



---

# CFD ANALYSIS OF A FORMULA SAE CAR MODEL

---

AERODYNAMICS



BY  
ADITYA SONWANE  
MARCH 2019

# **1. PRESENTATION**

Given a CAD model of a Formula SAE car we are to run a simulation. The simulation is to be meshed and modified as per required and is to be run from the beginning. We completely split the vehicle and then applied various conditions to it as it is an easy method for the simulation to run. The FSAE car has many aerodynamic parts and regions like wings, spoiler (not in this particular model), which are needed to be considered while generating the mesh and running the simulation. We have meshed the car model differently for plane surfaces of the car body, for the curves and edges, and also for the curved sections so that we get a good volumetric mesh. After the meshing the baseline simulation was run for 1000 iterations, the cells generated after meshing were 4171132 with 12466665 faces and 4410962 vertices. Required plots and reports were monitored like mass flow, residuals, down forces, lifts, etc. The simulation was then modified and improved by changing the parameters of the meshing and of the simulation so that we get more accurate results for the FSAE car. The improved simulation was run for 1500 iteration. This improved simulation had 6253006 cells with 18697683 faces and 6565881 vertices. The graphs and images of the improved simulation are also presented and compared to see the difference in the baseline simulation and the improved simulation.

# **2. SUMMARY**

This is a simulation of a FSAE car model. We are to learn how to derive various aerodynamic properties after setting up the model, meshing, surface wrapping applying initial conditions, boundary conditions, creating different volume sources so that we get refined mesh at the particular parts or regions like the nose of the car, wings, etc because it is difficult to obtain results at such complicated regions if same conditions are applied over whole body. After applying these parameters we set the global meshing parameters which are corresponded to the free stream velocity of 50 mph, a specific turbulence intensity, air density of 1.2 kg/m<sup>3</sup> and the dynamic viscosity. Along with that we are needed to specify the

boundary mesh parameters so that we execute a good volumetric mesh. We are to check the mesh quality after the mesh execution. This is necessary because it will not create any problems further while the simulation is run and with the results. To properly check the mesh around the car we created some section planes at specific regions like the wing, the wheel and the side pod duct. After completing the simulation reports were plotted to check the solution of the simulation, plots like Lifts, Drags, Downforces, mass flow are created and monitored. Also to check the proper meshing after running the simulation we created various scalar scenes to check the velocity magnitude, pressure coefficients and the wall  $y+$  and many more parameters and reported the results.

### 3. PROBLEM DESCRIPTION

In this study of running the CFD simulation, we are needed to run the simulation for capturing critical flow over the car body. Over the critical parts of the wings and the curves as well. Analyse the flow fields in the mentioned areas of car. The simulation that is to be run is a baseline simulation. After monitoring the reports and plots, we can modify and improve the mesh and the simulation to give us more detailed and better results. This can help in understanding the airflow and aerodynamic properties of the FSAE car in a free stream velocity.

### 4. CASE SETUP

- ***Pre-processing: Generation of surface and volume mesh***

We imported the cad model of the car. Then made them to split by non-continuous so that we can differentiate the wheels and the car body. To the car body we used split by patch to properly get the differentiated parts from body like sidepod, wings, hook, endplates, curved surfaces.

- ***Surface wrapping***

Since the car body is split into many parts it will be hard to use the previous mesh so surface wrapping operation is performed to get generate another

mesh of the car. The base size in the default controls was 8mm, the target surface size was 100%, the minimum surface size was 5%.

Special custom controls were added while surface wrapping for the curves and edges of the car. The base percentage was 25% and the minimum surface size value was 5%.

Also custom controls were created for the hook like structure, for lower and upper wings , inner and outer end plates as well.

To prevent the overlapping of wing contact surfaces contact prevention was created with minimum size of 0.5mm.

And the surface wrapper was executed.

- ***Wind tunnel***

Wind tunnel was created as a block of following dimensions

	X	Y	Z
Corner 1	-9.5 m	0.0 m	0.004 m
Corner 2	37.0 m	6.0 m	8.0 m

This wind tunnel still needs boundary conditions and the block surfaces are needed to be named as per the tunnel. Using split by angle with 89 degree we created inlet, outlet, symmetry plane, far field and the road.

Then we used the subtract operation to get the air wind tunnel.

- We now created some more blocks for mesh refinement at particular sections like at the wing, under tray very near to the car, and in middle of the near car block and the wind tunnel block using following dimensions

Name	Areas	Corner 1(x,y,z)			Corner 2(x,y,z)		
Near	Near the car	-1	0	0	5	1.2	1.5
Middle	One level coarser than near field	-3	0	0	8	1.5	2
Under tray	Under tray region	-1	0	0	5	1	0.1
Wing	Wing region	-0.95	0	0	0	0.8	0.4

- Now boundary conditions were set by creating a fluid region. In this fluid region the inlet, outlet, symmetry plane, wall these conditions were applied to the wind tunnel.
- Now we created an automated mesh which will help un surface remeshing, automatic surface repair, trimmed cell meshing, prism layer meshing.
- ***Setting Global mesh parameters***

This is needed to set the prism layer properties for the simulation.

In the default controls we selected the base size of 24mm, number of prism layers to 6, percentage of base for total wall thickness as 33 and prism layer near wall thickness as 0.1mm.

- ***Boundary specific parameters***

For the block of mesh refinement we created we are to specify some specific parameter.

For the wind tunnel prism layers were disabled and minimum surface size was made custom. The target surface size was made 2000 and the minimum surface size was made 200.

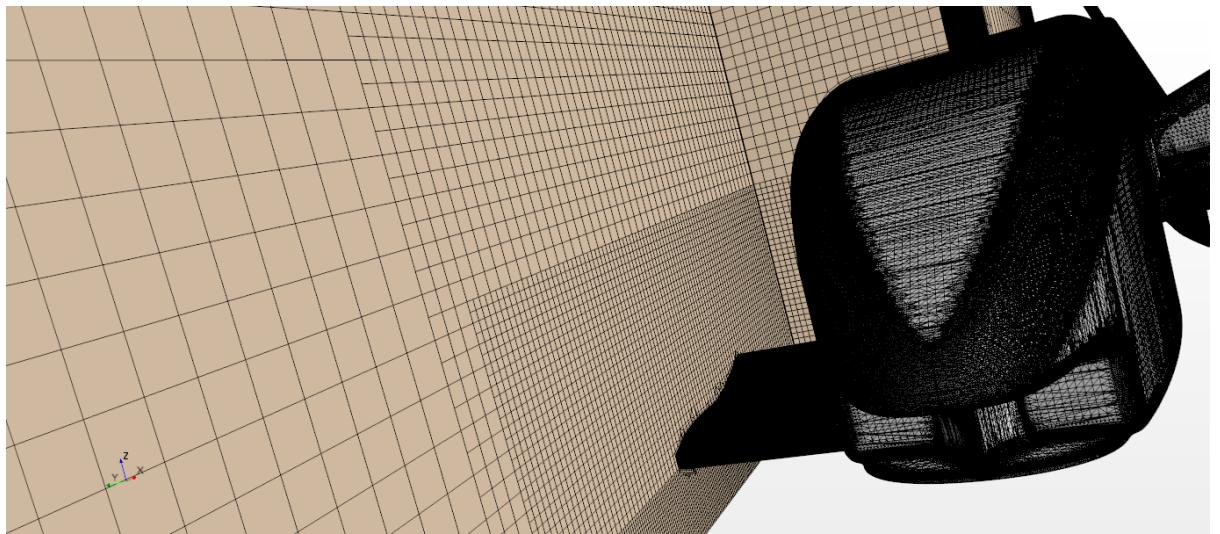
For the near block relative size was 100, for the middle block the relative size was 200, for the block at the wing the relative size was 50 and at the under tray the relative size was 33.

- Completing the above steps the volume mesh was executed.
- Now after the volume meshing is completed we are to check the quality of mesh. Doing this we will be able to know if there is need for remeshing before the simulation is run.
- To check the meshing around complicated parts we create section planes and cut them through the component giving us the required mesh around the component.

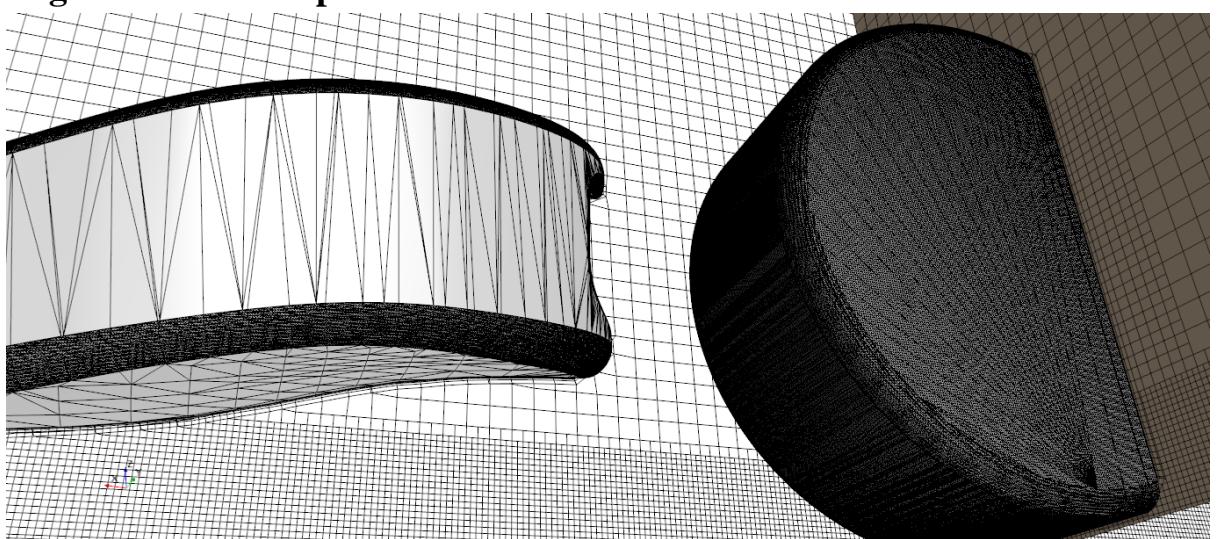
The 1<sup>st</sup> section plane was made normal to Y at (13.75,0.375,0.4) this plane showed the mesh around the wings of the car.

2<sup>nd</sup> section plane was also normal to Y at (0,-5,10) this plane showed the mesh around the sidepod duct and sidepod.

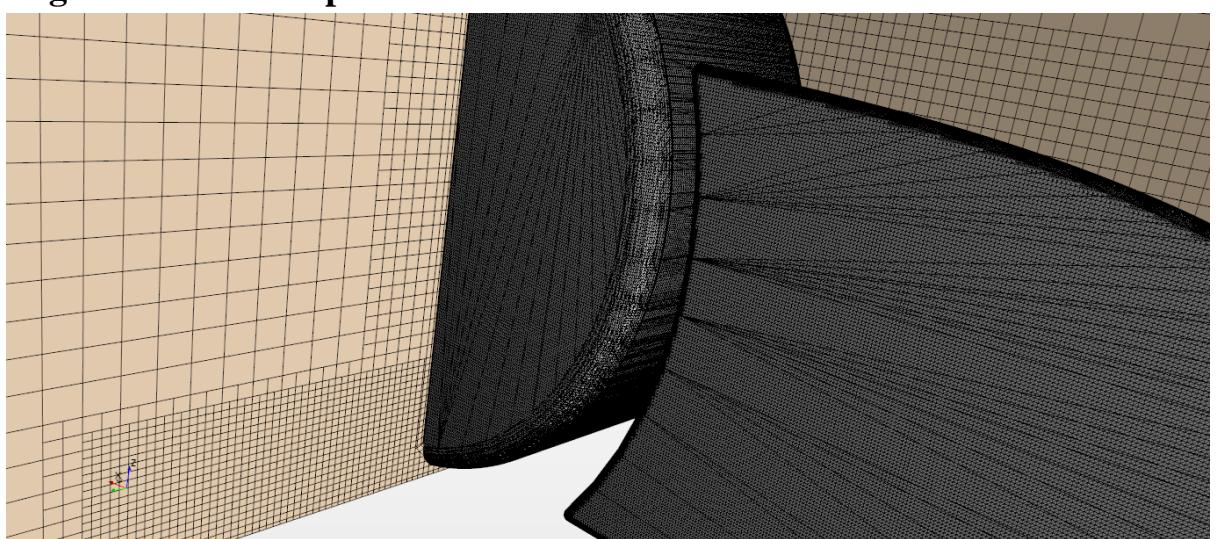
3<sup>rd</sup> section plane was made normal to X at (0.588,0.5,0.50044) this plane showed the mesh around the front wheel of the car.



**Fig. 4.1. 1<sup>st</sup> section plane normal to Y**



**Fig. 4.2. 2<sup>nd</sup> section plane normal to Y**



**Fig. 4.3. 3<sup>rd</sup> section plane normal to X**

- ***Physics setup***

The volume that is intended to analyse has discretized into many small volumes that we can now solve.

The required physics setup from the physics node selecting model is as follows:

- i. Stationary model
- ii. Gas
- iii. Segregated flow
- iv. Constant density
- v. Steady state
- vi. Turbulent flow
- vii. K-omega turbulence model
- viii. Selecting the cell quality remediation

- ***Initial conditions and boundary conditions***

The most important initial condition here is defining the inlet velocity in X-direction as 50mph.

Set the outlet pressure of 0 gauge pressure.

Set the symmetry plane and far fields as symmetry plane parameter.

Set grounds wall parameter to wall.

Set sidepod duct pressure as 0 gauge pressure.

Now for the front and rear wheel we are to set the axis of rotation as (0,-1,0). The rotation rate of 660 rpm.

The only difference in the front wheel and rear wheel is of the origin, origin for front wheel is (0.07,0,0.25) and for rear wheel is (1.765,0,0.25)

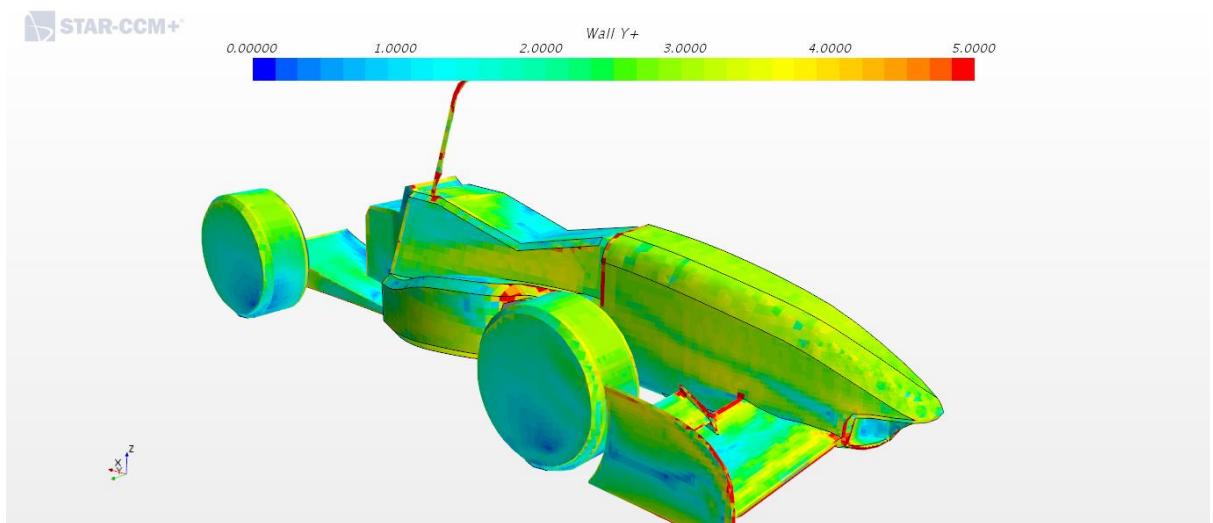
## 5. COMMENT ON OVER ALL QUALITY OF MESH

The quality of mesh of the baseline solution is good but not much refined as it has 4.9 million cells. This mesh effects on the monitor plots and the scenes of the car simulation. The parameters that capture the flow can not properly capture and distinguish the near by regions which are being highly affected by the flow than to those which are not. The baseline simulation mesh can be modified and improved to get better results in the analysis of the model car. After examining the baseline mesh it was needed to be

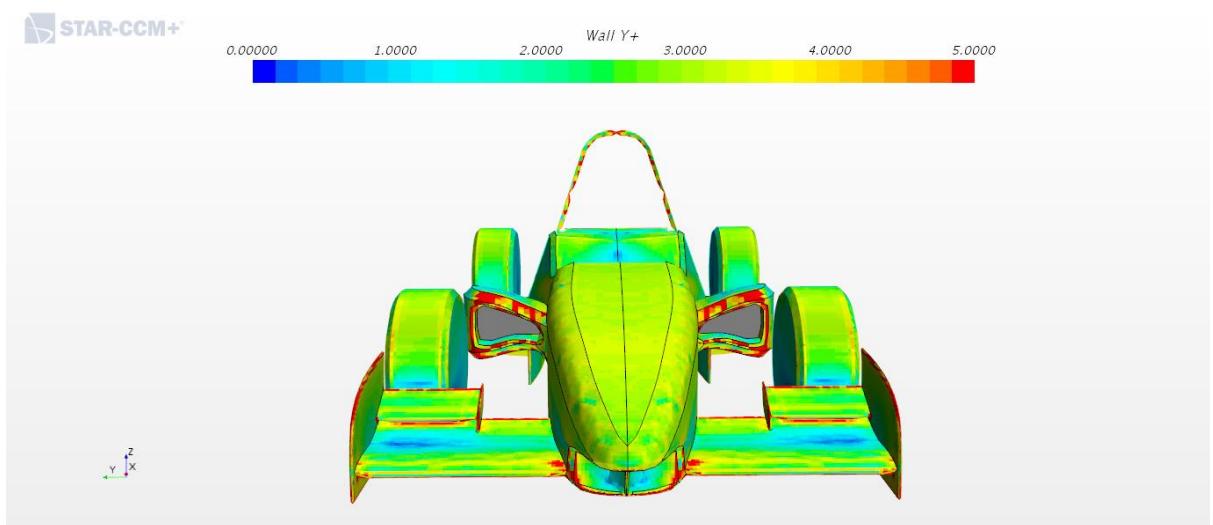
modified to get more fine mesh at the wheels at the curves sections like hook like structure, sidepod ducts ,etc The modified solution had around 6.2 million cells. As these many cells cannot make the mesh coarse we are to get better results. To check the quality of mesh I have added two section planes adjacent to each other normal to the Y and also a section plane normal to X to check the quality of mesh at wings and the sidepod. The coordinates at which the section plane is added are mentioned in the physics set up

## 6. COMMENT ON EFFECT OF MESH

Poor mesh quality in a simulation will result in inaccurate solution or very slow convergence of the model. Even after the model with poor quality mesh is converged it will not be as good as the solution of a good quality mesh. We have used custom mesh at the curve sections of the car body. If the mesh at these regions is not proper. This can result in false results. Most of the errors in the mesh occur because of following reasons: if the mesh is coarse, it has high mesh skewness, there are large jumps in the volumes of the adjacent cells, if there is an inappropriate boundary layer mesh. All these errors in mesh may ruin the simulation, analysis and the post processing of the solution. Even this can mis calculate the results as well. If we generate a fine and more accurate mesh as it should be we can get great results but because the mesh is too fine it may require a lot of time and memory to run. To check the mesh quality we can compare the wall  $y+$  scene generated in the simulation, for baseline simulation you will see a little different mesh as compared to the improved one. In my condition I ran 3 baseline simulation and 4 improved and updated simulations. The problem with the baseline simulations was it generated just 1.9 million cells at first then for 2<sup>nd</sup> time it generated about 3.8 million cells no matter the simulation ran for 2<sup>nd</sup> time but the wall  $Y+$  seemed really messed up so in the 3<sup>rd</sup> baseline simulation I remeshed and got about 4.6 million cells which executed good results. For the improved quality simulation at first my cell size was not much improved, for 2<sup>nd</sup> time it got about 4.9 millions cells later 5.8 million and at last about 6.2 million simulation if we compare the results we can see good refinement and accuracy in results.



**Fig. 6.1.** Wall  $y+$  in the baseline simulation.



**Fig. 6.2.** Wall  $y+$  in the improved simulation.

## **7. HOW TO MAKE IMPROVEMENTS IN SIMULATION**

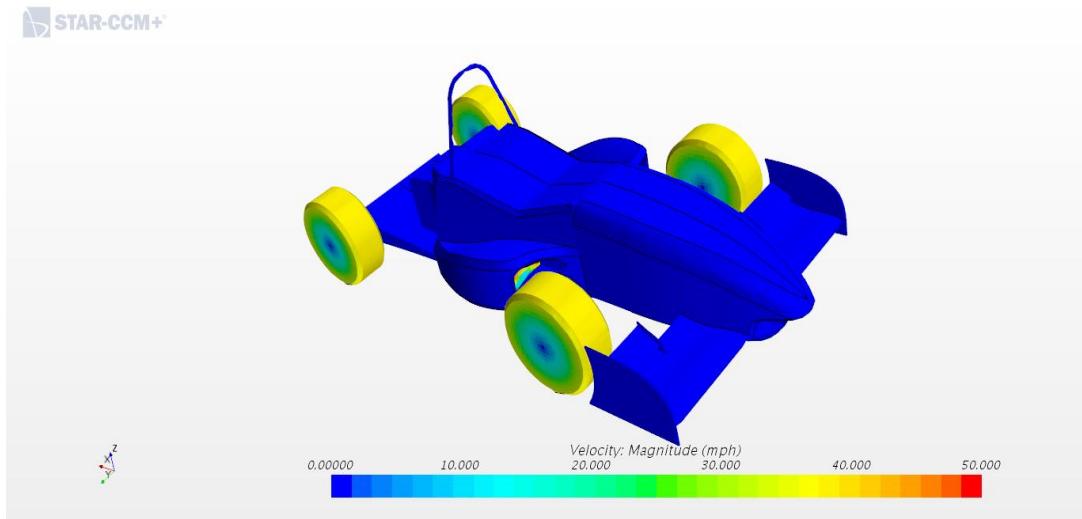
For getting better results we improved the mesh and ran the simulation again. The changes done in the simulation are listed below:

- The base size in the default controls was changed from 24 mm to 21 mm.
- The prism layer near wall thickness for wheels was changed from 0.1 mm to 0.05 mm
- The prism layer near wall thickness for body was not changed it was kept constant at 0.1 mm.
- For the surface wrapper the percent of base was changed from 25 to 20.
- Also the minimum surface size node in the surface wrapper was changed from 5% to 3%.
- The baseline simulation was run for 1000 iterations and now the improved simulation was ran for 1500 iterations to obtain better accuracy
- All the surfaces and curves also the custom controls were modified to get the improvements.
- Changing the values like above has resulted in increasing the cell up to 1.5 millions and hence getting a better result.

## **8. ANALYSIS**

- After completing the 1500 iteration steps we have created monitors for downforce of wing and body, drag of wing and body, lift of wing and body, mass flow over the sidepod duct, residuals and also created various scenes to check the distribution of pressure, velocity in all 3 directions and these all scenes were also created on the centre plane i.e. the symmetry plane of the car model.

- After examining the results of the simulation, we can say that simulation has derived the proper results. Checking the flow over the body, the pressure coefficient, velocities etc.
- As it should be the velocity of the flow is proper it is more above the car and low below which is the reason for the generation of forces.



**Fig. 8.1. Velocity magnitude over the car body.**

- The pressure coefficient shows the pressure distribution which is different at the wheels and at the body.
- Along with that the wall  $y+$  scene is also analysed the prism layer near wall thickness for wheel was changed to 0.05mm hence we can see the difference in the wall  $y+$  of wheels and body.
- Also stream lines were generated to see the flow above and below the car body.
- We have determined the coefficient of down force and cross checked to see if the results we are getting are valid or not.

After running the downforce report we ran got the downforce of **63.5257N**. Now we created another reports for front wheel and rear wheel moment. After running those reports we get, moment on front wheel as **15.2659 N-m** and moment on rear wheel as **92.41017 N-m**.

Getting the total of these two moments we get:

$$15.2659 + 92.41017 = 107.67607 \text{ N-m}$$

The distance between the front and rear wheel is calculated by taking difference between the origin points

$$1.765 - 0.07 = 1.695 \text{ m}$$

Now dividing the total moment by the distance between wheels will give us the down force which should be equal to the downforce from reports or else there is error in simulation.

$$107.67 / 1.695 = 63.51 \text{ N}$$

**Since the answers are equal, the simulation is valid.**

- Another way to check the simulation is by the force coefficients. Creating force coefficients for total down force, for rear wheel and for front wheel.
- The sum of the coefficients of downforce on front and rear wheel should be equal to the total down force coefficient.

$$\mathbf{C\_DF(total) = C\_DF\ rear(at\ front\ wheel) + C\_DF\ front(at\ rear\ wheel)}$$

Running report for downforce force coefficient we get 0.58019

For the down force at front wheel we get 0.08225

And for the down force at rear wheel we get 0.49784

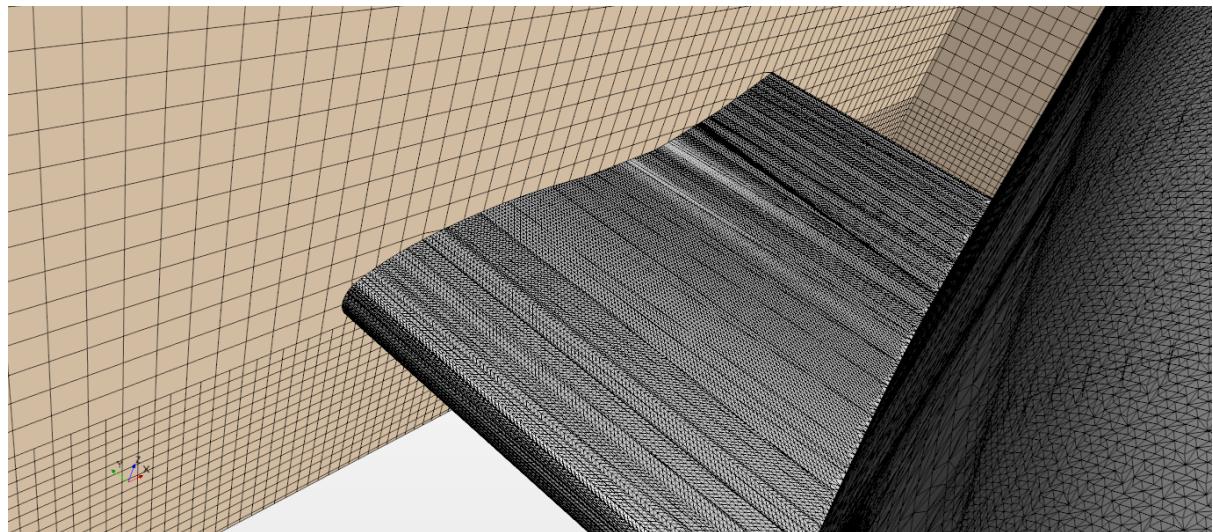
Doing the sum of the front and rear downforce force coefficients we get

$$\mathbf{0.08225 + 0.49794 = 0.580189}$$

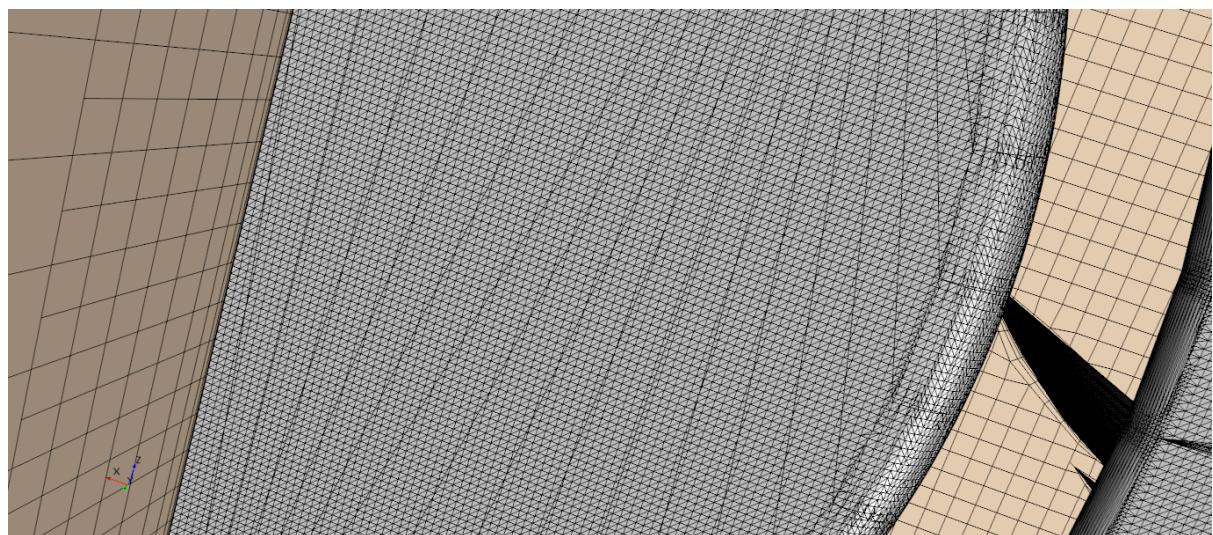
This value is equal to the total downforce value so the simulation is valid.

## 9. Scenes and plots

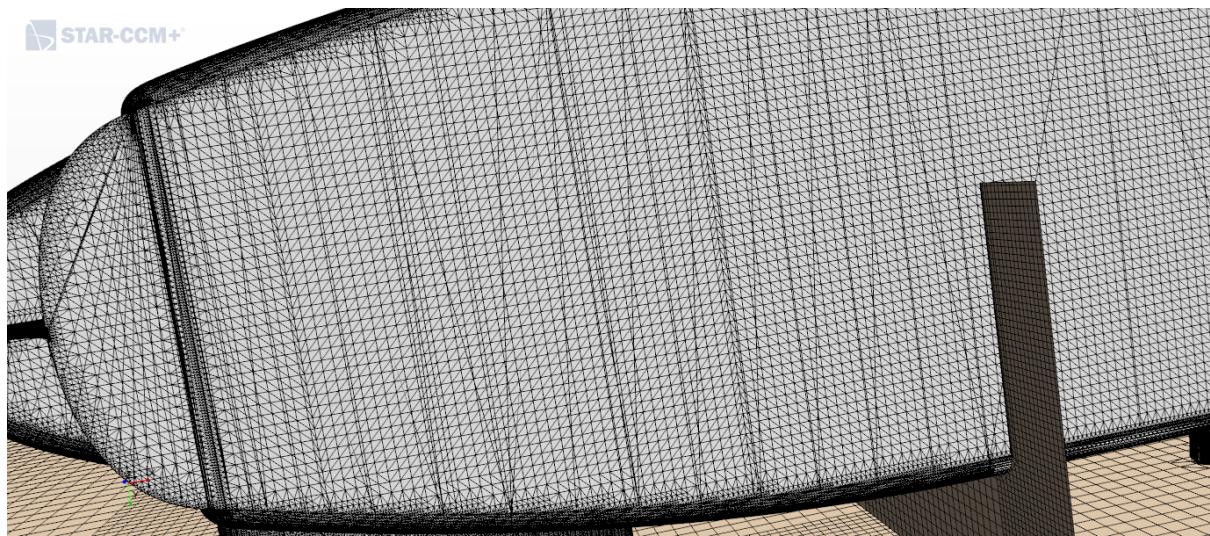
### A. Mesh scene



**Fig. 9.A.1.** Zoomed mesh scene near wing.

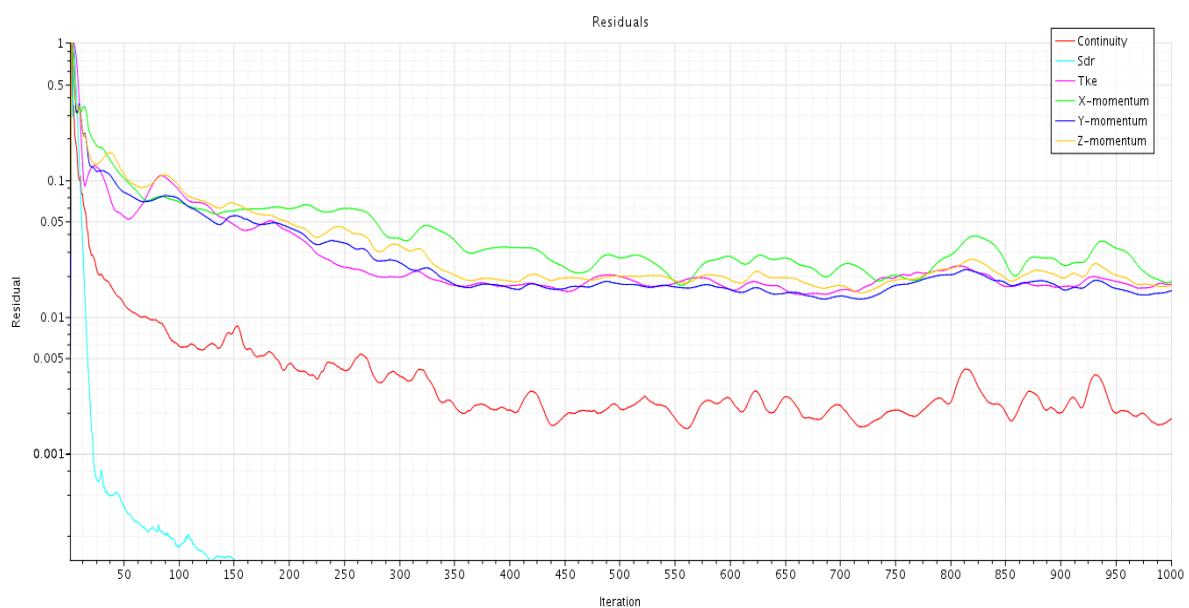


**Fig. 9.A.2.** Zoomed mesh scene near the wheel of car.

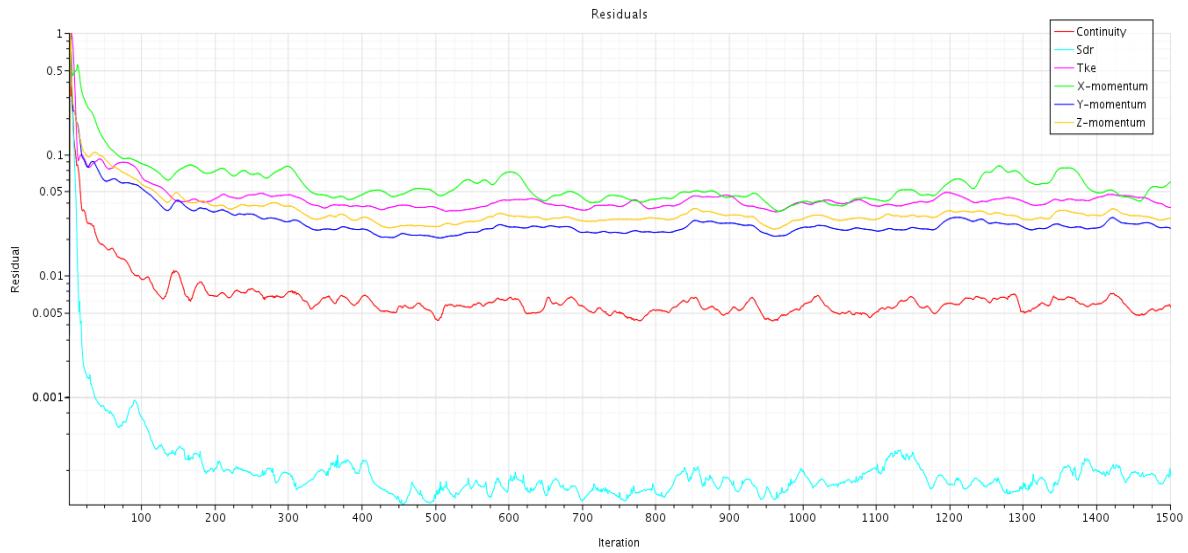


**Fig. 9.A.3. Zoomed mesh scene under the car body.**

## B. Residual graph

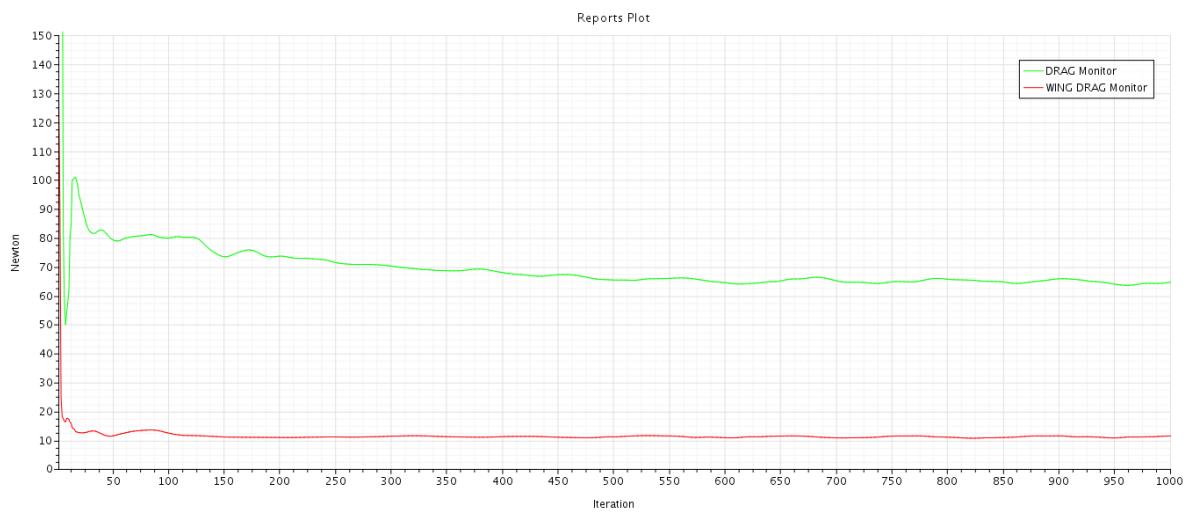


**Fig. 9.B.1. Residual graph for the baseline simulation.**

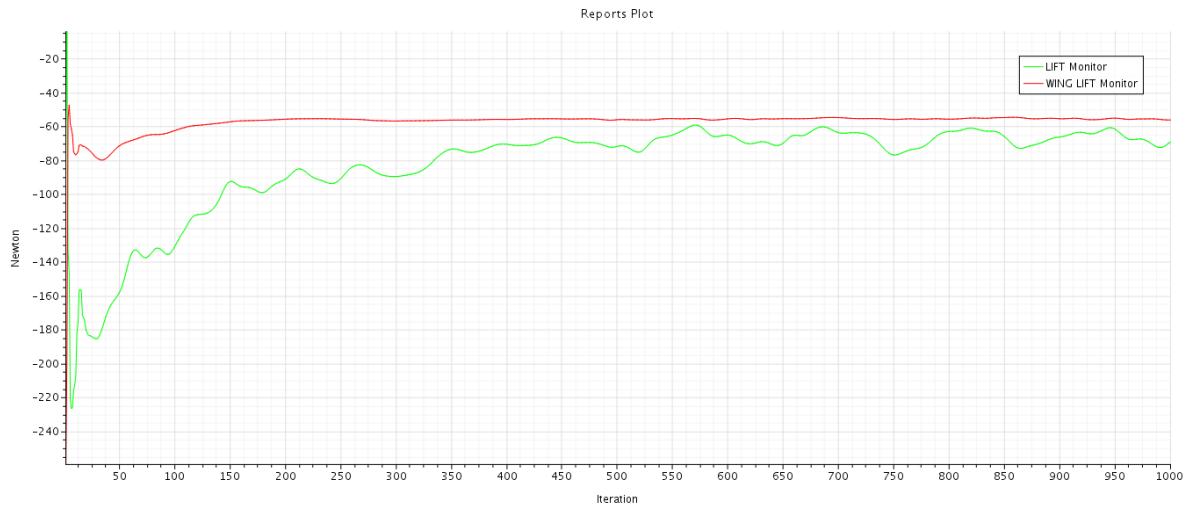


**Fig. 9.B.2. Residual graph for the improved simulation.**

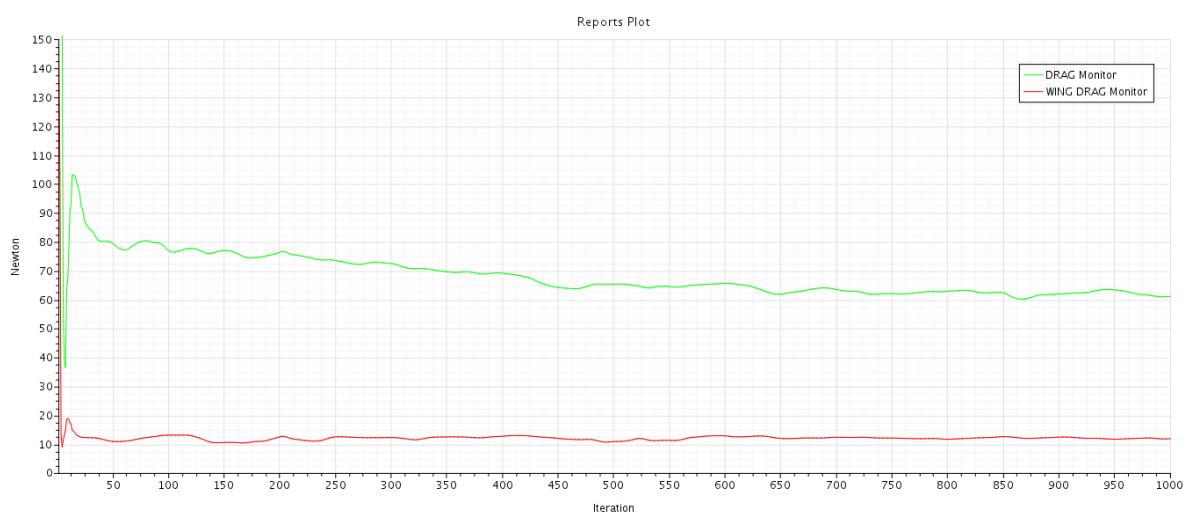
## C. Monitors of drag and lift



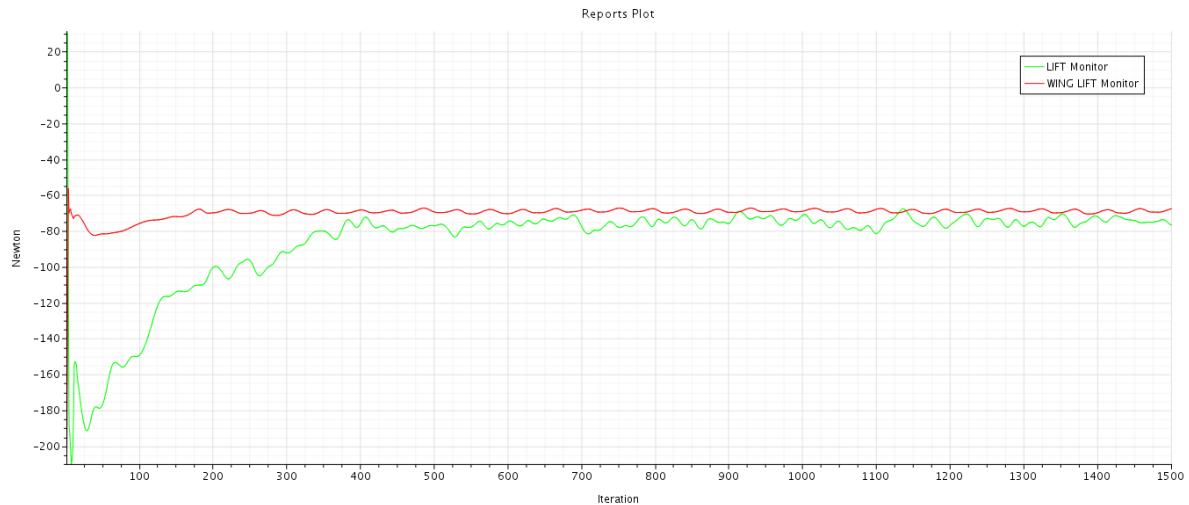
**Fig. 9.C.1. Body drag and wing drag for the baseline simulation.**



**Fig. 9.C.2. Body lift and wing lift for the baseline simulation.**

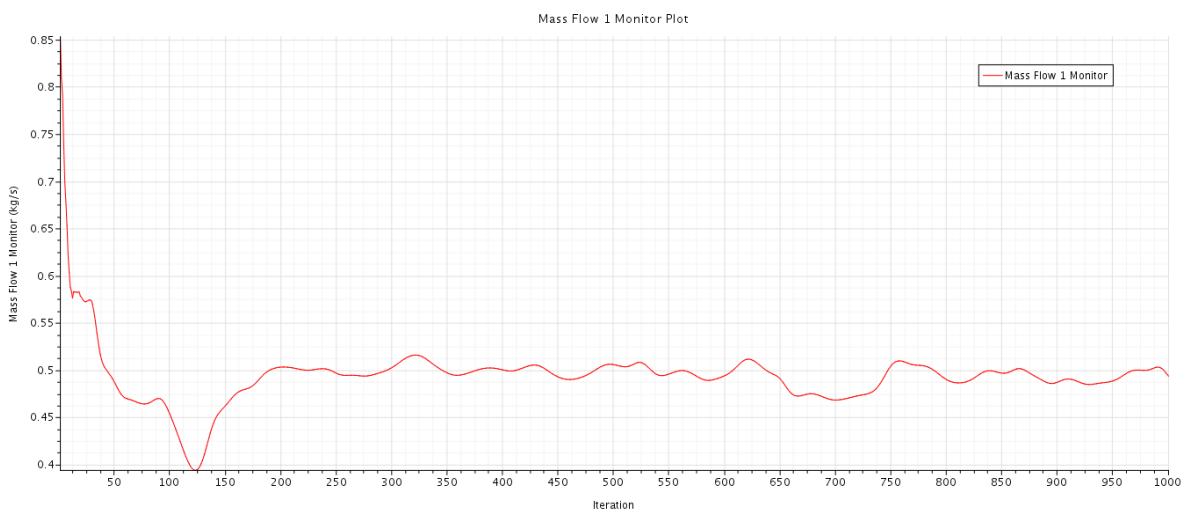


**Fig. 9.C.3. Body drag and wing drag for the improved simulations.**

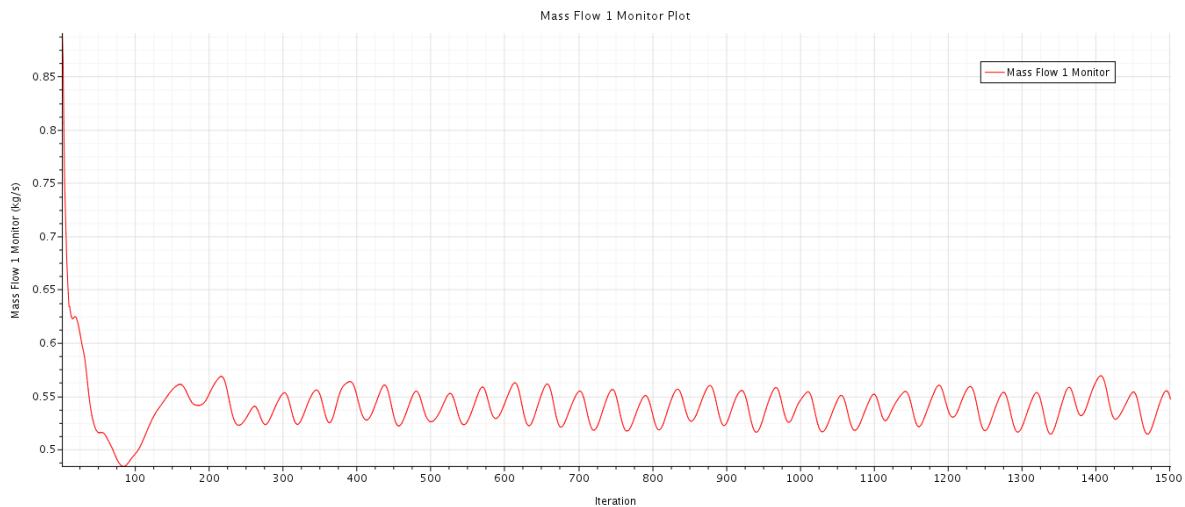


**Fig. 9.C.4. Body lift and wing lift for the improved simulation.**

## D. Mass flow over sidepod duct

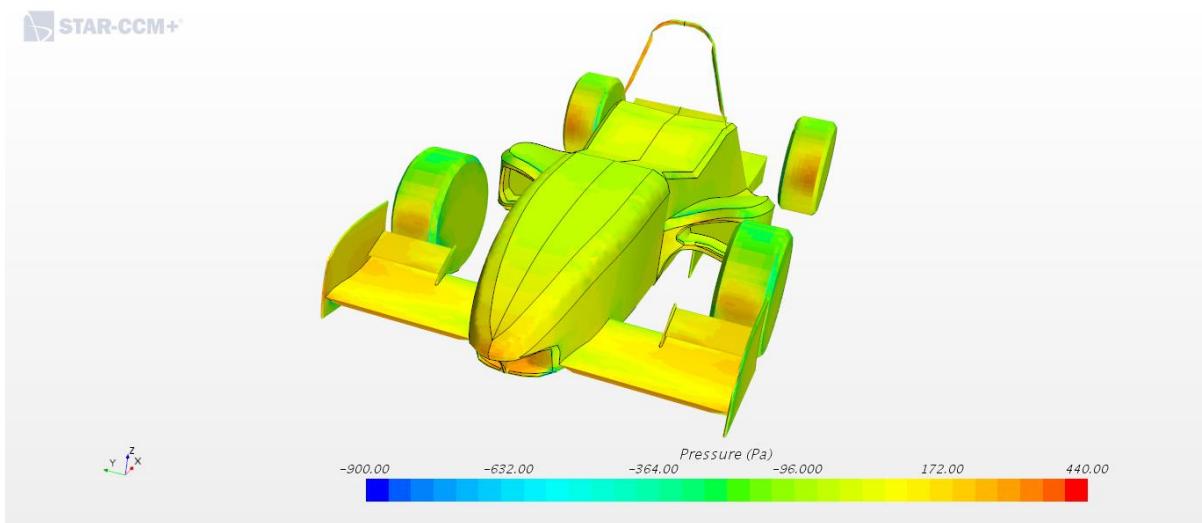


**Fig. 9.D.1. Mass flow in the sidepod duct of the baseline simulation.**

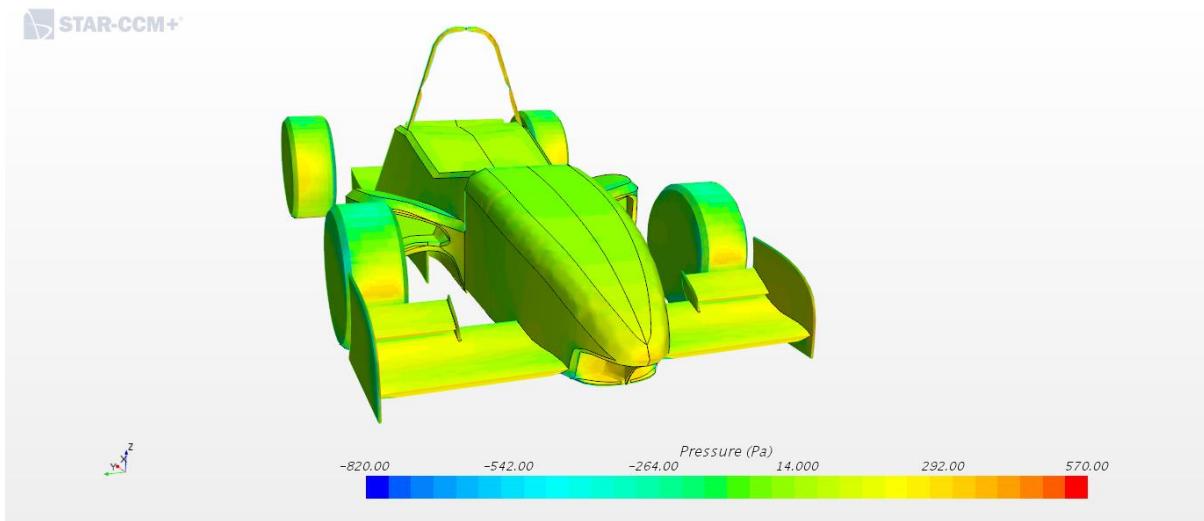


**Fig. 9.D.2. Mass flow in the sidepod duct of the improved simulation.**

## E. Surface pressure

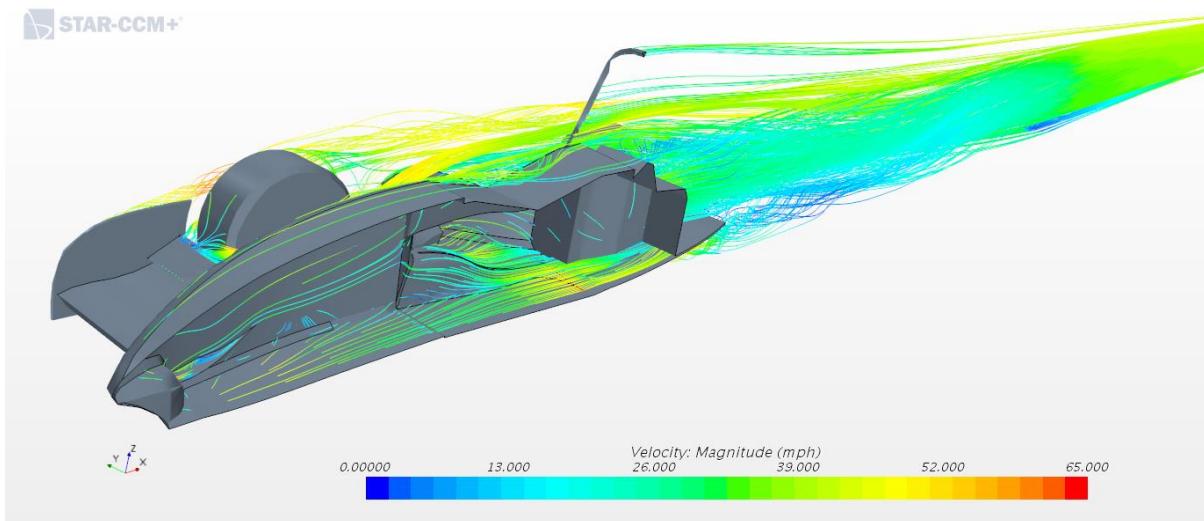


**Fig. 9.E.1. Surface pressure over car body in the baseline simulation.**



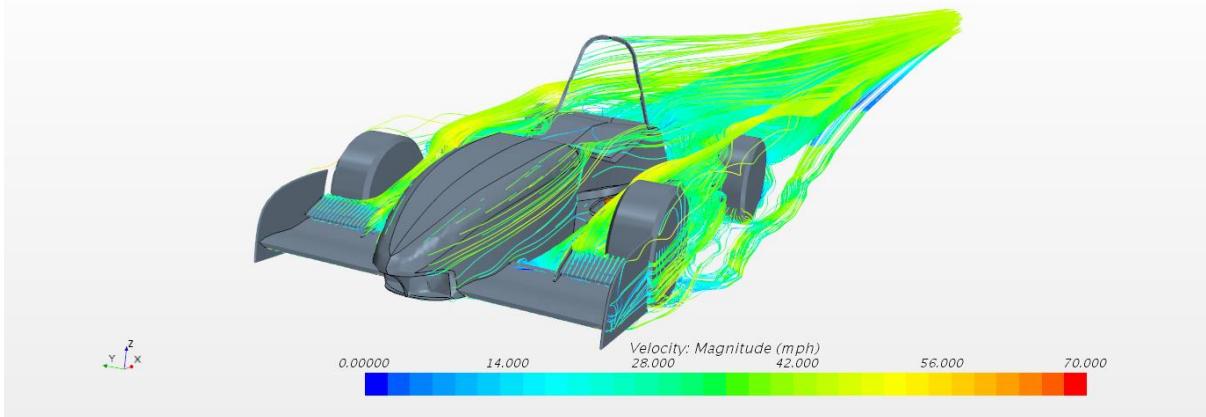
**Fig. 9.E.2. Surface pressure over car body in the improved simulation.**

## F. 3D streamlines



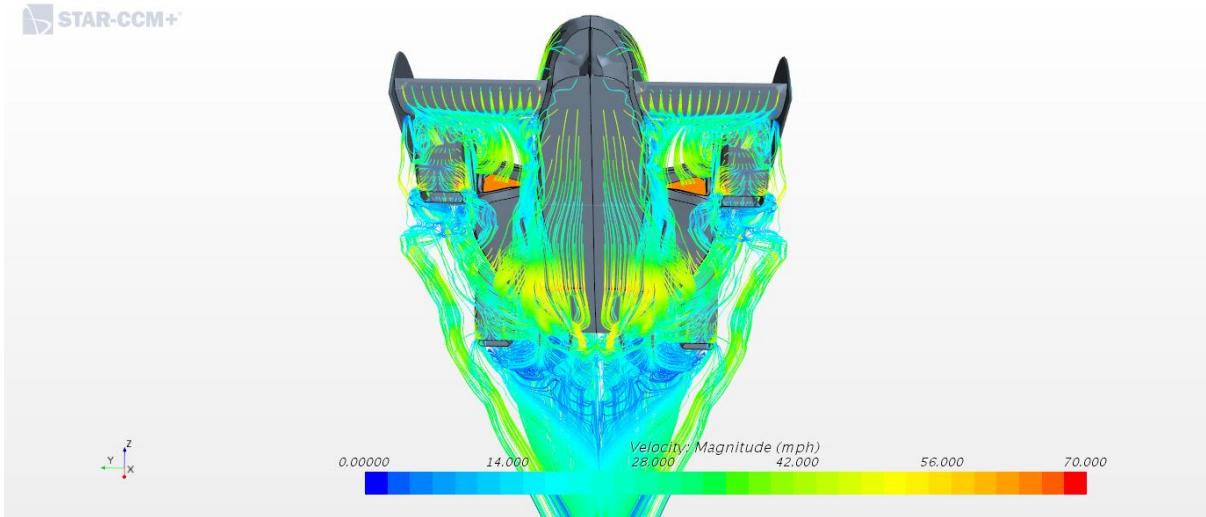
**Fig. 9.F.1. 3D streamlines in the baseline simulation.**

 STAR-CCM+



(a)

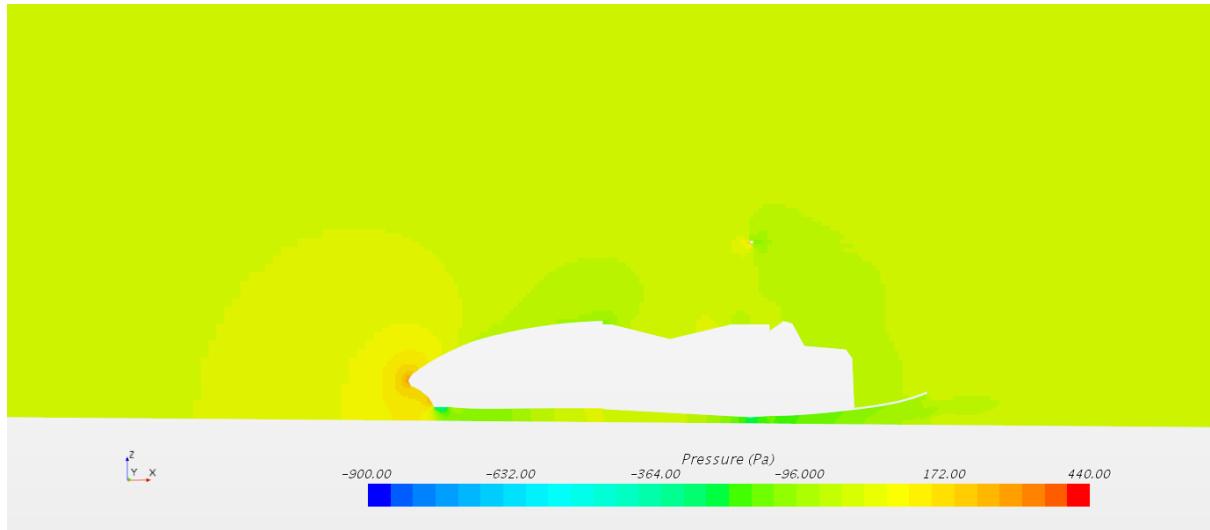
 STAR-CCM+



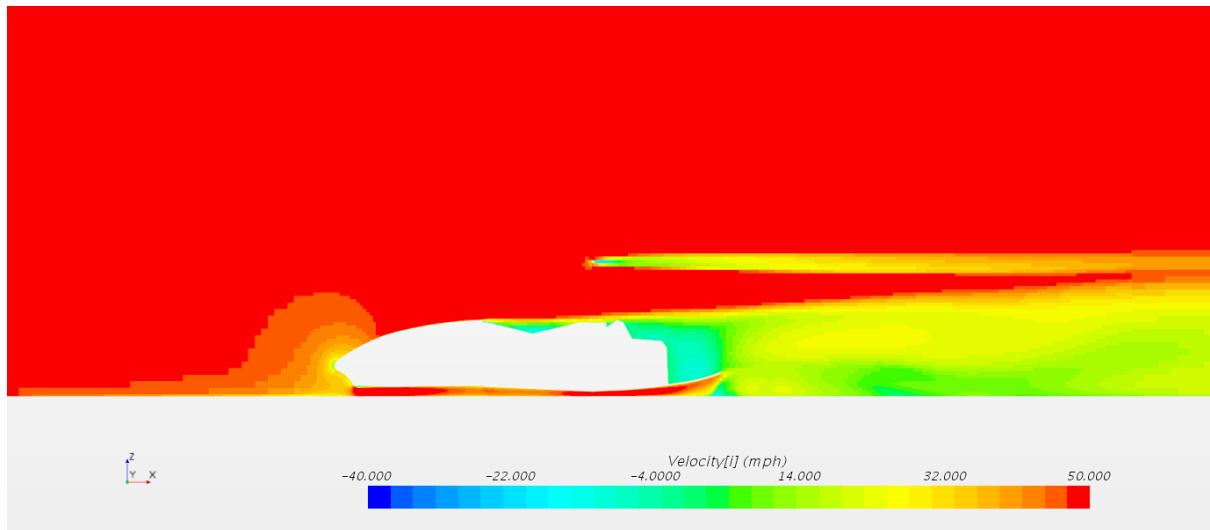
(b)

**Fig. 9.F.2. (a) streamlines in the improved simulation, (b)stream lines under the car body in improved simulation.**

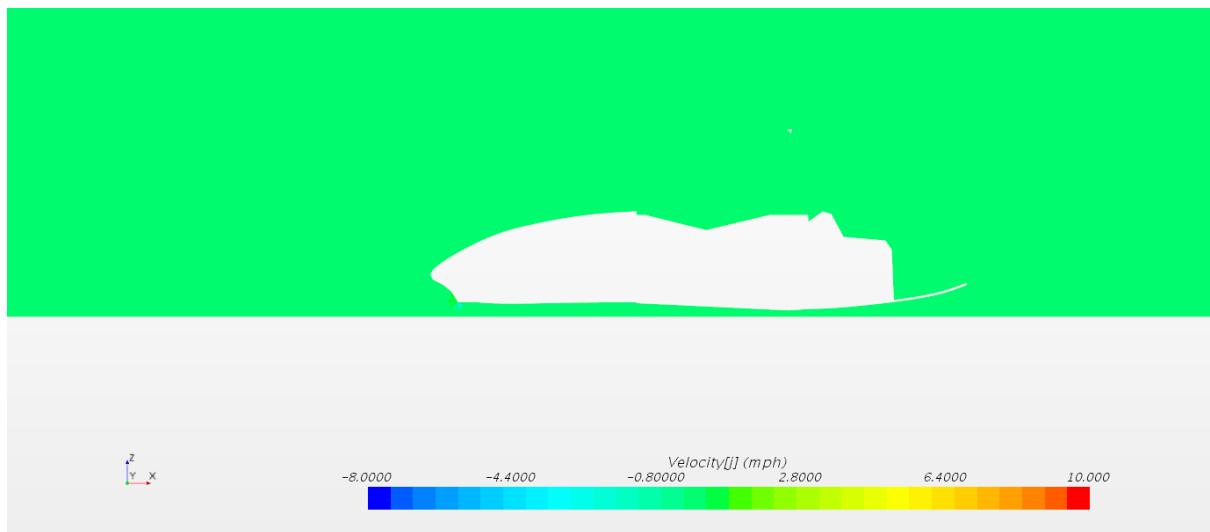
## G. Centre plane images



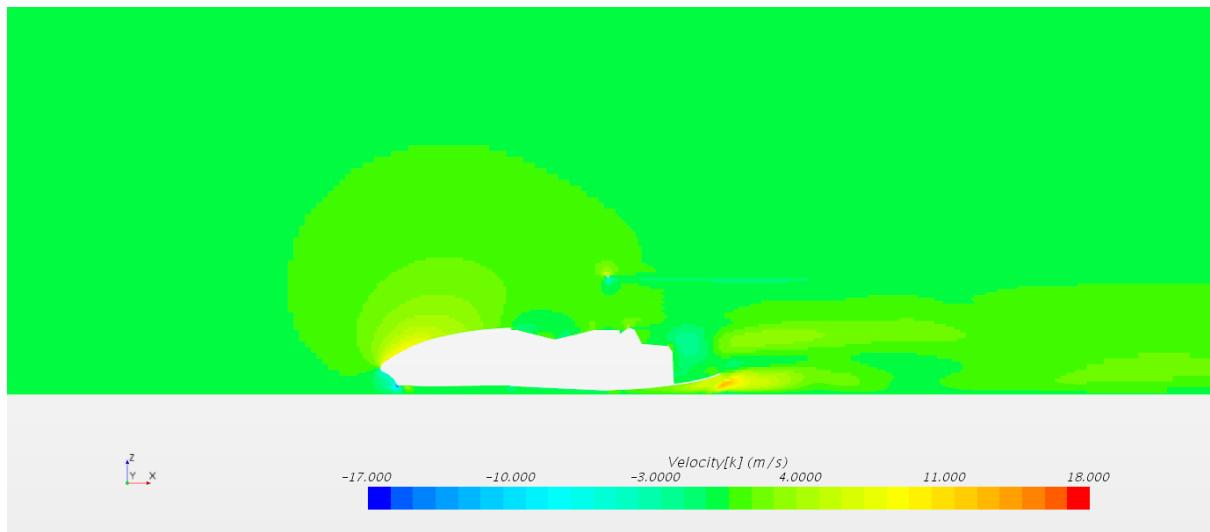
**Fig. 9.G.1. Centre plane pressure distribution in baseline simulation.**



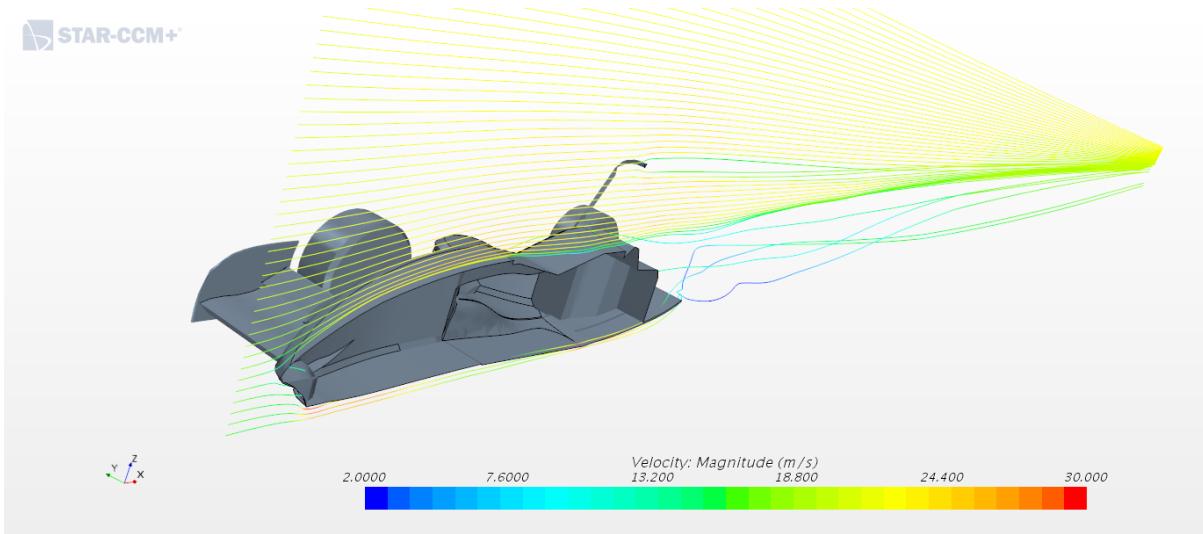
**Fig. 9.G.2. Centre plane velocity in X-direction in baseline simulation.**



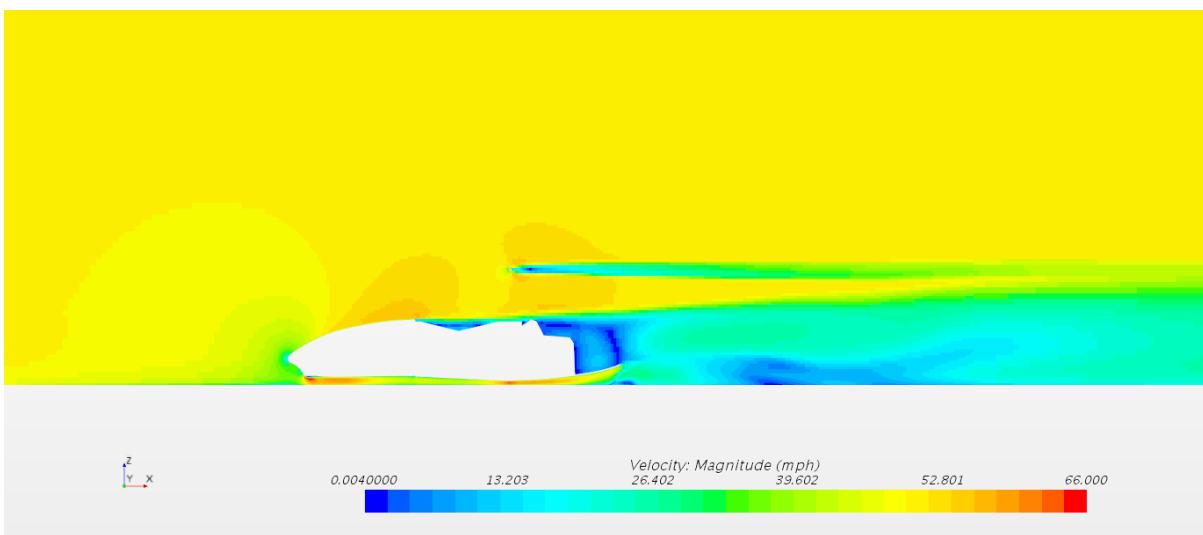
**Fig. 9.G.3. Centre plane velocity in Y-direction in baseline simulation.**



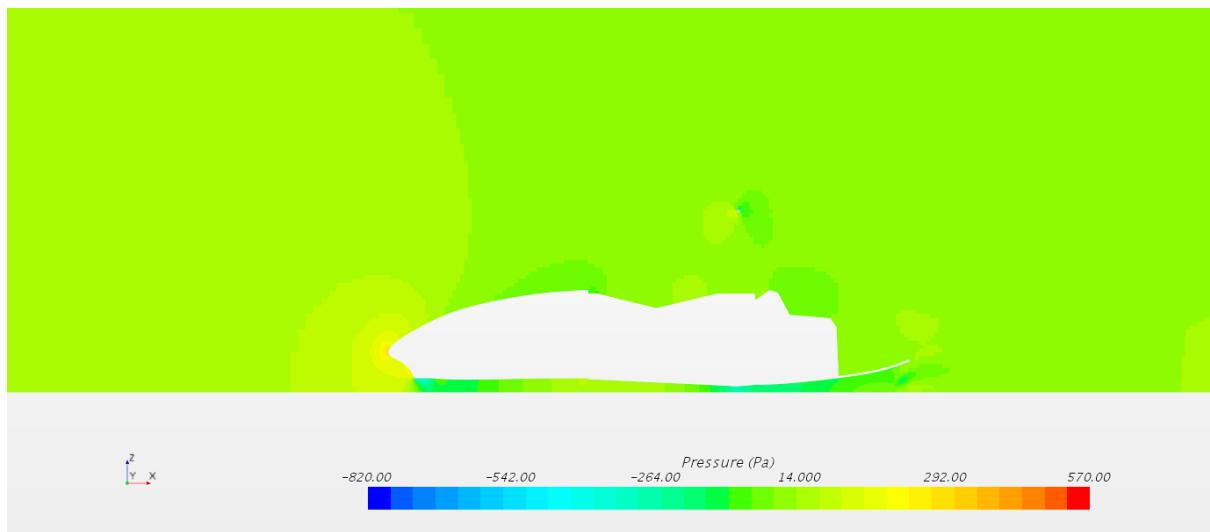
**Fig. 9.G.4. Centre plane velocity in Z-direction in baseline simulation.**



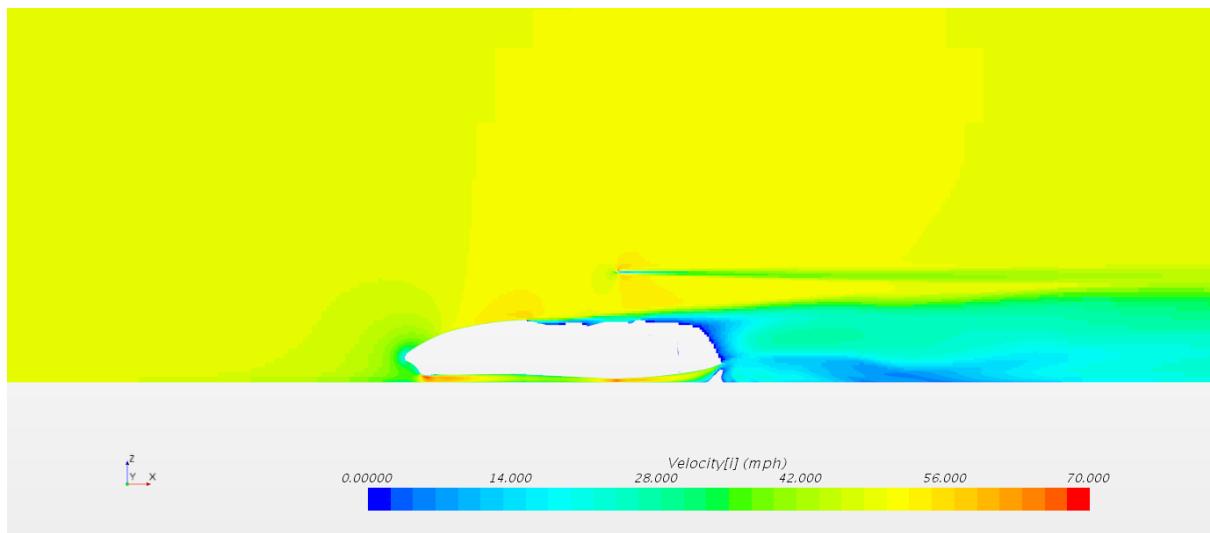
**Fig. 9.G.5. Centre plane streamlines in baseline simulation.**



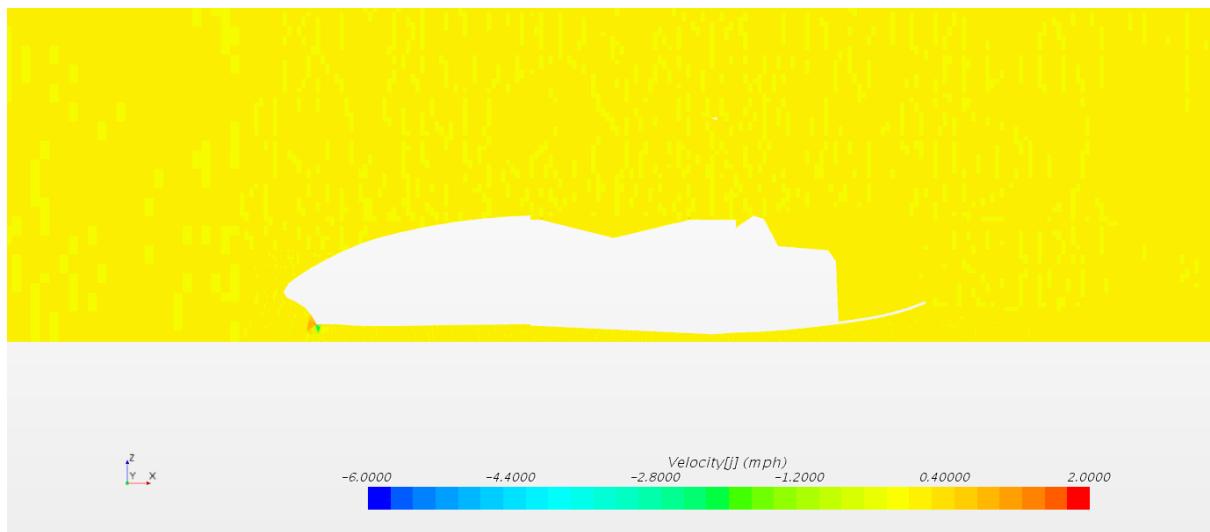
**Fig. 9.G.6. Centre plane velocity magnitude in baseline simulation.**



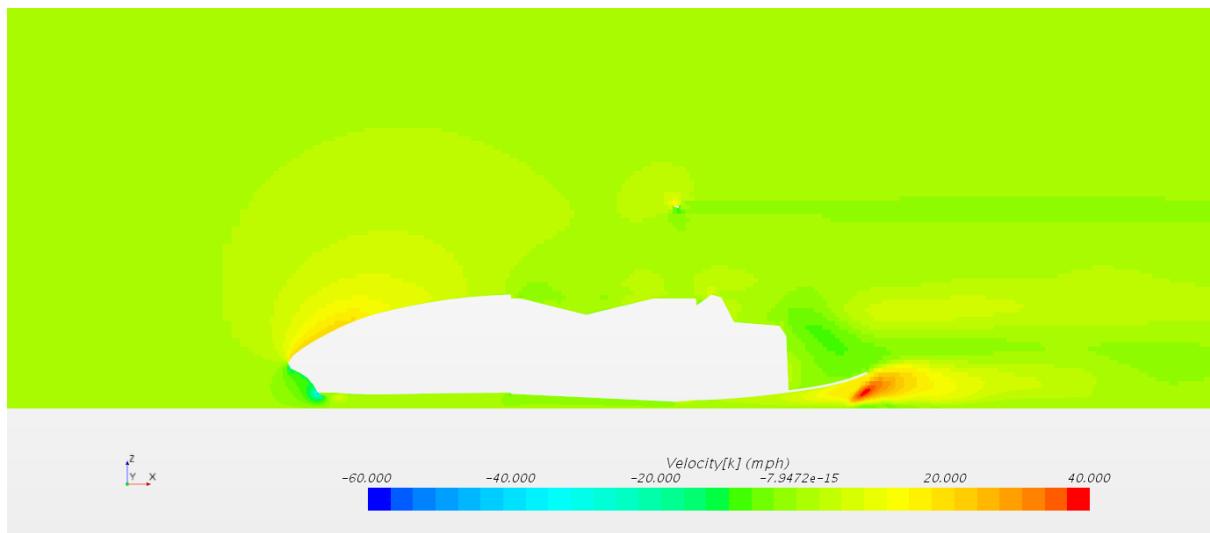
**Fig. 9.G.7. Centre plane pressure distribution in improved simulation.**



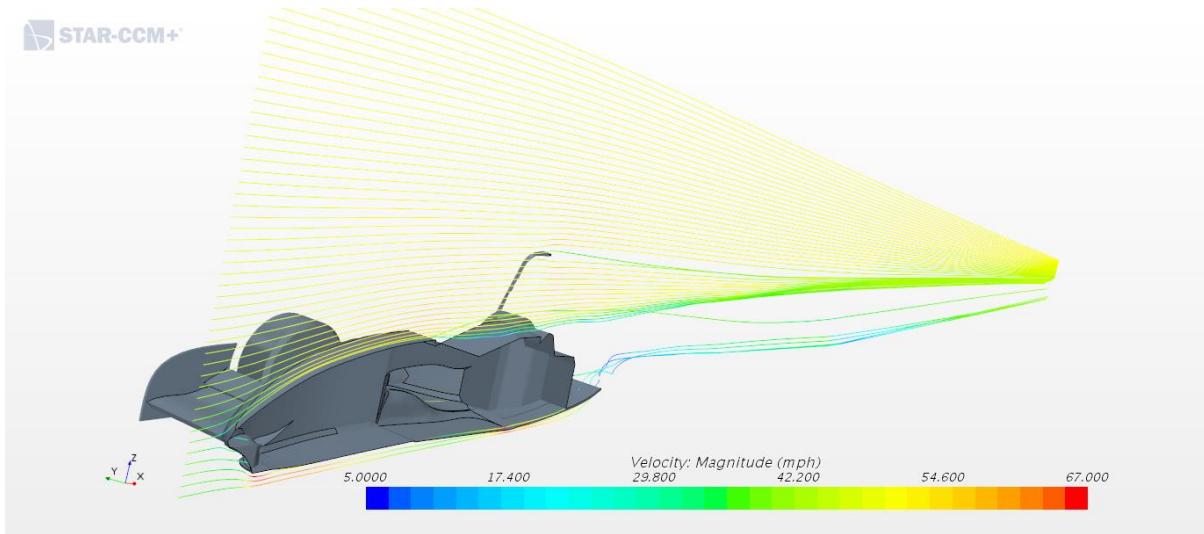
**Fig. 9.G.8. Centre plane velocity in X-direction in improved simulation.**



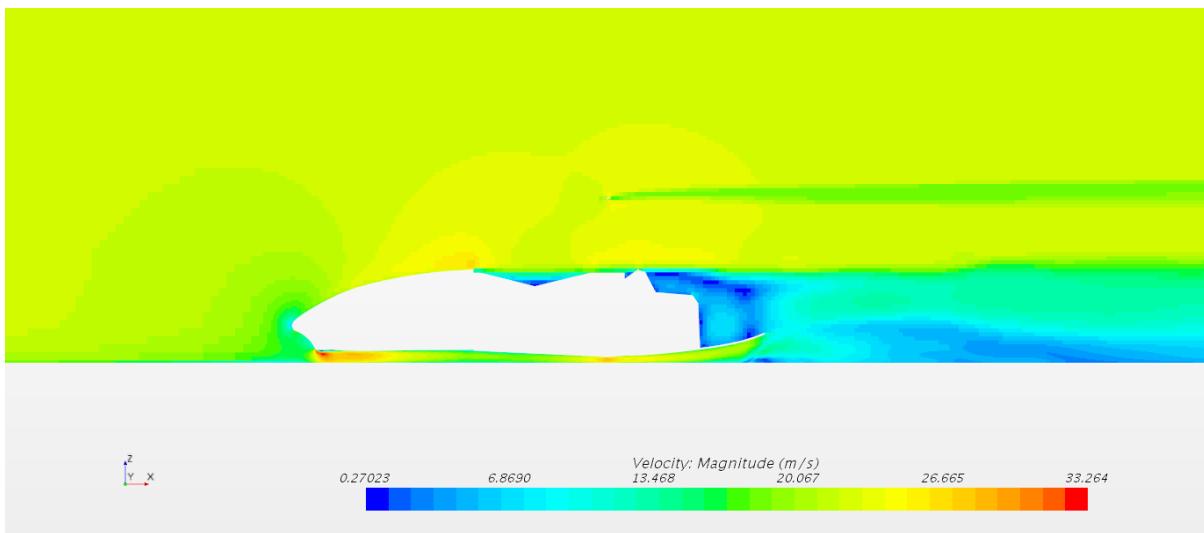
**Fig. 9.G.9. Centre plane velocity in Y-direction in improved simulation.**



**Fig. 9.G.10. Centre plane velocity in Z-direction in improved simulation.**

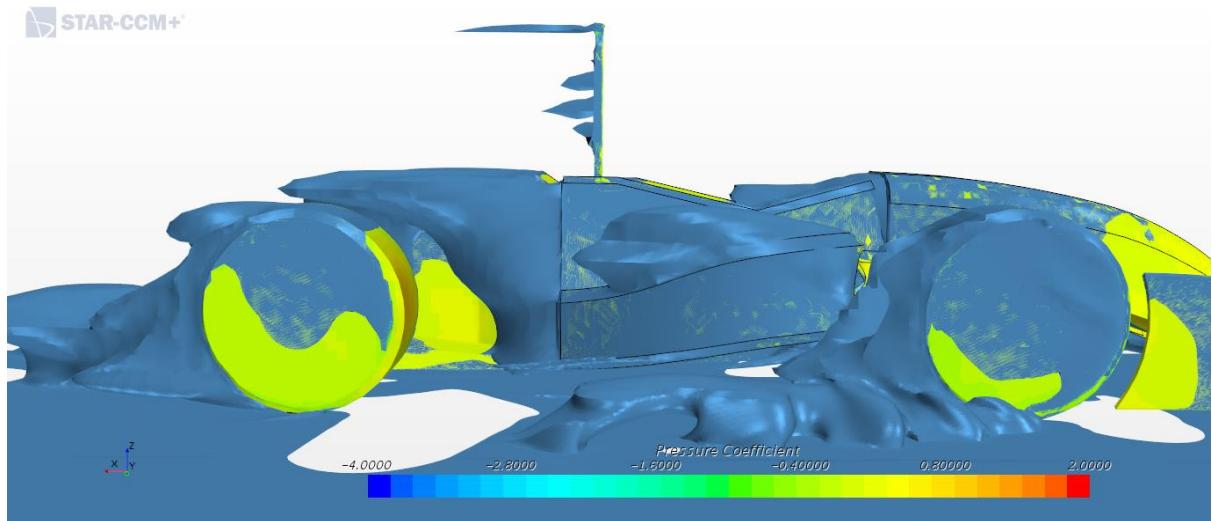


**Fig. 9.G.11. Centre plane streamlines in improved simulation.**

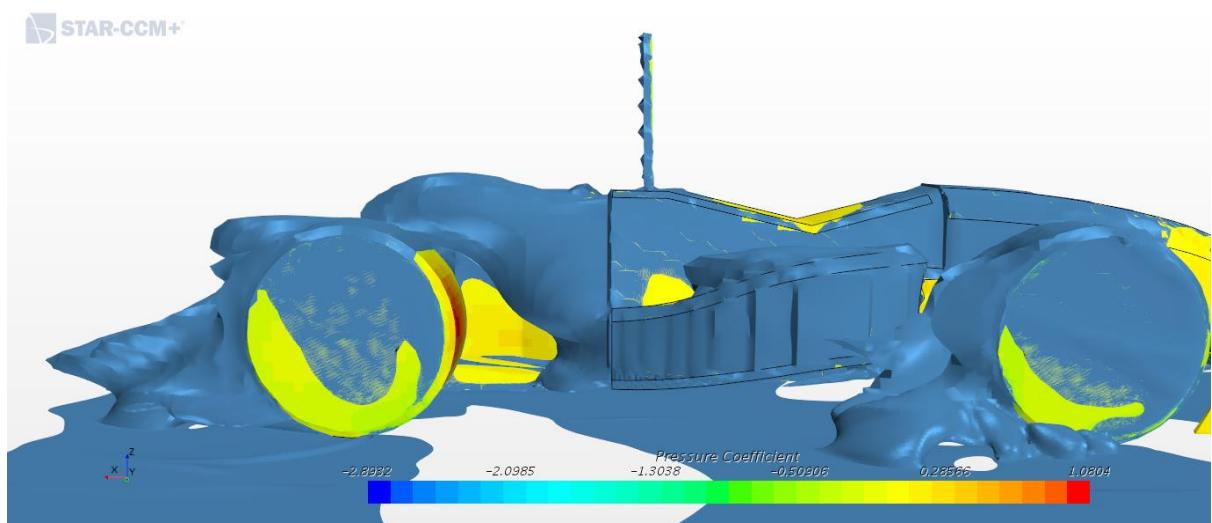


**Fig. 9.G.12. Centre plane velocity magnitude in improved simulation.**

## H. ISO SURFACE

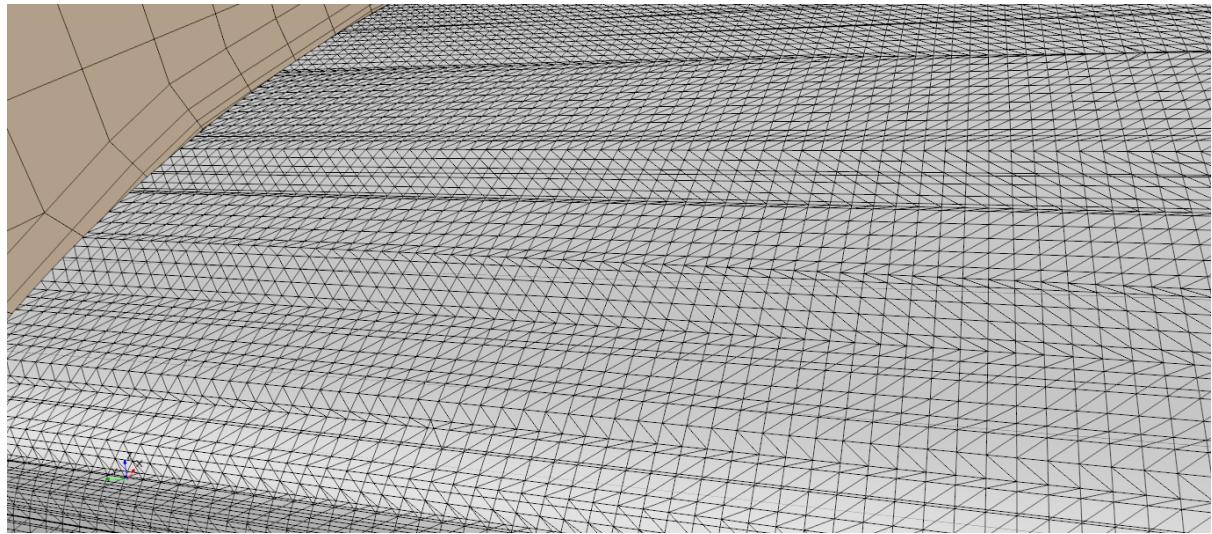


**Fig. 9.H.1. Iso surface in the baseline simulation.**

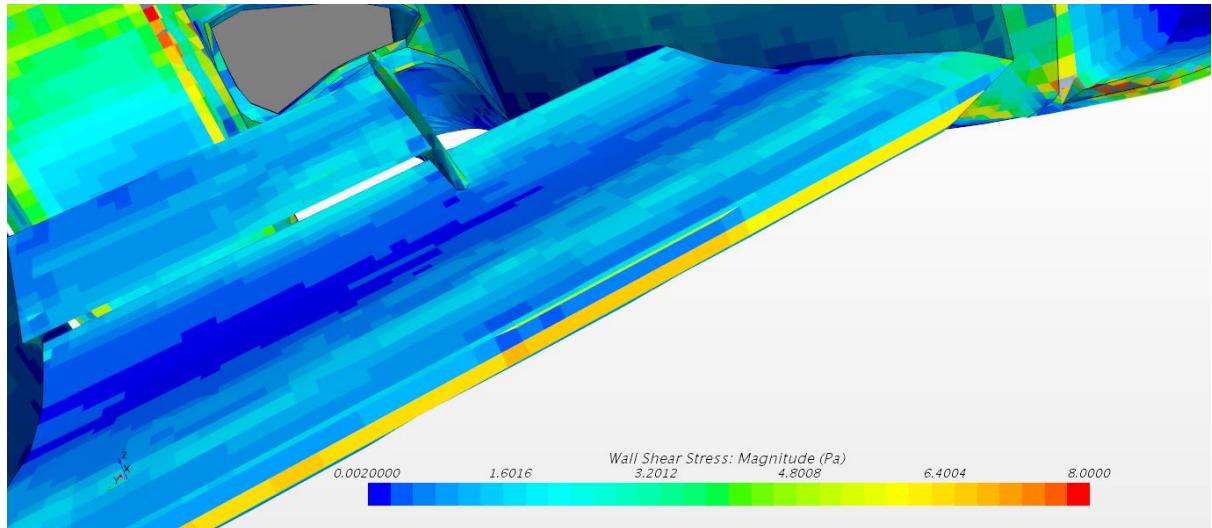


**Fig. 9.H.2. Iso surface in the improved simulation.**

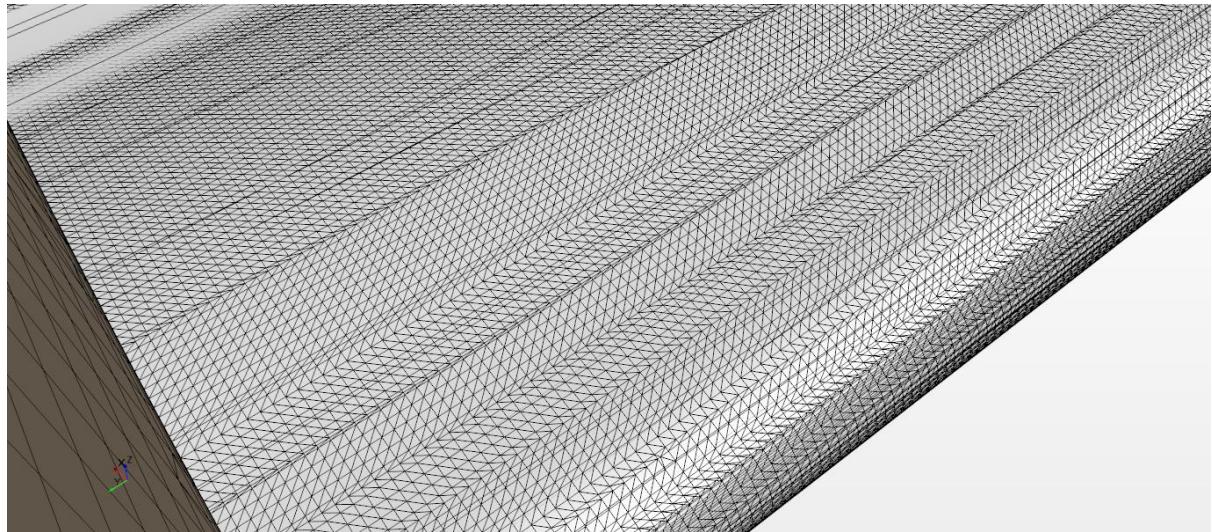
## I. ZOOMED AREA OF WING



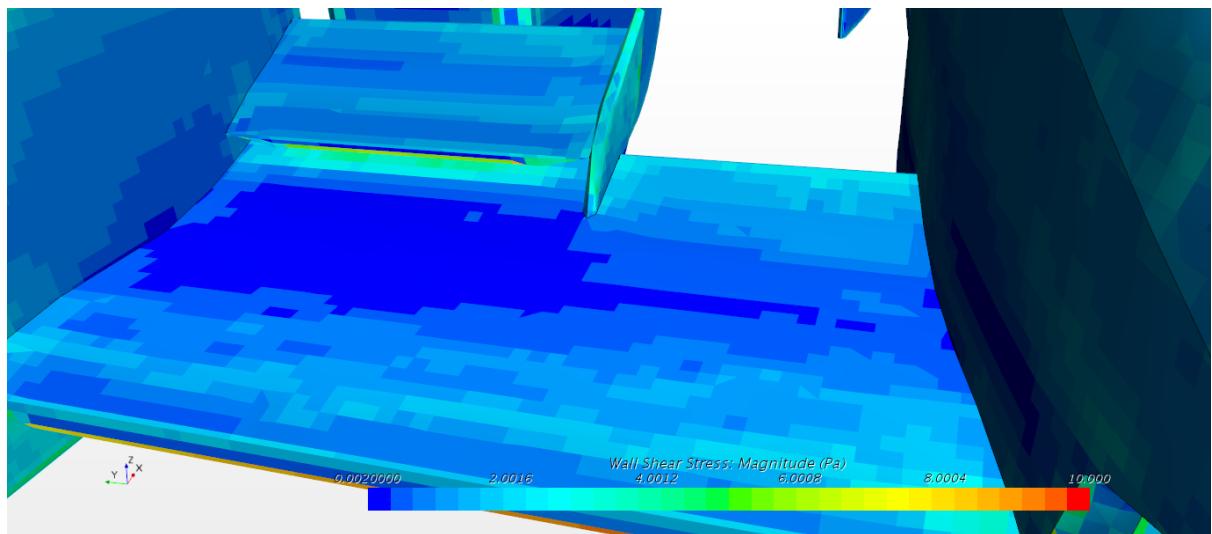
**Fig. 9.I.1. Mesh area of wing for the baseline simulation.**



**Fig. 9.I.2. Wall shear stress area of wing for the baseline simulation.**

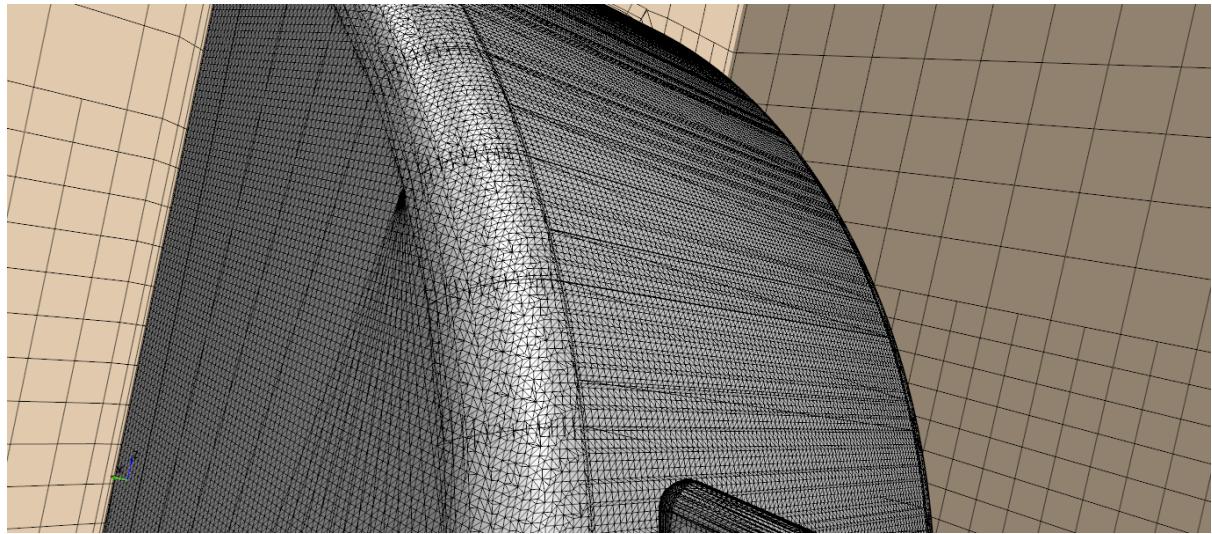


**Fig. 9.I.3. Mesh area of wing for the improved simulation.**

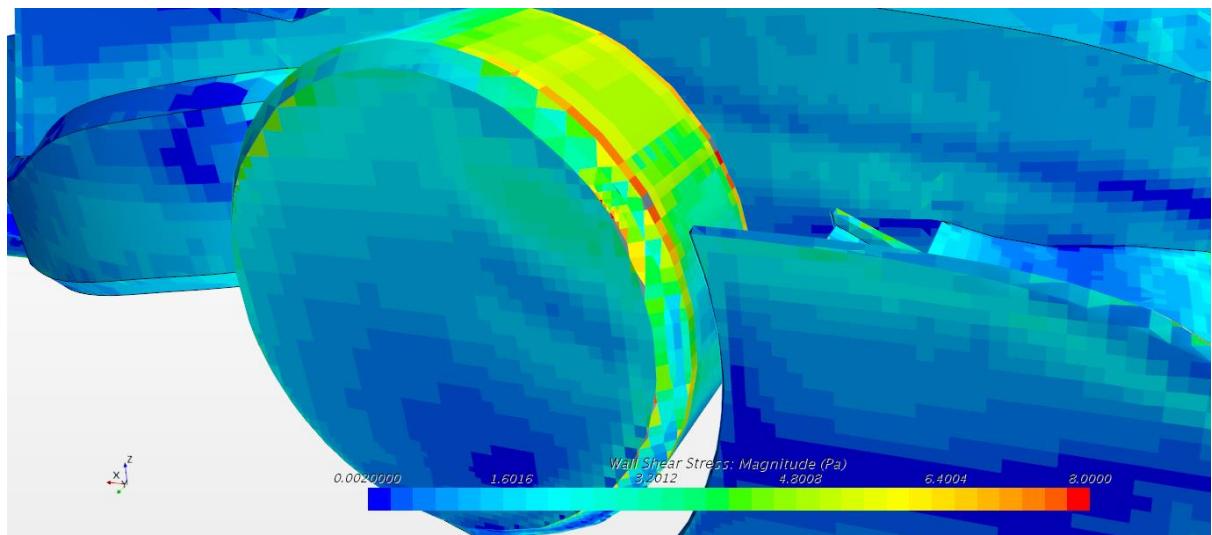


**Fig. 9.I.4. Wall shear stress area of wing for the improved simulation.**

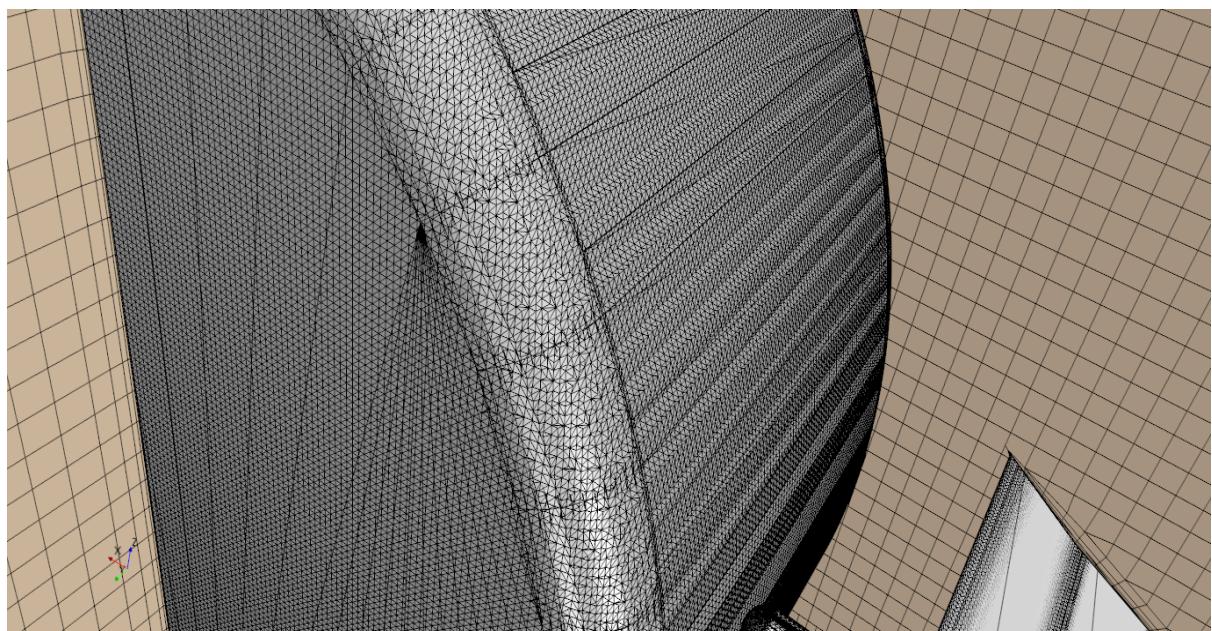
## J. ZOOMED AREA OF TIRES



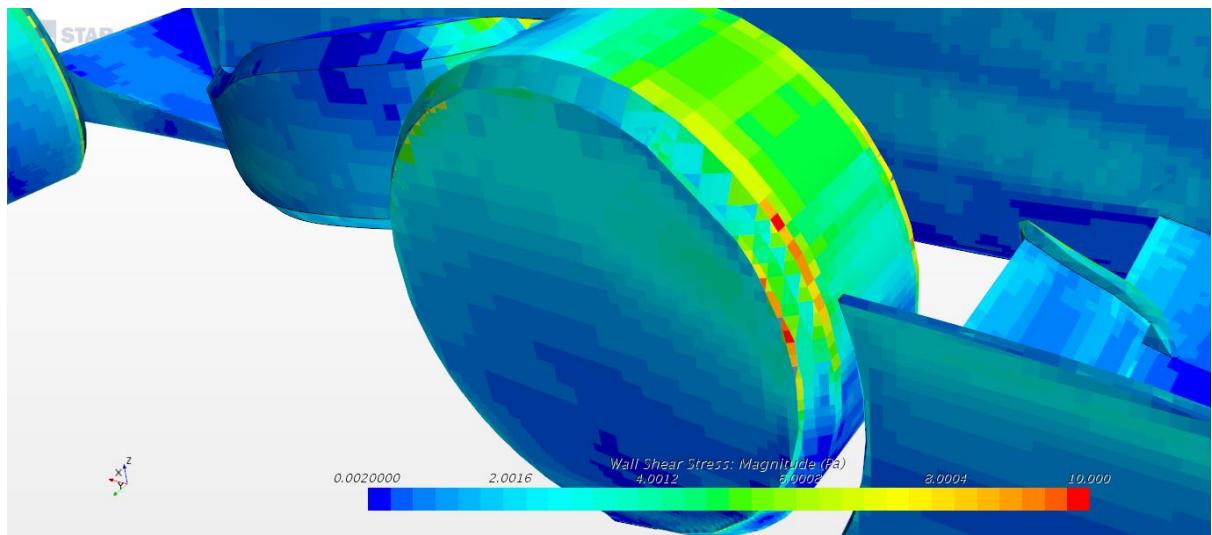
**Fig. 9.J.1.** Mesh area of tire of the baseline simulation.



**Fig. 9.J.2.** Wall shear stress of tire of baseline simulation.



**Fig. 9.J.3.** Mesh area of tire of the improved simulation.



**Fig. 9.J.2.** Wall shear stress of tire of improved simulation.