



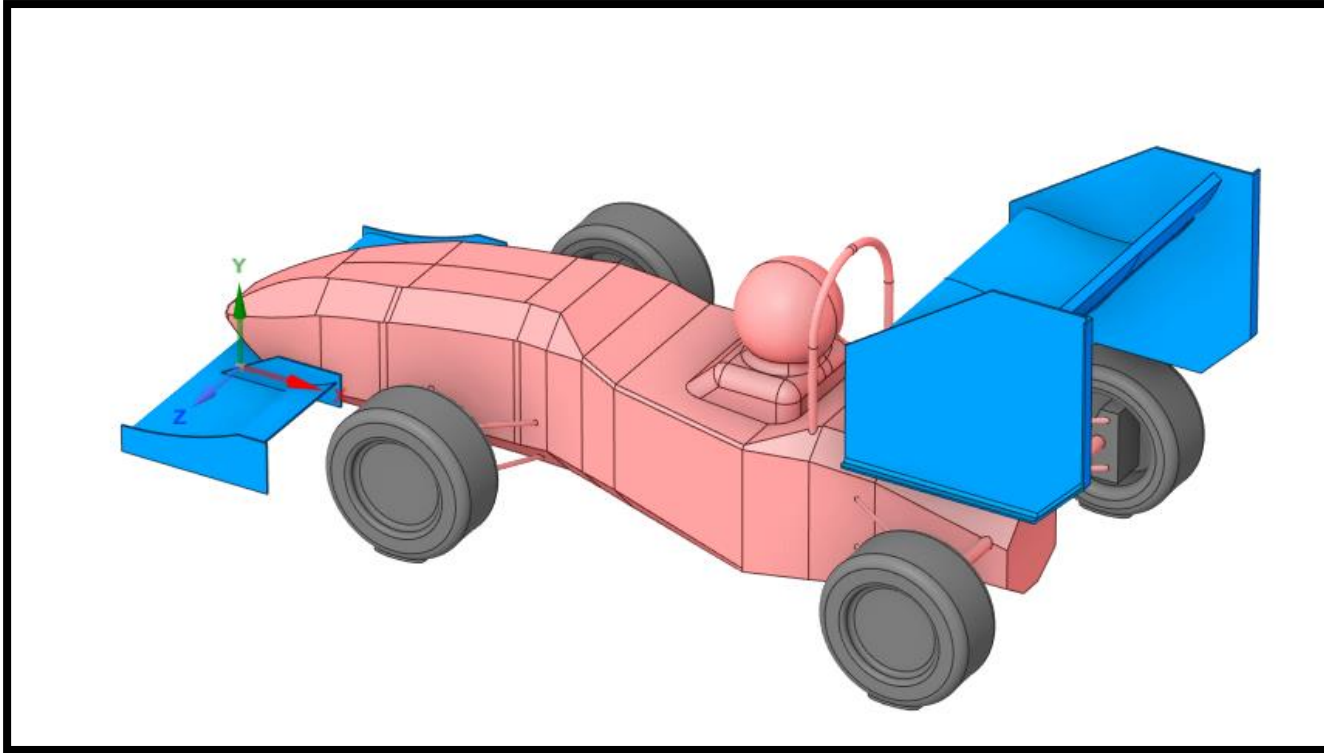
SYRACUSE UNIVERSITY

ENGINEERING & COMPUTER SCIENCE

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.

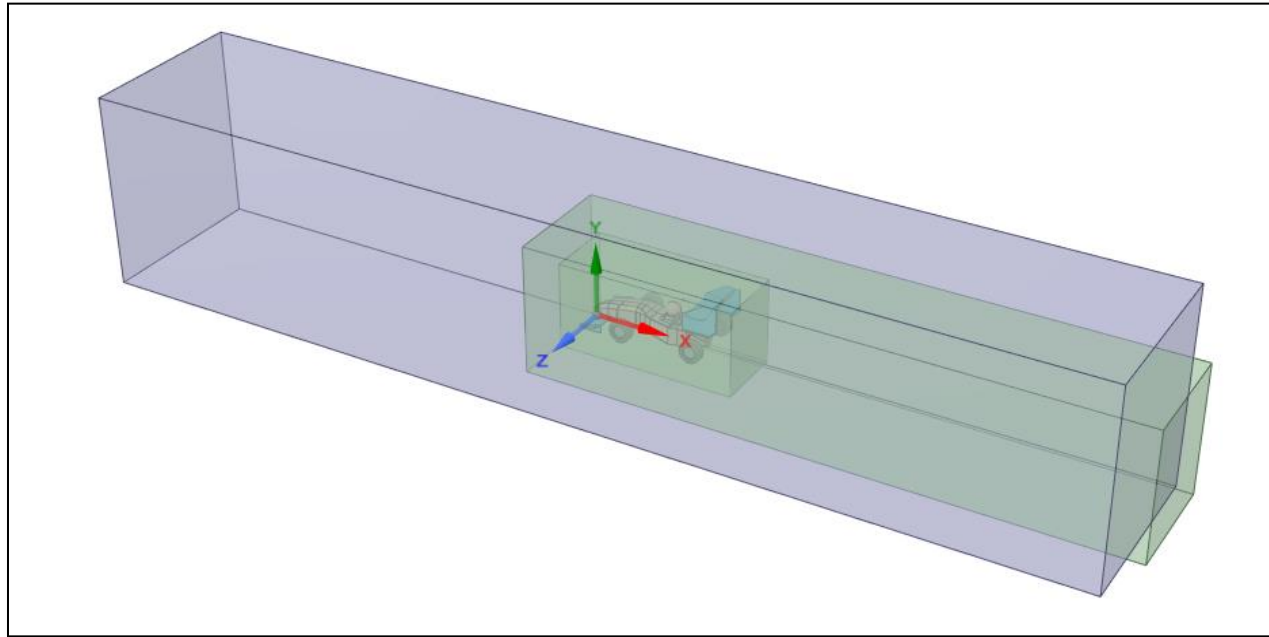


FIG. A. Enclosure and BOI

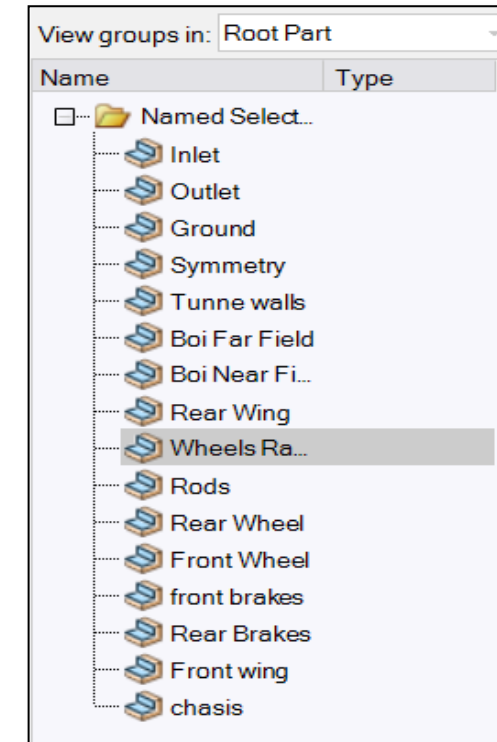
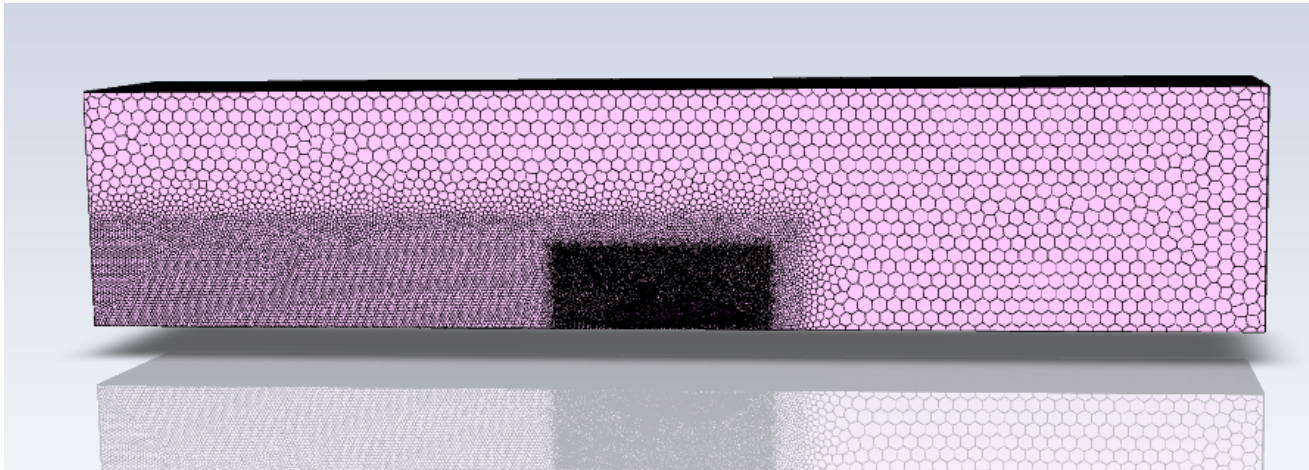


FIG. B. Name selection panel

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.

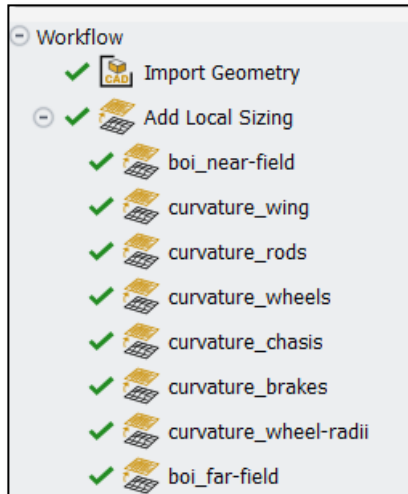


FIG. C. Workflow pattern

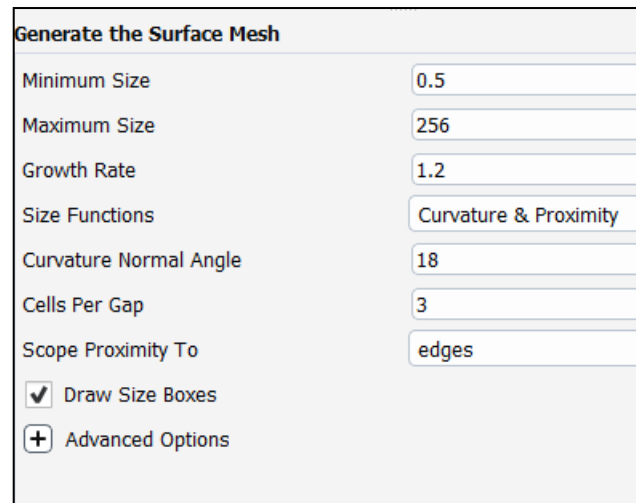


FIG. D. Mesh settings

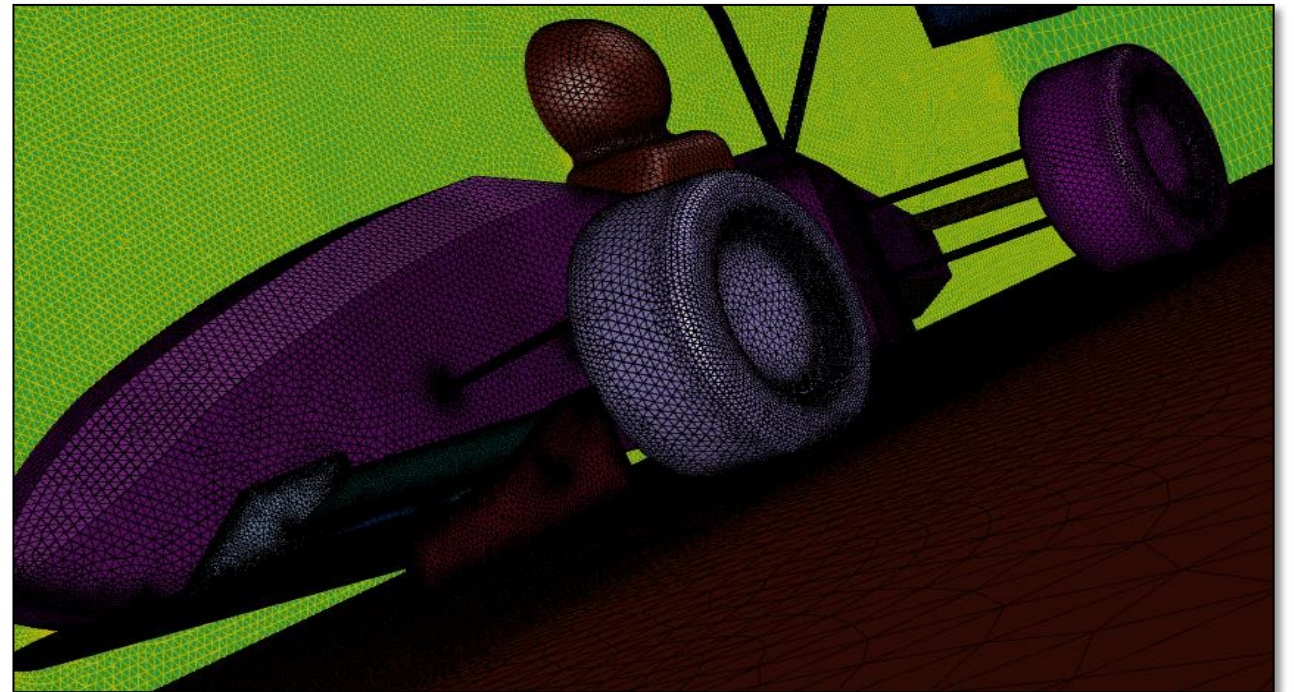
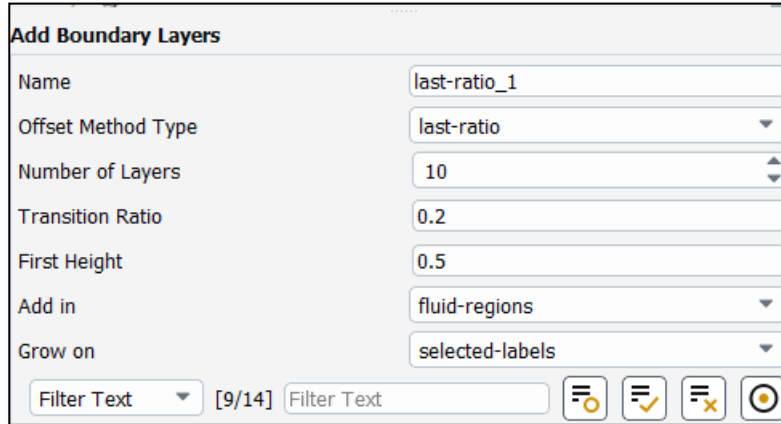


FIG. E. Surface Mesh with skewness 0.59

- 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.

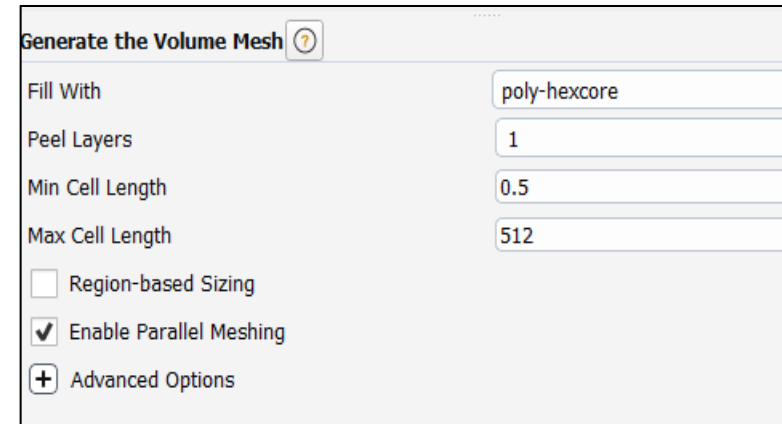


Add Boundary Layers

Name	last-ratio_1
Offset Method Type	last-ratio
Number of Layers	10
Transition Ratio	0.2
First Height	0.5
Add in	fluid-regions
Grow on	selected-labels

Filter Text [9/14] Filter Text

FIG. F. Boundary Layer conditions



Generate the Volume Mesh

Fill With	poly-hexcore
Peel Layers	1
Min Cell Length	0.5
Max Cell Length	512

☐ Region-based Sizing

☒ Enable Parallel Meshing

Advanced Options

FIG. G. volume mesh conditions

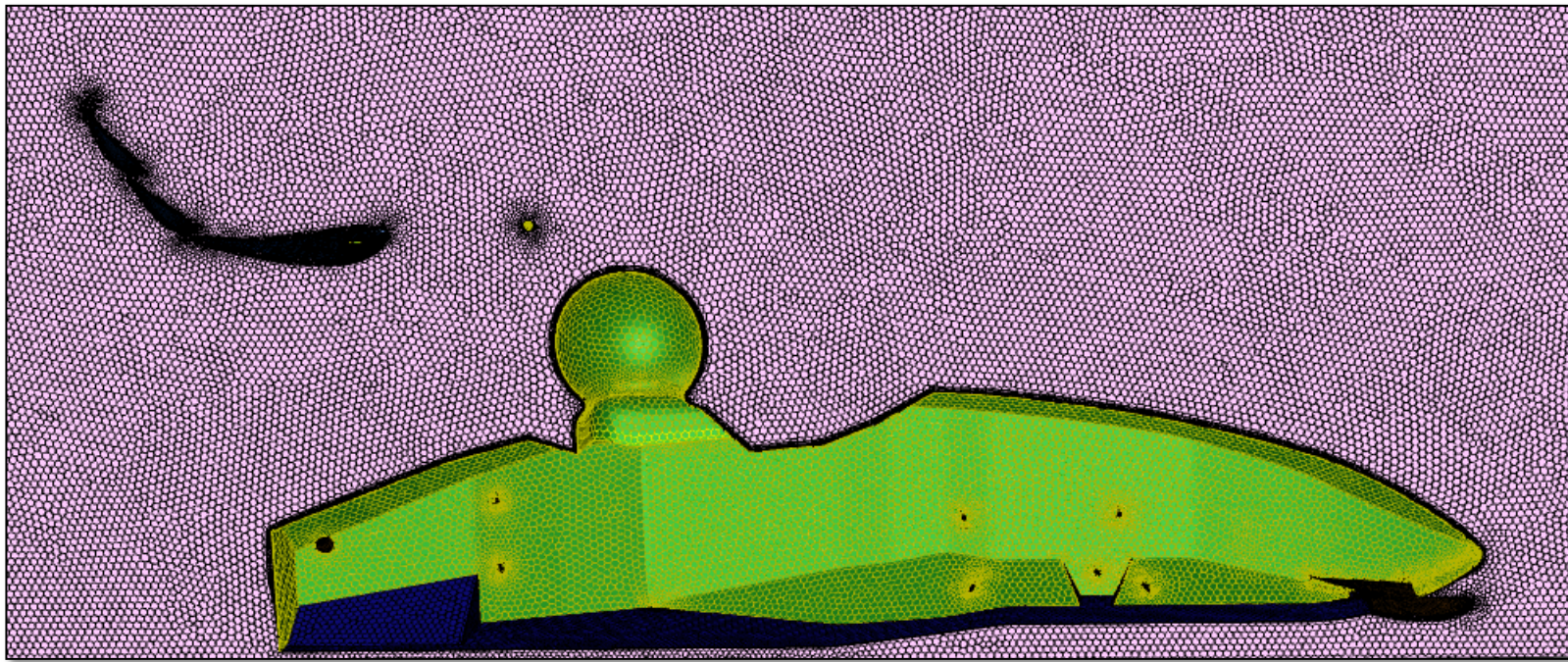


FIG. H1. volume mesh

```

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: chassis, type: wall, count: 23168
  
```

FIG. H2. Orthogonal Quality

```

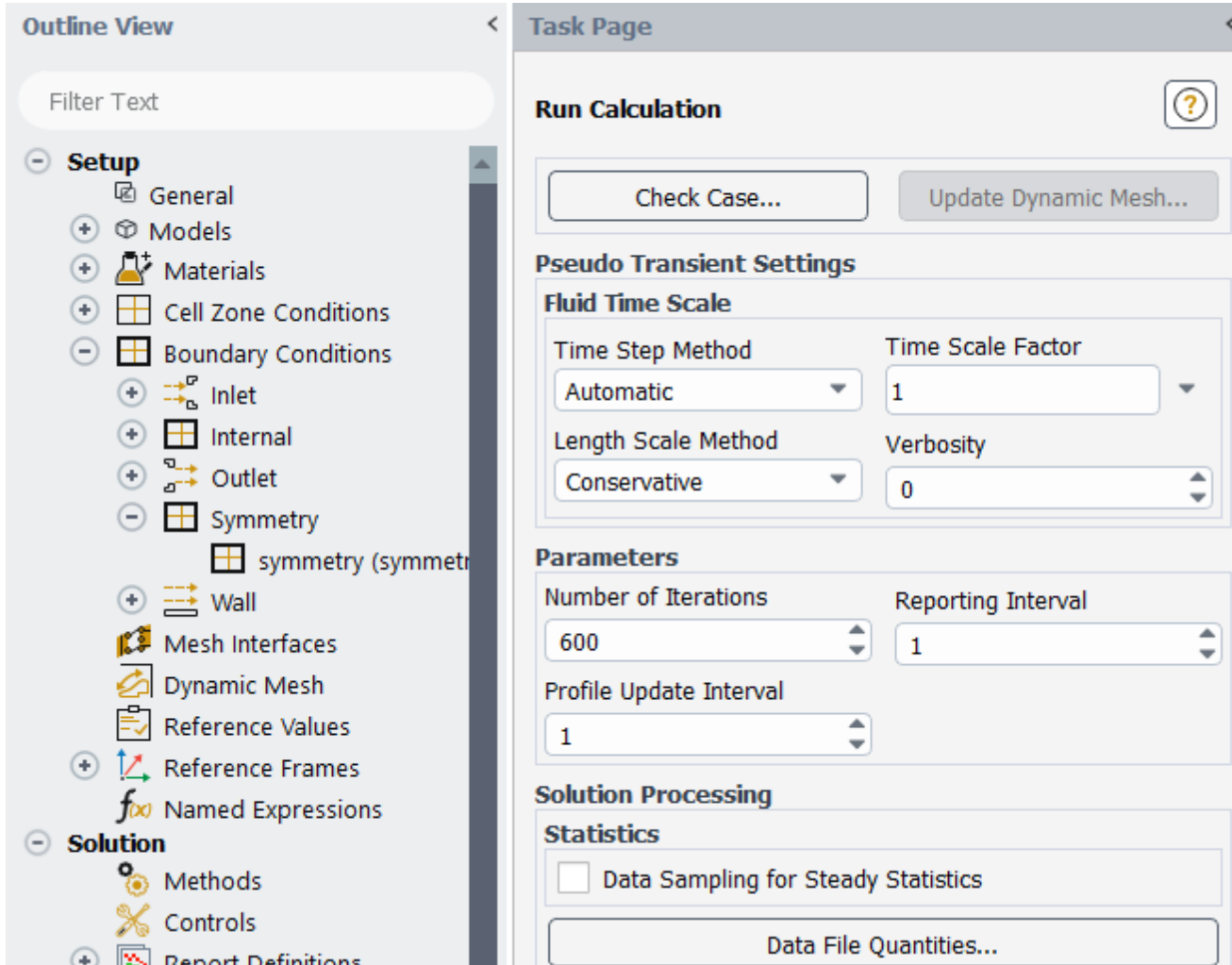
Mesh Quality:

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

Starting 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.

Task Page

Reference Values

?

Compute from

Reference Values

Area [m²]

0.5008232

Density [kg/m³]

1.225

Enthalpy [J/kg]

0

Length [m]

1

Pressure [Pa]

0

Temperature [K]

288.16

Velocity [m/s]

20

Viscosity [kg/(m s)]

1.7894e-05

Ratio of Specific Heats

1.4

Yplus for Heat Tran. Coef.

300

Reference Zone

FIG. J. Reference Conditions

Viscous Model

×

Model

☐ Inviscid

☐ Laminar

☐ Spalart-Allmaras (1 eqn)

☐ k-epsilon (2 eqn)

☒ k-omega (2 eqn)

☐ Transition k-kl-omega (3 eqn)

☐ Transition SST (4 eqn)

☐ Reynolds Stress (7 eqn)

☐ Scale-Adaptive Simulation (SAS)

☐ Detached Eddy Simulation (DES)

☐ Large Eddy Simulation (LES)

k-omega Model

☐ Standard

☐ GEKO

☐ BSL

☒ SST

k-omega Options

☐ Low-Re Corrections

Options

☒ Curvature Correction

☐ Corner Flow Correction

☐ Production Kato-Launder

☒ Production Limiter

Transition Options

Transition Model

none

Curvature Correction Options

CCURV

constant

1

Model Constants

Alpha*_inf

1

Alpha_inf

0.52

Beta*_inf

0.09

a1

0.31

Beta_i (Inner)

0.075

Beta_i (Outer)

0.0828

TKE (Inner) Prandtl #

1.176

TKE (Outer) Prandtl #

1

SDR (Inner) Prandtl #

2

SDR (Outer) Prandtl #

1.168

Production Limiter Clip Factor

10

User-Defined Functions

Turbulent Viscosity

none

OK

Cancel

Help

FIG. K. Viscous model

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

Create/Edit Materials

Name:

Material Type:

Chemical Formula:

Fluent Fluid Materials:

Mixture:

Order Materials by:
☒ Name
☐ Chemical Formula

Fluent Database...
GRANTA MDS Database...
User-Defined Database...

Properties

Density [kg/m³]:

Viscosity [kg/(m s)]:

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.

Create/Edit Materials

Name: air

Material Type: fluid

Chemical Formula:

Fluent Fluid Materials: air

Mixture: none

Order Materials by:
☒ Name
☐ Chemical Formula

Fluent Database...
GRANTA MDS Database...
User-Defined Database...

Properties

Density [kg/m³]: constant (1.225) [Edit...]

Viscosity [kg/(m s)]: constant (1.7894e-05) [Edit...]

Change/Create Delete Close Help

FIG. K. Material Conditions

Velocity Inlet

Name: inlet

Momentum Thermal Radiation Species DPM Mixture

Velocity Specification Method: Magnitude, Normal to Boundary

Reference Frame: Absolute

Velocity Magnitude [m/s]: 20

Supersonic/Initial Gauge Pressure [Pa]: 0

Turbulence

Specification Method: Intensity and Viscosity Ratio

Turbulent Intensity [%]: 0.5

Turbulent Viscosity Ratio: 2

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.

F

Pressure Outlet

×

Zone Name

outlet

Momentum

Thermal

Radiation

Species

DPM

Multiphase

Potential

UDS

Backflow Reference Frame

Absolute

Gauge Pressure [Pa]

0

Pressure Profile Multiplier

1

Backflow Direction Specification Method

Normal to Boundary

Backflow Pressure Specification

Total Pressure

☐ Prevent Reverse Flow

☐ Radial Equilibrium Pressure Distribution

☐ Average Pressure Specification

☐ Target Mass Flow Rate

Turbulence

Specification Method

Intensity and Viscosity Ratio

Backflow Turbulent Intensity [%]

0.5

Backflow Turbulent Viscosity Ratio

2

Apply

Close

Help

FIG. M. Pressure outlet

F

Wall

×

Zone Name

rear-wheel

Adjacent Cell Zone

enclosure-enclosure11

Momentum

Thermal

Radiation

Species

DPM

Multiphase

UDS

Wall Motion

☐ Stationary Wall

☒ Moving Wall

Motion

☒ Relative to Adjacent Cell Zone

☐ Absolute

Speed [rad/s]

87.489

☐ Translational

☒ Rotational

☐ Components

Rotation-Axis Origin

X [m]

2.46959

Y [m]

0.2254

Z [m]

0.58795

Rotation-Axis Direction

X

0

Y

0

Z

1

Shear Condition

☒ No Slip

☐ Specified Shear

☐ Specularity Coefficient

☐ Marangoni Stress

Wall Roughness

Roughness Models

☒ Standard

☐ High Roughness (Icing)

Sand-Grain Roughness

Roughness Height [m]

0

Roughness Constant

0.5

Apply

Close

Help

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.

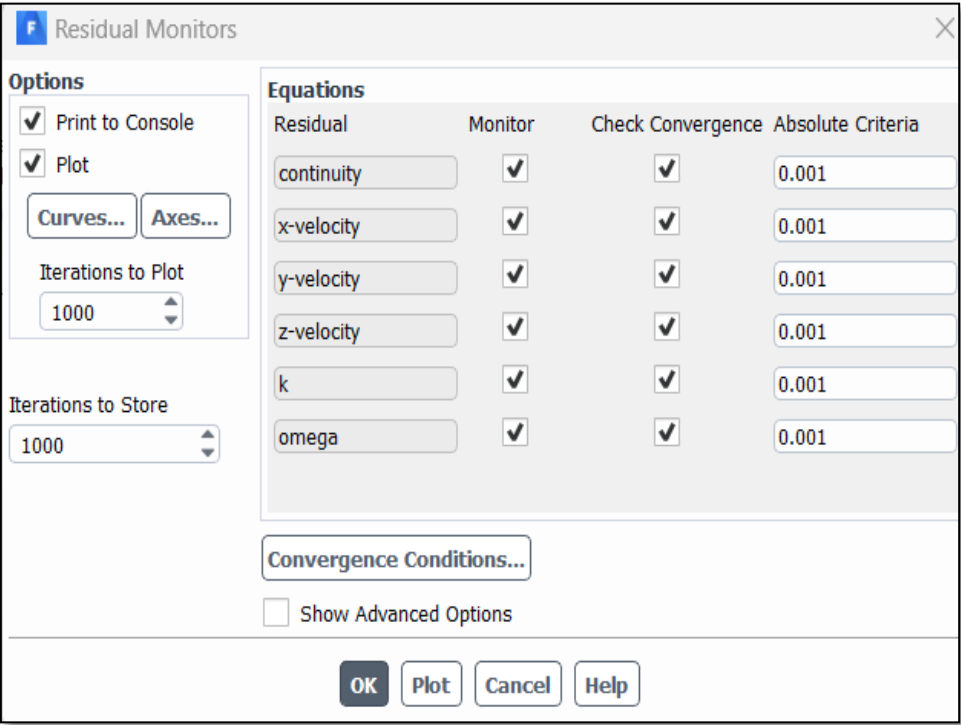


FIG. O. Residual Monitors

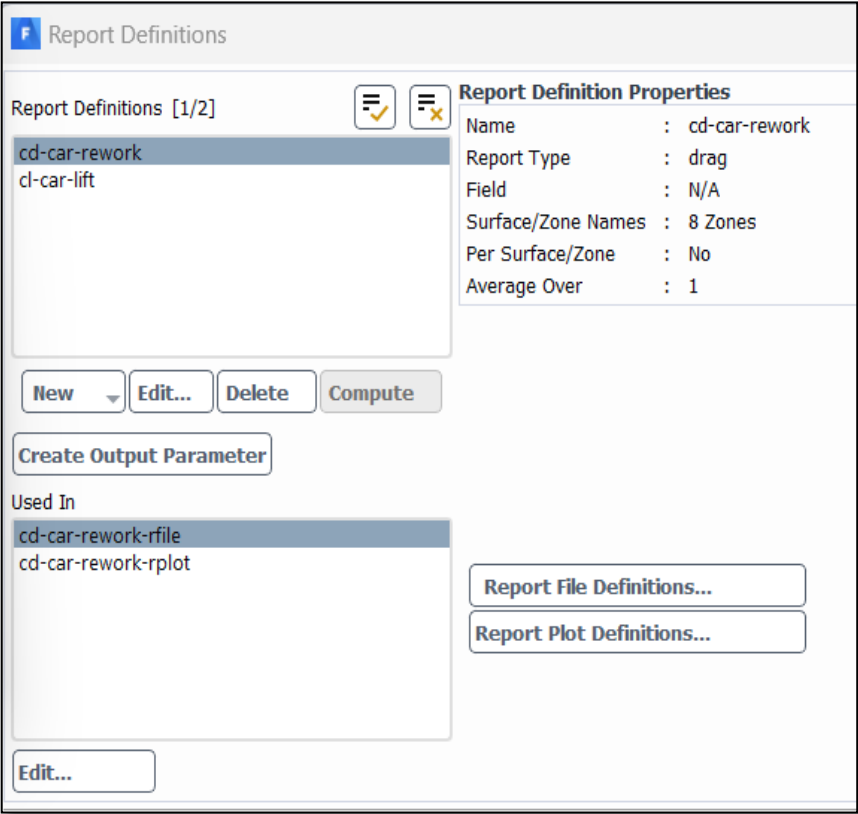


FIG. P. Definition Reports

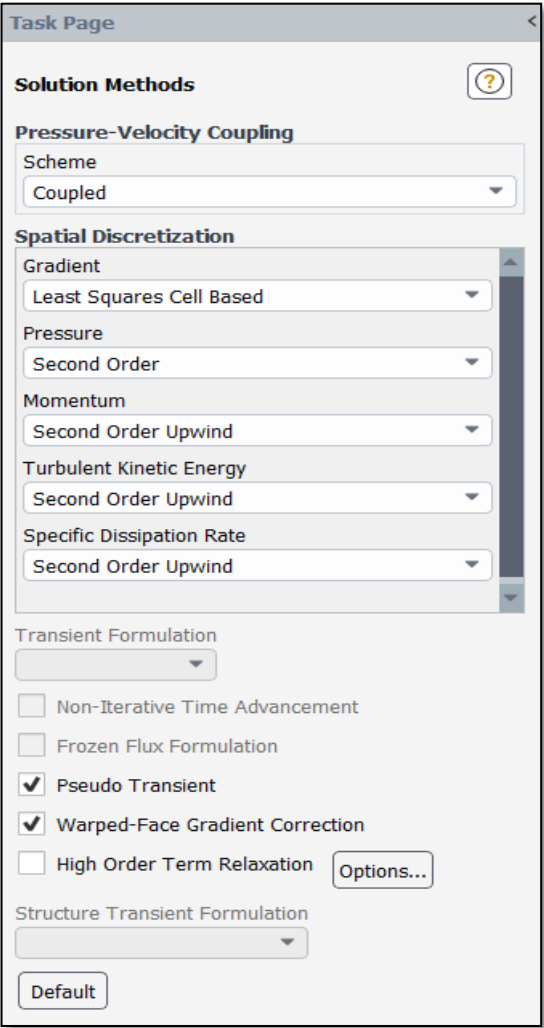
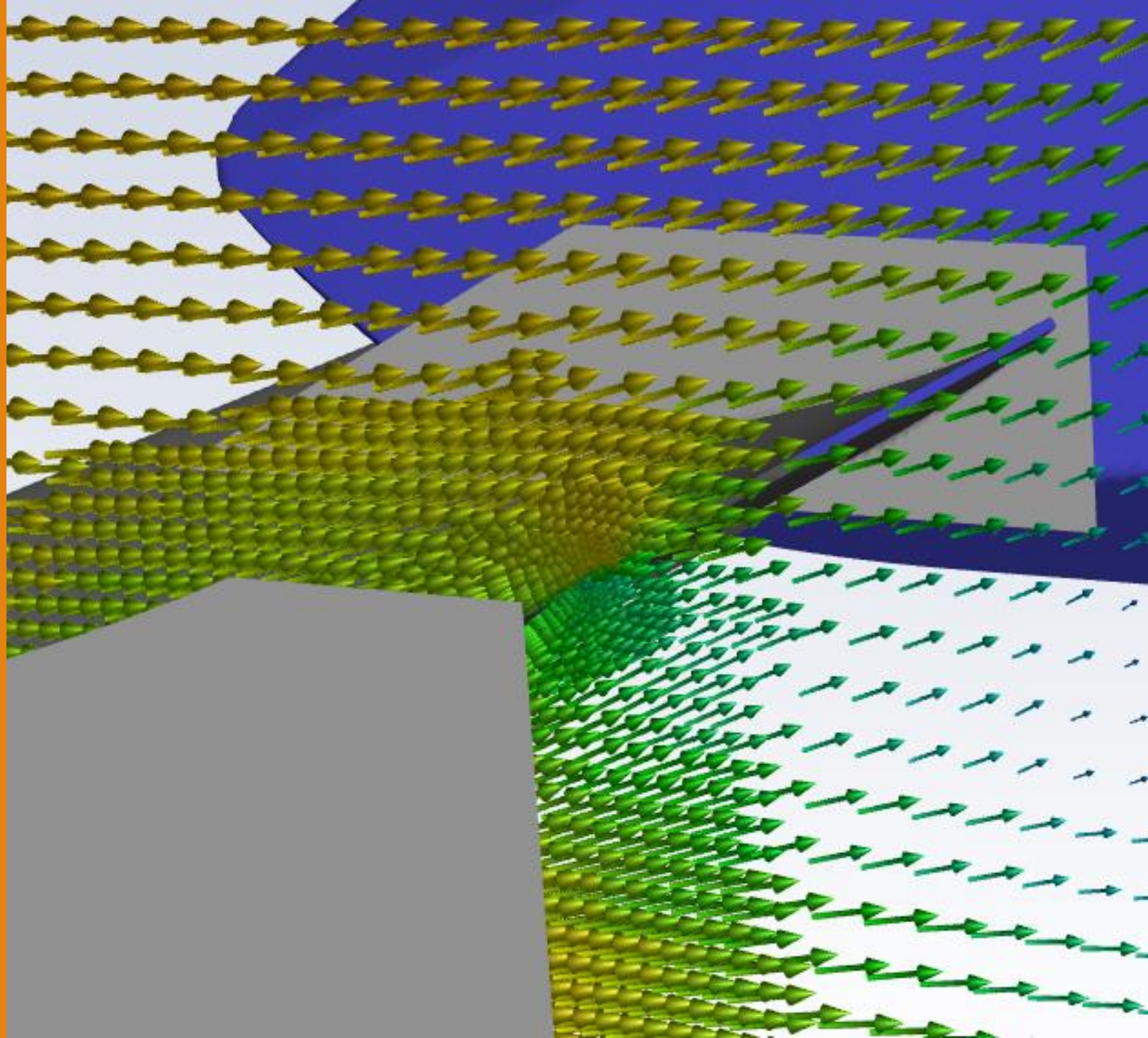


FIG. Q. Solution Methods

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)						
Zone	Forces [N]			Coefficients		
	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)						
Zone	Forces [N]			Coefficients		
	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.

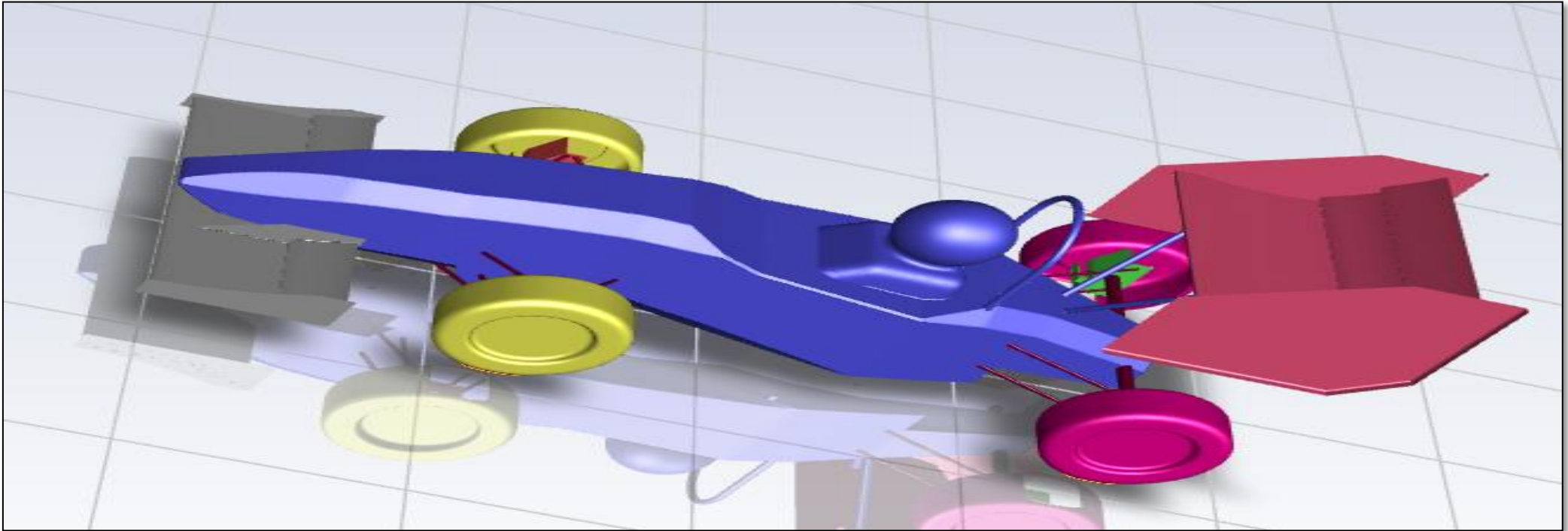


FIG. T. Graphical representation of the full car

Here we see the velocity contour of the car:

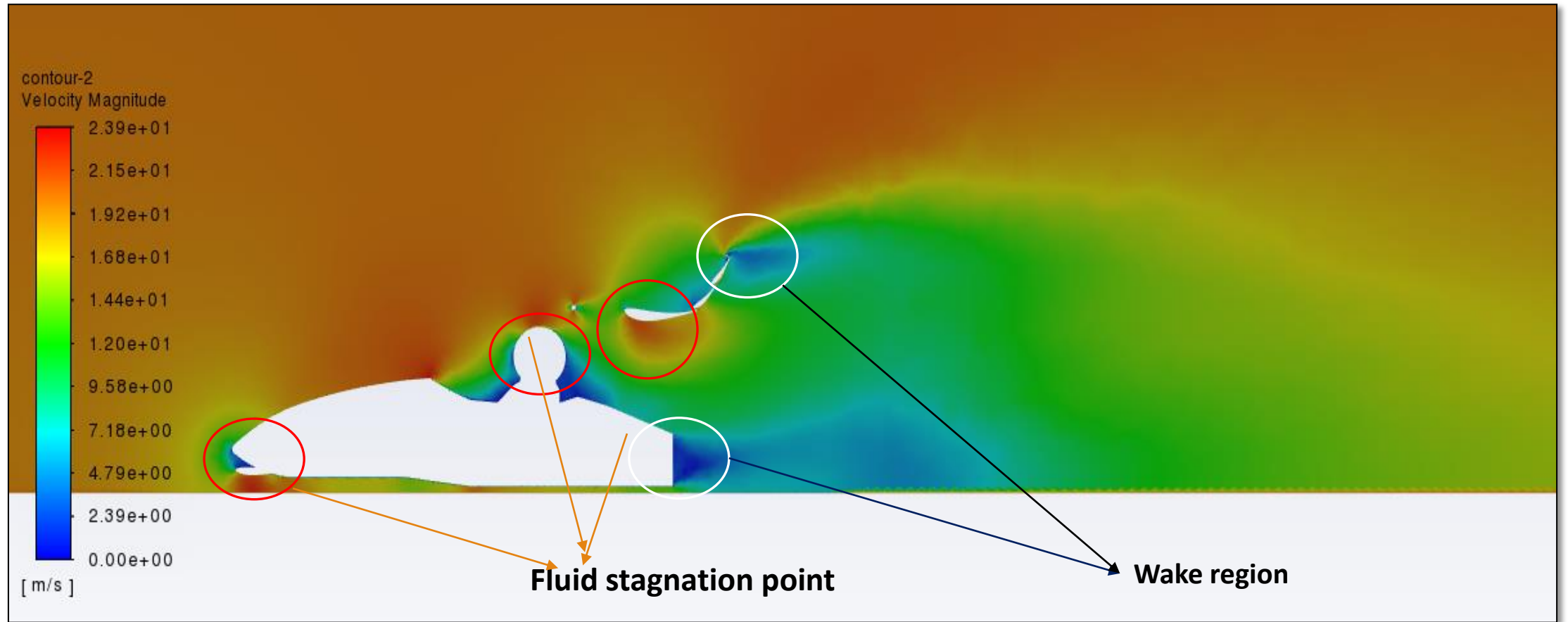


FIG. U. Velocity contour of the car

Here we see the pressure contour of the car:

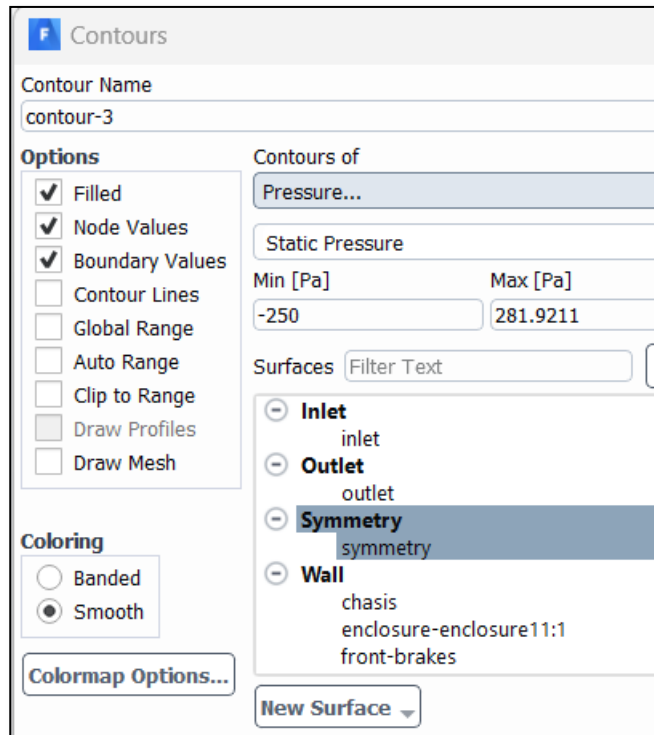
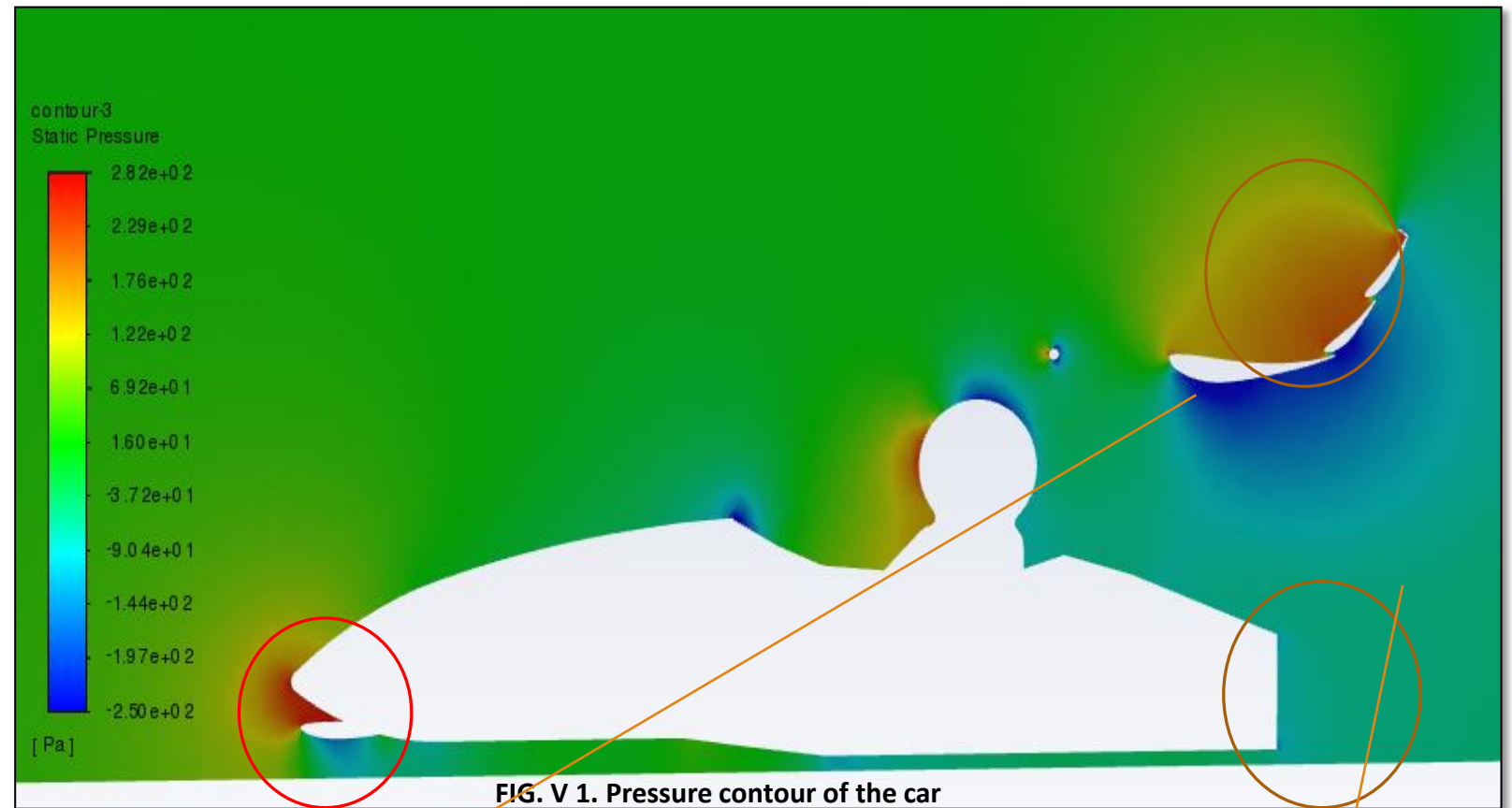


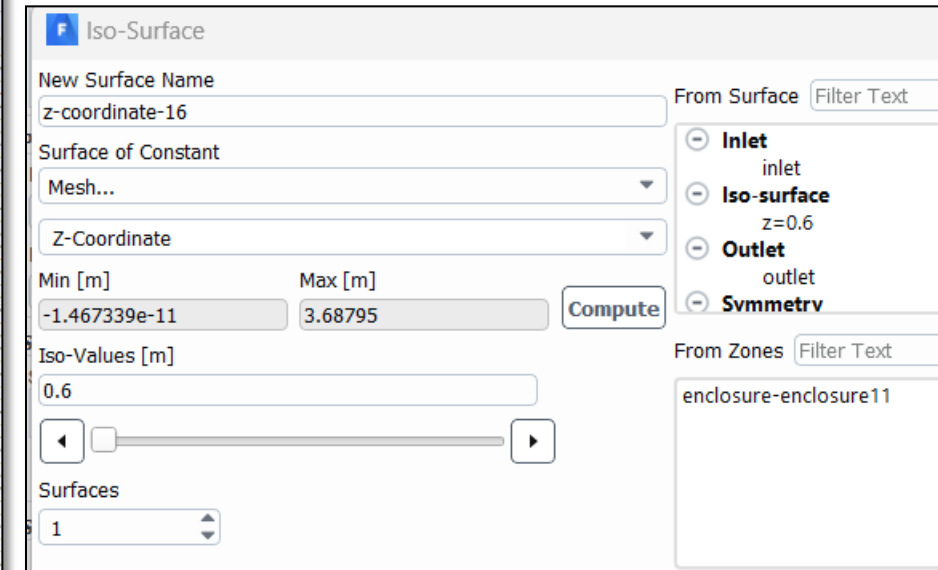
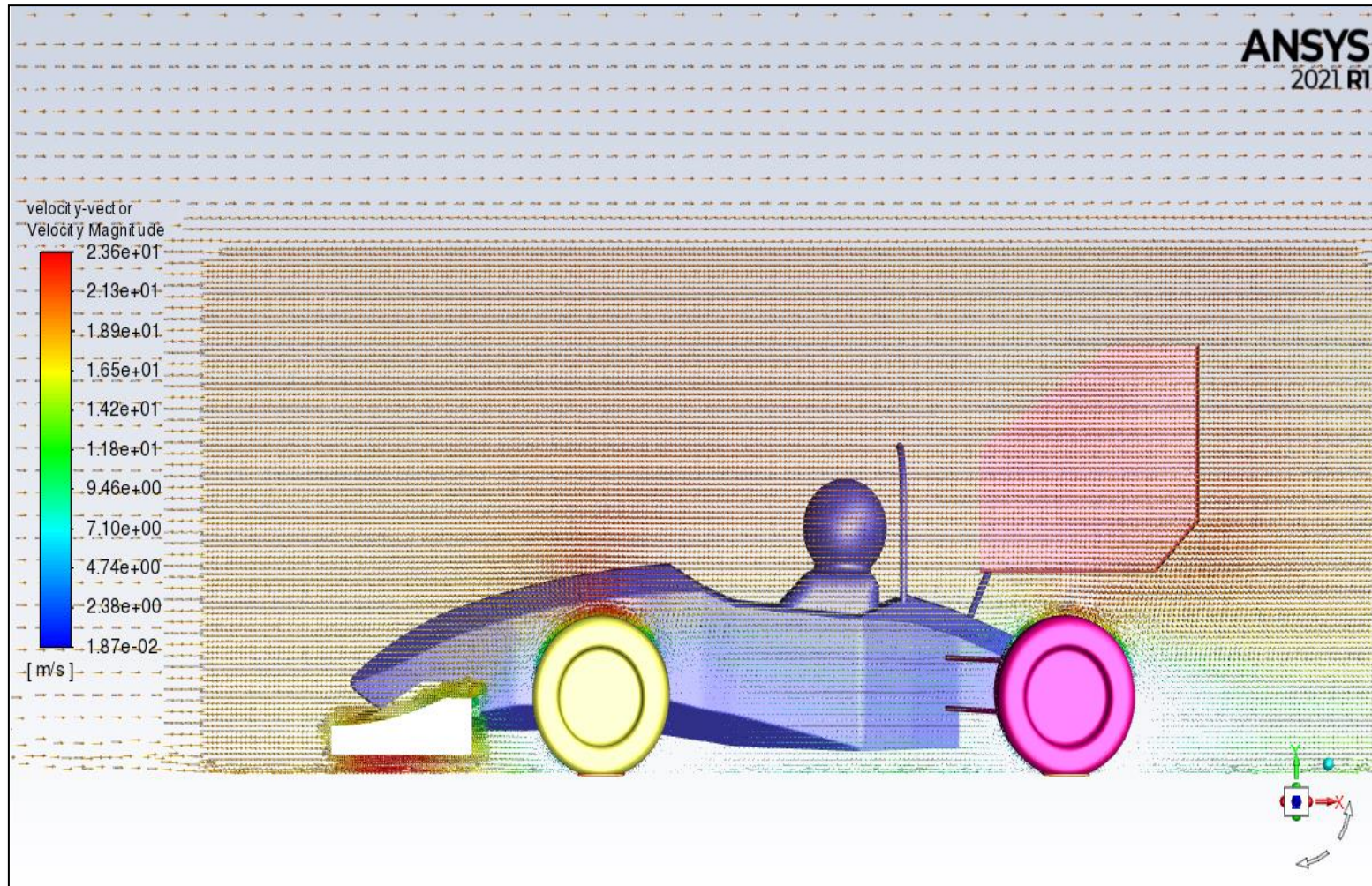
FIG. V. Pressure contour of the car



HIGH PRESSURE due to flow stagnation

LOW PRESSURE due to flow separation

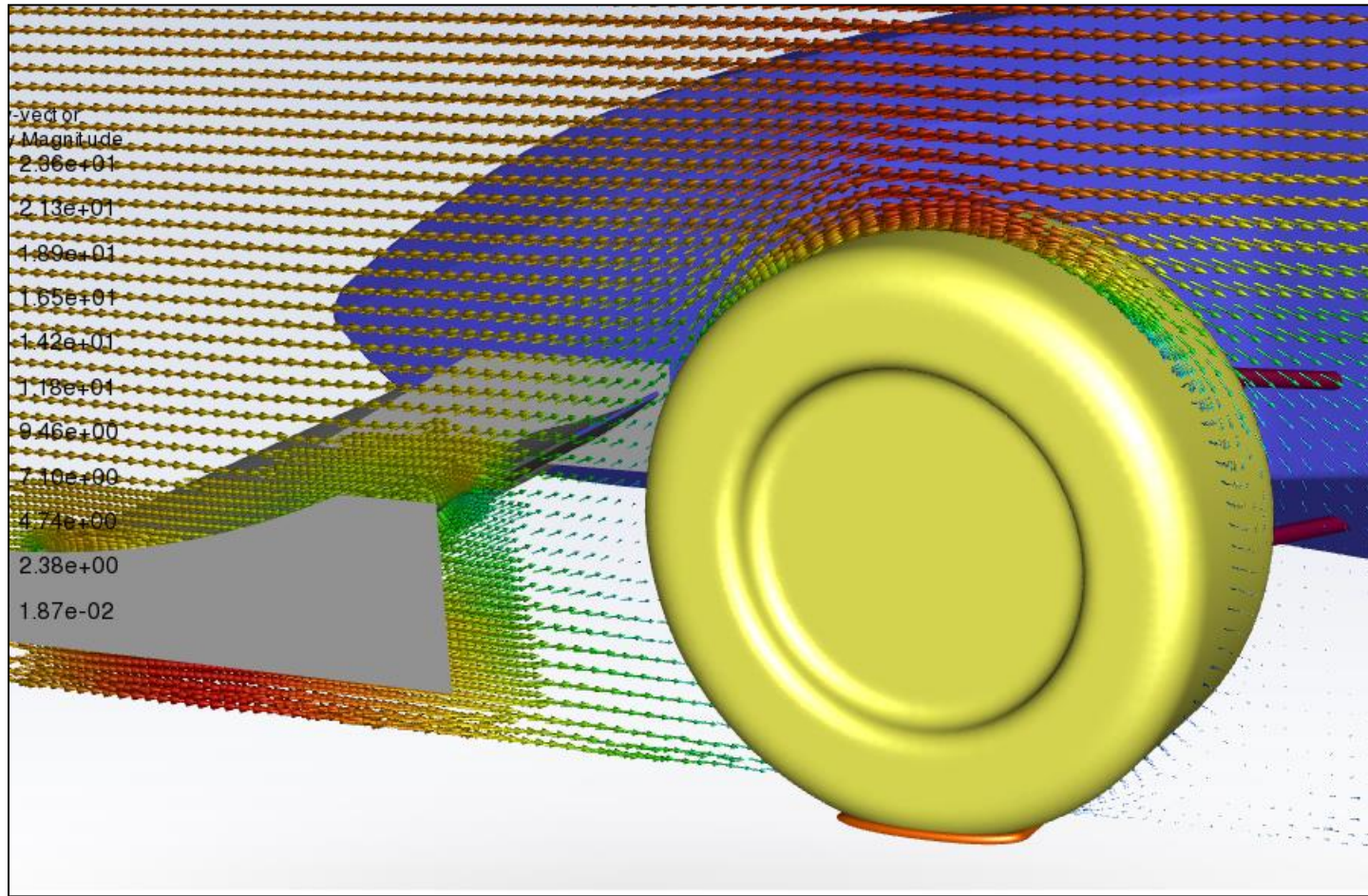
Here we see the velocity vector of the car:



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

FIG. W. Velocity vector of the car

Here we see the velocity vector of the car:

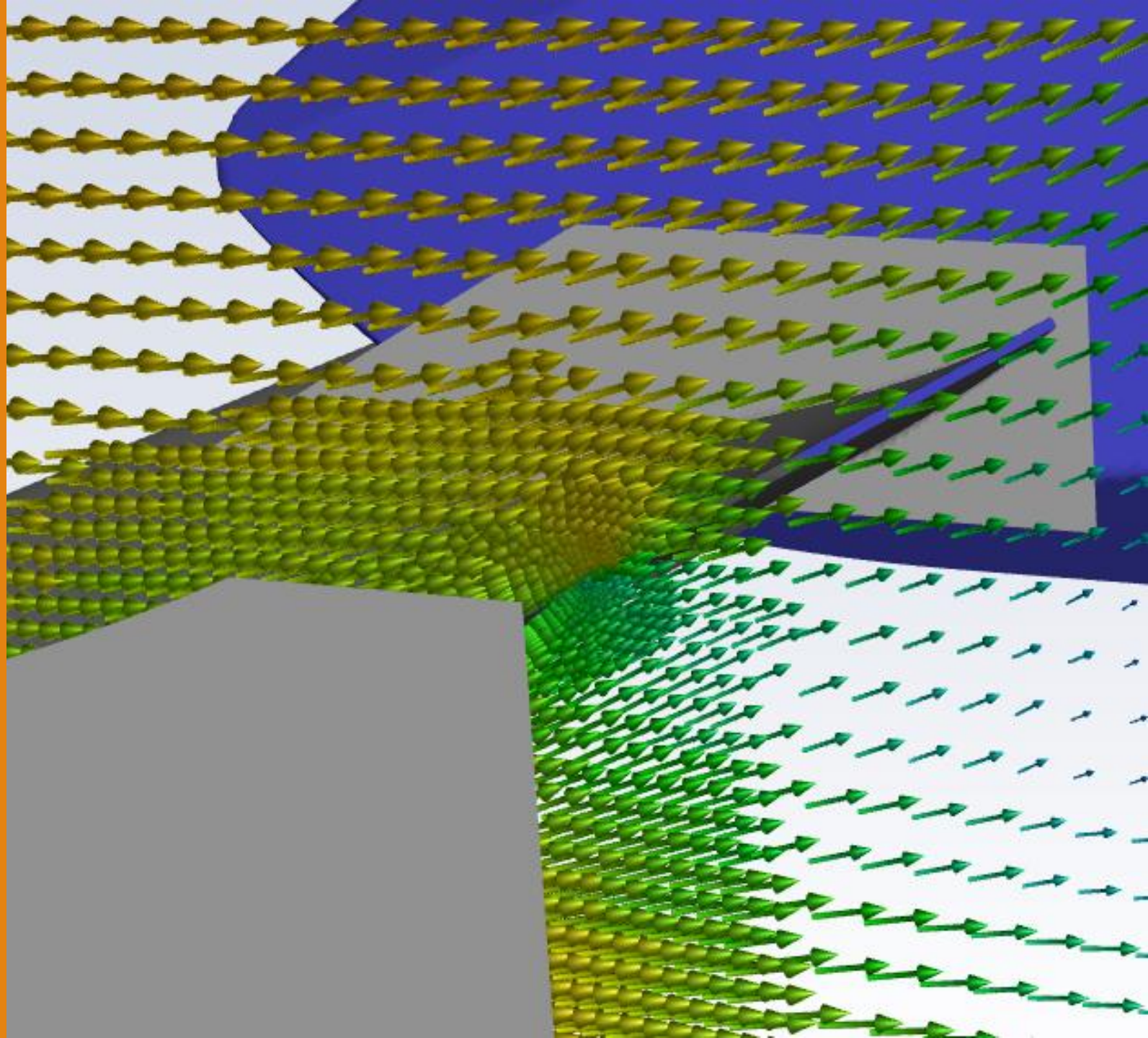


Air flow near the wheels also has strong rotational component and flow vertices where it shows the region of the rotating flow.

FIG. W1. Velocity vector of the car

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.

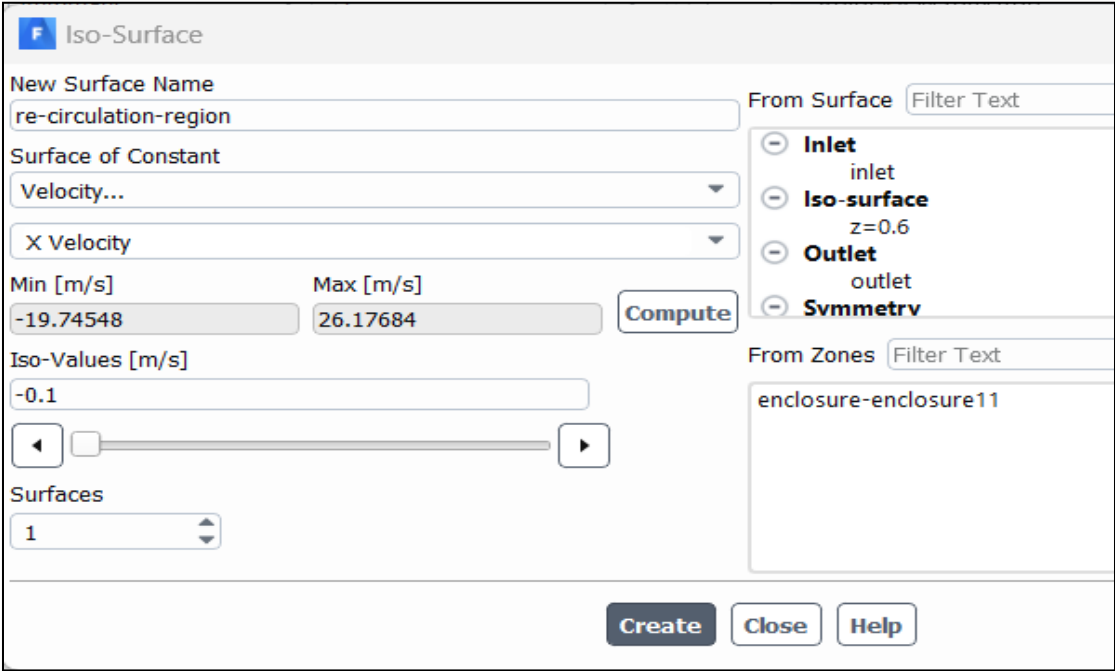


FIG. X – Velocity of X - Axis under ISO - SURFACE

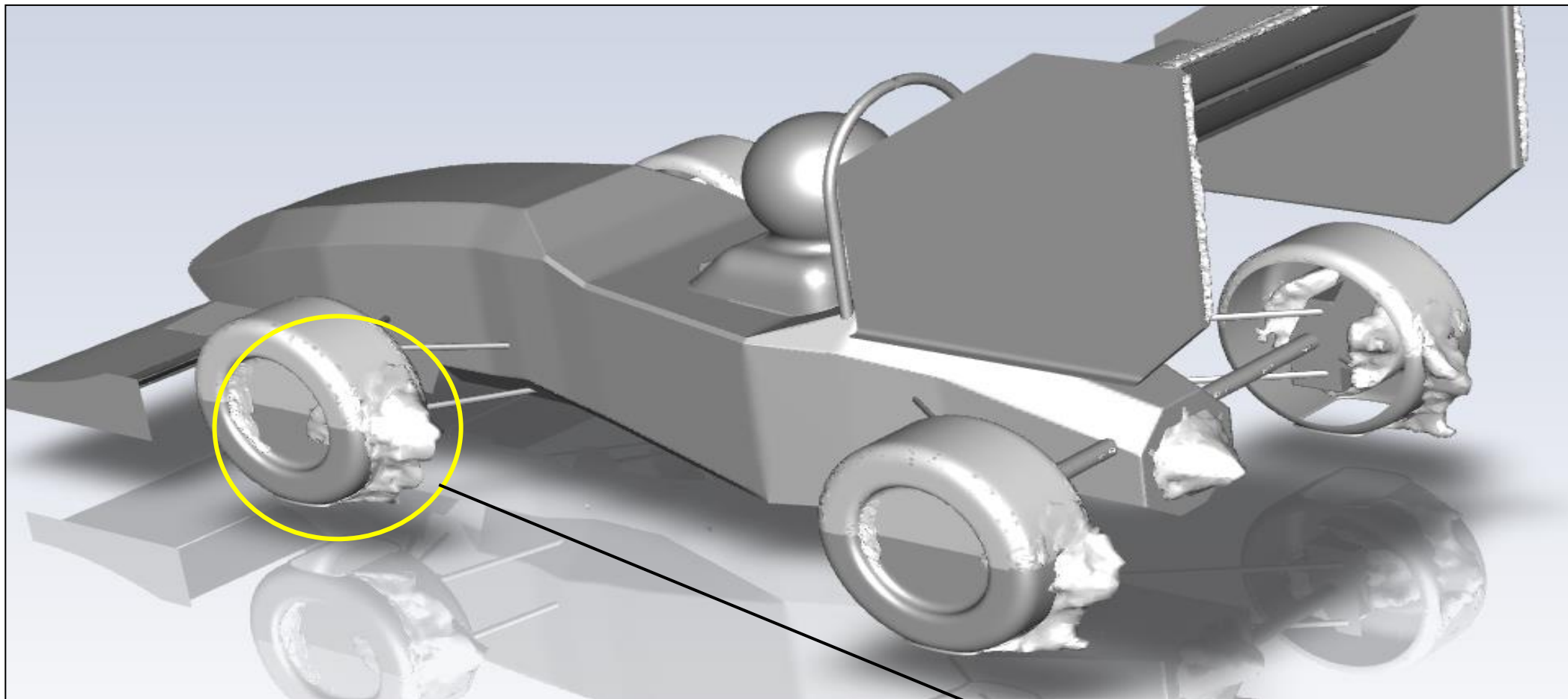


FIG. X 1– Velocity of X - Axis under ISO - SURFACE

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

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

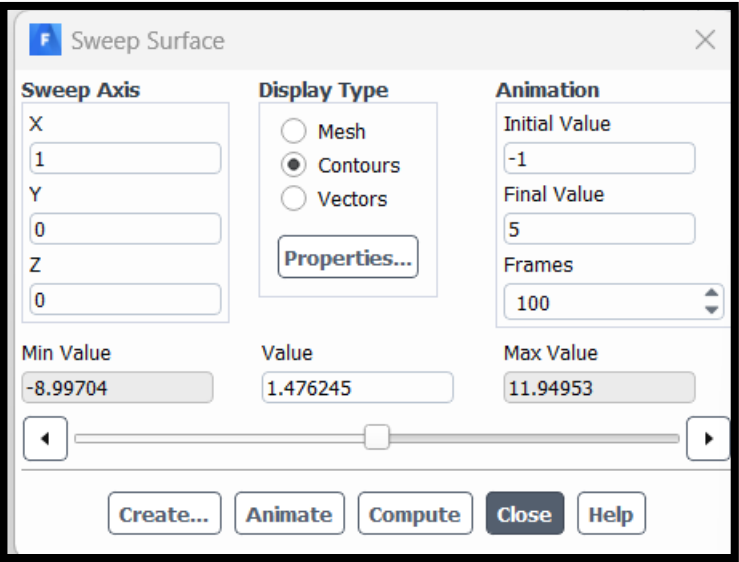


FIG. Y1– Sweep surface settings

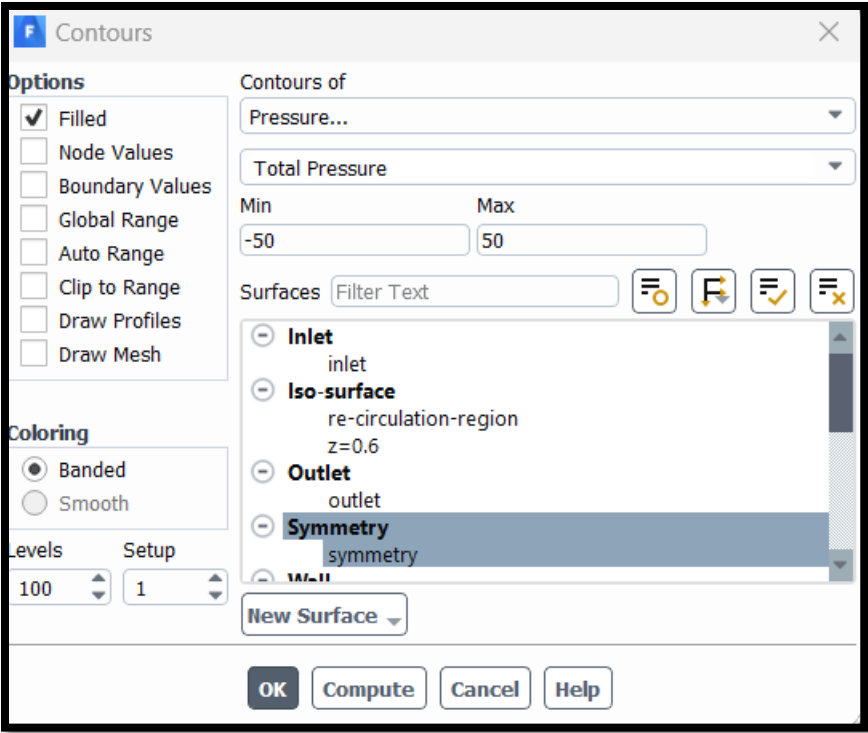


FIG. Y2– Sweep surface settings

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.

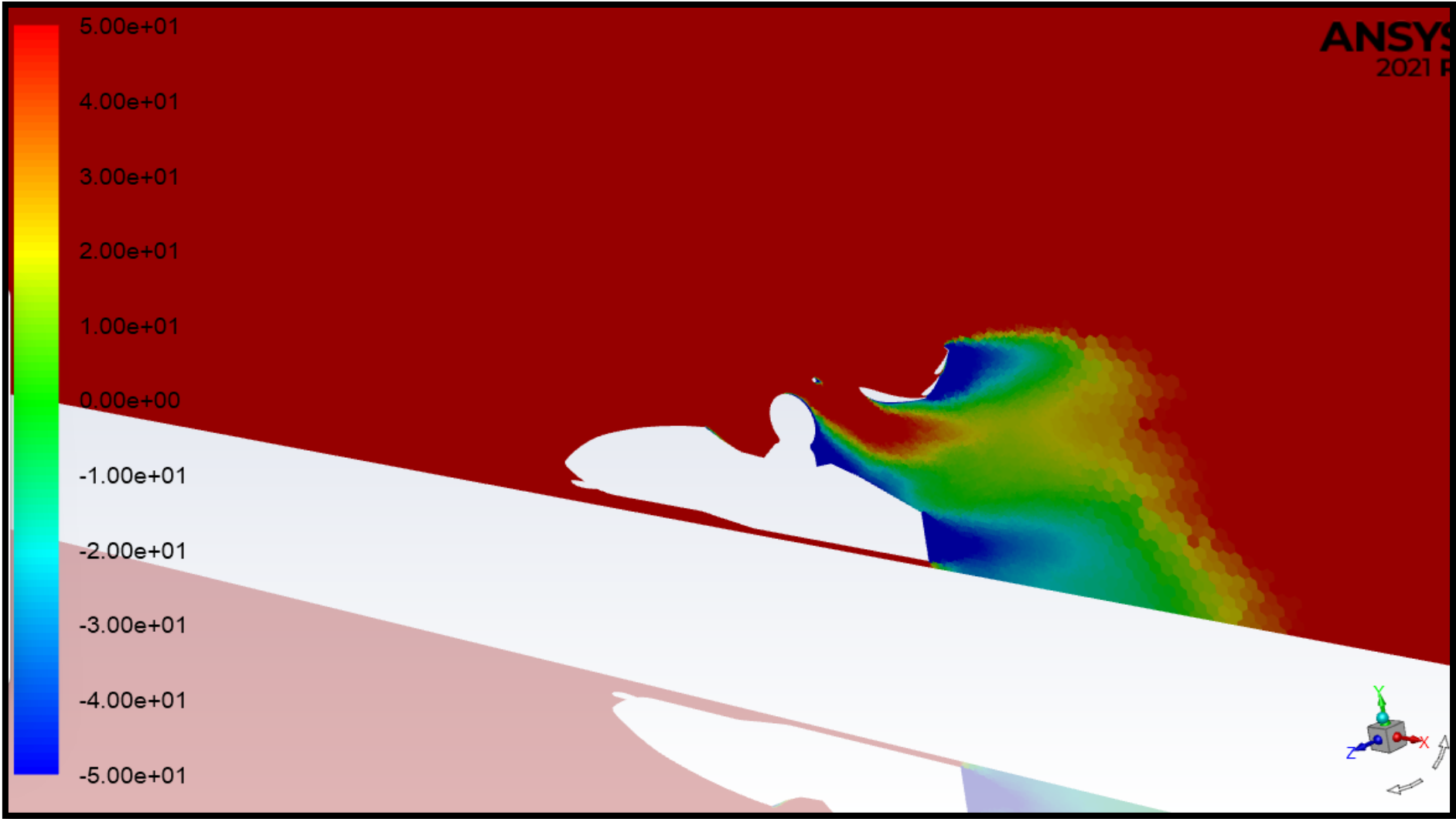


FIG. Y3– Sweep surface contour plot

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

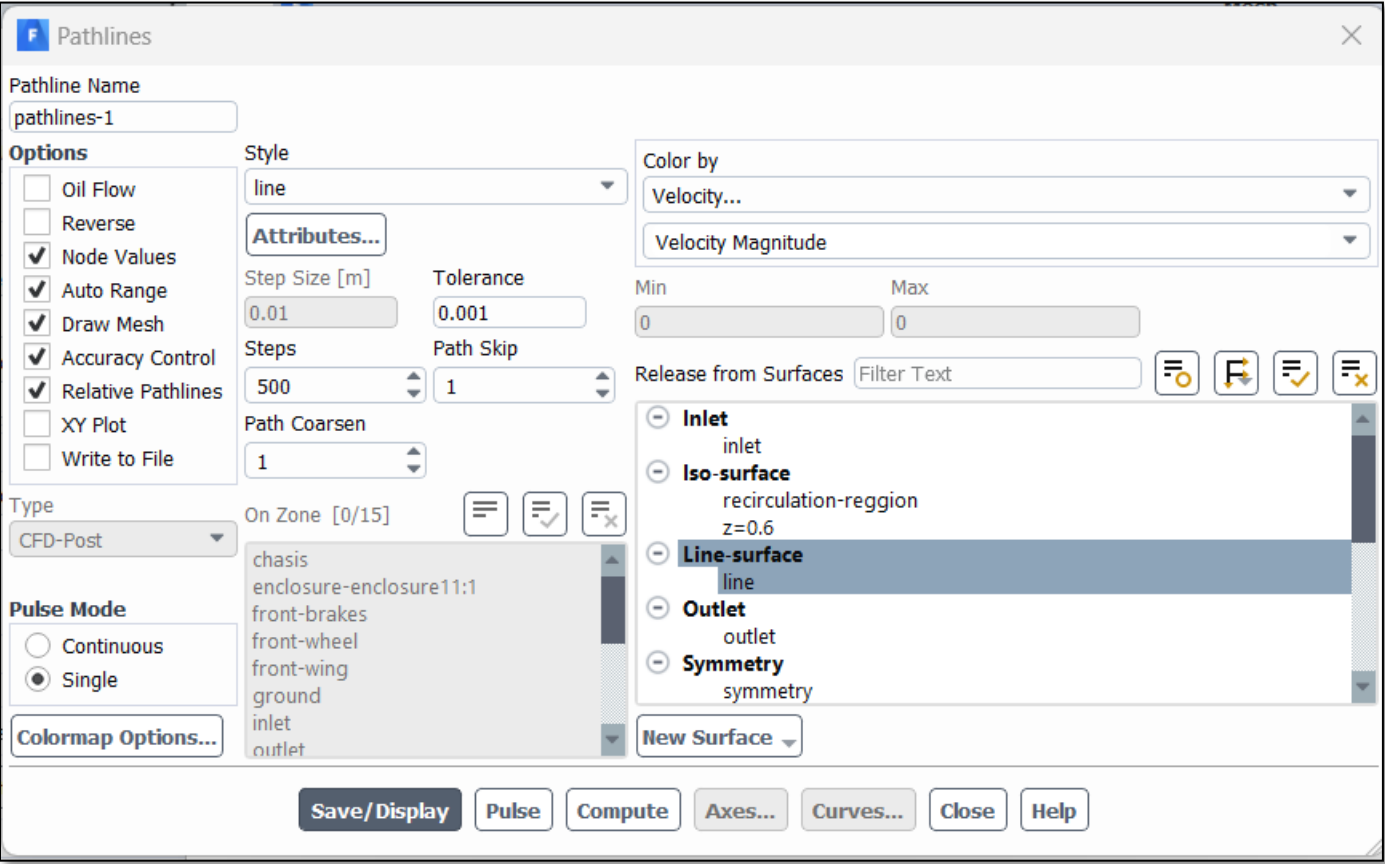


FIG. Z1– path line settings

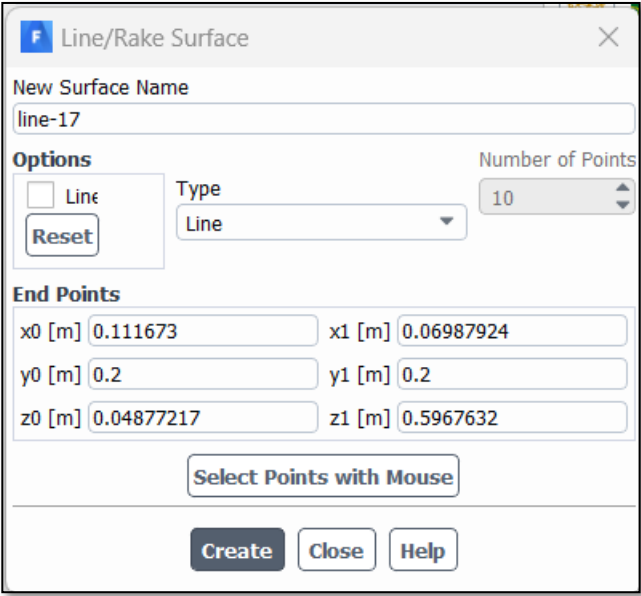
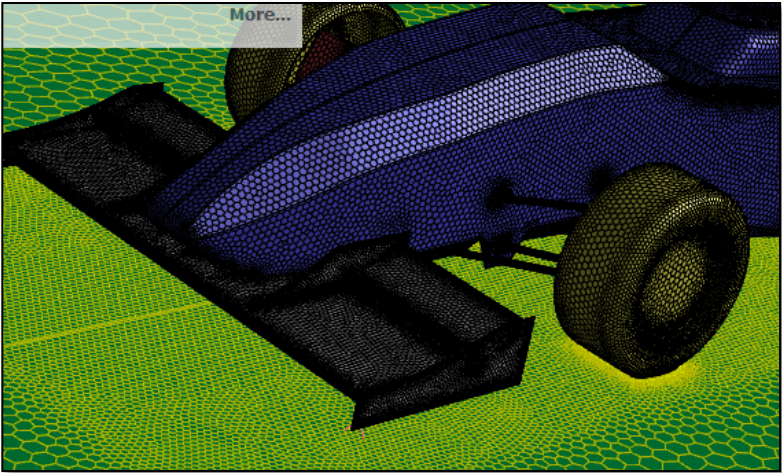


FIG. Z2– Line and rake surface



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

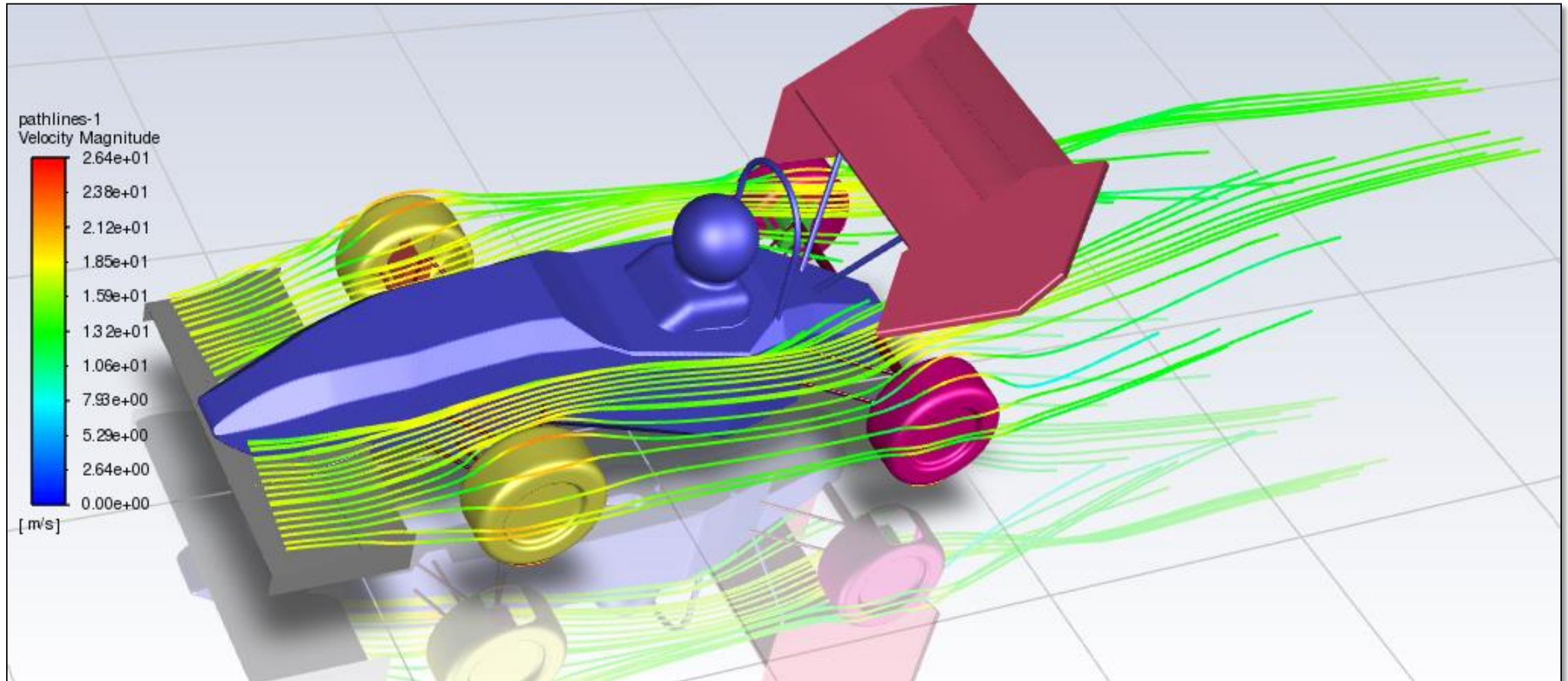


FIG. Z3– Path Lines

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

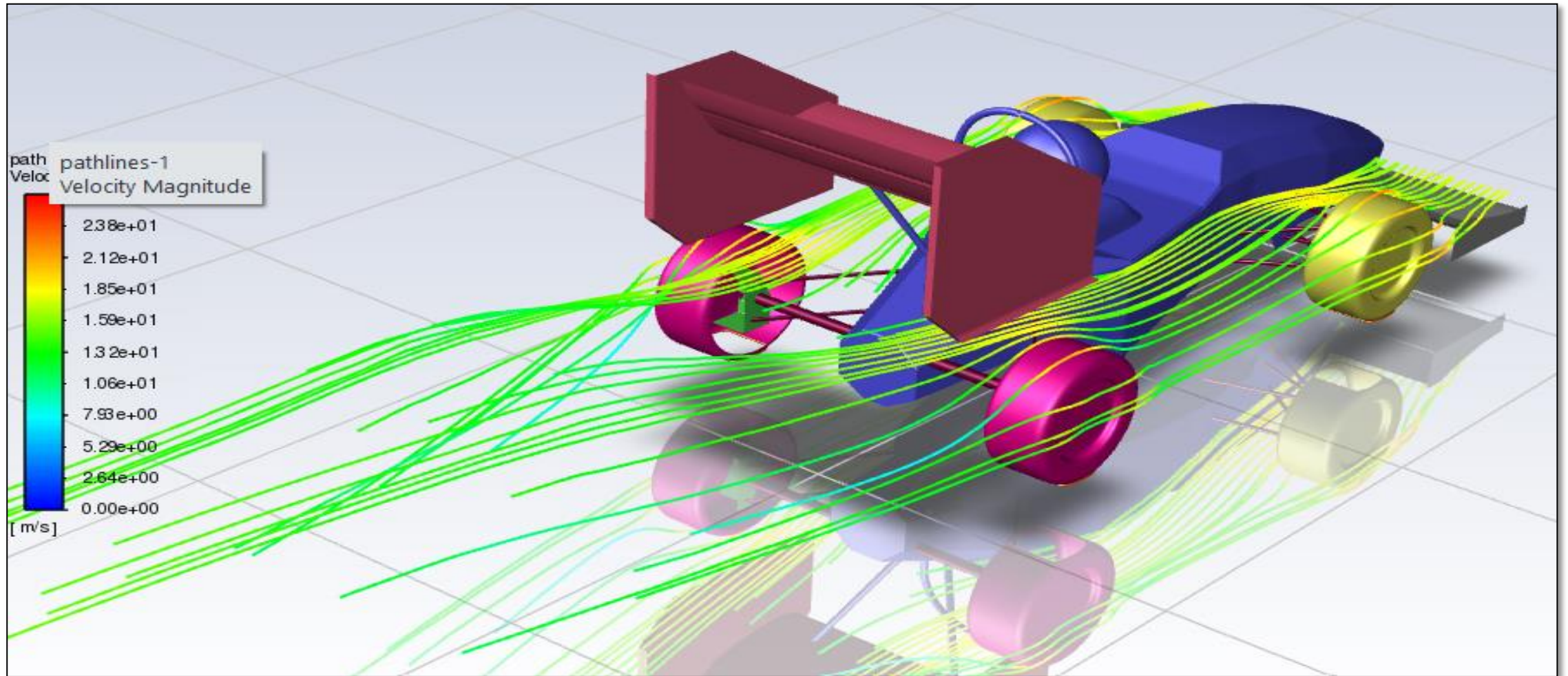


FIG. Z4– Rear Path Lines

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!