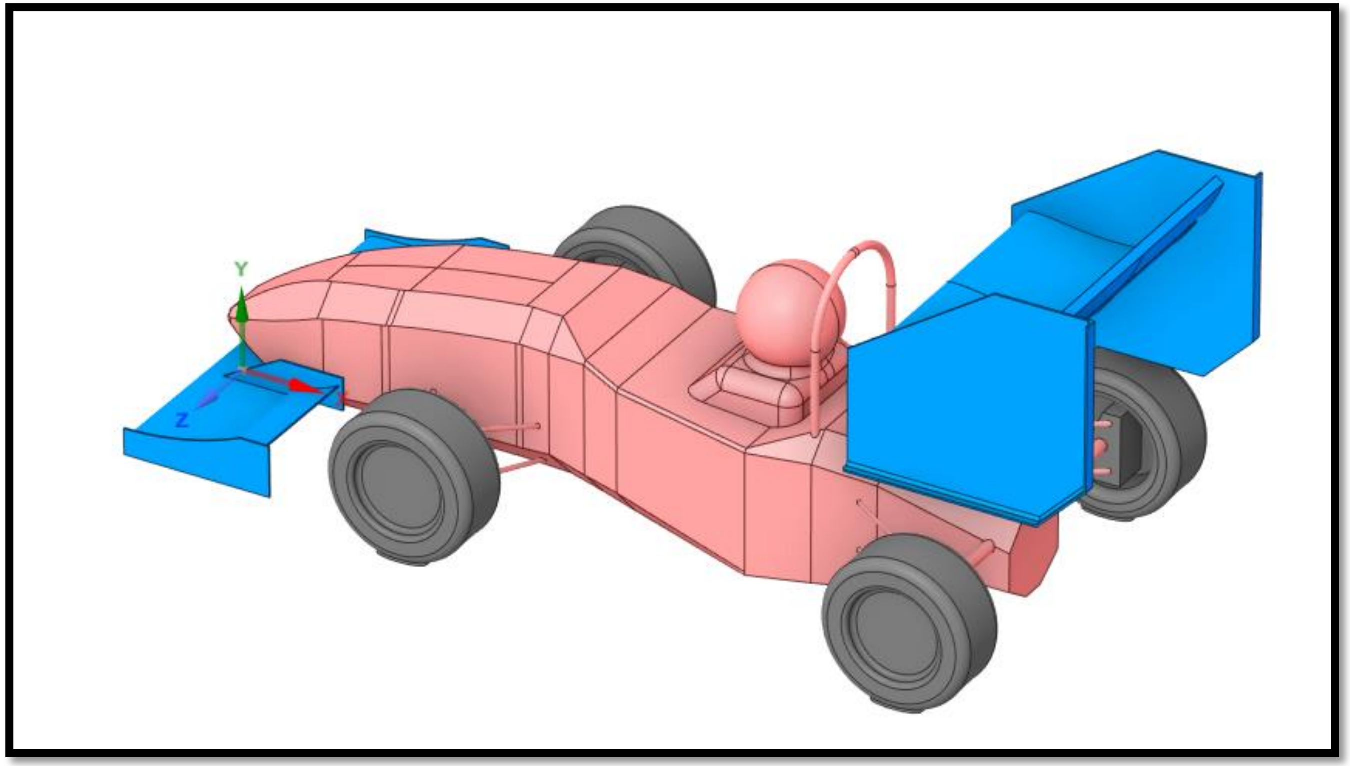
AERODYNAMICS OF FASE CAR – CFD PROJECT

GUIDED BY: DR. MEHMET SARIMURAT

-JAYAPRAKASH CHANDRAN
522242685



PHASE – 1

PREPARATION OF FASE CAR IN ANSYS SPACE CLAIM

- In order to prepare the body, we are creating the Enclosure three times the car to produce the fluent result that seems to be tested in the wind tunnel.

- Body of influence is done to control the mesh size and refine the region of thrust, where it is used to have accurate results in the vicinity of the car.

- Name selection is prepared for the car to run the boundary conditions during the mesh phase.

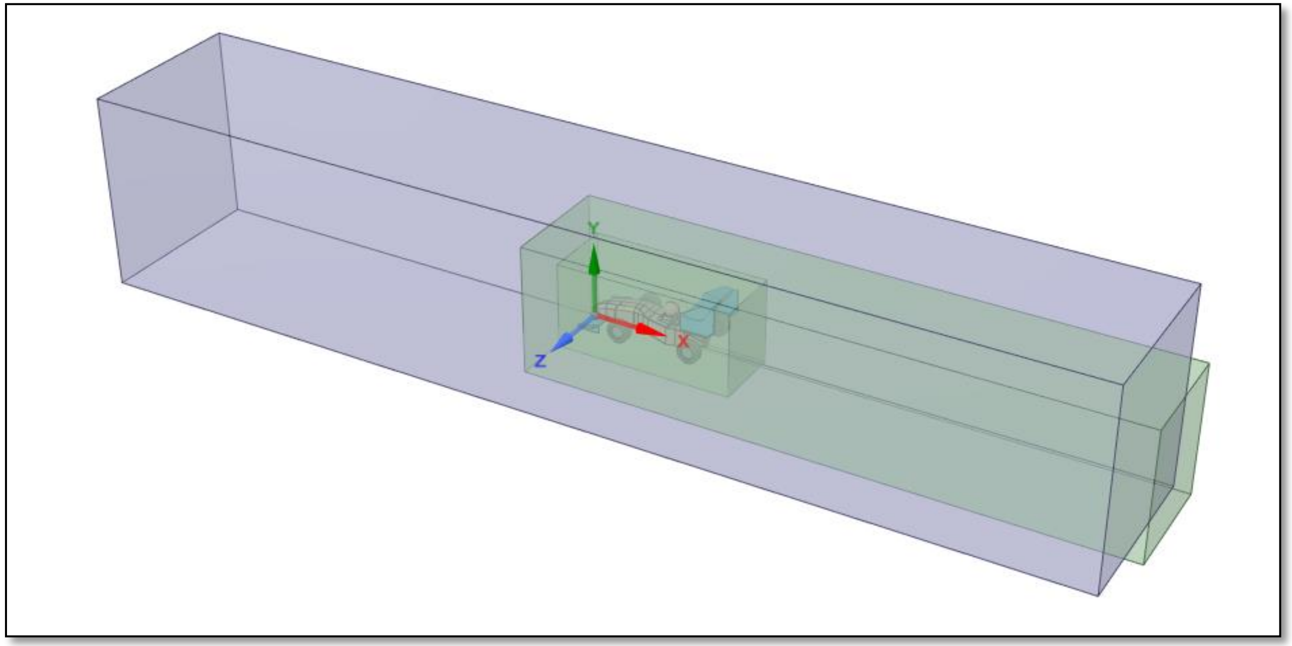


FIG. A. Enclosure and BOI

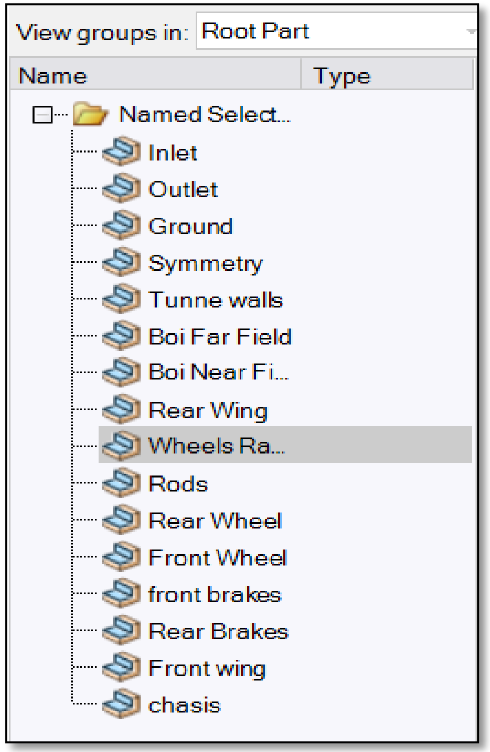
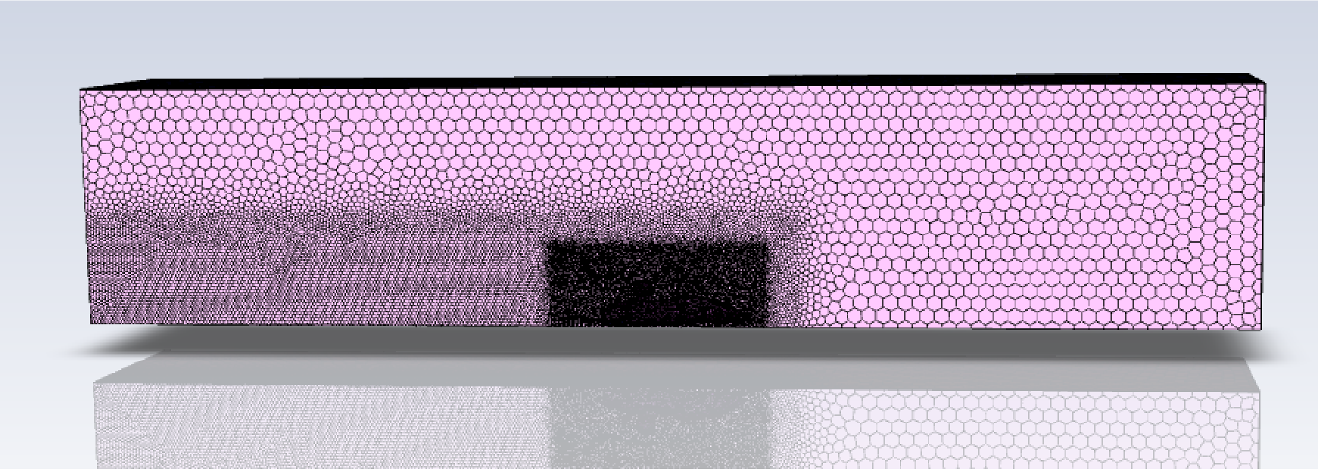


FIG. B. Name selection panel



PHASE – 2

TO CREATE THE COMPUTATIONAL MESH

- Using Ansys fluent meshing we are creating the following properties such as surface mesh, boundary layer mesh, and poly hexacore volume mesh to analyze the flow of the FSAE Car.

- To create precise meshing, I have created the local sizing for the body of influence and curvature to analyze the flow in the wake region and kept the desired settings for the surface mesh.

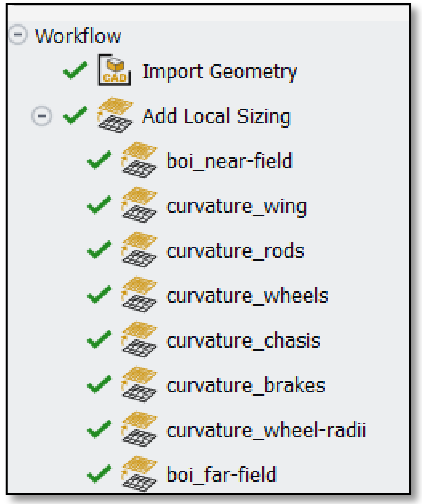


FIG. C. Workflow pattern

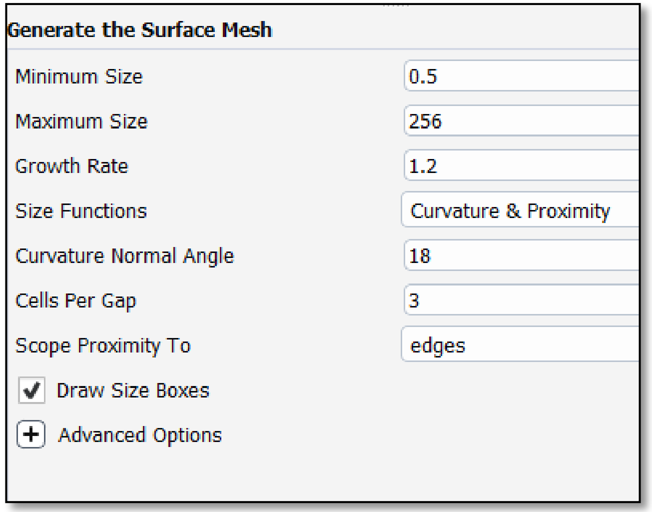


FIG. D. Mesh settings

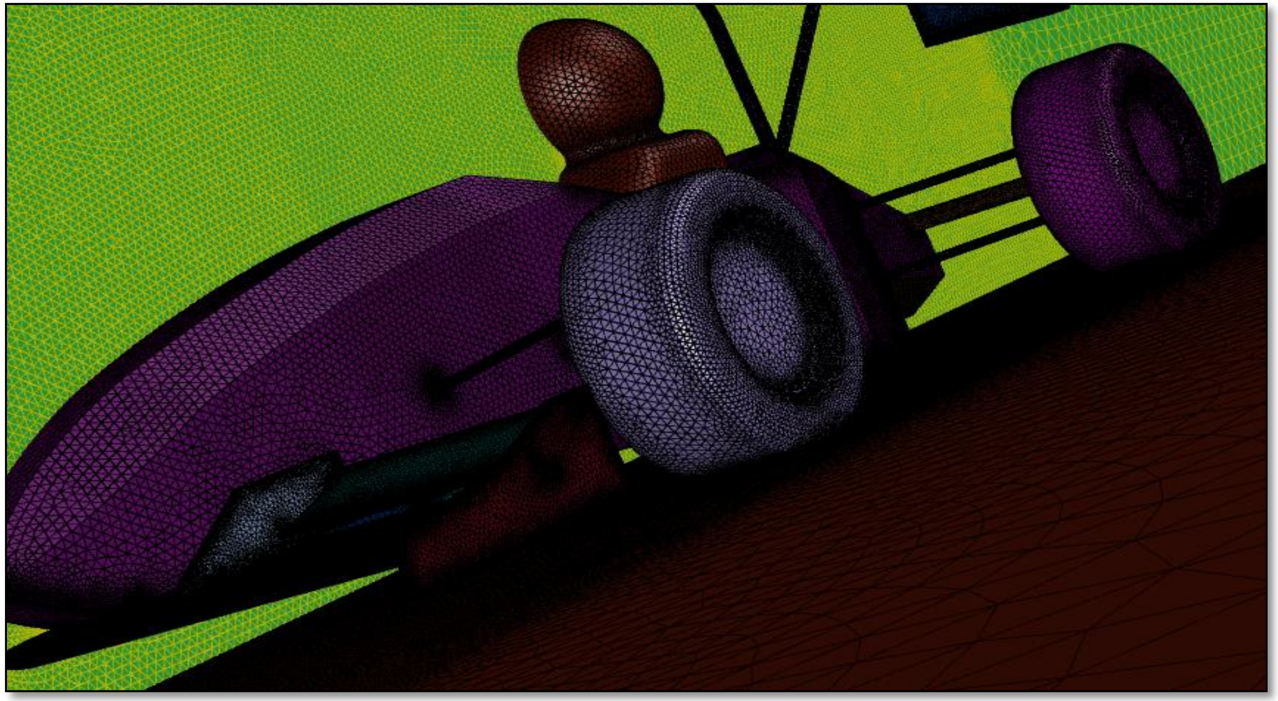


FIG. E. Surface Mesh with skewness 0.59

- Now, the boundary layer mesh is created to accurately capture the flow physics. Moreover, additional mesh requirements are needed to understand the prism layers required for the external flow problems.

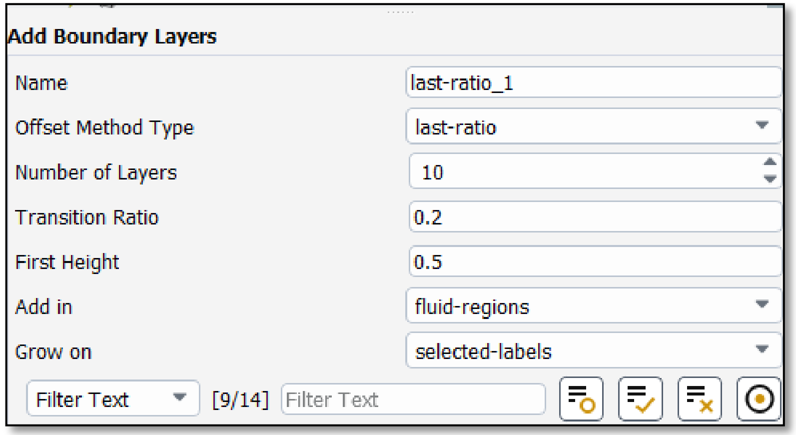


FIG. F. Boundary Layer conditions

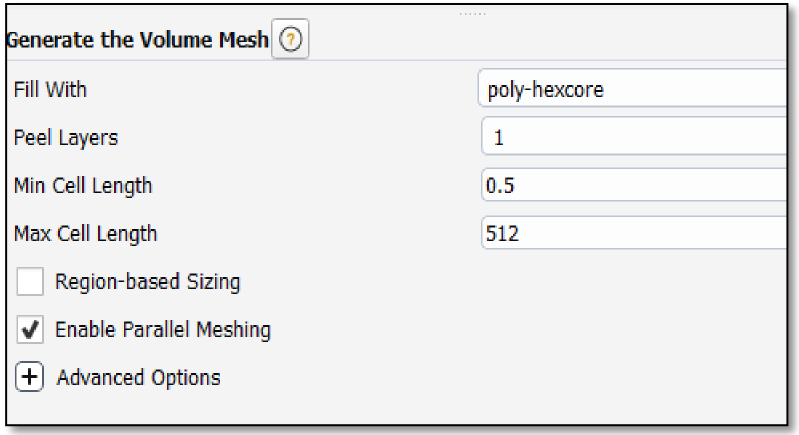


FIG. G. volume mesh conditions

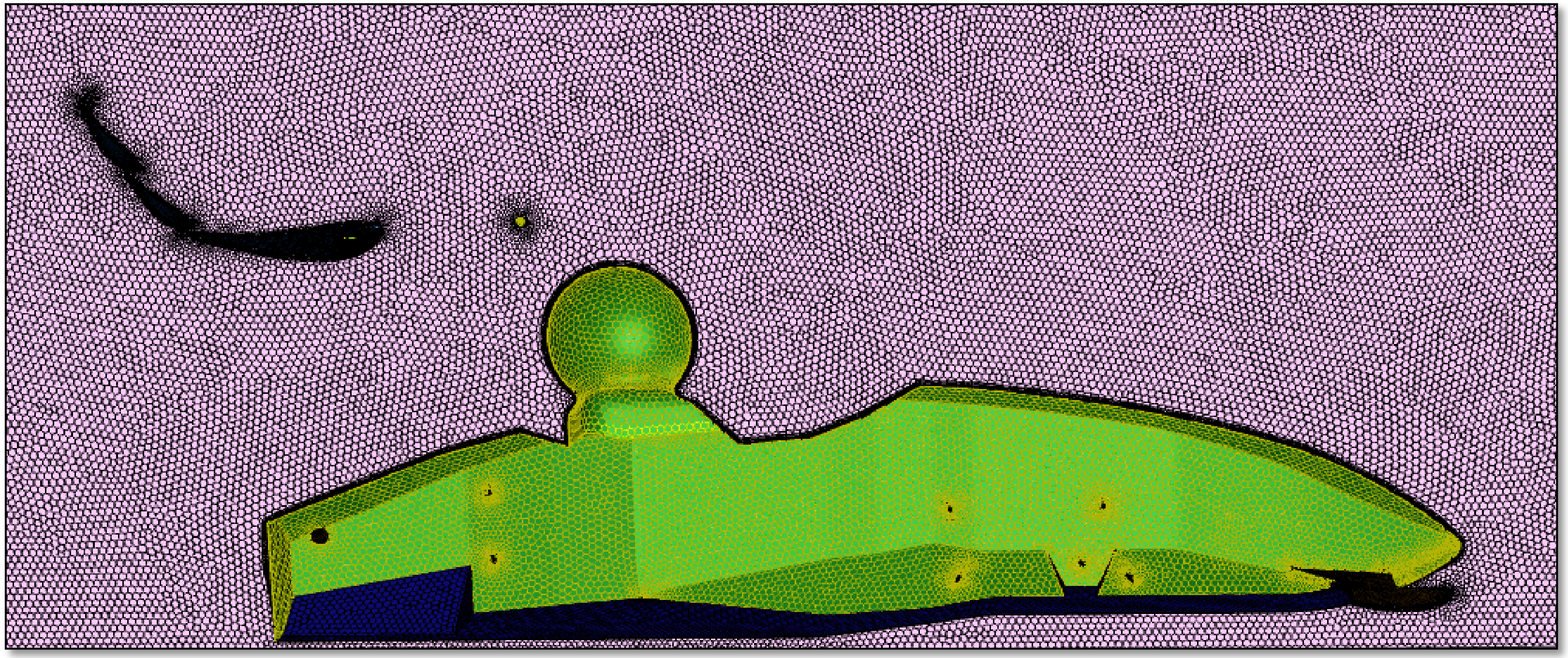


FIG. H1. volume mesh

```
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. H2. Orthogonal Quality

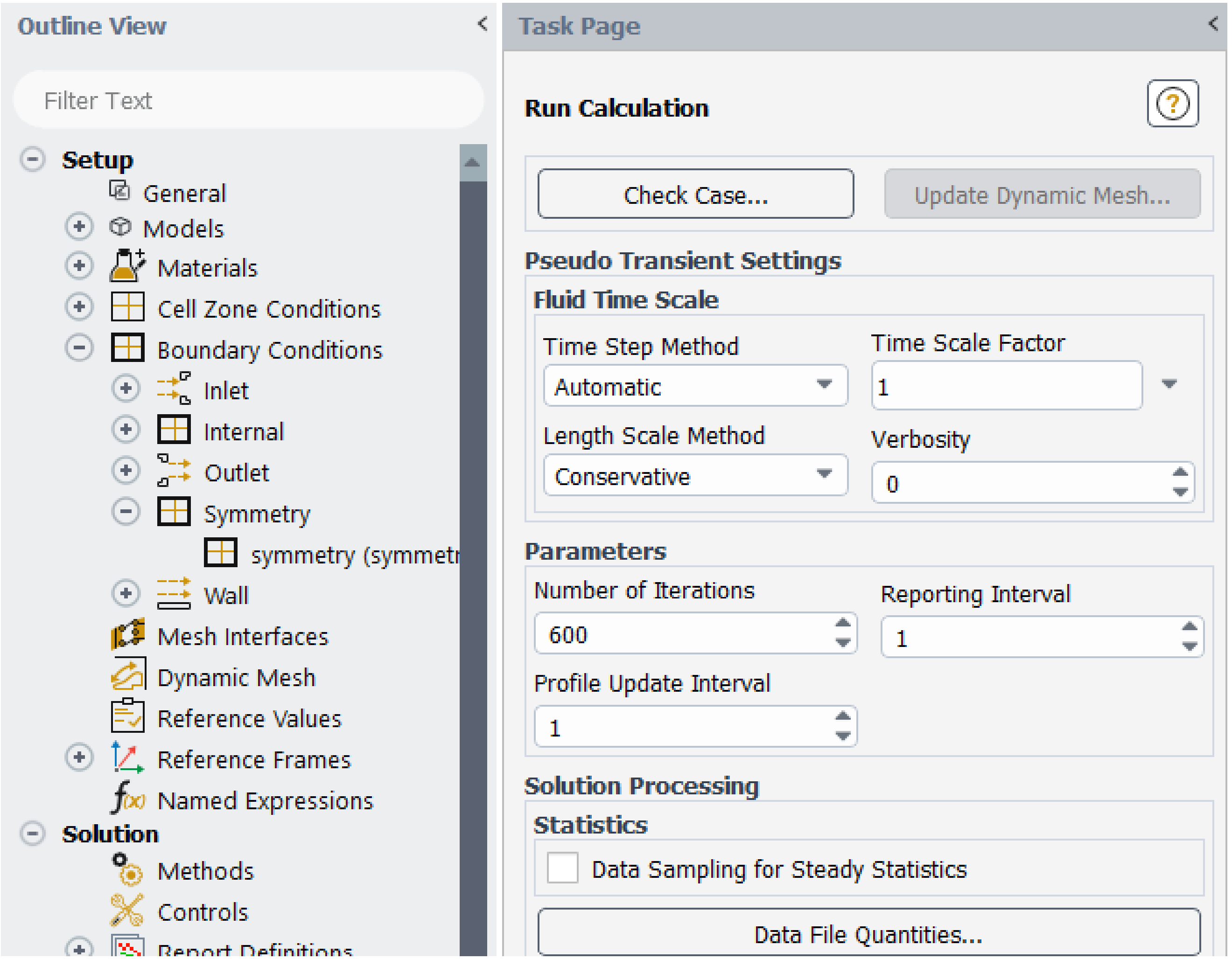
```
Mesh Quality:

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. H3. Mesh Quality

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



PHASE – 3

SOLVER SETUP

Staring with mesh check and quality check to ensure everything is in good standing. After that, we intend to perform physics conditions under pressure-based and steady-state solvers. Finally setting up a turbulence module, since the Reynolds number based on the car length is said to be 4 million, so the flow is assumed to be turbulent.

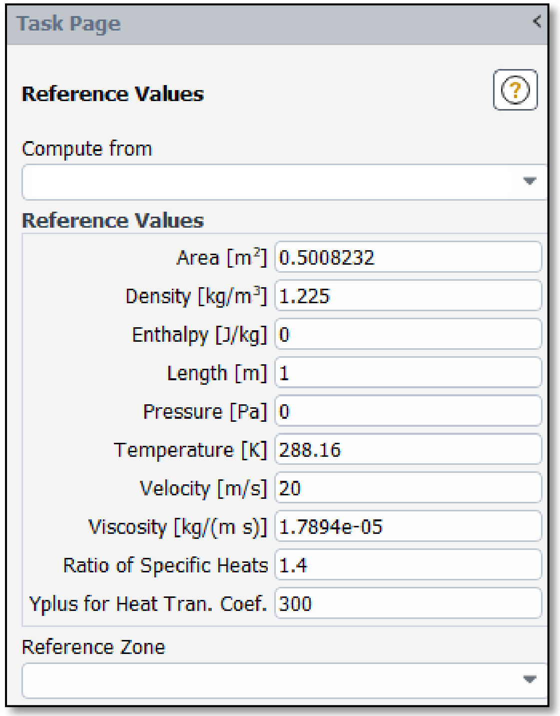


FIG. J. Reference Conditions

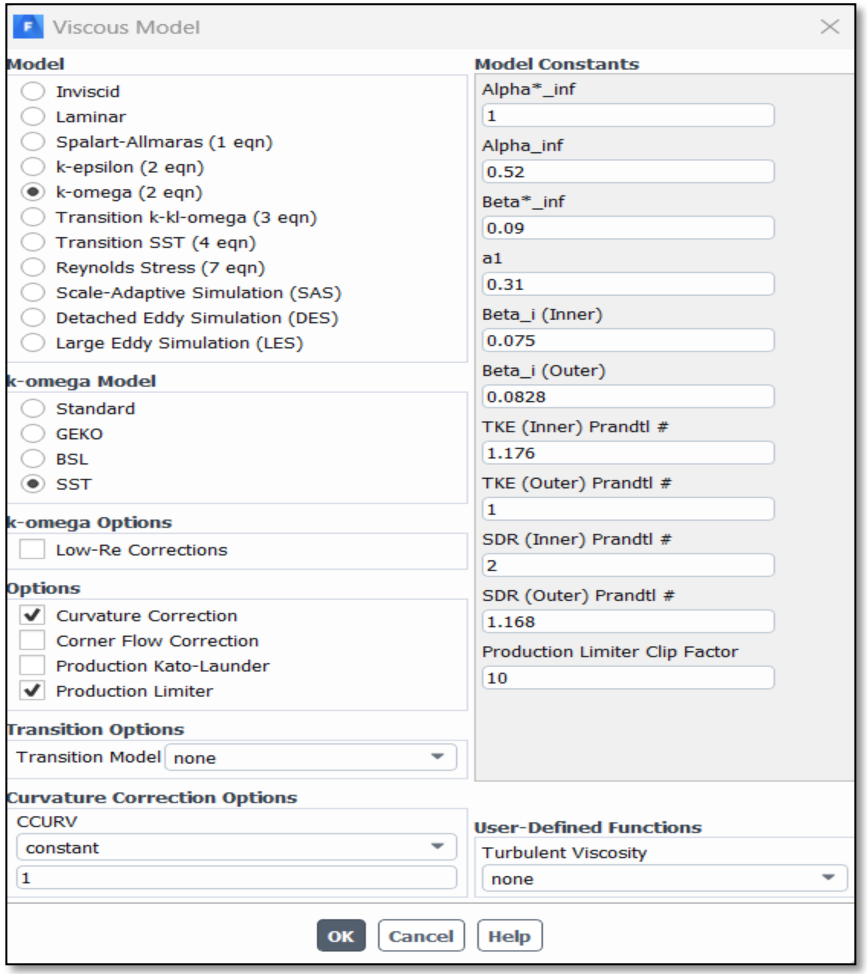


FIG. K. Viscous model

Since the flow in this simulation is expected to be incompressible, so we go with the default settings.

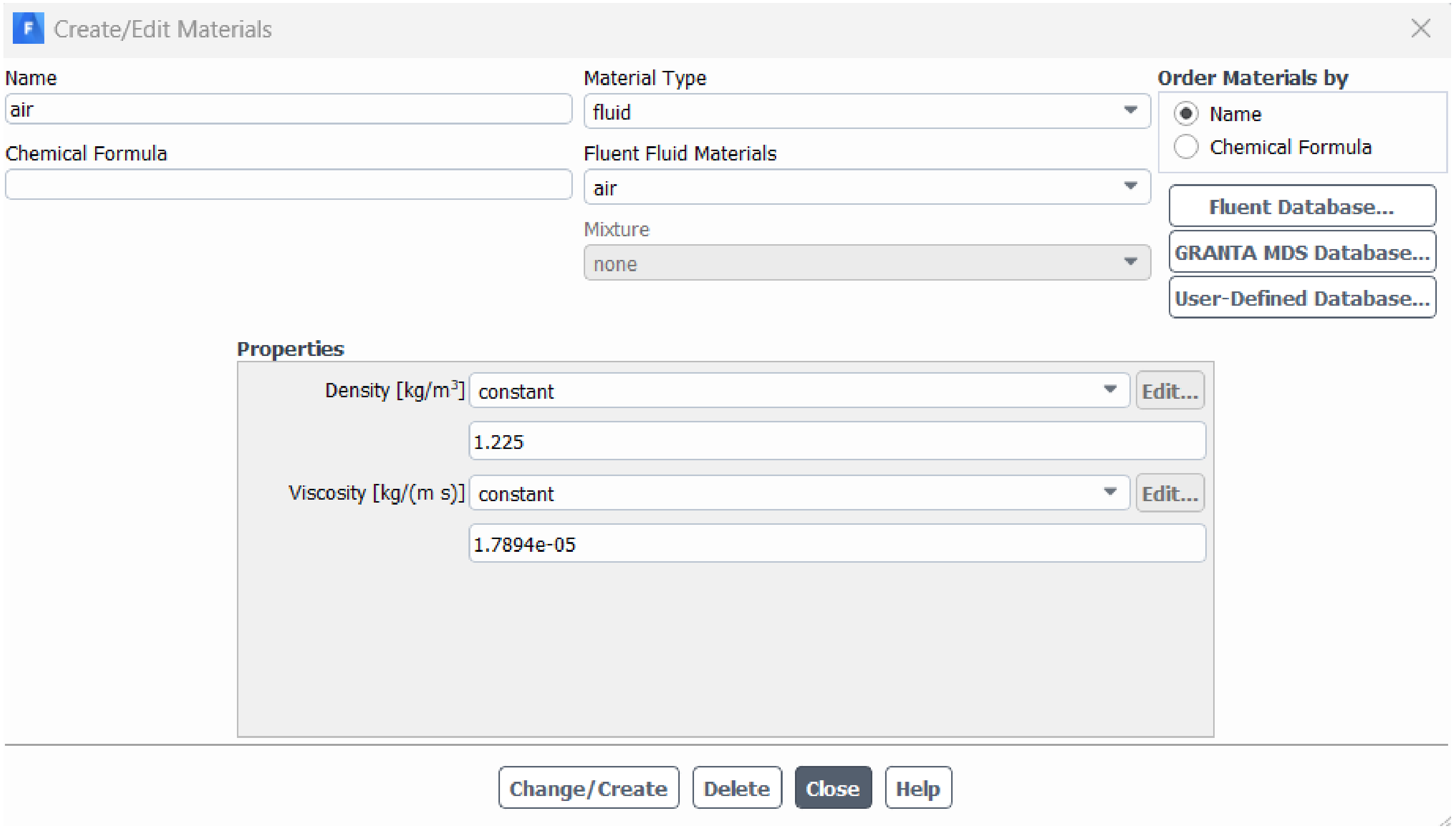


FIG. J. Reference Conditions

- Since the flow in this simulation is expected to be incompressible, so we go with the default settings.
- Since the inlet of the wind tunnel is usually very low, so we set the intensity as 0.5 and the turbulent ratio as 2.

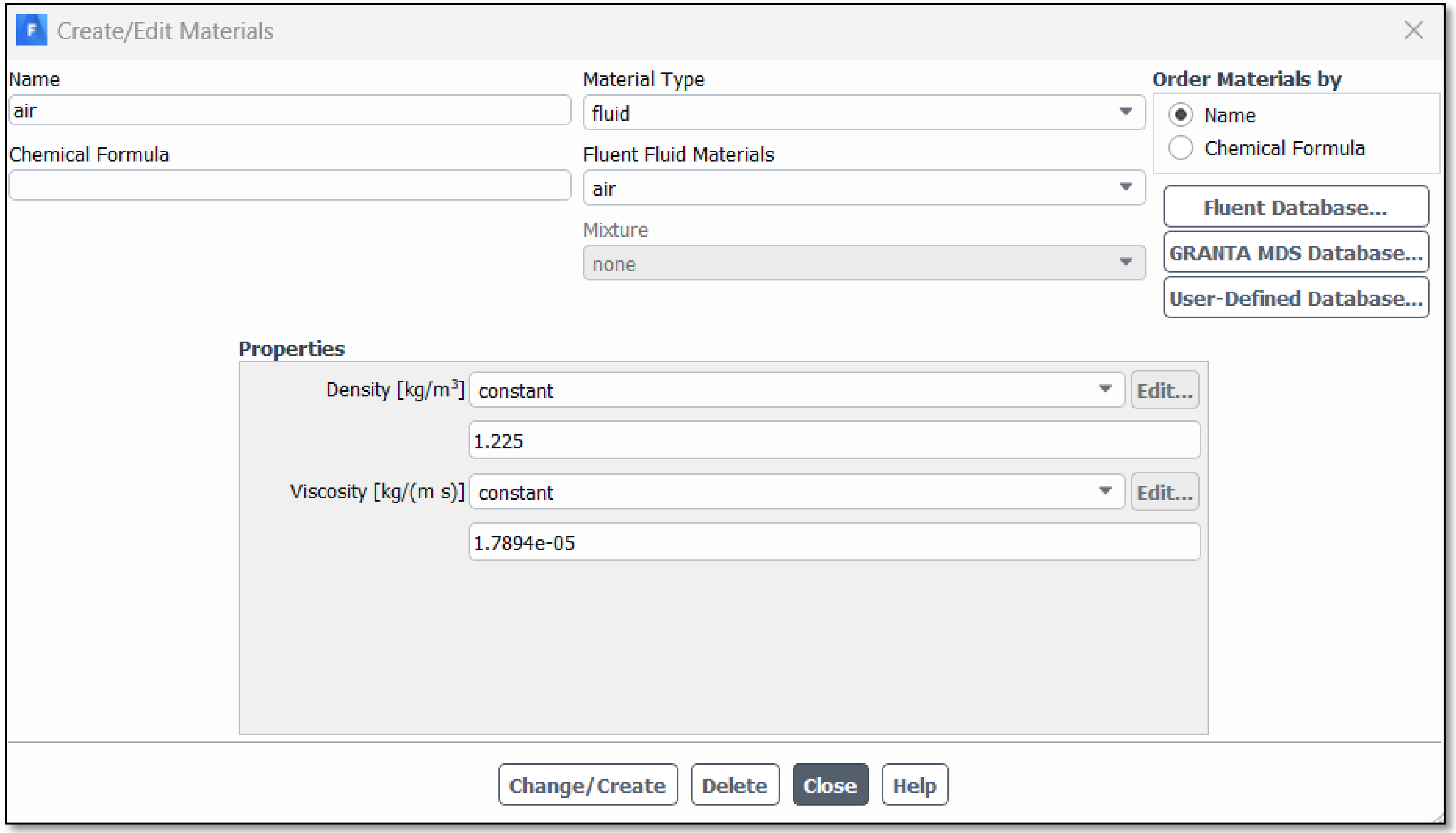


FIG. K. Material Conditions

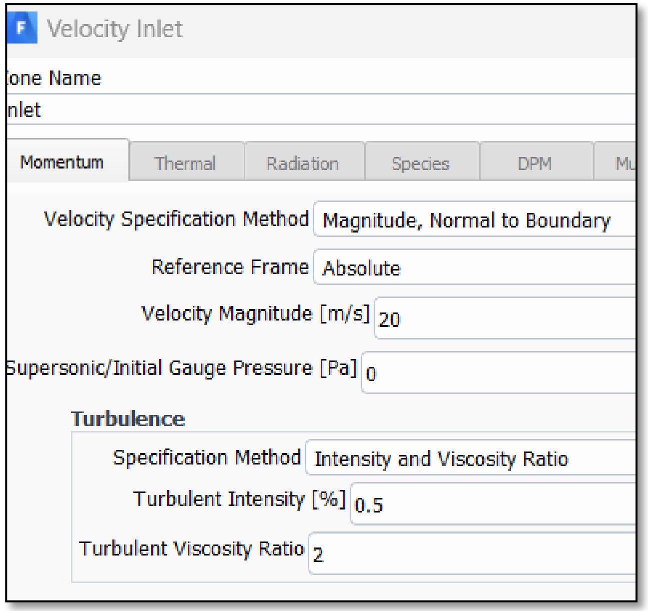


FIG. L. Velocity Inlet

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.

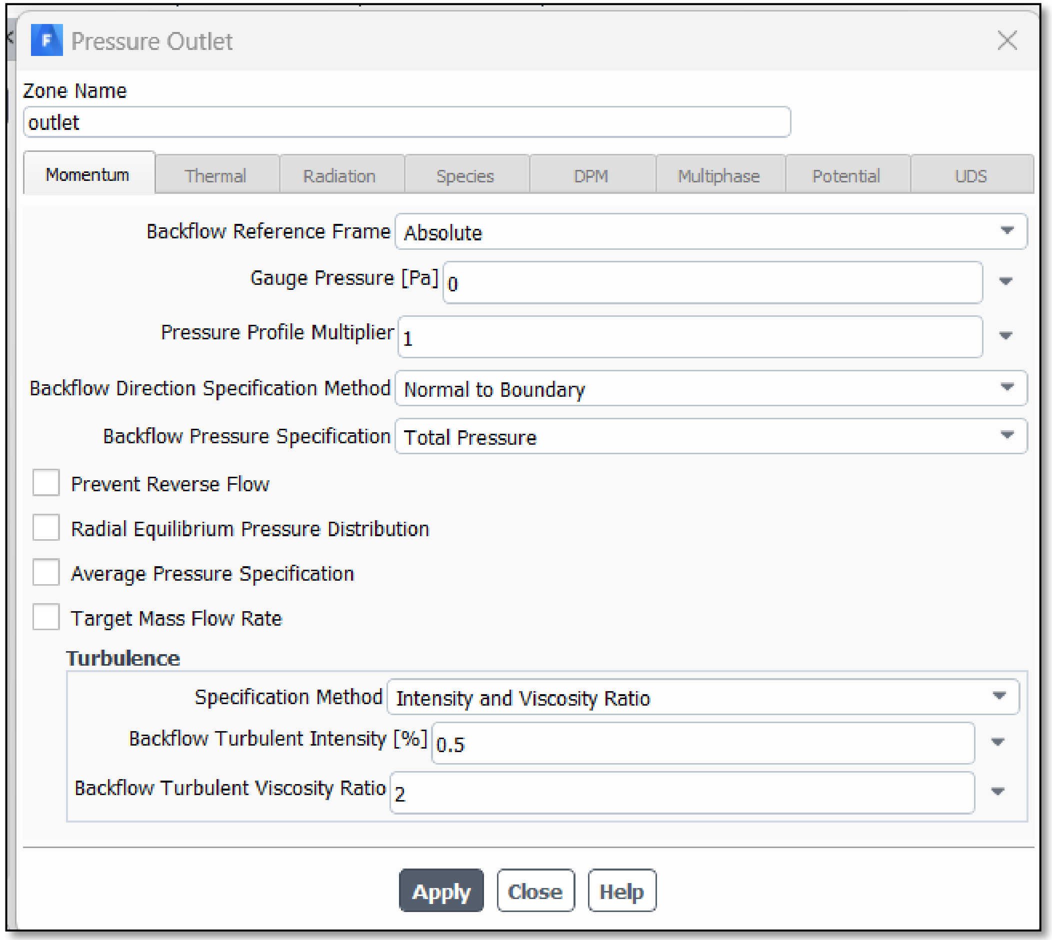


FIG. M. Pressure outlet

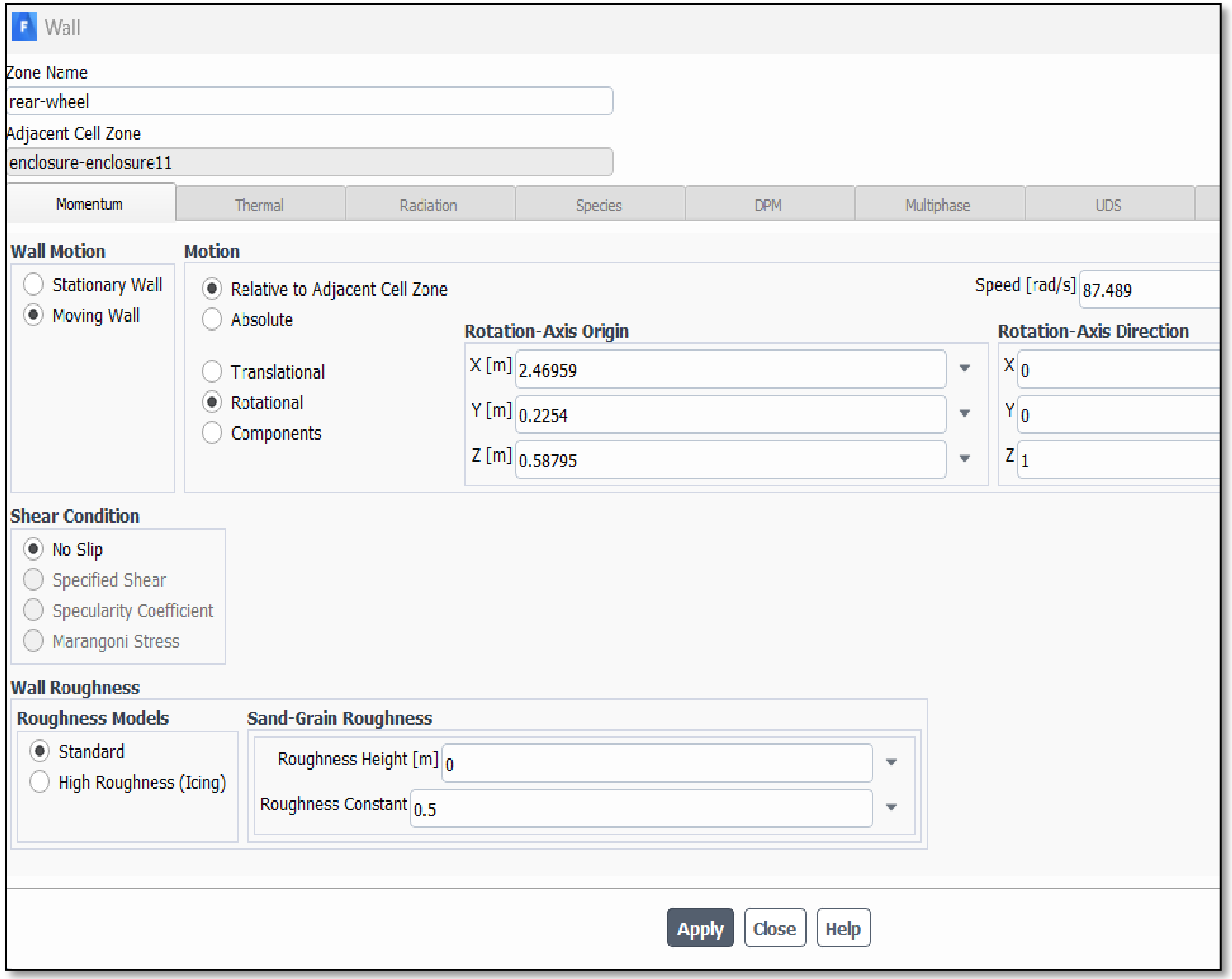


FIG. N. wall Conditions

To run the iteration in the solver setup, certain conditions are needed to be set up for the fluent to run along with FMG initialization. In these mesh checks, we analyzed the material, settled up the proper boundary conditions, and monitored the solution.

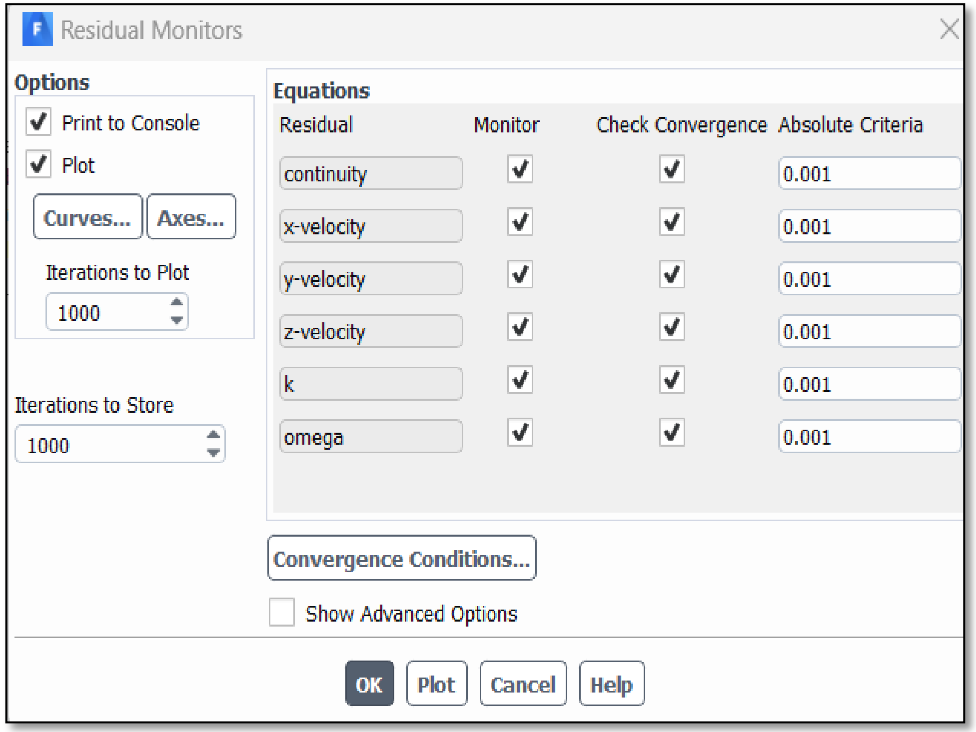


FIG. O. Residual Monitors

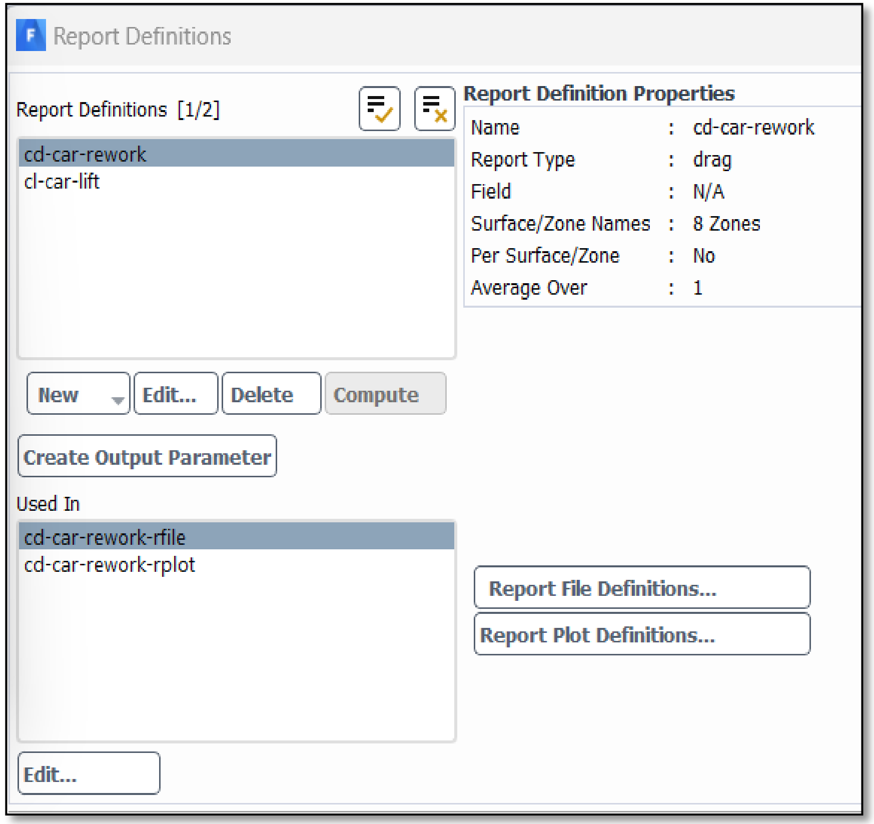


FIG. P. Definition Reports

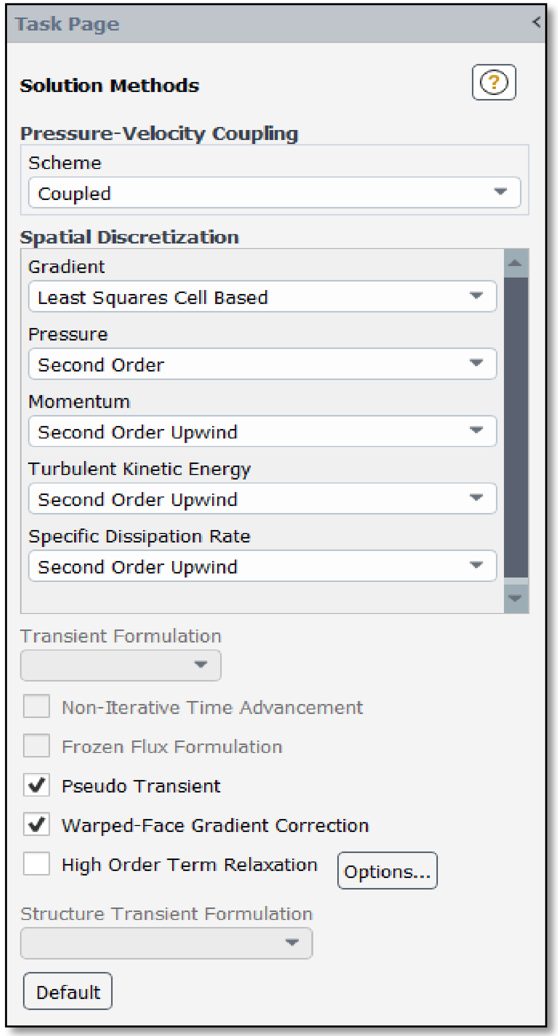
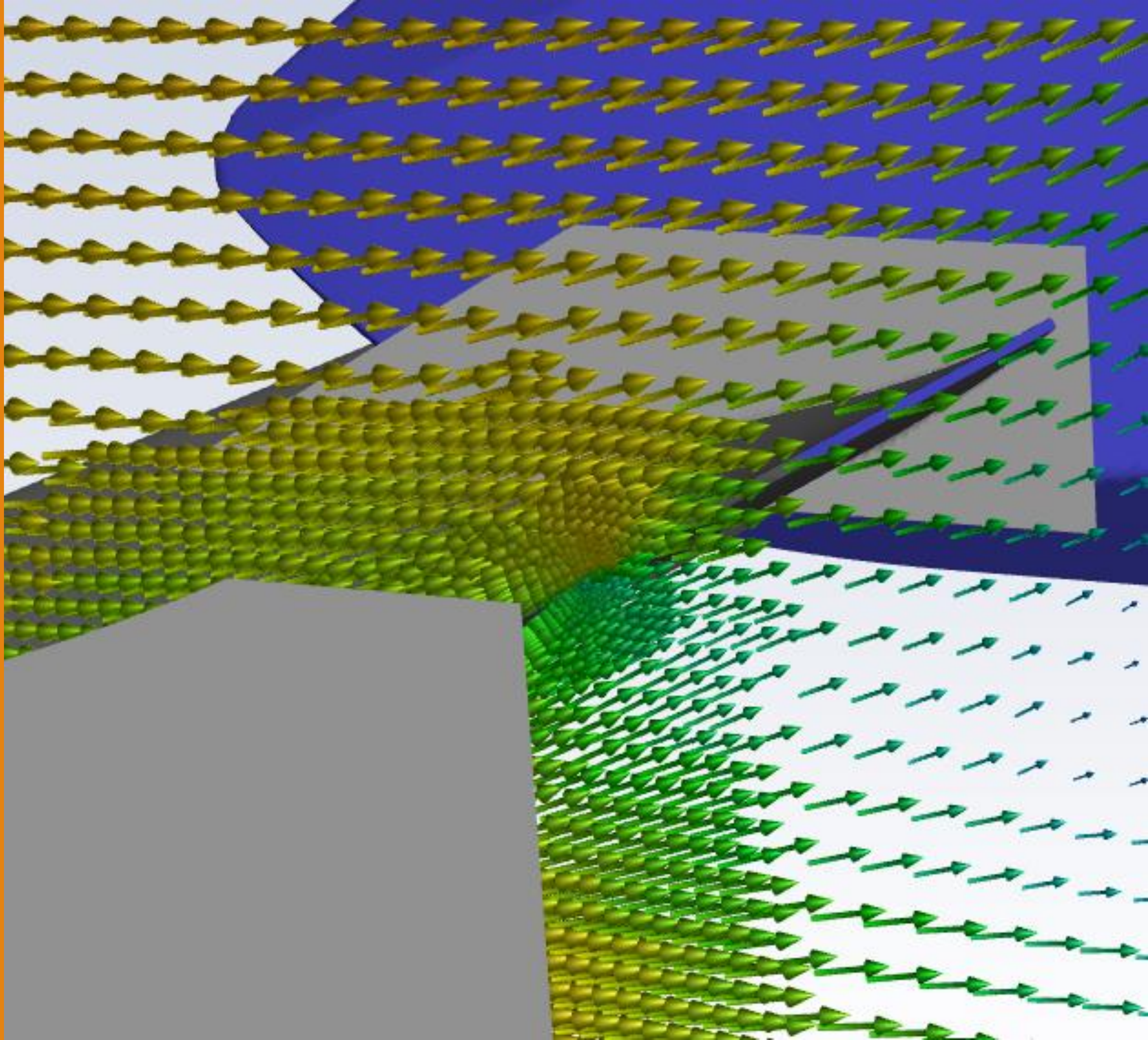


FIG. Q. Solution Methods

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)

	Forces [N]			Coefficients		
Zone	Pressure	Viscous	Total	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. R. Forces direction – X axis

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 (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. S. Forces direction Y axis

In this Y-axis console, the total forces are in a negative direction, where the force acts the car to keep on the ground, where it is determined as the downforce.

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

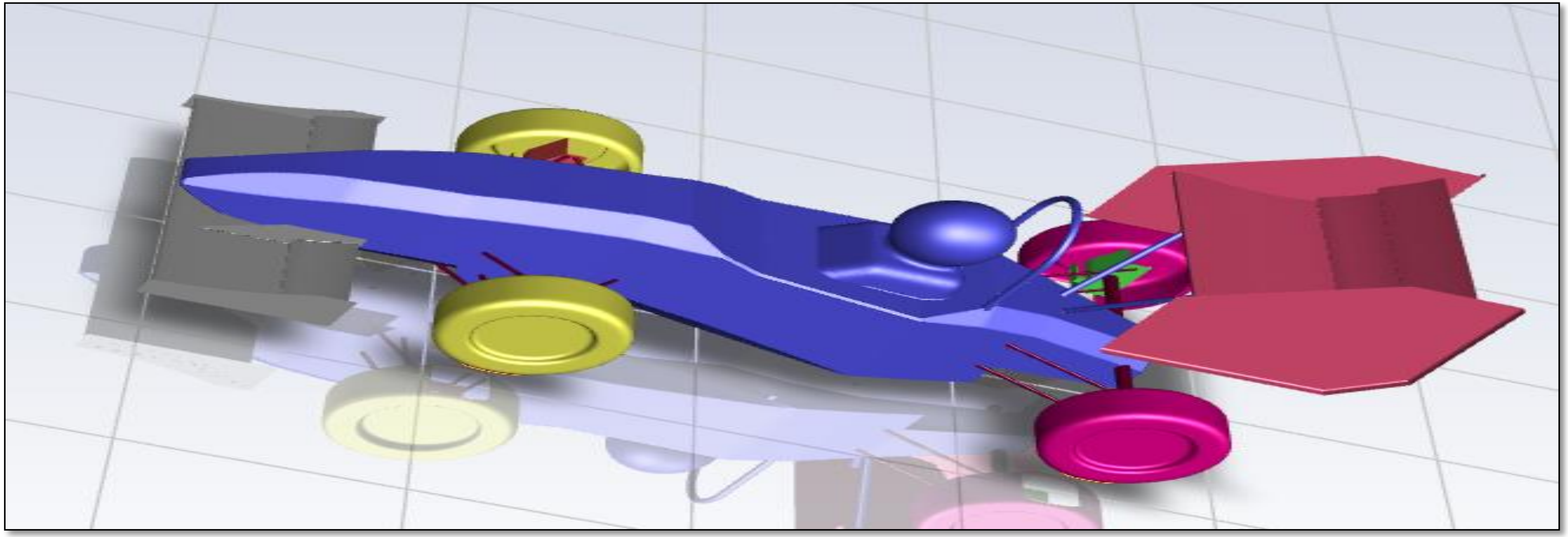


FIG. T. Graphical representation of the full car

Here we see the velocity contour of the car:

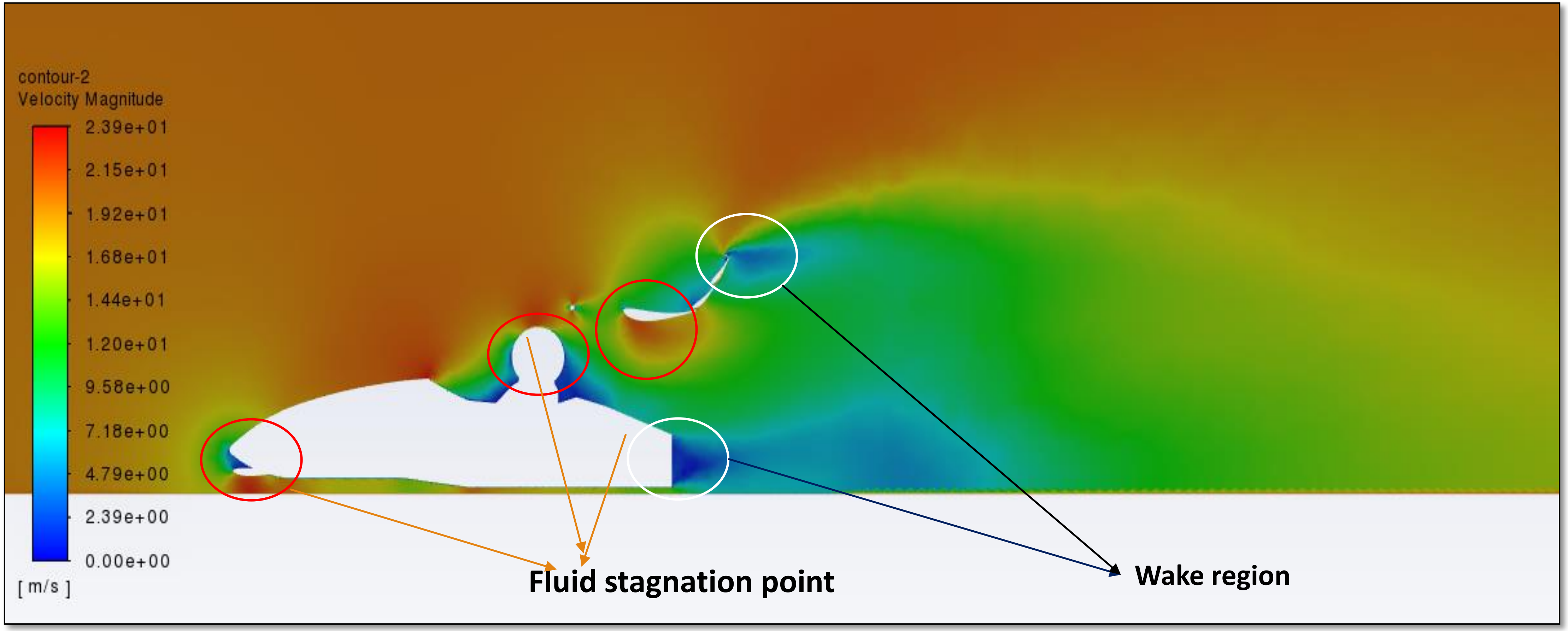


FIG. U. Velocity contour of the car

Here we see the pressure contour of the car:

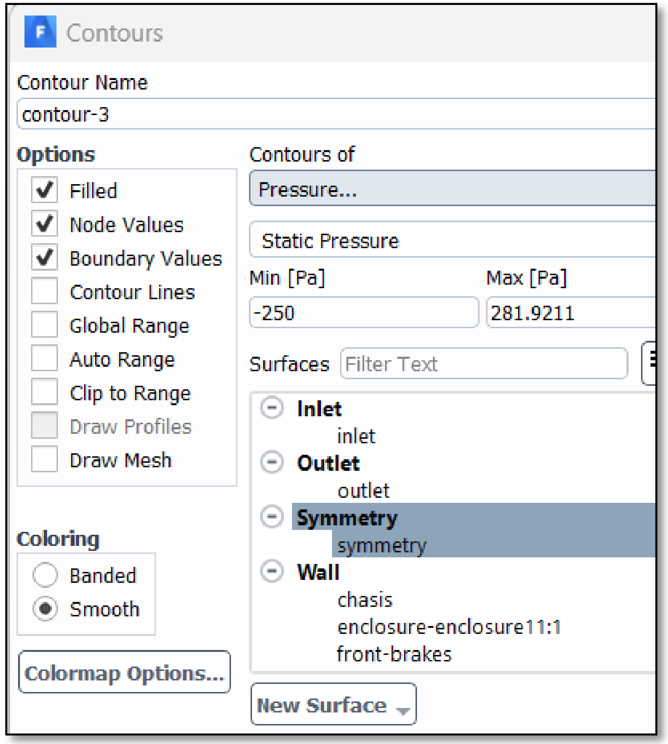


FIG. V. Pressure contour of the car

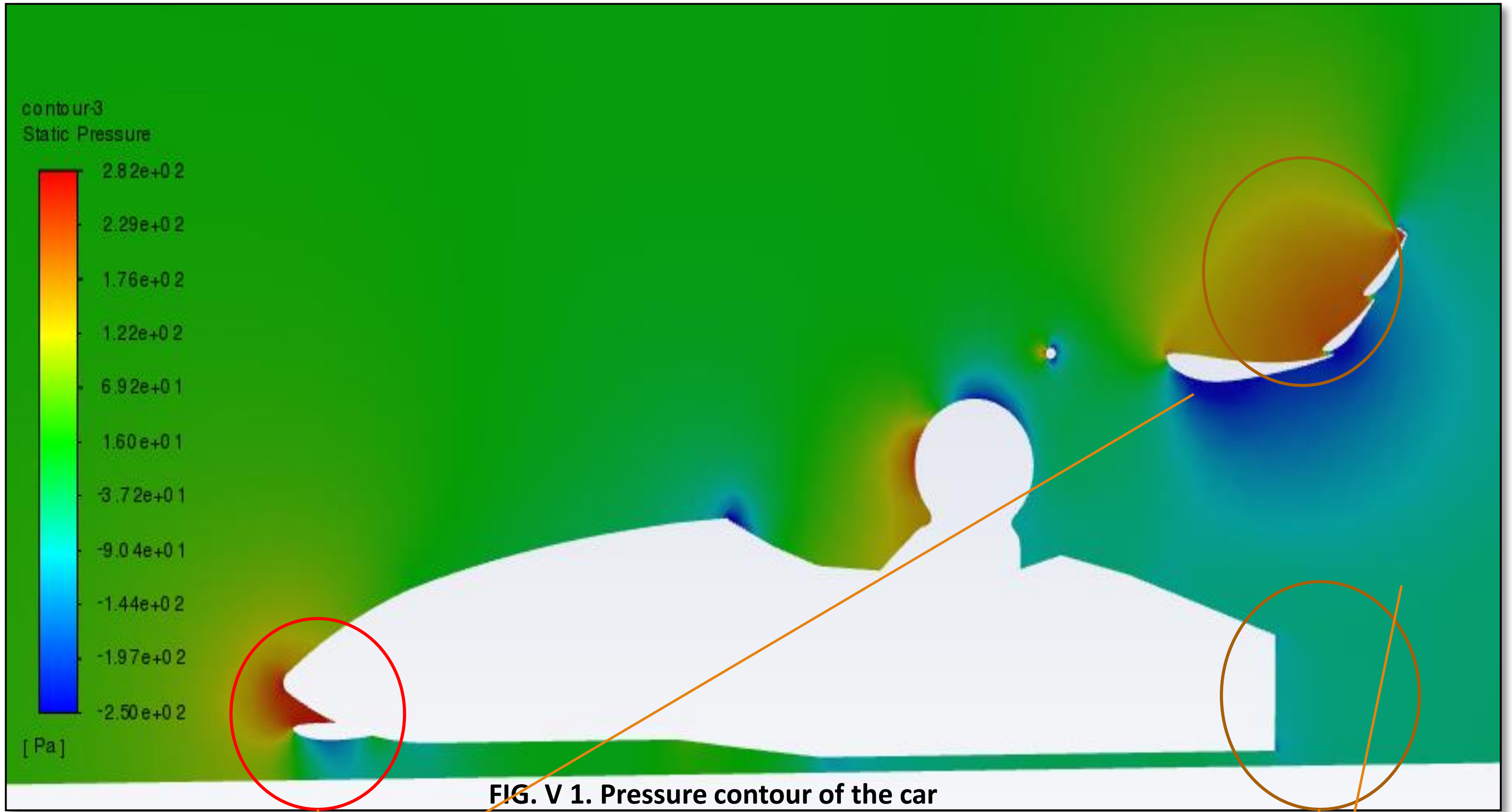


FIG. V 1. Pressure contour of the car

HIGH PRESSURE due to flow stagnation

LOW PRESSURE due to flow separation

Here we see the velocity vector of the car:

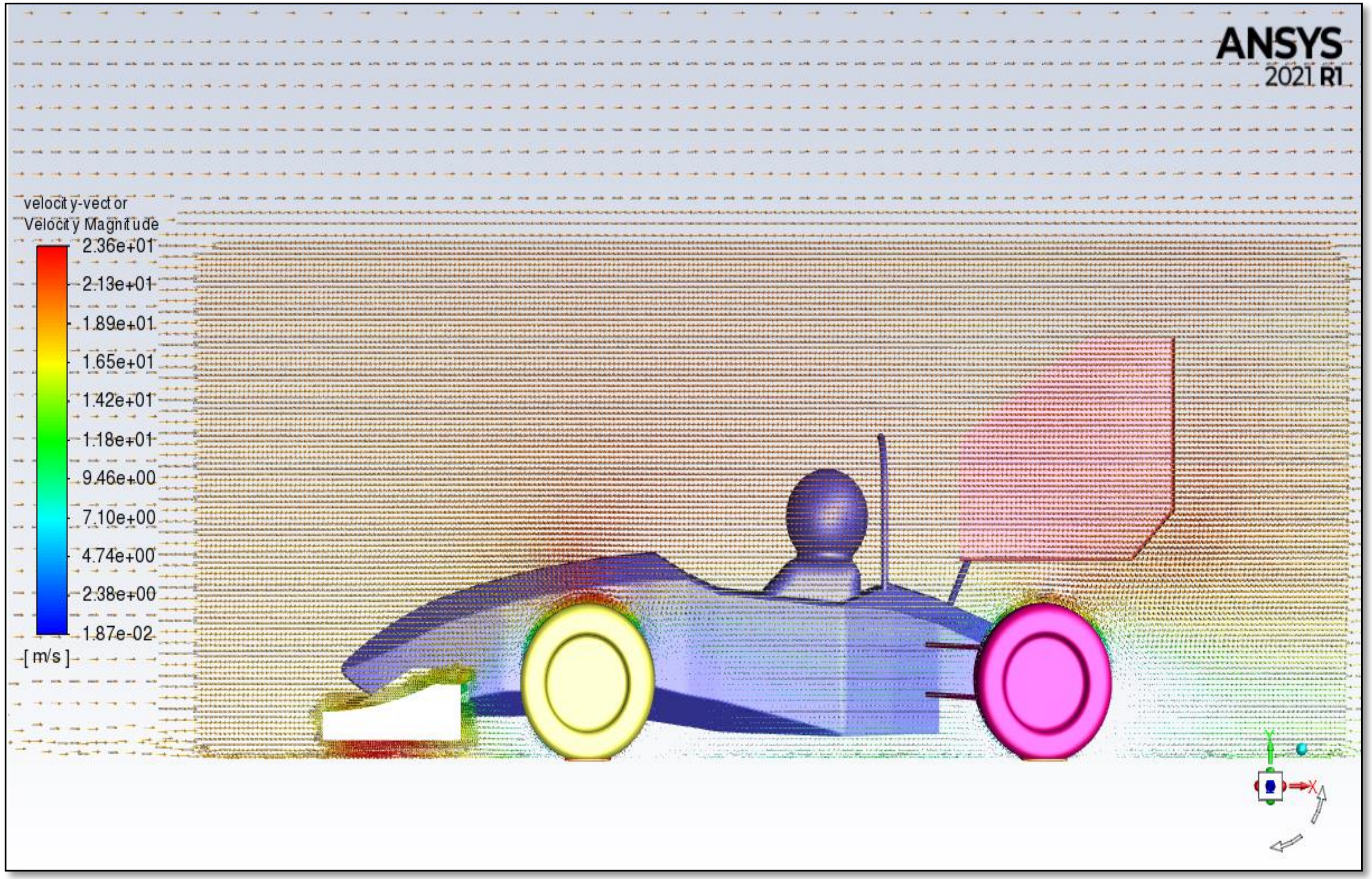
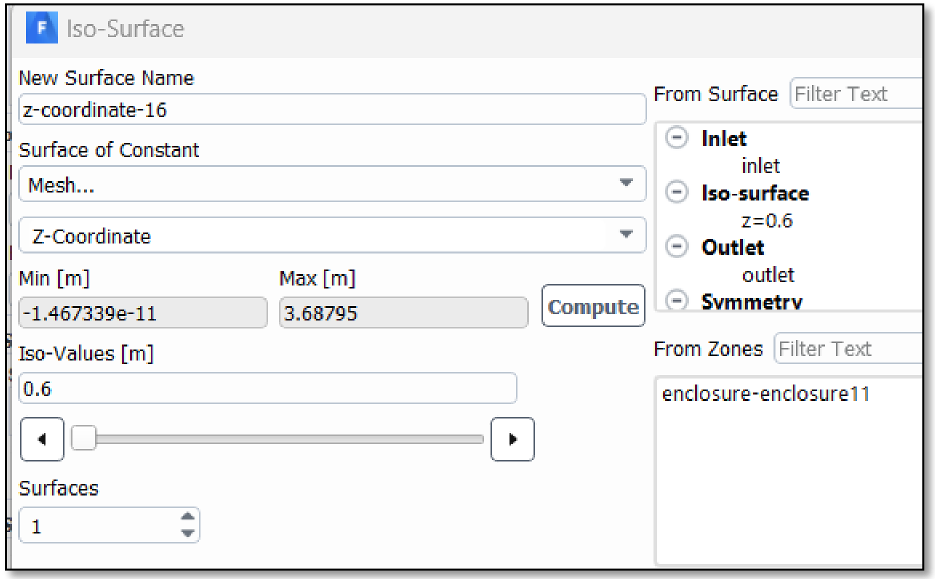


FIG. W. Velocity vector of the car



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

Here we see the velocity vector of the car:

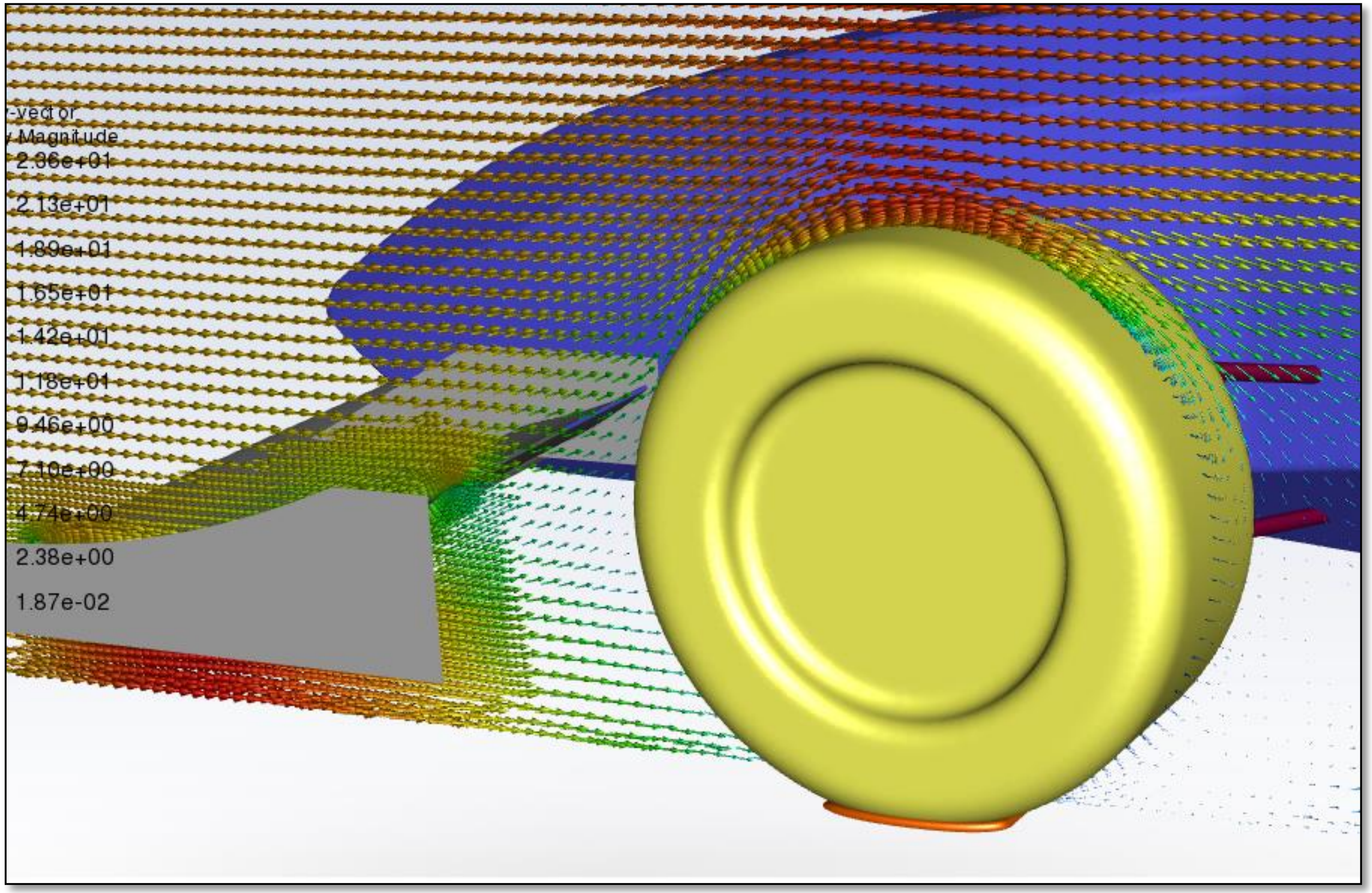
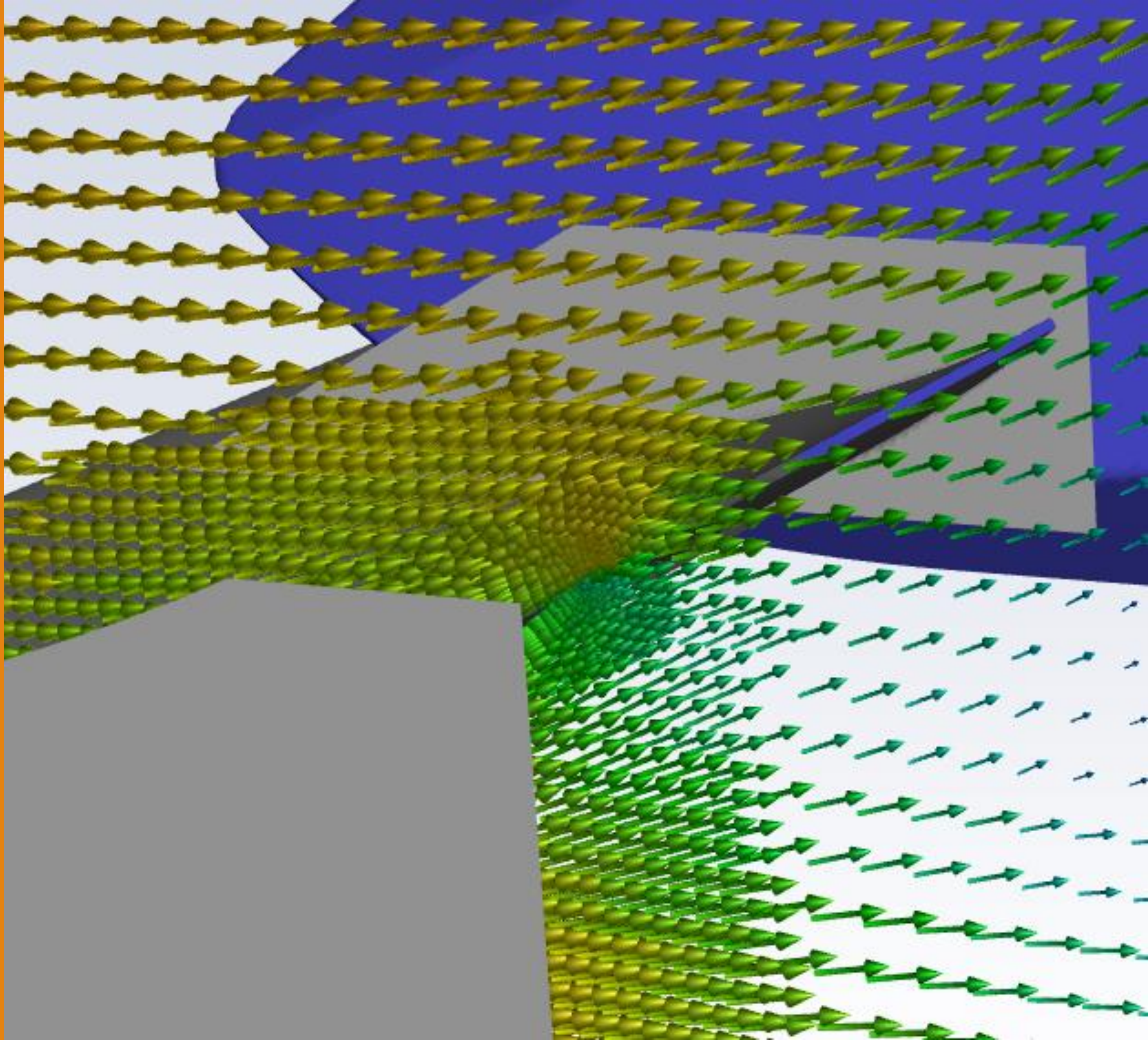


FIG. W1. Velocity vector of the car

Air flow near the wheels also has strong rotational component and flow vertices where it shows the region of the rotating flow.

PHASE – 5

POST PROCESSING - 2



Here we create the Iso surfaces of the X velocity of ISO SURFACE of the car as well as sweep surfaces animations that are used to understand flow vertices around the car geometry, and the path line traveled by fluid particles around the car. From this modulation, we will go through the Adverse pressure gradient where it detaches from the surfaces.

Flow separation should be avoided where it creates a significant effect to drag where it will be seen in the iso surface creation.

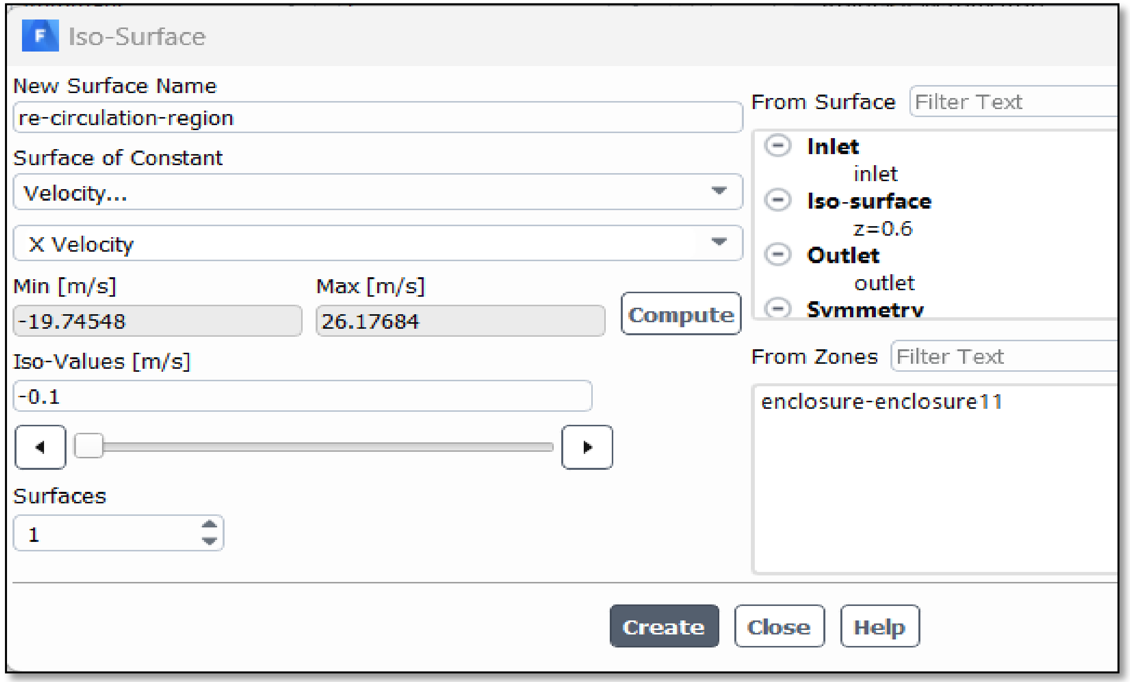


FIG. X – Velocity of X - Axis under ISO - SURFACE

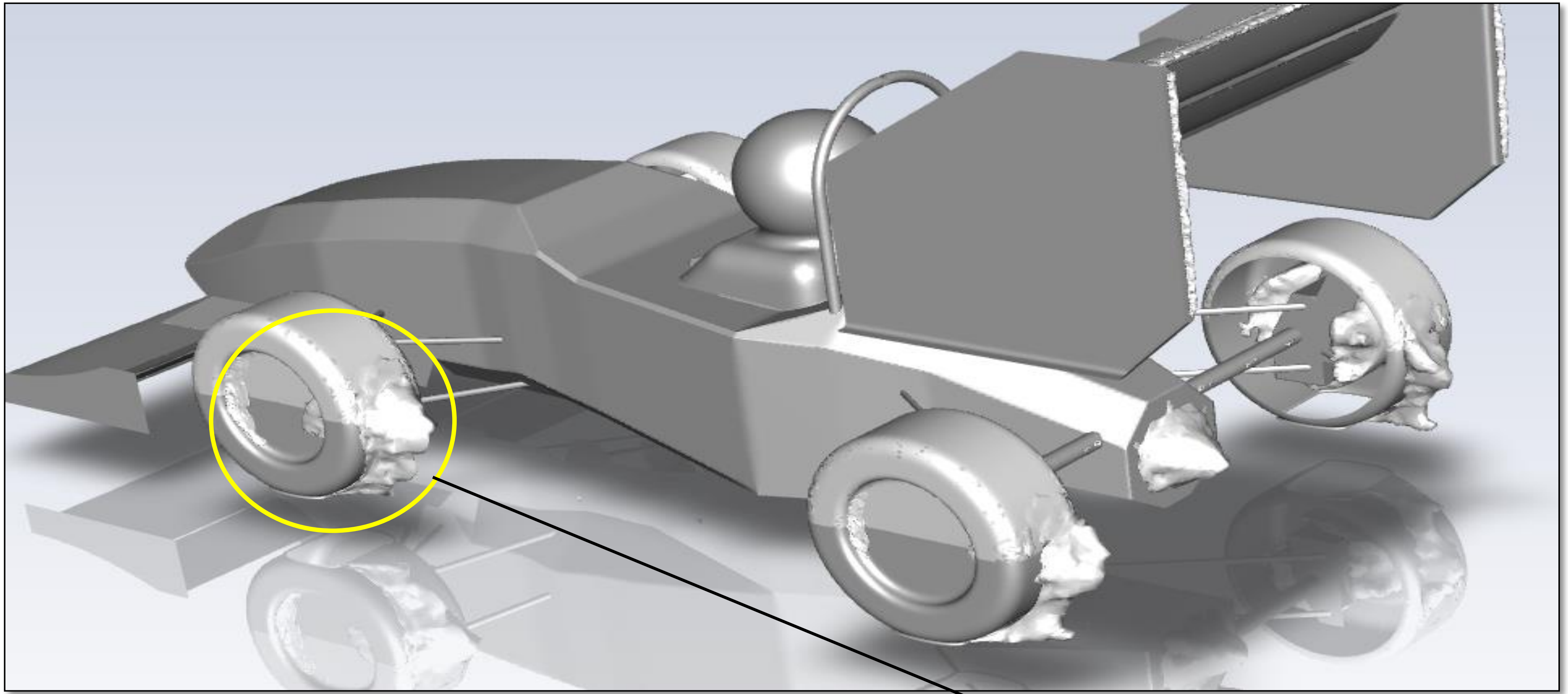


FIG. X 1– Velocity of X - Axis under ISO - SURFACE

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

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

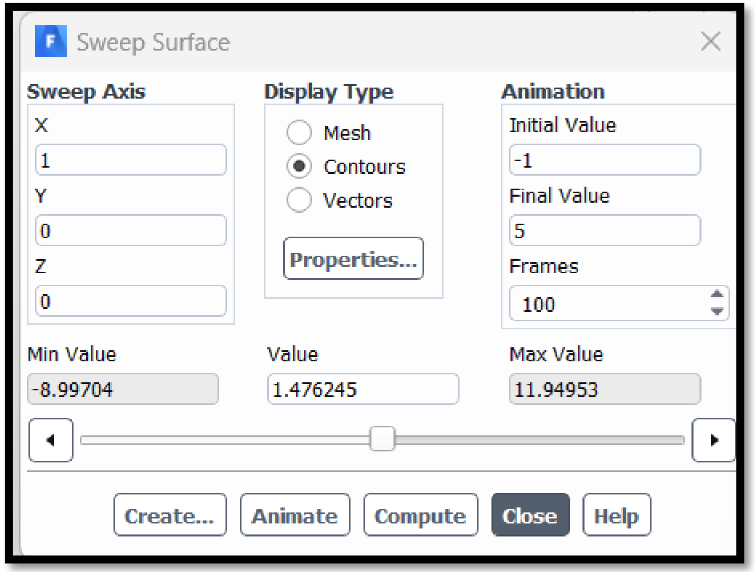


FIG. Y1– Sweep surface settings

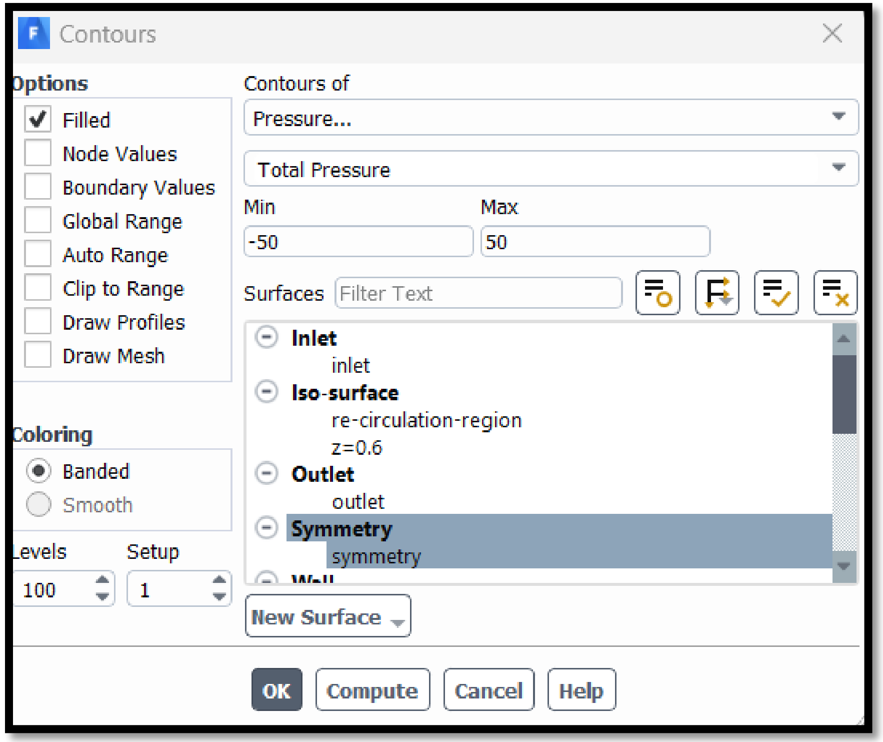


FIG. Y2– Sweep surface settings

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.

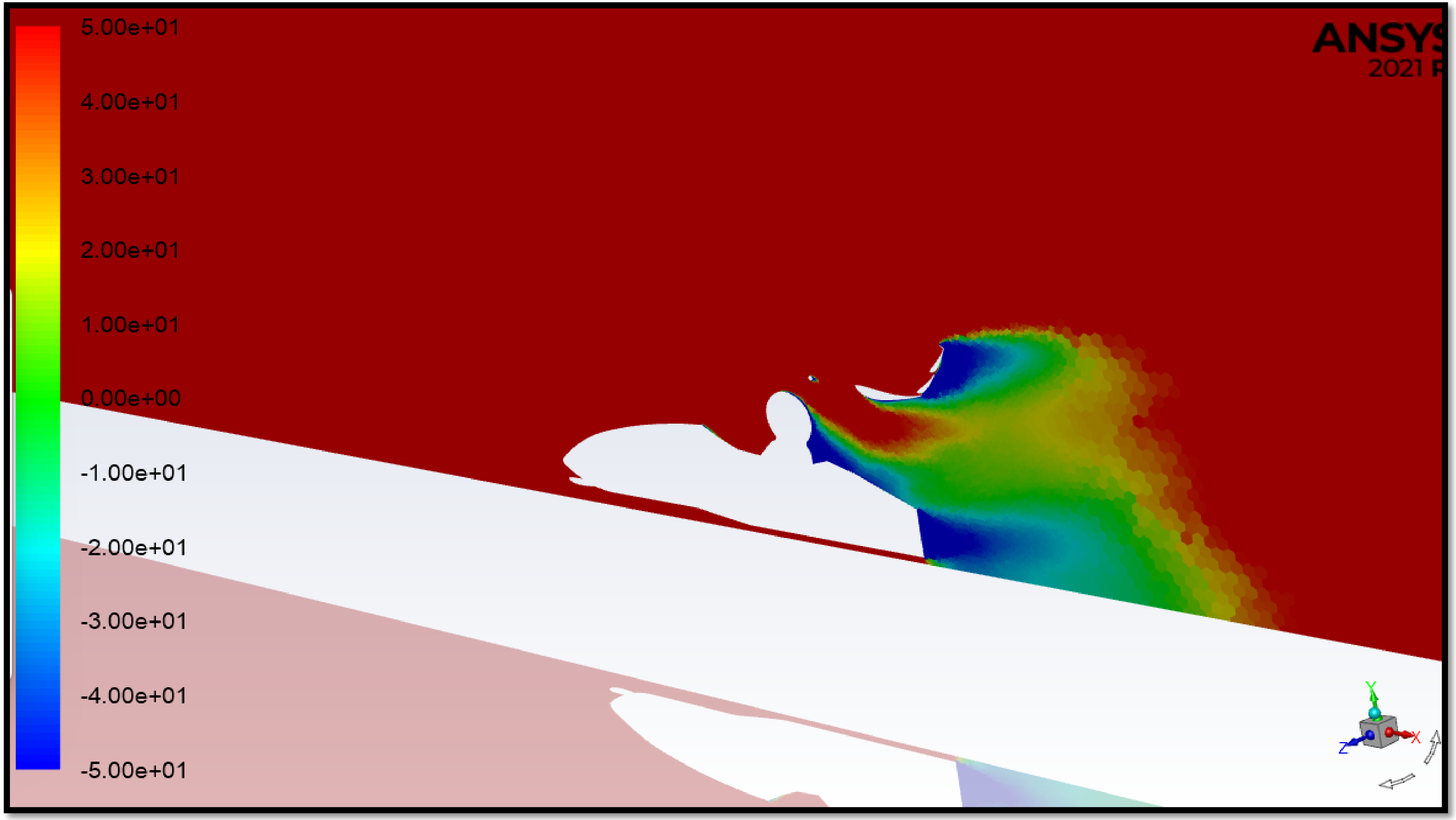


FIG. Y3– Sweep surface contour plot

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

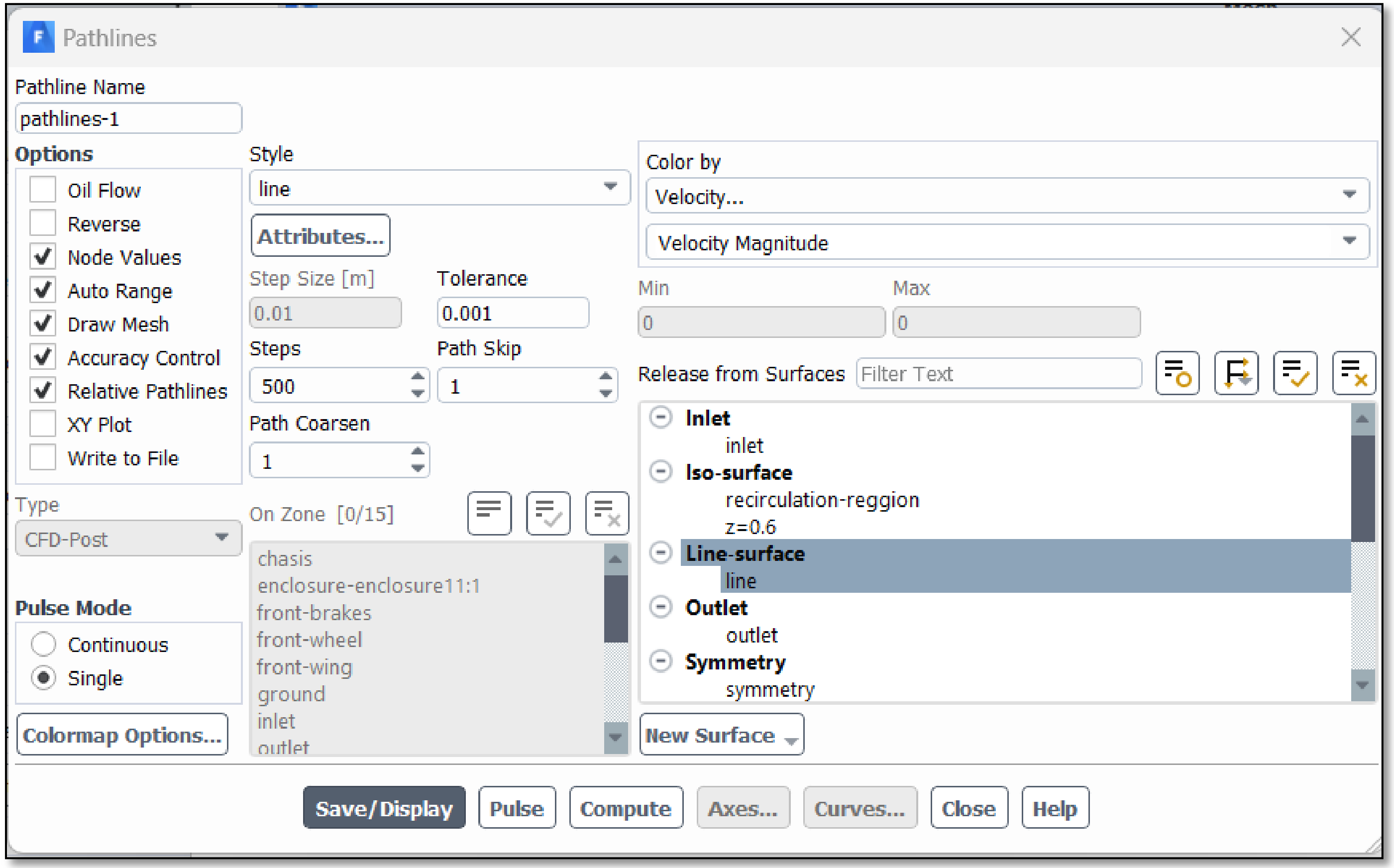


FIG. Z1– path line settings

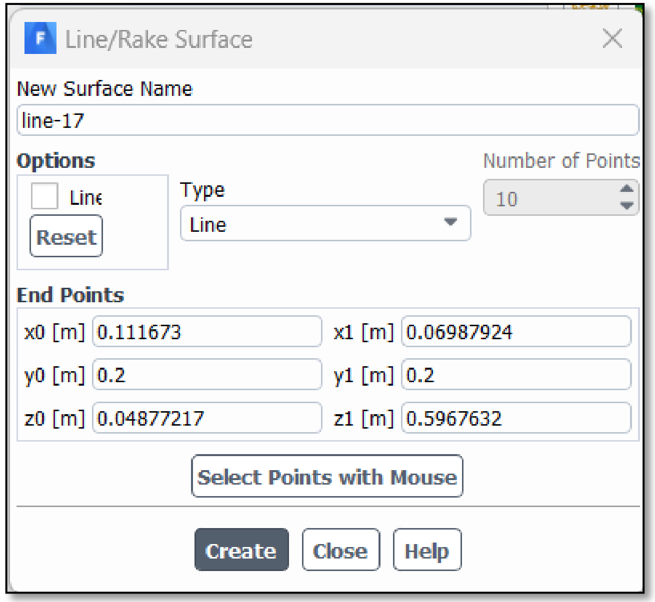
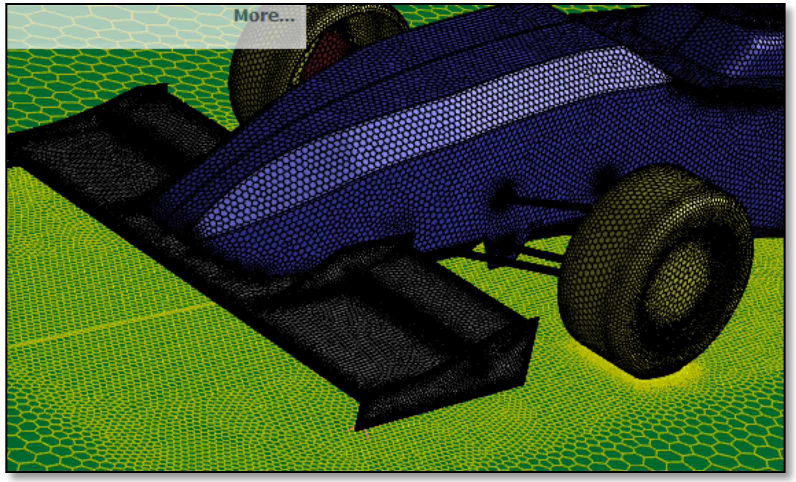


FIG. Z2– Line and rake surface



To understand the quality of the flow around the car, these path lines are used to analyze, the surfaces of the car.

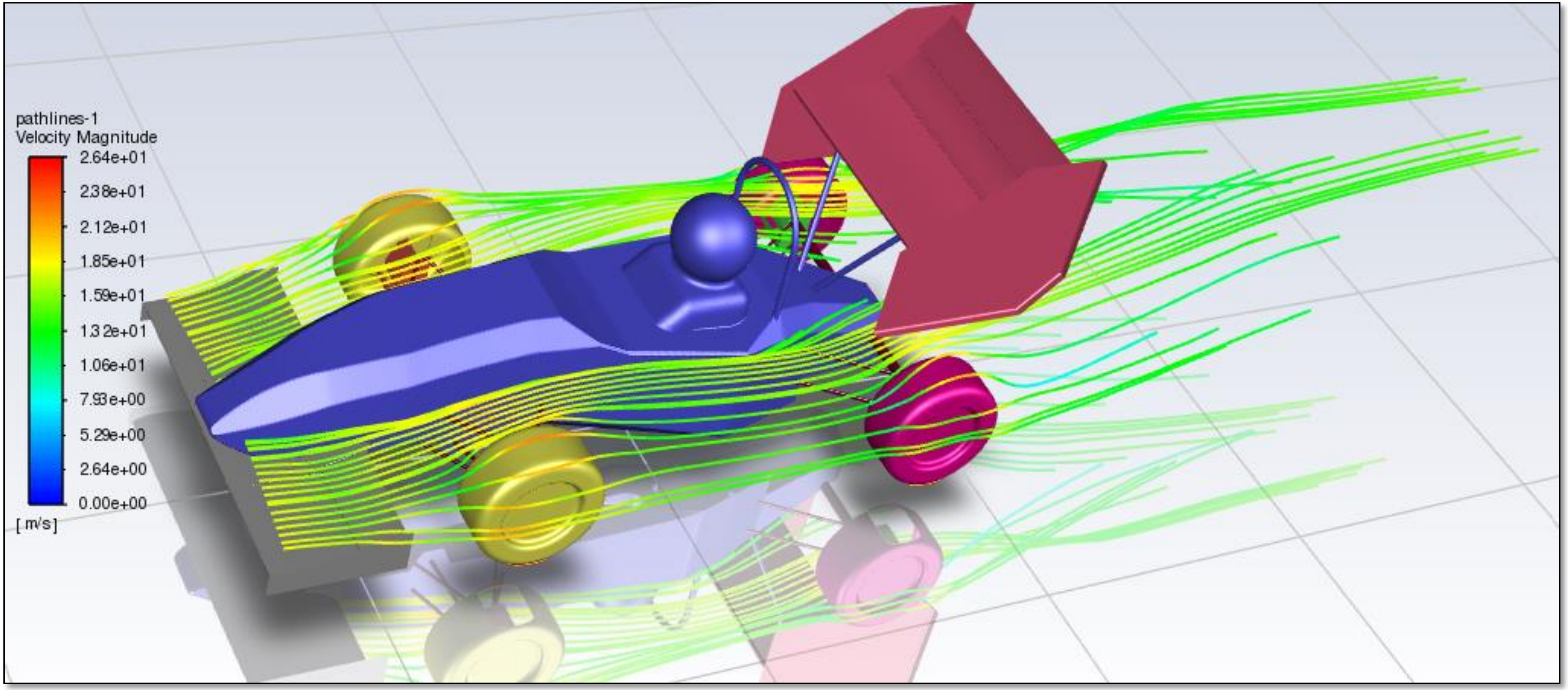


FIG. Z3– Path Lines

The quality of the path line, in the rear side of the wing.

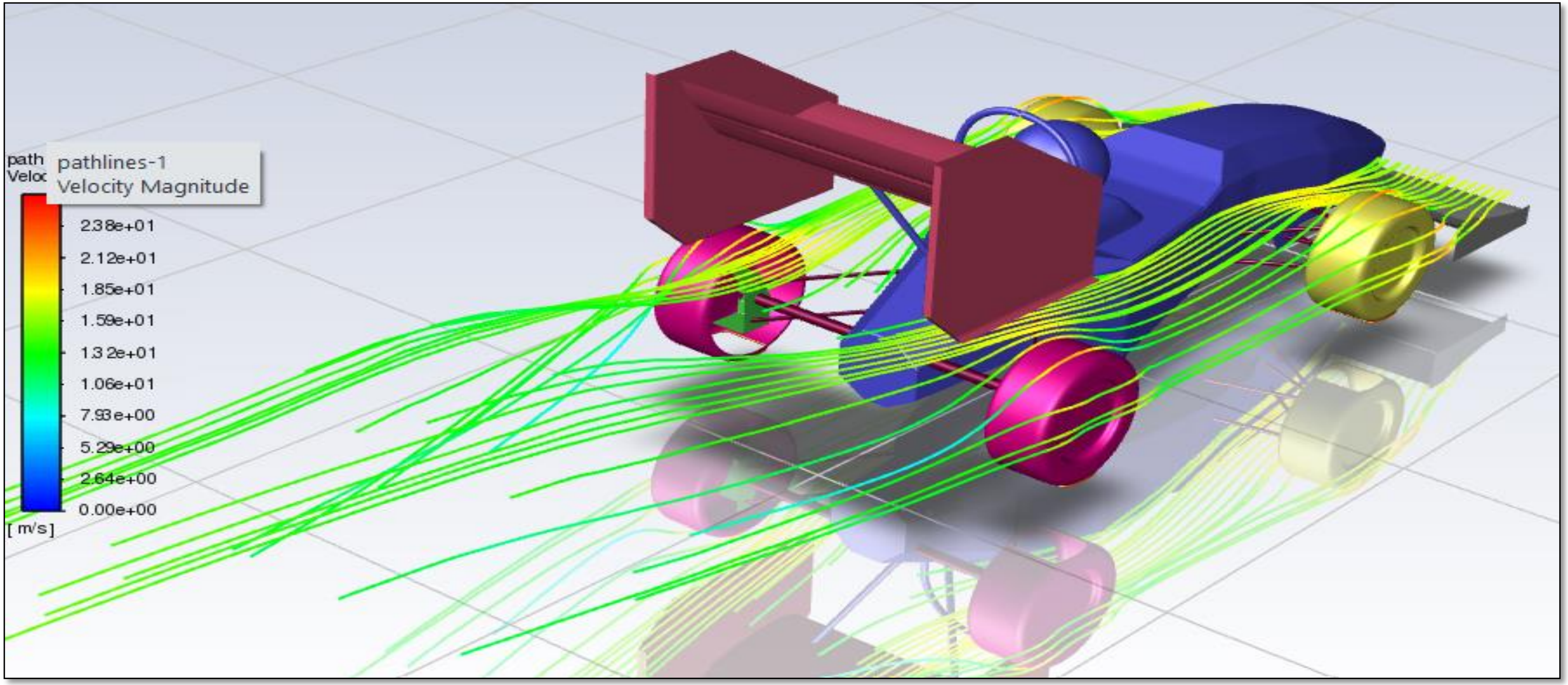


FIG. Z4– Rear Path Lines

SUMMARY:

- In order to analyze the aerodynamics of the FSAE car, we have gone through a certain phase to analyze.
- Came with the body of influence and boundary conditions to create the meshing process.
- Having used the surface mesh, boundary layer mesh, and poly hexacore volume mesh creation for different analyses of meshing.
- Local sizing is used for precise meshing, especially in the wake region. Mesh and quality checks are performed, followed by setting up pressure-based, steady-state solvers, and a turbulence module.
- Forces are analyzed, and contours of velocity magnitude, static pressure, and velocity vector plots are created. The rear wing and chassis contribute significantly to the force in the x-axis, while the total forces act in the negative y-axis direction, providing downforce to keep the car on the ground.
- Additionally, in the analysis process, iso surfaces of the X velocity are created to visualize and understand the flow patterns around the car. Sweep surface animations are also generated to examine the flow vertices and track the path lines of fluid particles around the car. These visualizations help in identifying adverse pressure gradients, where the flow detaches from the surfaces, leading to flow separation.

THANK YOU !