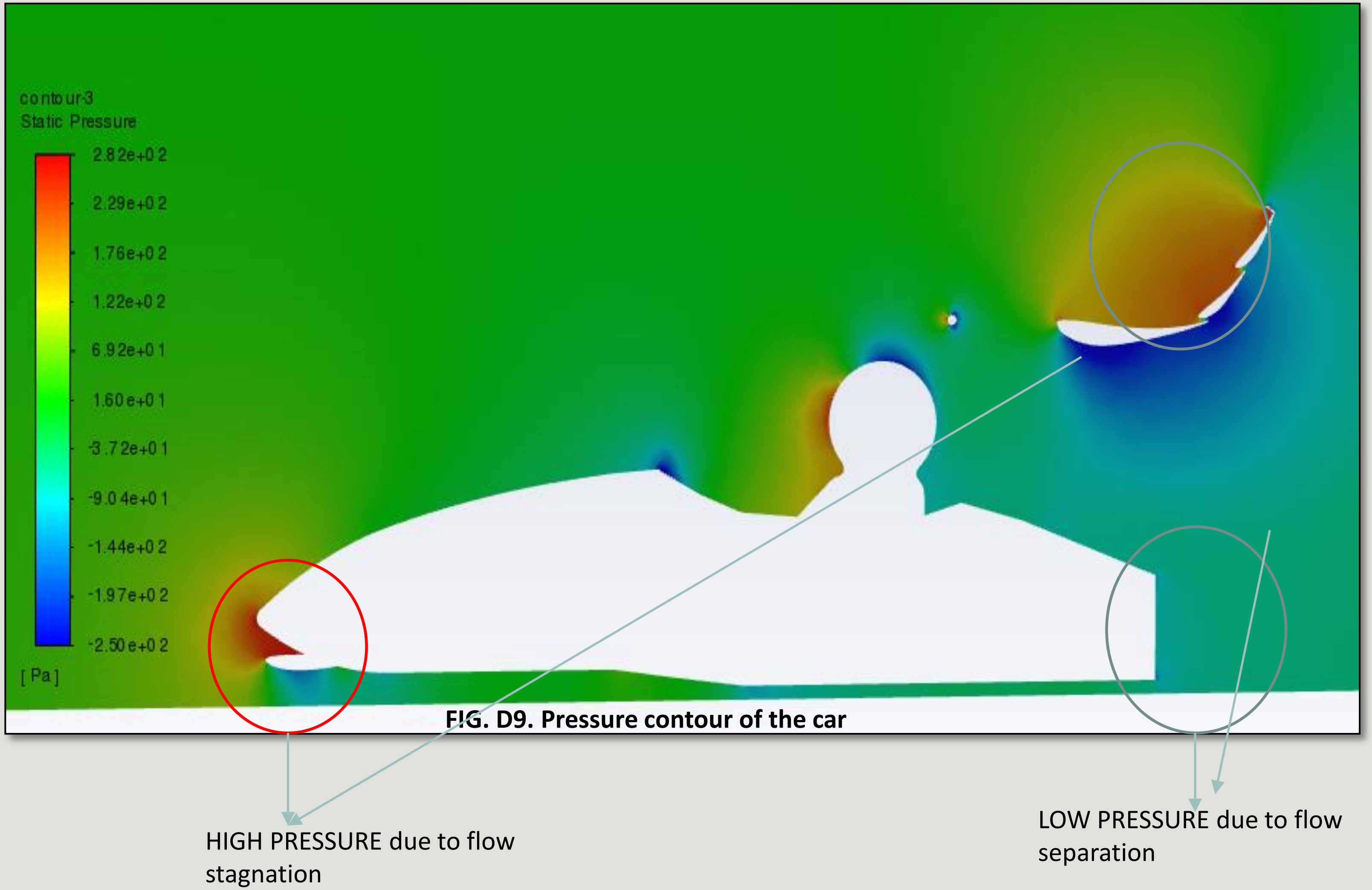


FIG. D9. Pressure contour of the car

Pressure contour of Single wing

As the force is non-negative, it indicates an excess of pressure in all directions. This suggests that the flow is exerting a higher pressure on the surfaces of the car, creating a favorable aerodynamic condition. The balanced pressure distribution allows for enhanced stability and efficient airflow management around the vehicle. This positive pressure indicates that the design is effectively utilizing aerodynamic principles to generate the desired forces and optimize performance.

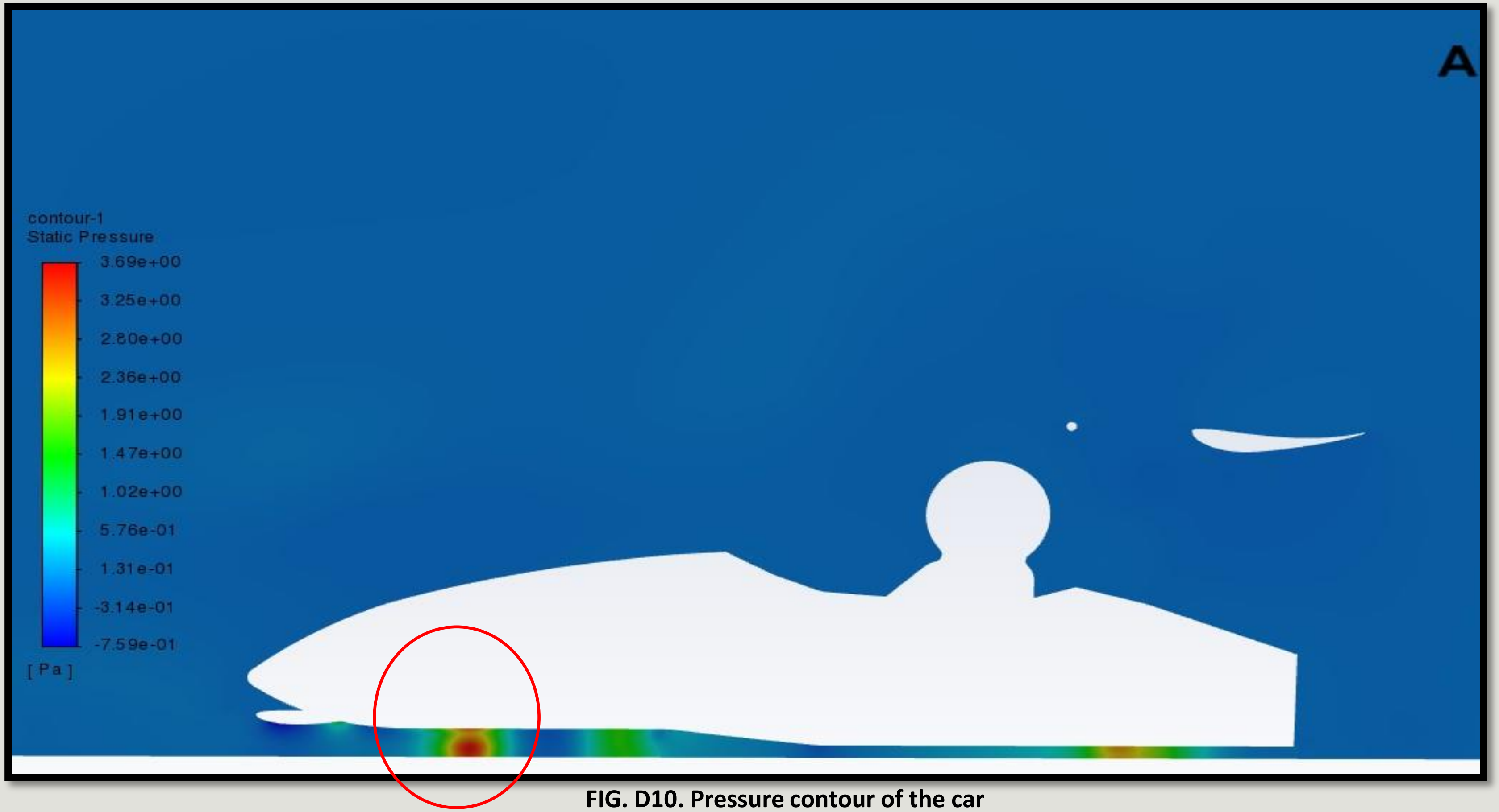


FIG. D10. Pressure contour of the car

High pressure is created only in the downside

HERE WE SEE THE VELOCITY VECTOR OF THE CAR FOR A FULL WING

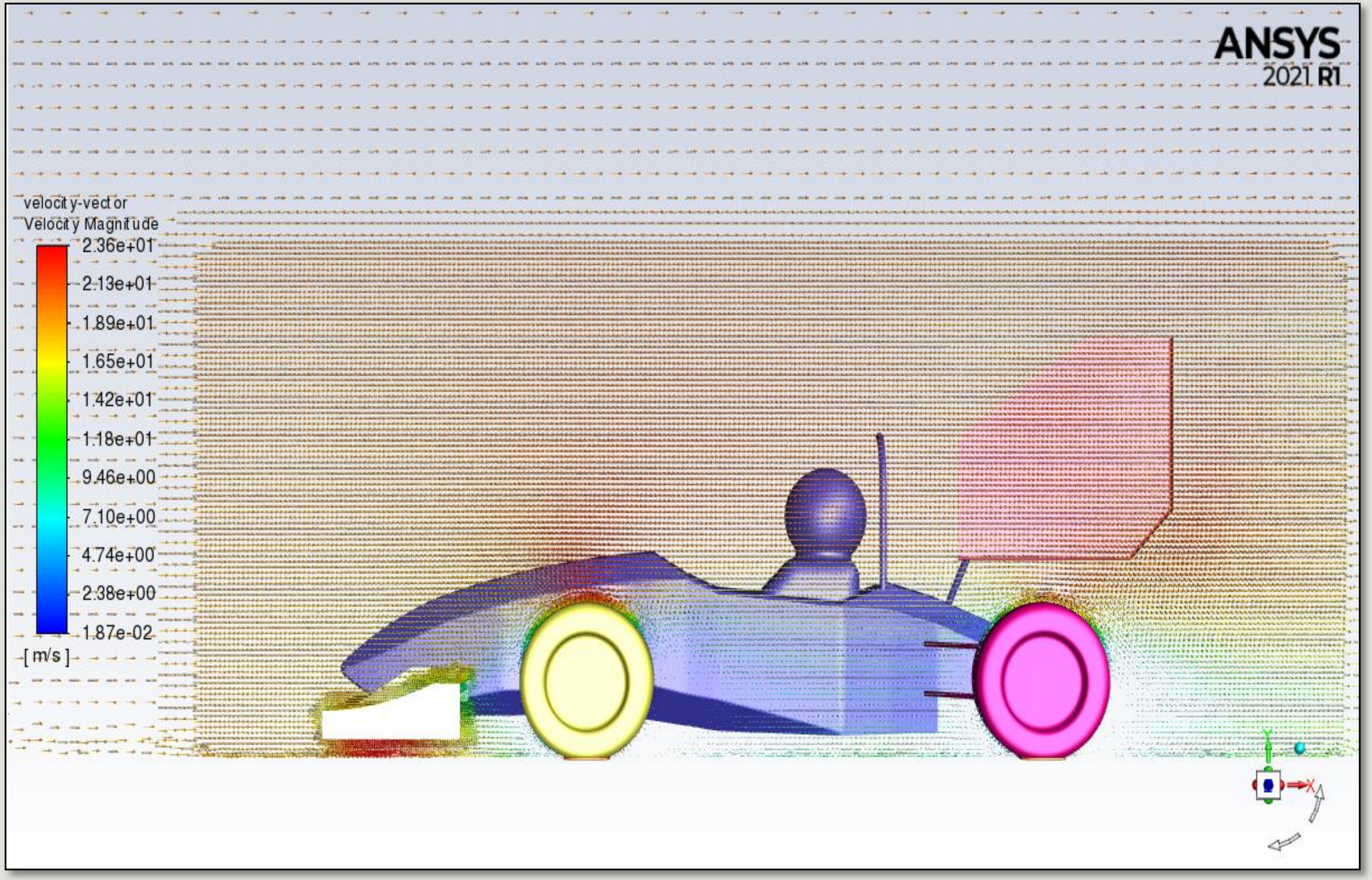
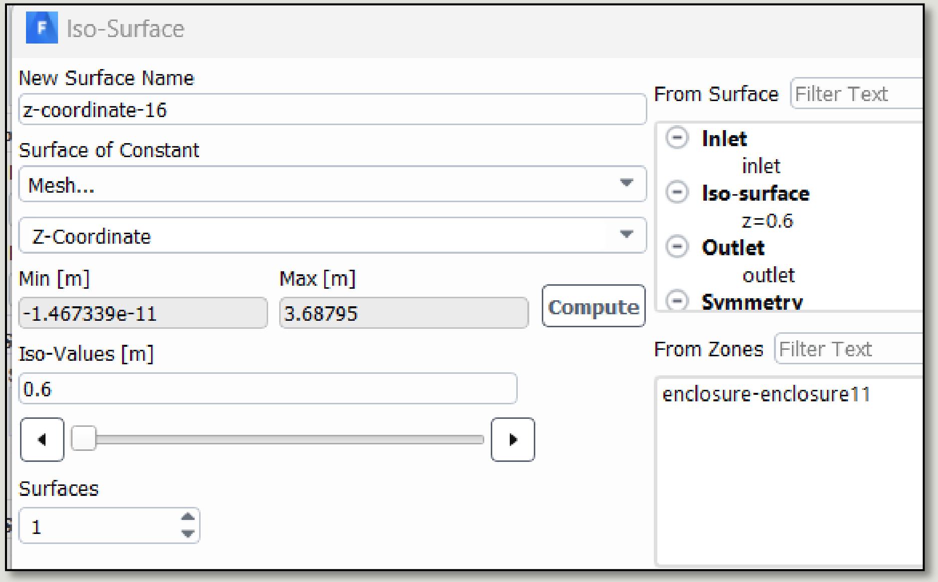
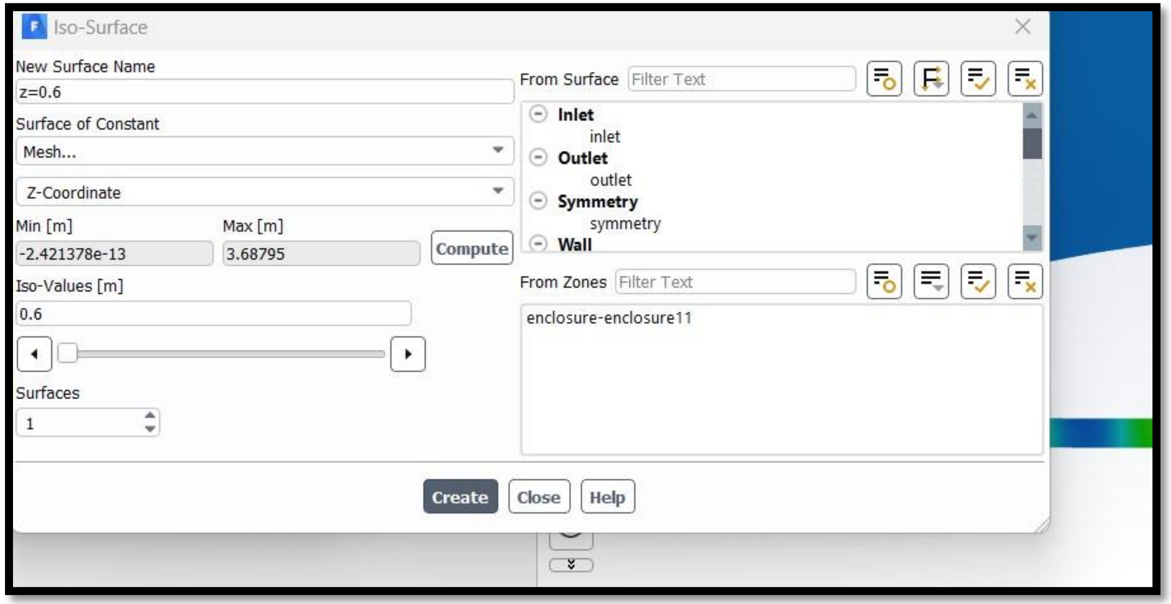
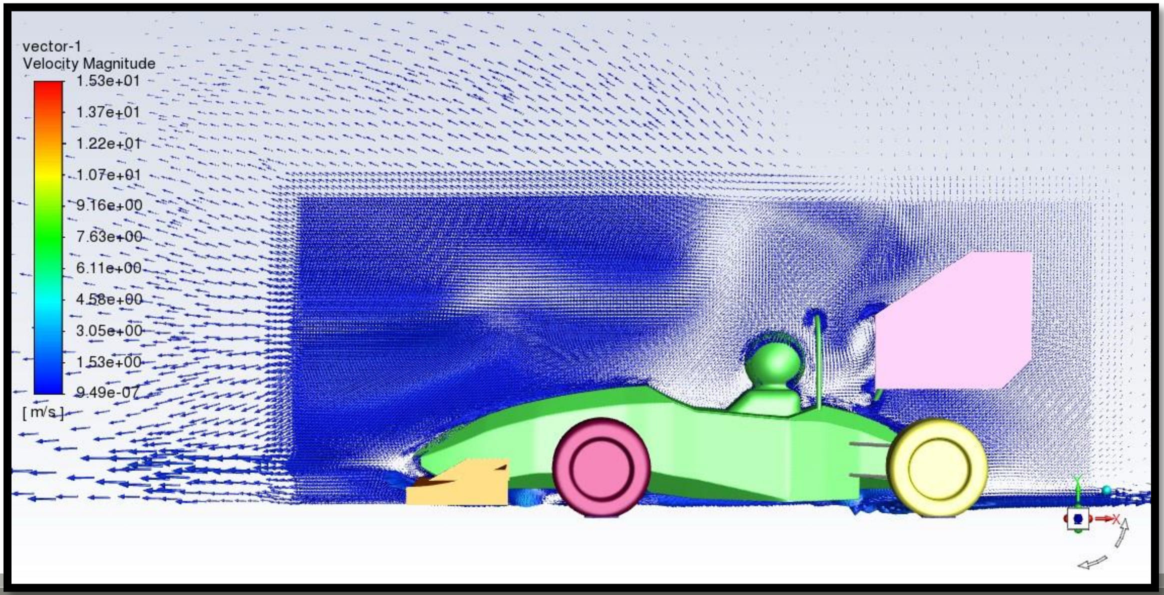


FIG. D11. Velocity vector of the car

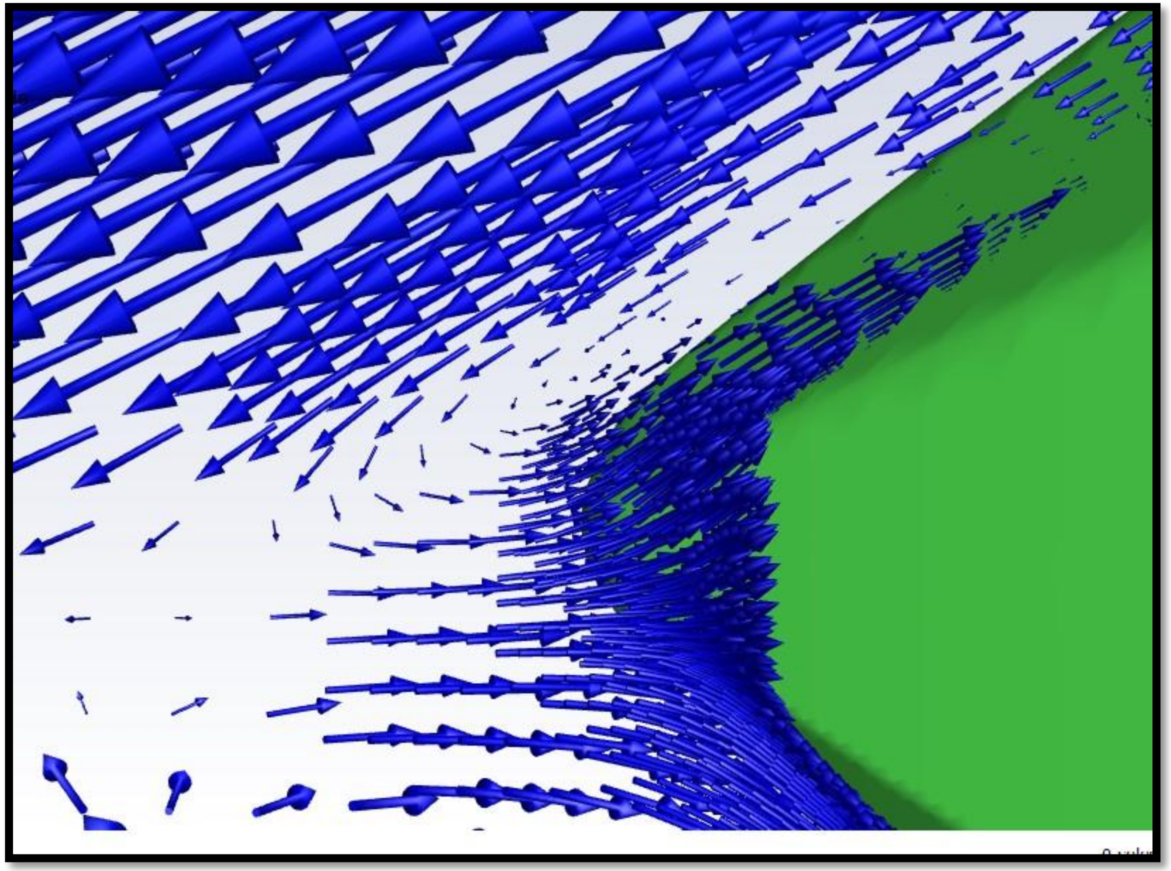


Used Iso surface for the Z coordinates for the velocity vector of the car.





Here we see the velocity vector of the car for a single wing and also used iso surface as per the full wing.



FIG'S. D12. The velocity vector of the car

To generate the path lines, analyze the flow through the path of the car body, with the following conditions FW.

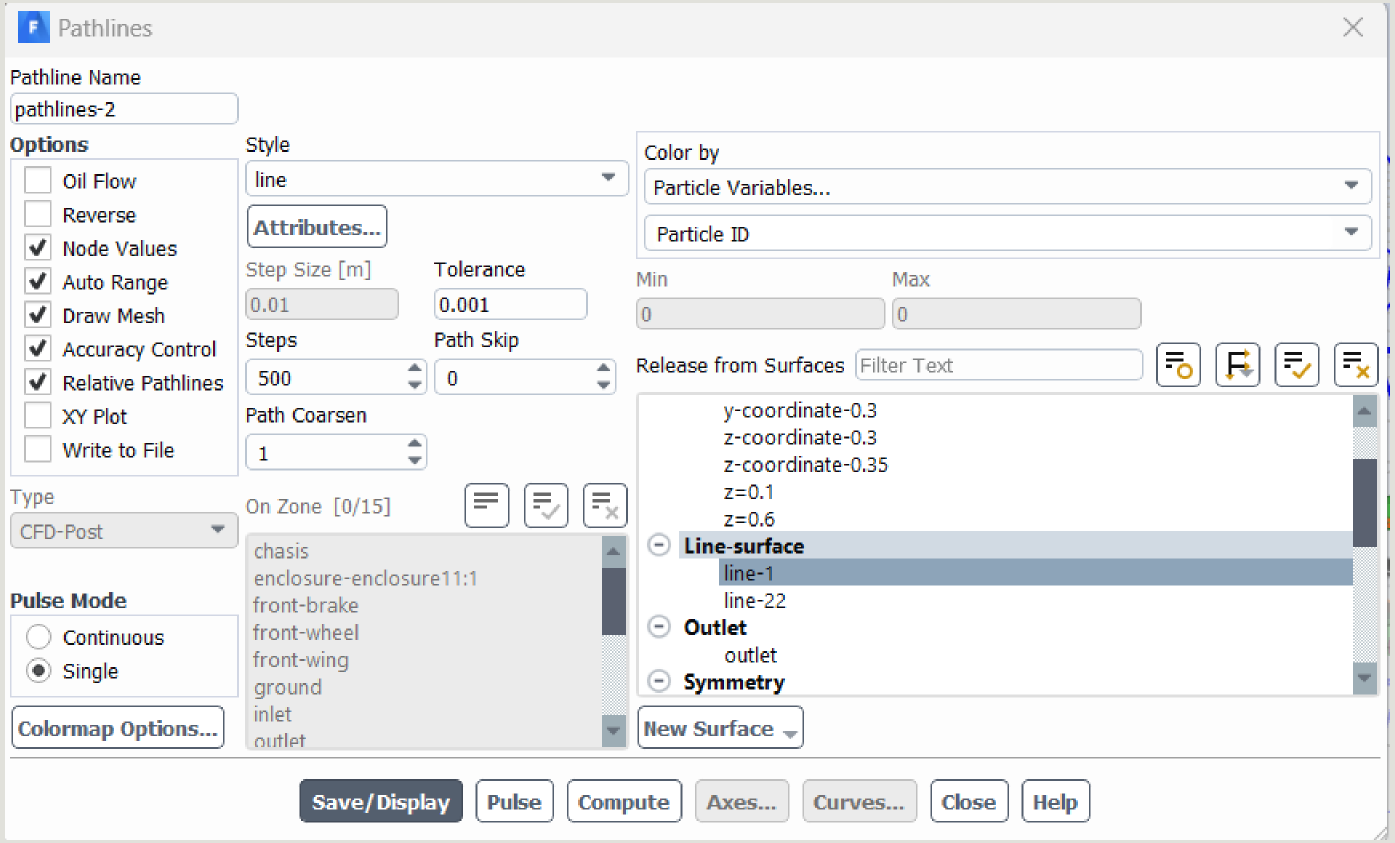


FIG. D13– path line settings

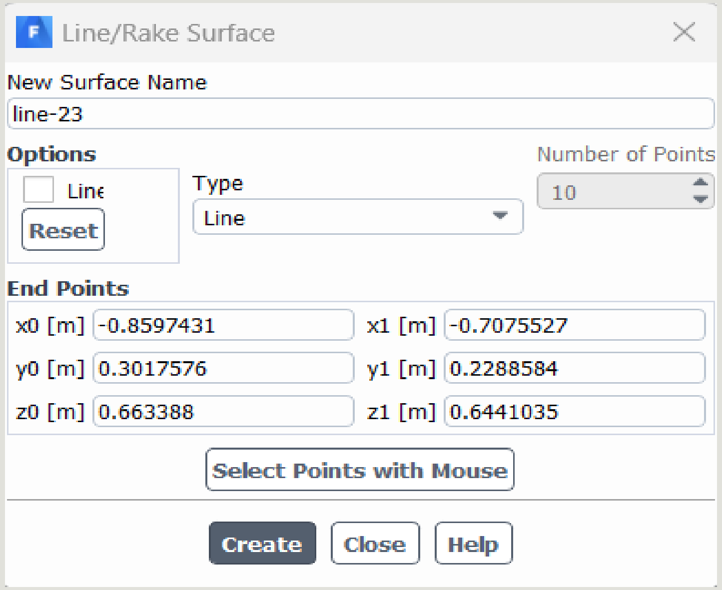
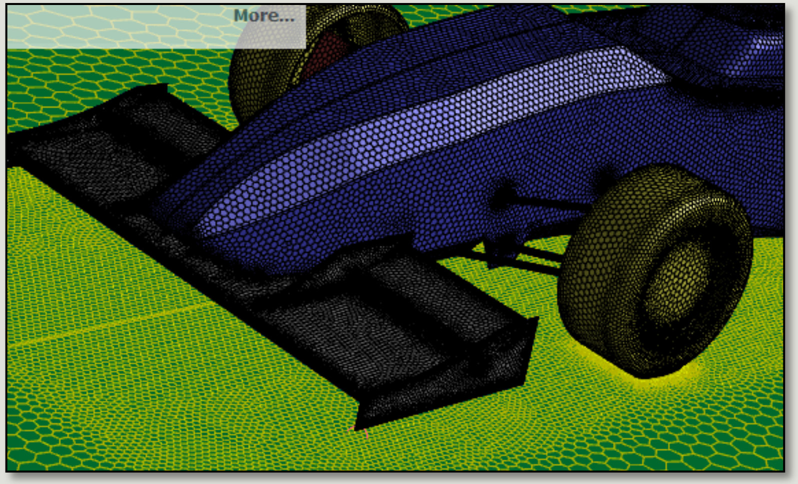


FIG. D14– Line and rake surface



To generate the path lines, analyze the flow through the path of the car body, with the following conditions forSINGLE WING

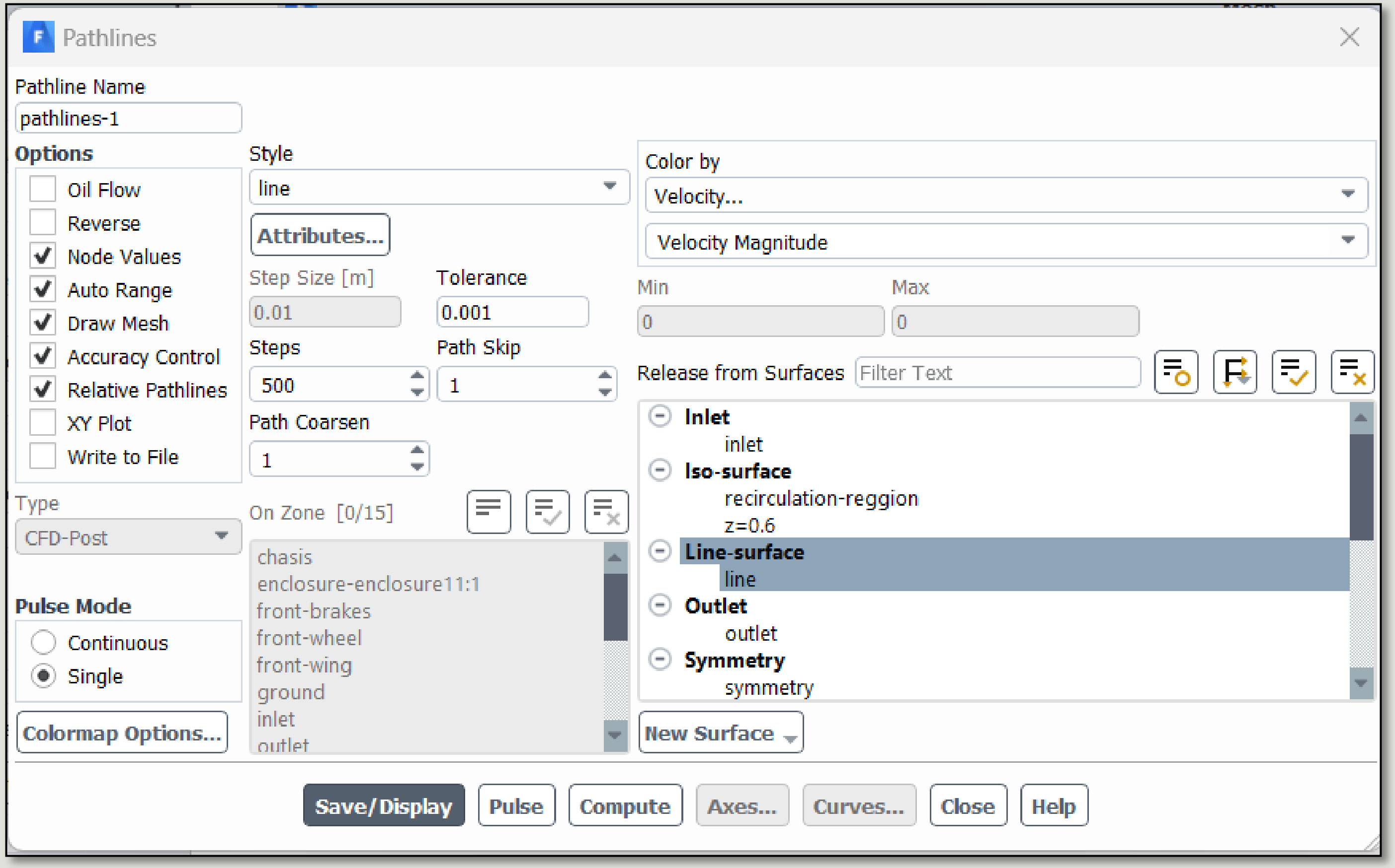


FIG. D15 – path line settings

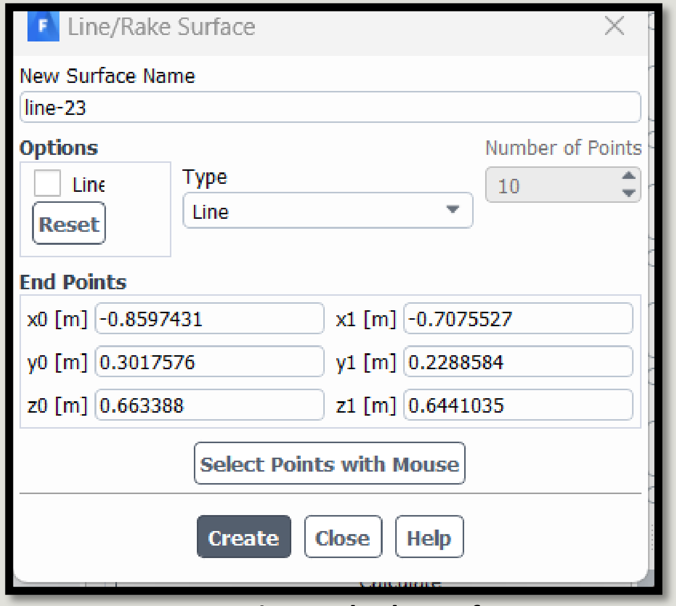


FIG. D16 – Line and rake surface

By examining the path lines, one can identify areas of flow stagnation, regions of high velocity, and the formation of vortices or turbulence. These flow characteristics play a crucial role in determining the aerodynamic forces acting on the vehicle and influencing its overall performance.

Analyzing the disparities in path lines allows for a deeper understanding of how design modifications impact the flow behavior and subsequent aerodynamic performance. It provides insights into areas where improvements can be made to optimize the flow patterns, minimize drag, enhance downforce, and improve the overall efficiency and stability of the vehicle.

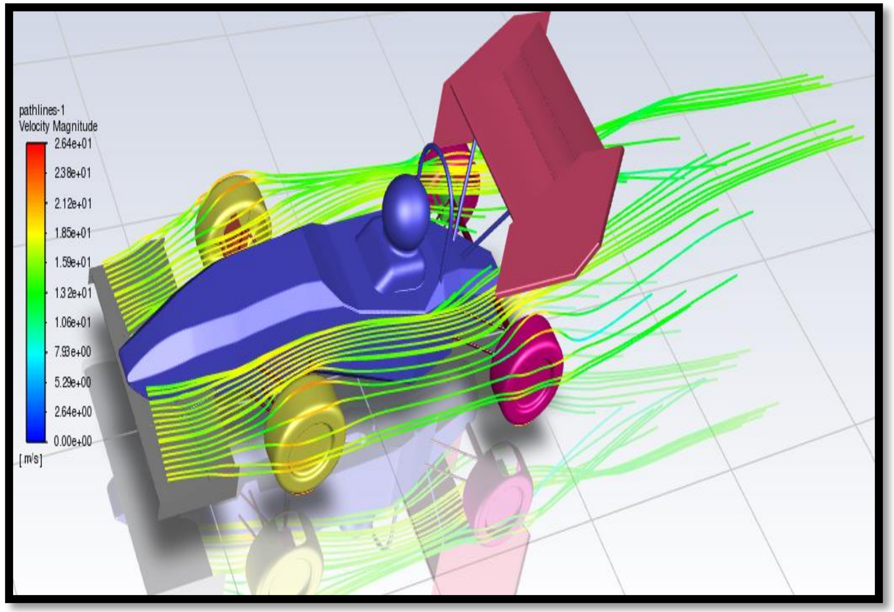


Fig D17 Path Line for FW

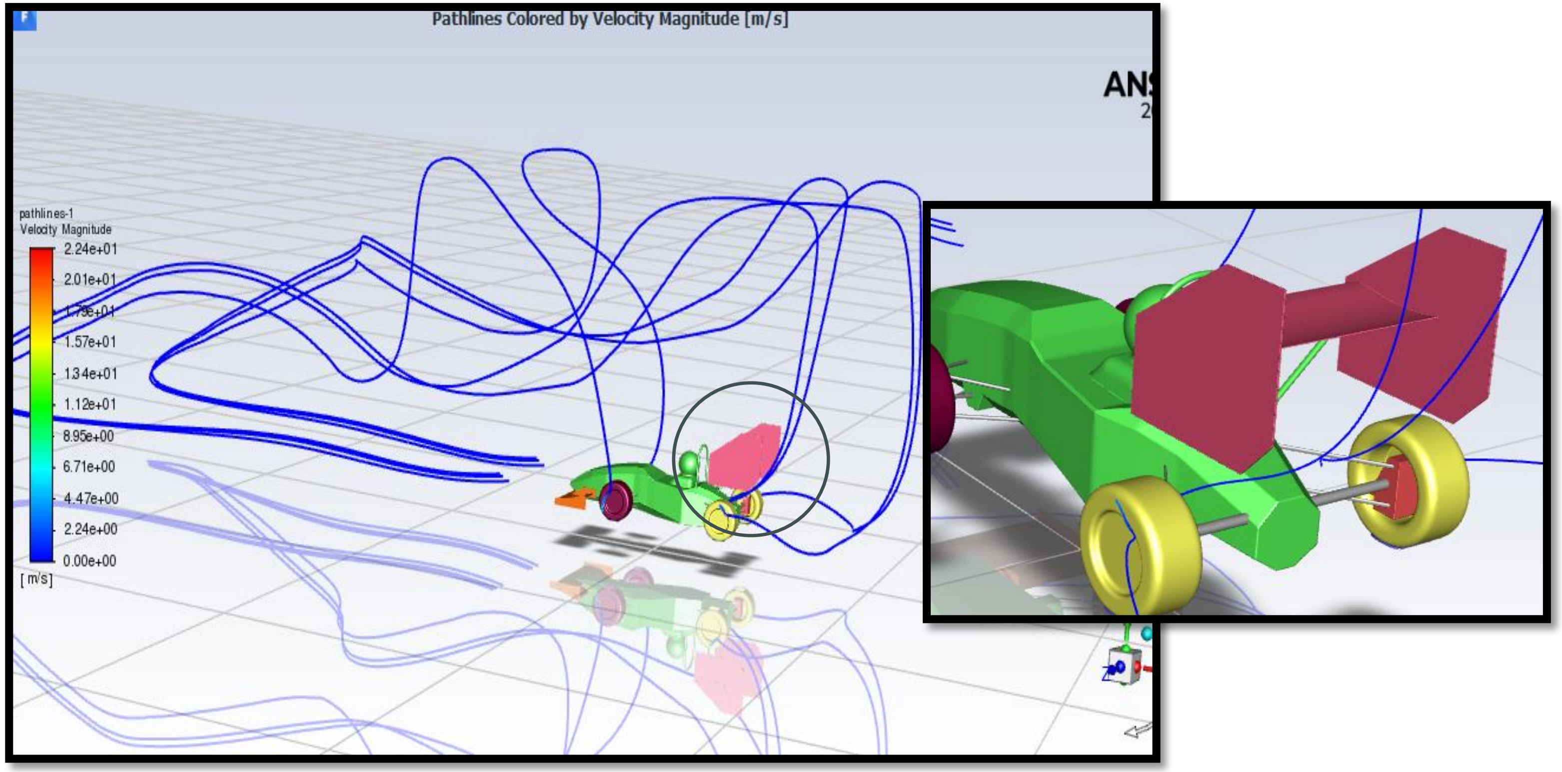


Fig D18 Path Line for SW

SUMMARY:

- In summary, the analysis of the FSAE car's aerodynamics involves several steps and considerations. The process begins with creating an enclosure around the car model, typically three times its size, to ensure accurate testing in a wind tunnel or virtual environment. The body of influence is defined to control the mesh size and refine the region of interest for precise results. Boundary conditions are set to establish the simulation parameters, and meshing is performed using ANSYS Fluent to create a surface mesh, boundary layer mesh, and poly hexacore volume mesh.

- For an incompressible flow simulation, default settings are typically used. Inlet conditions, such as turbulence intensity and turbulent ratio, are adjusted based on the expected characteristics of the wind tunnel. Solver iterations are conducted, with monitoring and analysis of the plotted data to identify steady-state conditions and convergence.

- Comparisons are made between different wing configurations, assessing variations in area, forces, and flow behavior. The presence of oscillations in the data is observed and analyzed for their significance and implications.

- The analysis also reveals the importance of generating negative forces towards the ground for compliance and stability. The presence of a wake region and its size in velocity contours provides insights into the aerodynamic performance and potential areas for improvement.

- Overall, the analysis of the FSAE car's aerodynamics helps in understanding its behavior, optimizing design, and enhancing performance by considering factors such as forces, pressure distribution, wake region, and flow characteristics.

THANK YOU!

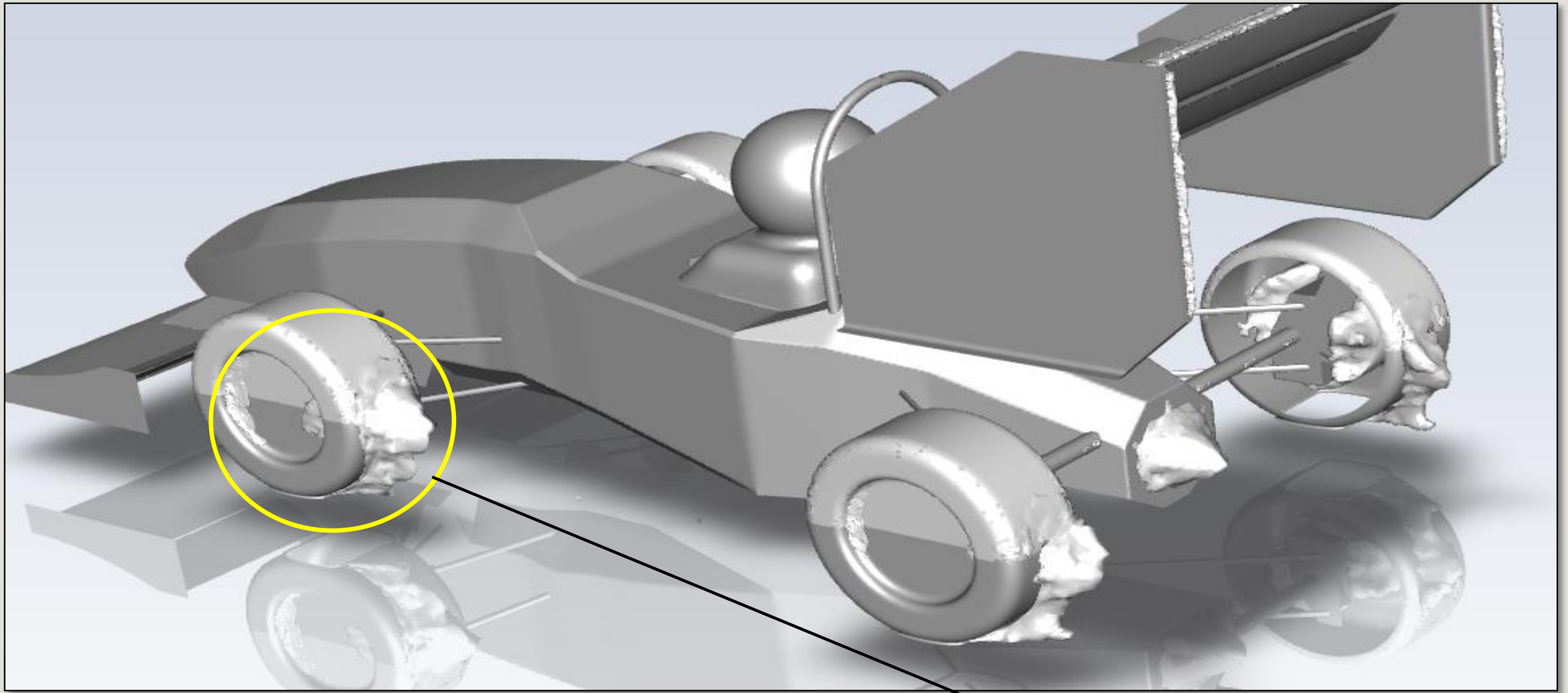


FIG. X 1– Velocity of X - Axis under ISO - SURFACE

Recirculation region in negative x velocity where the flow is reversed.

In order to run sweep surfaces, certain settings are made to do so.

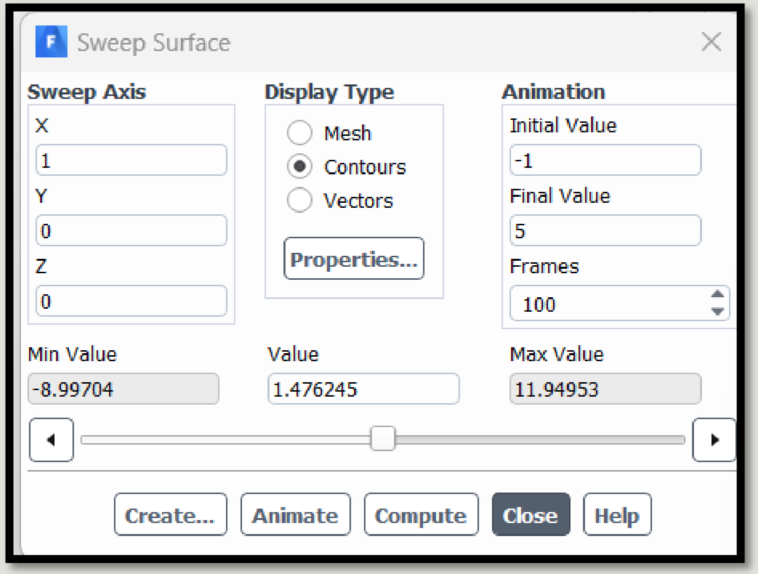


FIG. Y1– Sweep surface settings

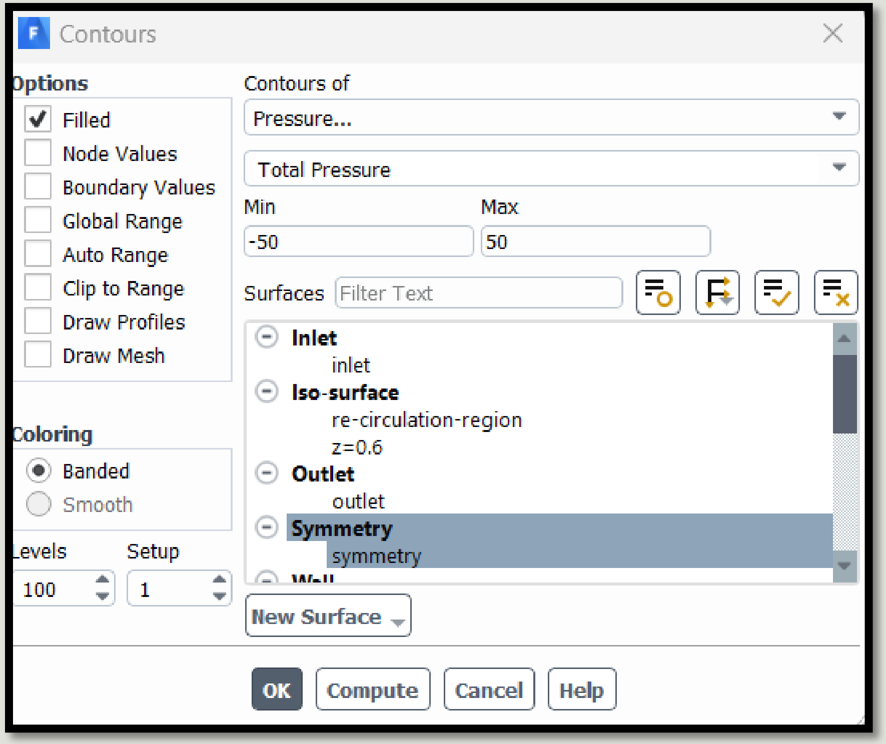


FIG. Y2– Sweep surface settings

We can see that low energy regions are associated with the vertices in the wake region as well as the rear region of the car, through the sweep surface plot.

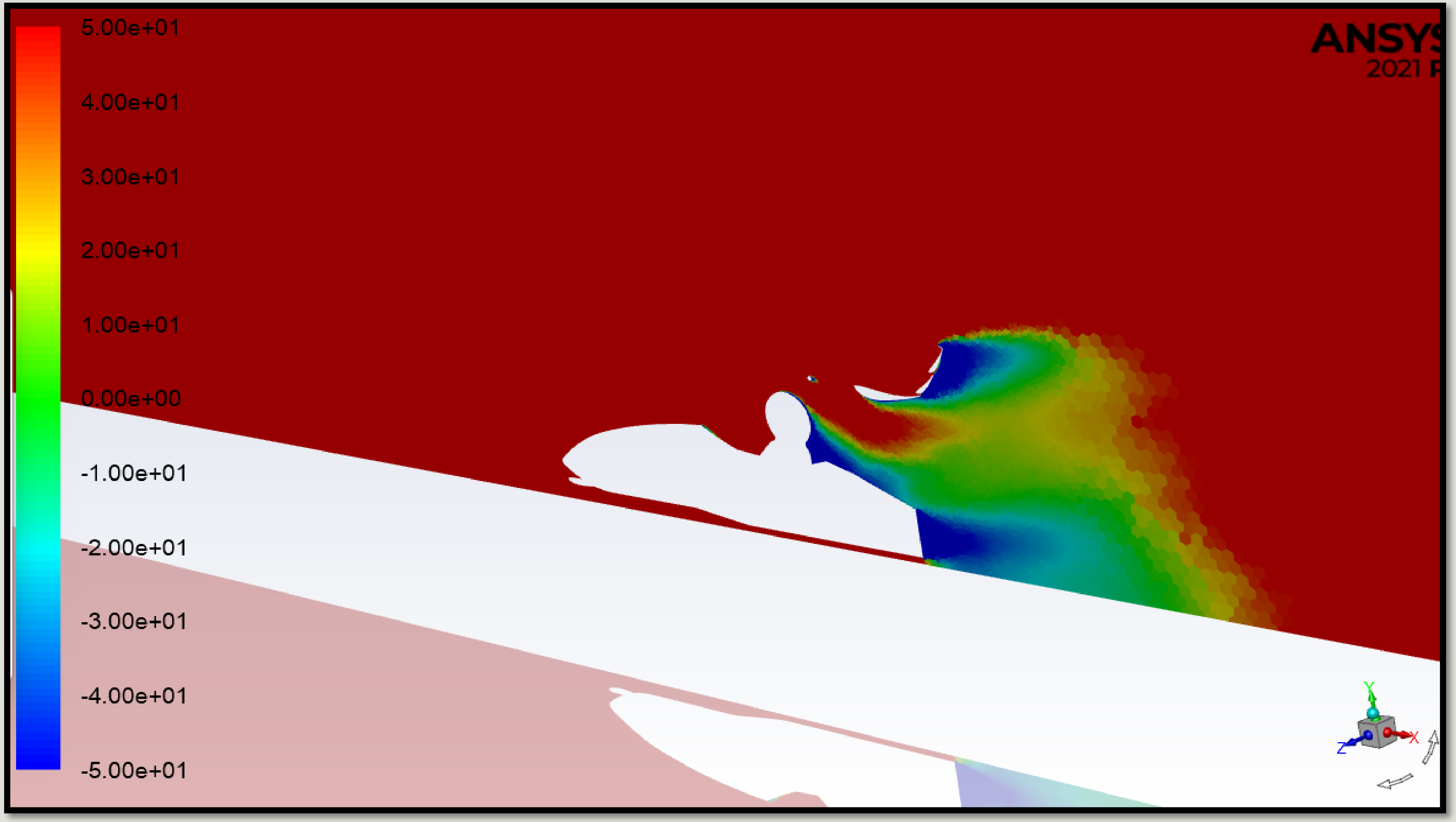


FIG. Y3– Sweep surface contour plot