4/8/2019

CFD ANALYSIS OF A PICKUP TRUCK MODEL WITH TAILGATE UP AND TAILGATE DOWN CASES

Aerodynamics, Project 2

Sonwane, Aditya

1. *Problem Description*

In this particular CFD simulation of the pickup truck model we are to run three simulations a : 1. A baseline simulation (either for tailgate up or for tailgate down case), 2. An Optimized tailgate up case and 3. An Optimized tailgate down case. For this we are to design a pickup truck. Post designing, simulations are to be run. These simulations are to be run for same atmospheric conditions and at 70mph. After running the baseline simulation some modifications are to be made to obtain better results. Doing the modifications the simulation is run again. Physical parameter are plotted to analyze the results.
By running these simulations we are to determine for which of the two cases either the tailgate up case or the tailgate down case the truck gets better gas mileage.

2. *CAD Model geometry*

This truck is designed in solidworks. This model is 6.02 m long from front bumper to rear bumper, it is 2.45 m wide without the side view mirrors and it is 2 m in height from the ground. The wheels of this model truck are 1.2 m in diameter and the 10 inches wide i.e 0.254 m wide. The carrier region of the truck is 2.38 m in length, 2.38 m wide and 0.75 m deep. The ground clearance of the truck is 0.325m.

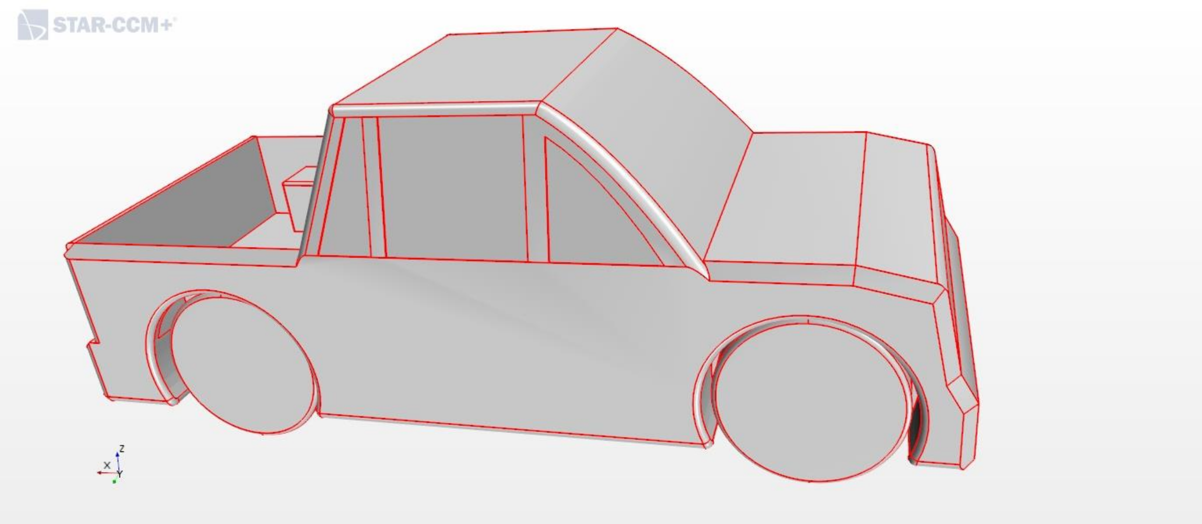

Fig. 2.1. side view tail gate up CAD geometry.

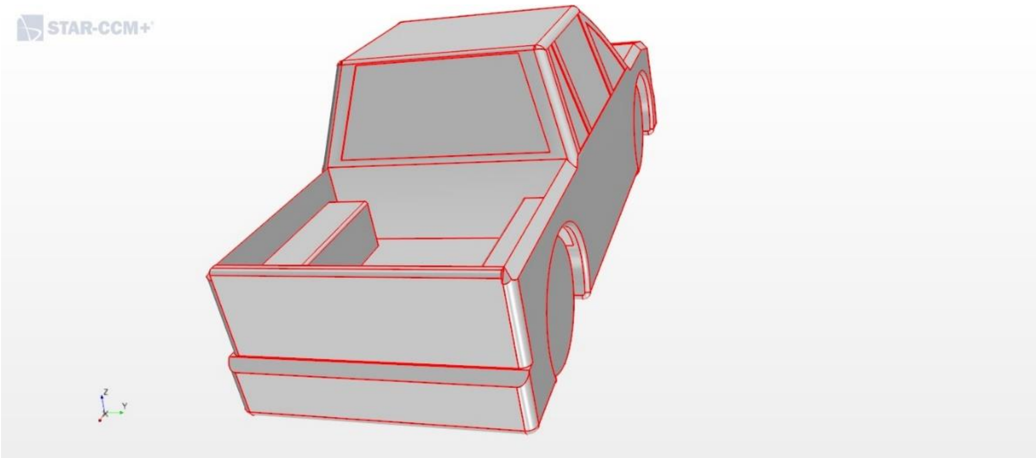

Fig. 2.2. Rear view tailgate up geometry.

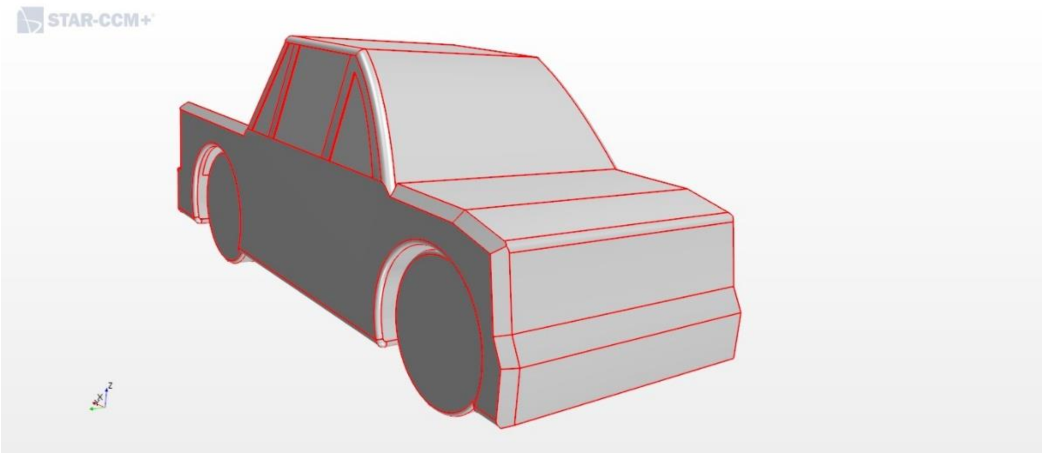

Fig. 2.3. isometric view for tail gate down CAD geometry.

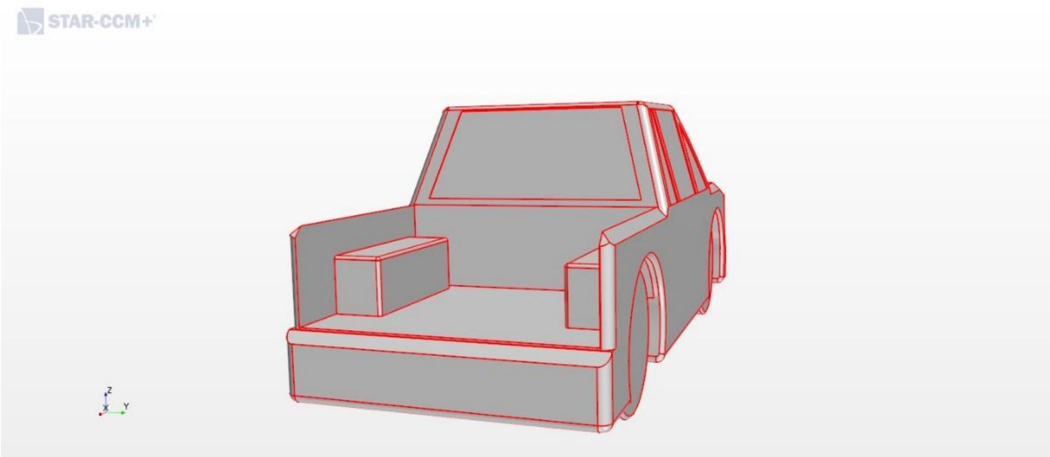

Fig. 2.4 Rear view of tailgate down CAD geometry.

3. *Baseline simulation*

The truck body is divided as

- Front tire, rear tire
- Tailgate
- Curves on the truck model
- The carrier region of the truck
- The windows and windshield
- Main body

The assembly was split using the split by non-continuous operation, in this we named the front and rear wheel. Then the truck body was split using split by patch operation, in this operation the body was divided as carrier region, curves, tailgate, etc. After determining the simvol it was split by angle to give it the required initial conditions.

During the wrapping operations the base size was set to 8mm, the target surface size was set to 100% and the minimum surface size was set to 5%.

The custom curve control was added for the computing the curve geometries on the truck and also the curves added on the tires. Target surface size was 25% and minimum size was 5%.

Contact prevention was added in between the tires or wheels and the body of the truck so that the mesh of these parts does not intersect since these parts are close.

Later on simvol block was created for the fluid. The dimensions of this simvol are:

	CORNER 1	**CORNER 2**
X	-15 m	42 m
Y	1.225 m	7.5 m
Z	1.6 m	10 m

It was suggested to create some more blocks around the truck model for volumetric mesh like, near truck, a 2nd level, under tray and the carrier region.

- The baseline size is specified to be 24 mm

- Number of prism layers added are 6
- Prism layer total thickness is 33%
- The prism layer near wall thickness was calculated as 0.11mm

The calculation for near wall thickness was calculated for 70mph, at 1% turbulent intensity, 1.2kg/m3 air density and 1.846e-5kg/m.s dynamic viscosity.

After specify the global parameters, boundary specific mesh parameters are selected in custom controls of the automated meshes.

For the tunnel wall the target surface size was set 2000 and minimum surface size was set 200.

Volumetric controls were created for the blocks we created earlier and the customized isotropic sizer were set as

Near Truck	100
Level 2	200
Carrier region	50
Under tray	50

Executing the automated mesh we got the volumetric mesh over the truck body.

Post execution the case had ***6019512 cells, 18049161 faces and 6466373 vertices***.

<u>The physics set up for baseline simulation is given as</u>:

- Stationary
- Gas
- Segregated flow
- Constant density
- Steady
- Turbulent
- K-omega turbulence model
- Cell quality remediation

Completing the physics setup we are to define the initial conditions for the boundary box.
In here the road is treated as wall, the far field and symmetry plane are treated as the symmetry plane, inlet as velocity inlet and outlet as pressure outlet.
The inlet velocity value was specified to be 70 mph and the outlet pressure value as 0 pa.

Here the wheels have motion so they are to be given the local rotation rate. This is same for the front and the rear wheels. The rotation rate was calculated as:
70miles per hour = 70 * 1609 meters per hour = 112630 meters per hour = 1877 meters per minute.
The circumference of the tire = Π*d = 3.142*1.2 = 3.768 m
Rotation rate = 1877/3.768 = 498.142 ~ 499 rpm
Since the rotation is in X-Z plane the axis of rotation is (0,1,0)
The origin for the: a. front wheel [2.645, 1.225, 2.2]
b. rear wheel [6.645, 1.225, 2.2]

4. *Mesh and mesh scenes*

Post executing the baseline simulation the mesh was optimized and made more refined and fine the changes made to make the mesh more fine are as follows:

- The target surface size and the minimum surface size from the surface wrapper was changed from 25 mm to 21 mm and 5% to 3% respectively.
- In the curve controls as well these sizes were changed to 21 mm and 3%.
- For making the undertray and carrier region fine the percentage of base was changed from 50% to 80% and 90% respectively.
- In the automated mesh the mesh base sizes were also changed to 26 mm making them more clear and fine yet giving more number of cells.

All these changes were made in both the tailgate up and tailgate down cases. Making these changes the refined cell count was:

	Cells	**Faces**	**Vertices**
Tailgate down case	6643185	19889387	6964899
Tailgate up case	7044385	21522190	7168103

Giving the better understanding of the meshes it was advised to add some section planes cutting the truck so that we can see the mesh at different regions of truck. So in the optimized solution 3 section planes were added, two in planes normal to Y axis and a plane normal to X axis cutting the tires and the carrier section of the truck model.

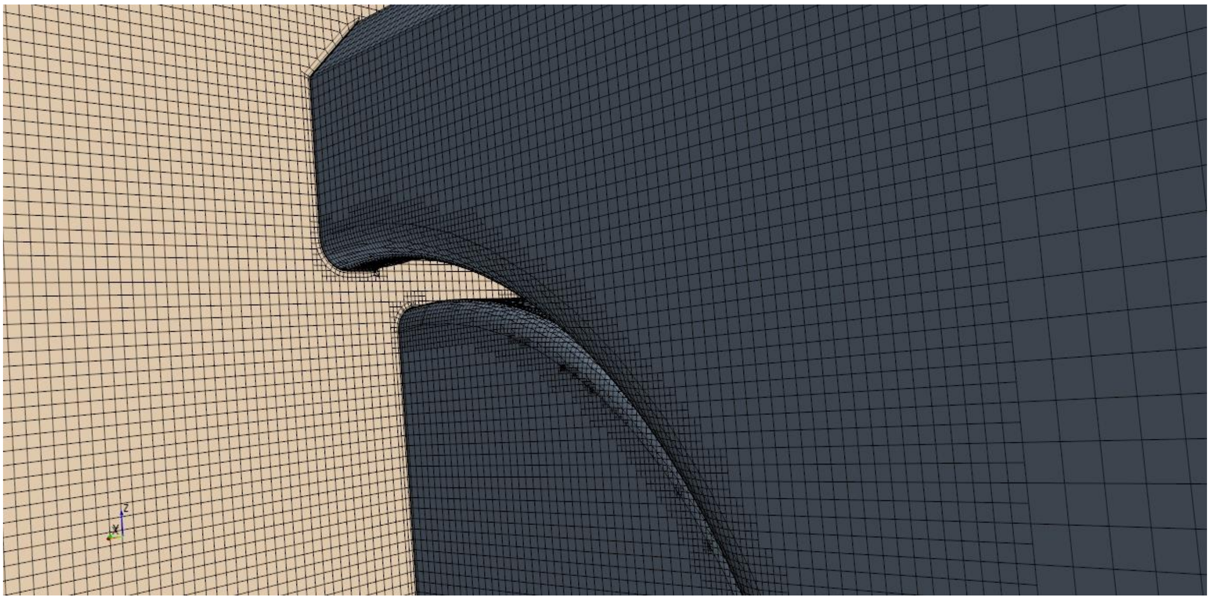

Fig. 4.1. Mesh at the section normal to X axis.

Fig. 4.2. Improved mesh of the truck in tailgate up case.

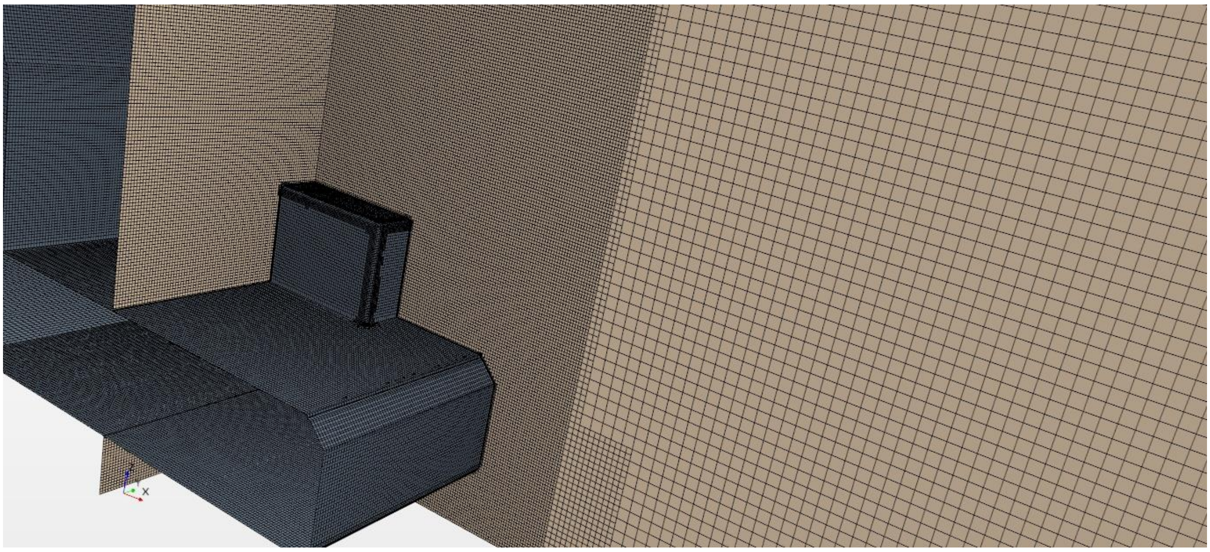

Fig. 4.3. mesh at the tail gate section.

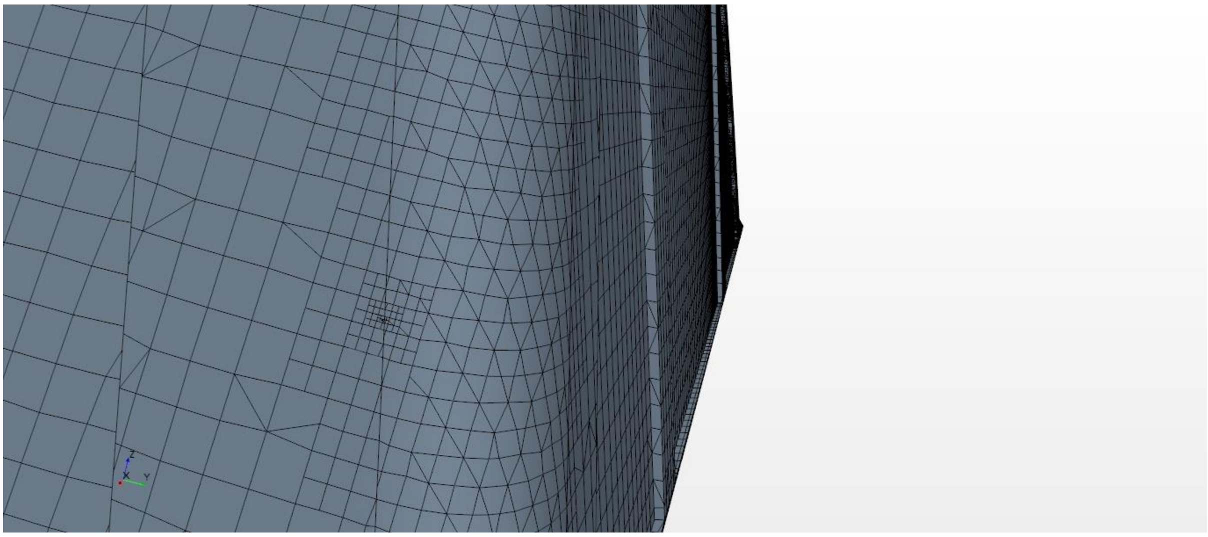

Fig. 4.4. Baseline simulation mesh at curve region.

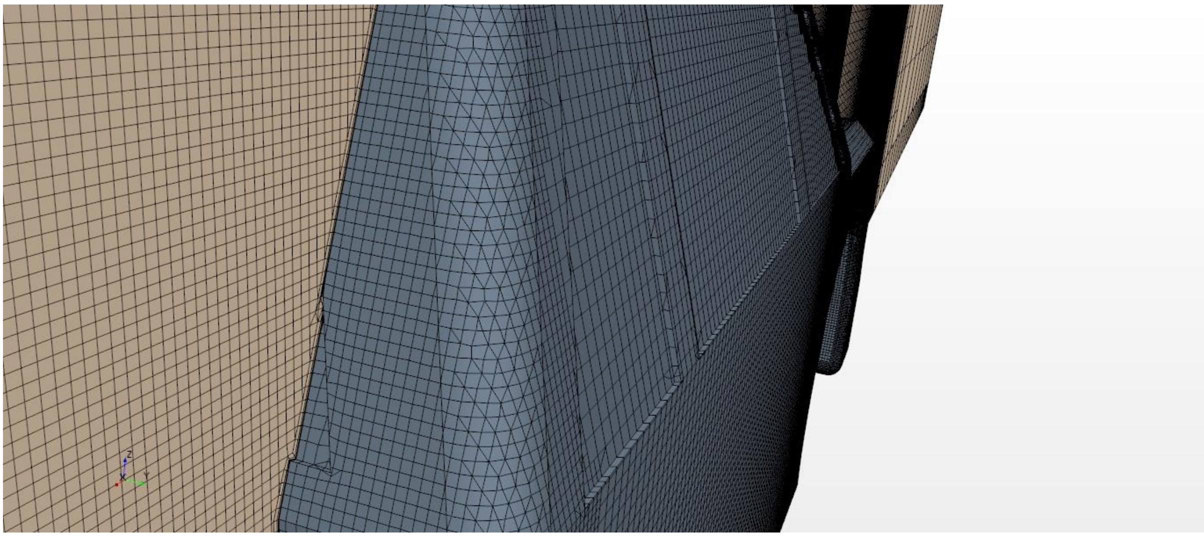

Fig. 4.5. Improved mesh in optimized simulation same region.

5. *Drag and lift value chart*

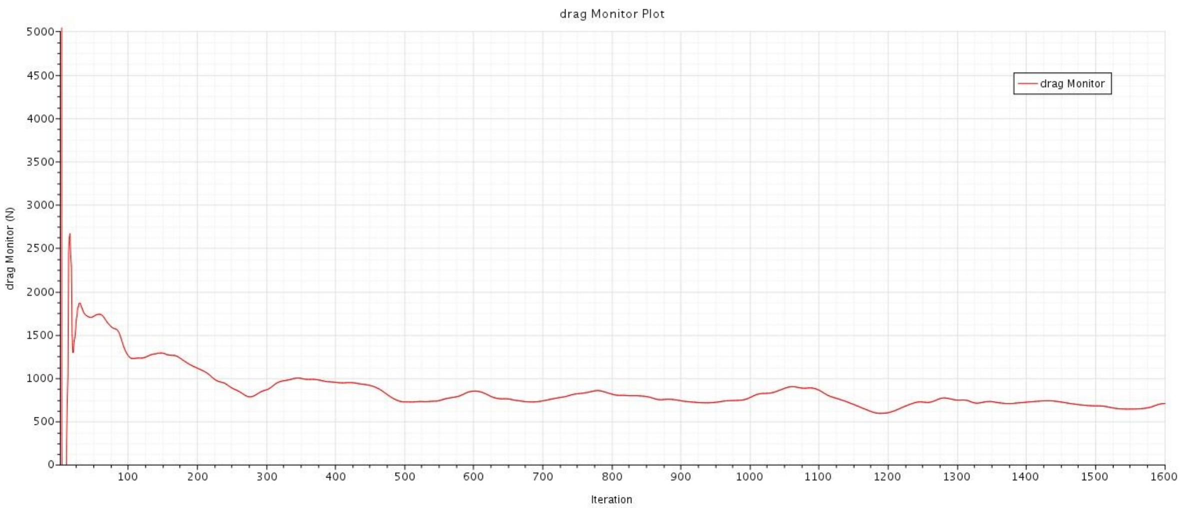

Fig. 5.1. Drag v/s iteration in tailgate up baseline simulation.

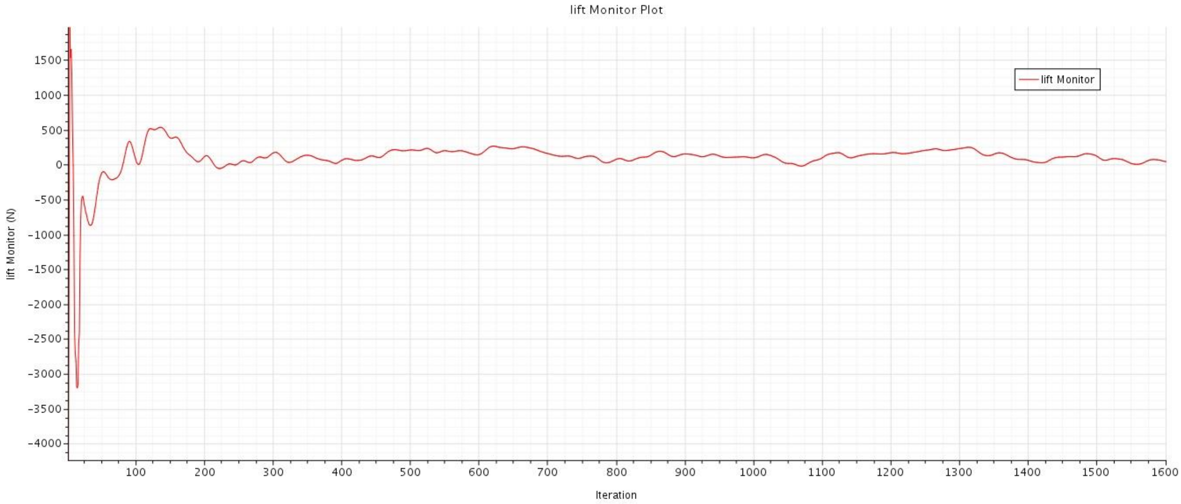

Fig. 5.2. Lift v/s iteration in tailgate up baseline simulation.

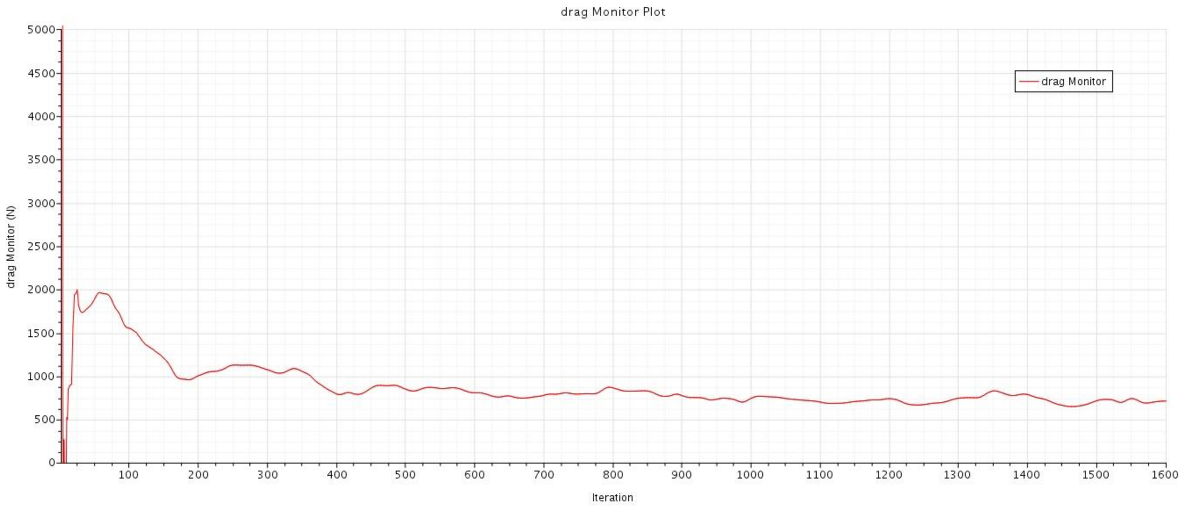

Fig. 5.3. Drag v/s iteration in tailgate up optimized simulation.

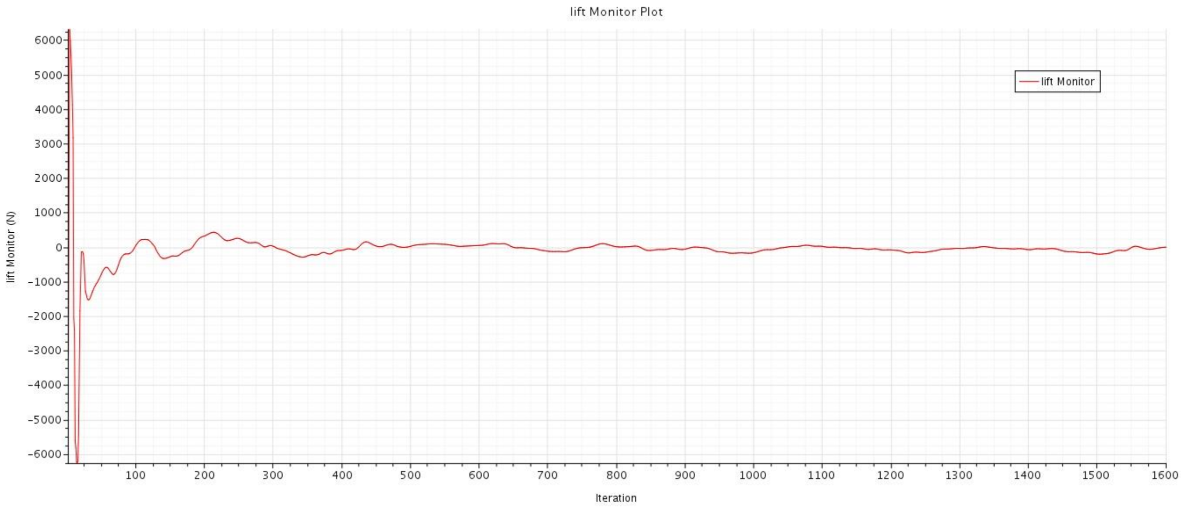

Fig. 5.4. Lift v/s iteration in tailgate up improved simulation.

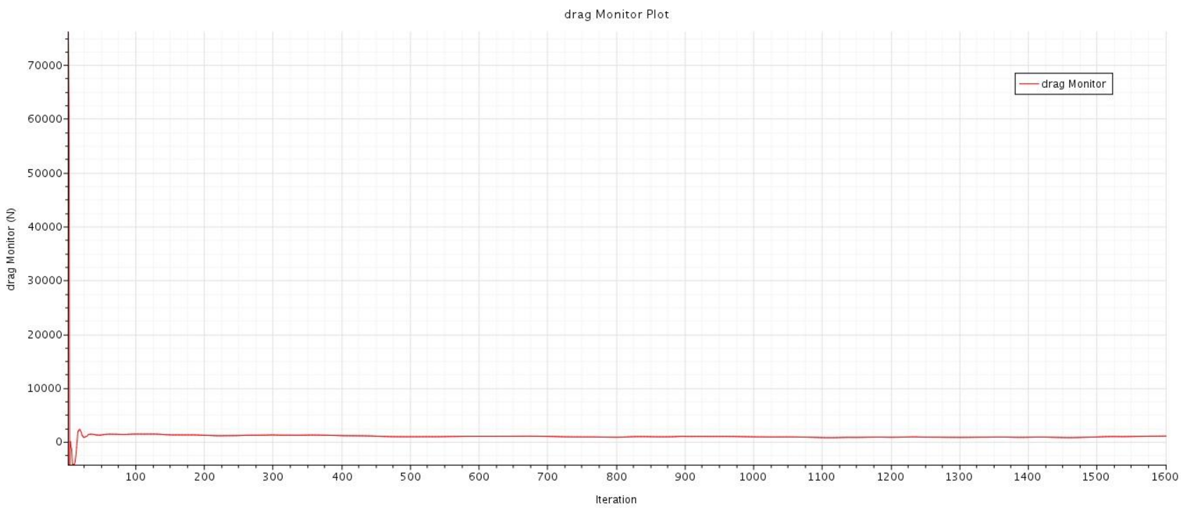

Fig. 5.5. Drag v/s iteration in tailgate down optimized simulation.

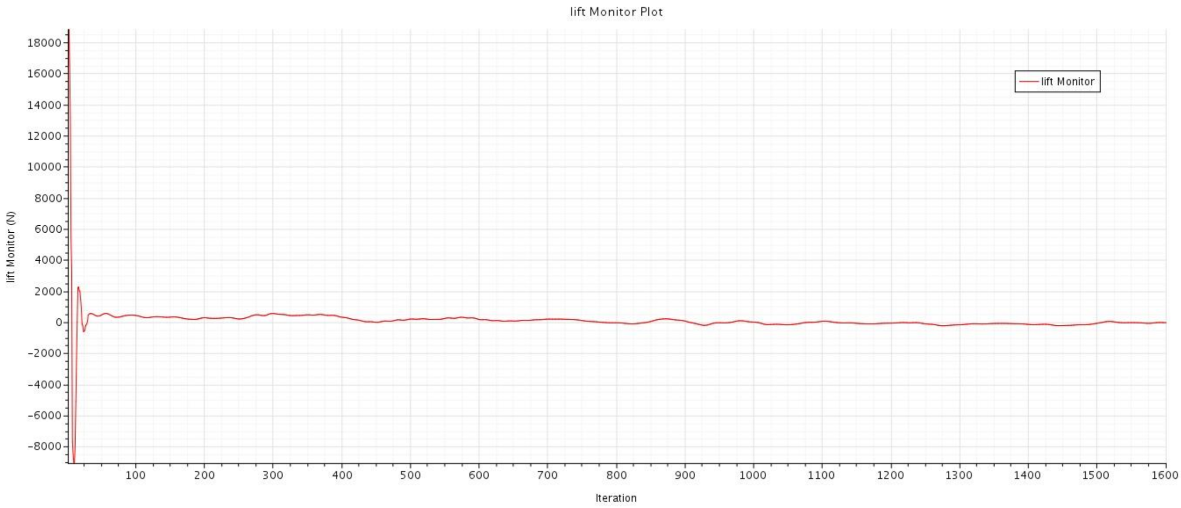

Fig. 5.6. Lift v/s iteration in tailgate down optimized simulation.

6. *Scenes*

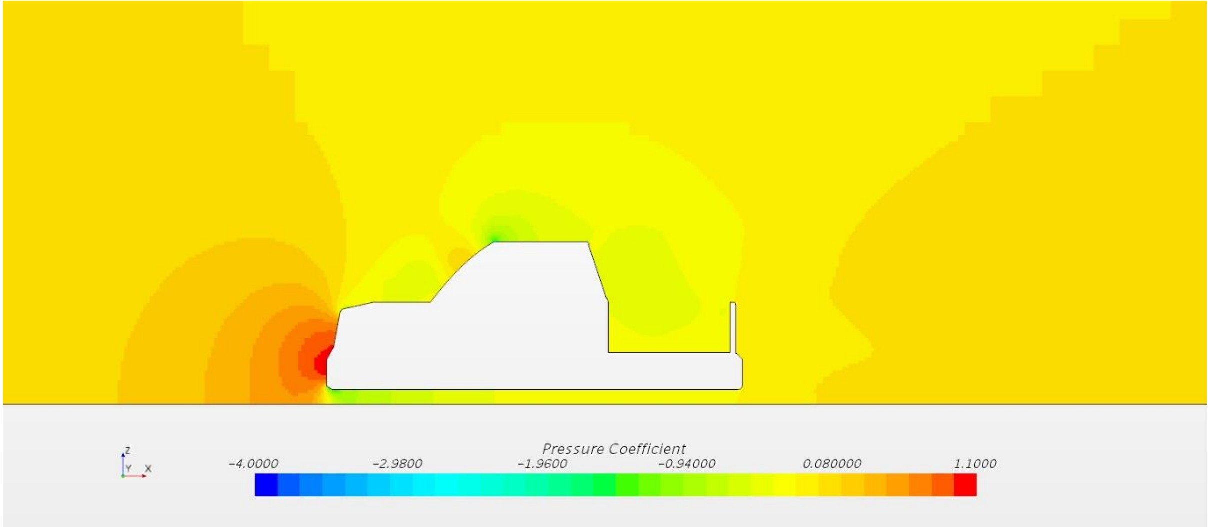

Fig. 6.1. Pressure coefficient of baseline tailgate up case.

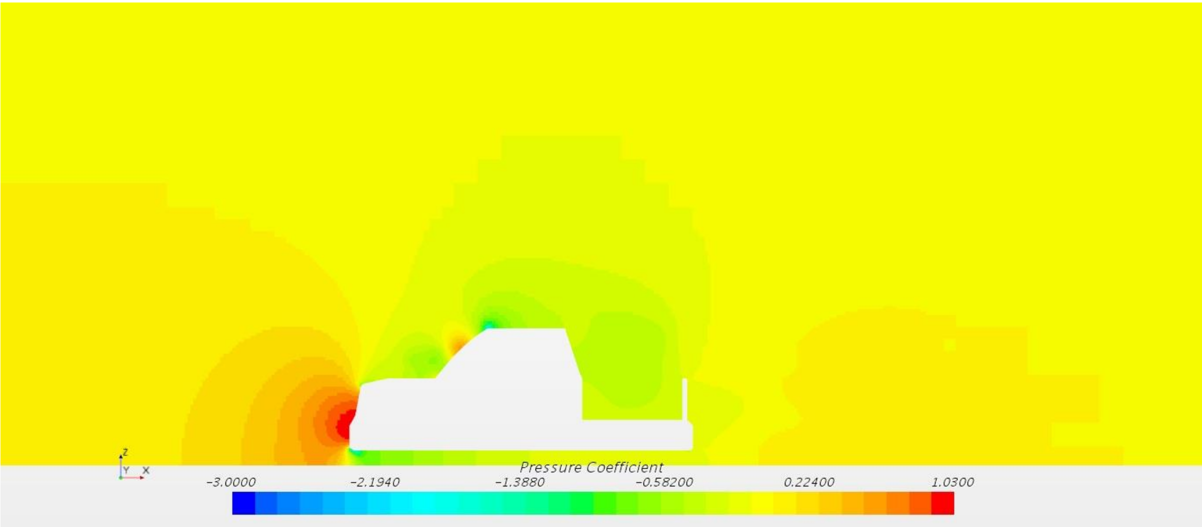

Fig. 6.2. Pressure coefficient of improvised tailgate up case.

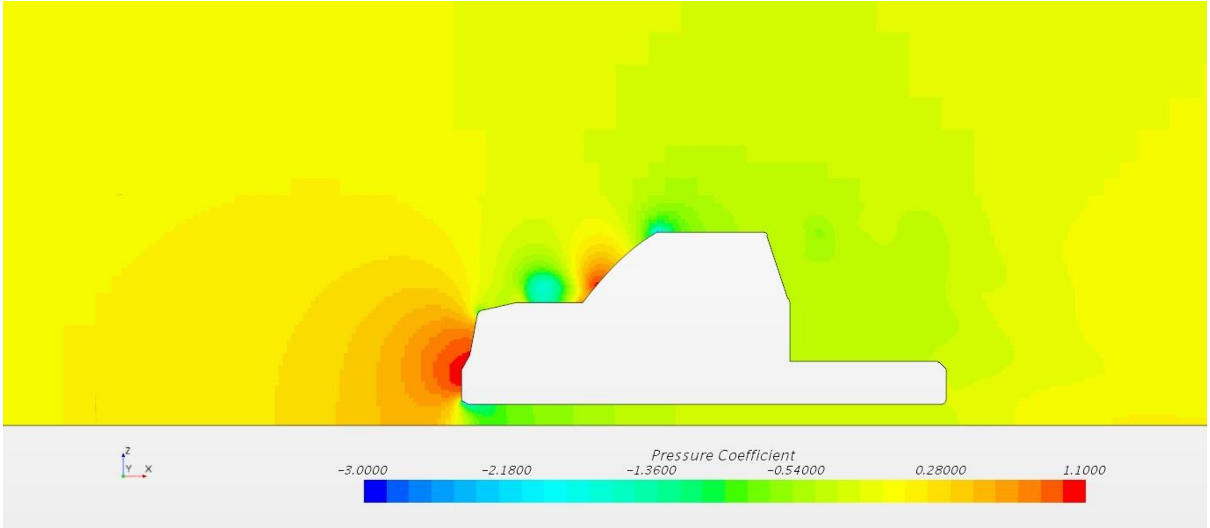

Fig. 6.3. Pressure coefficient of optimized tailgate down case.

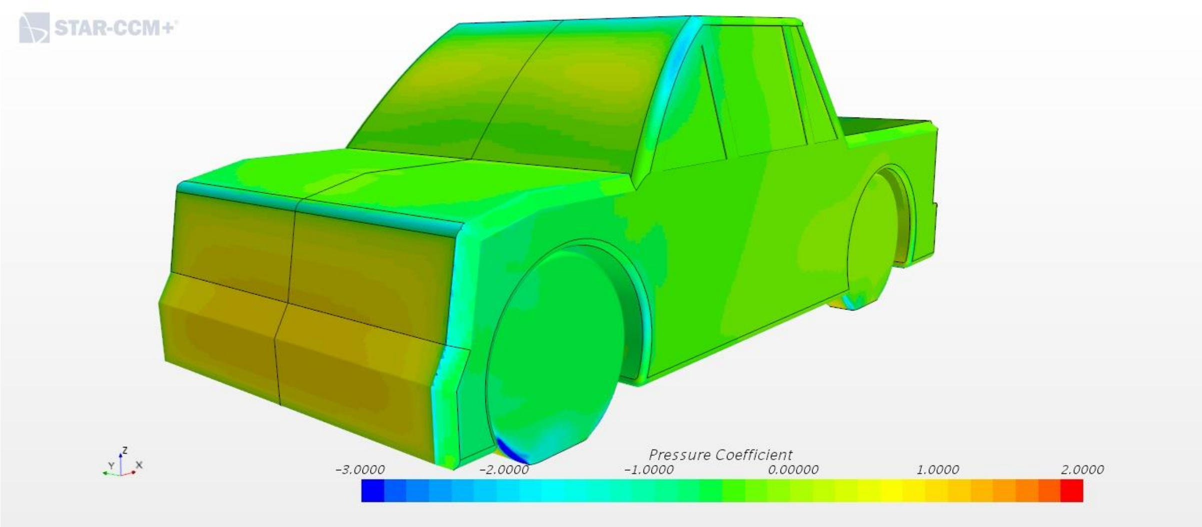

Fig. 6.4. Pressure coefficient over truck in baseline tailgate up simulation.

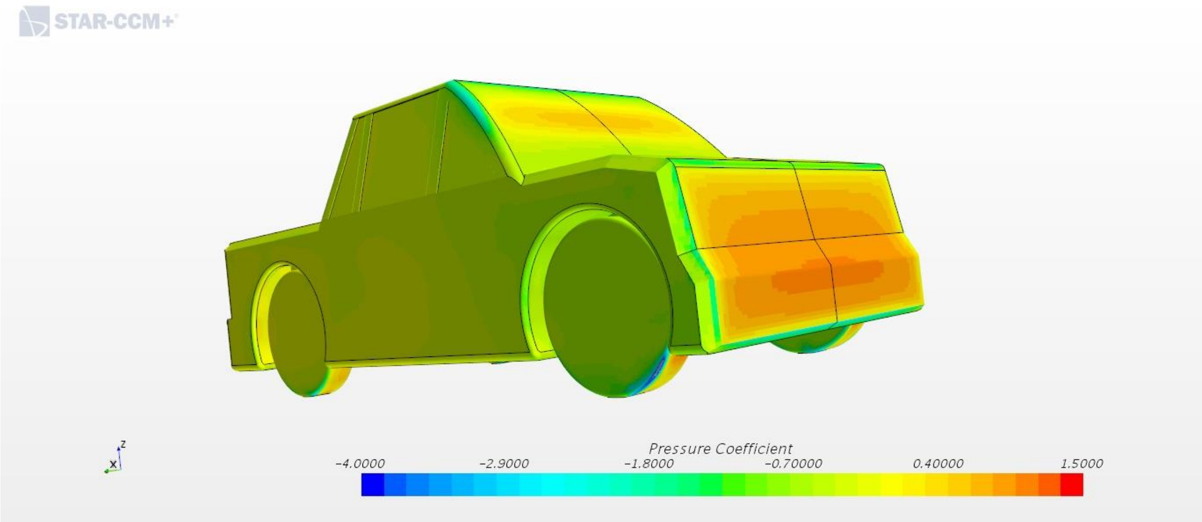

Fig. 6.5. Pressure coefficient over truck in optimized tailgate up case.

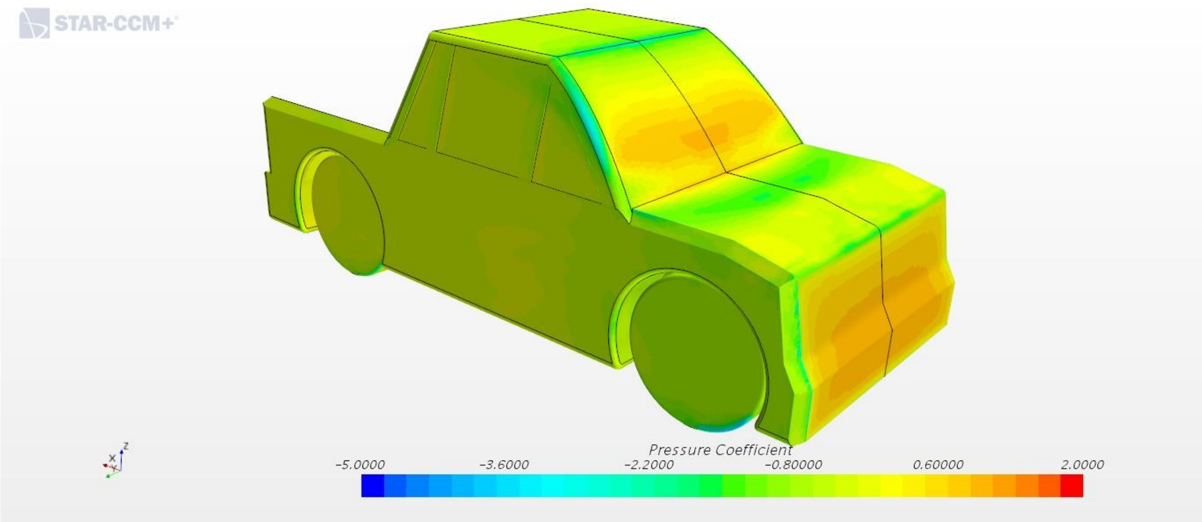

Fig. 6.6. Pressure coefficient over truck in improvised tail gate down case.

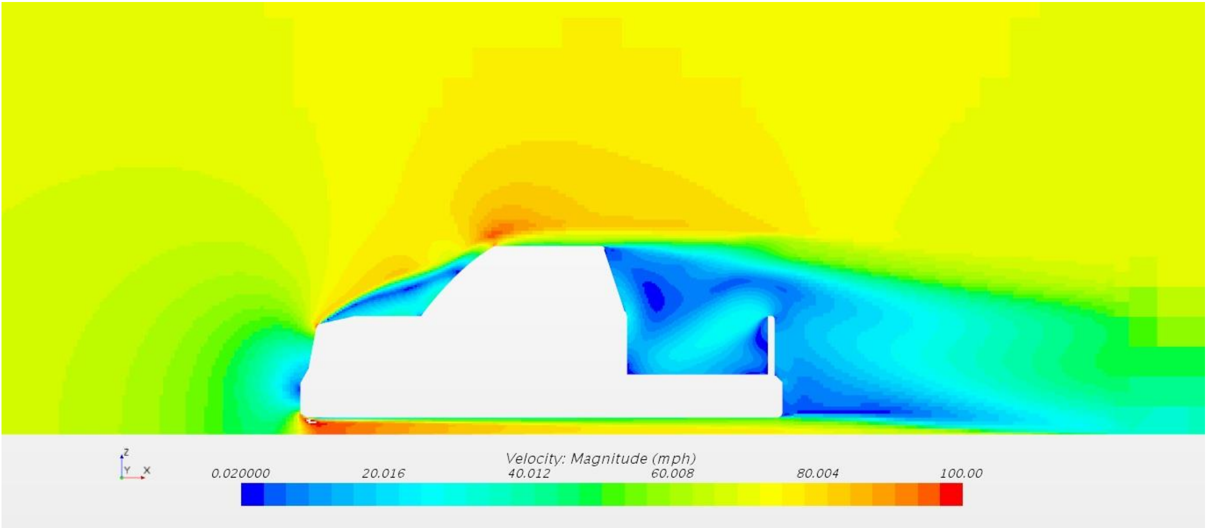

Fig. 6.7. Velocity magnitude in baseline simulation.

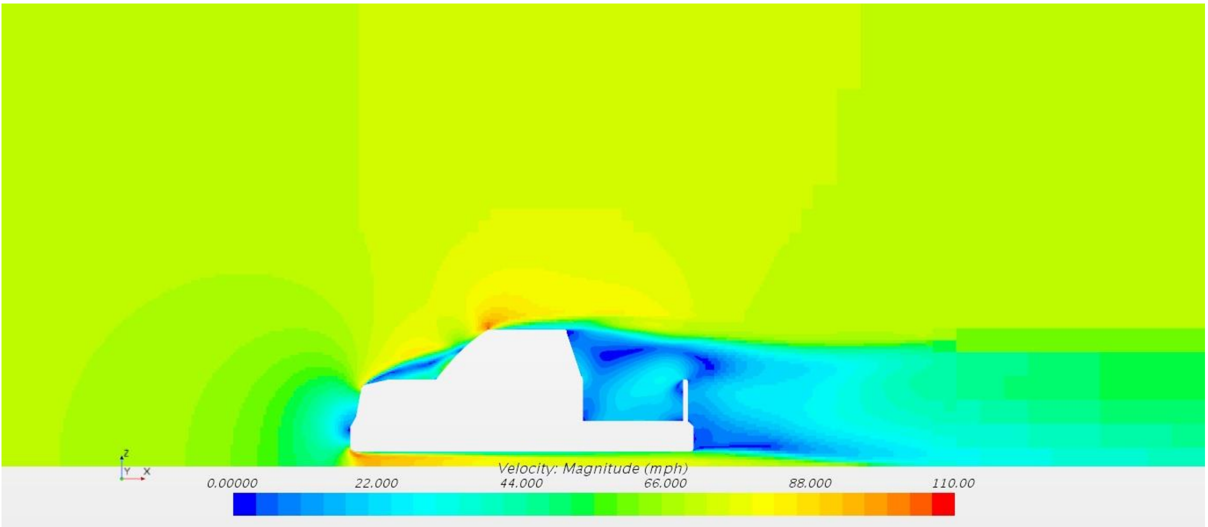

Fig. 6.8. Velocity magnitude in optimized tailgate up simulation.

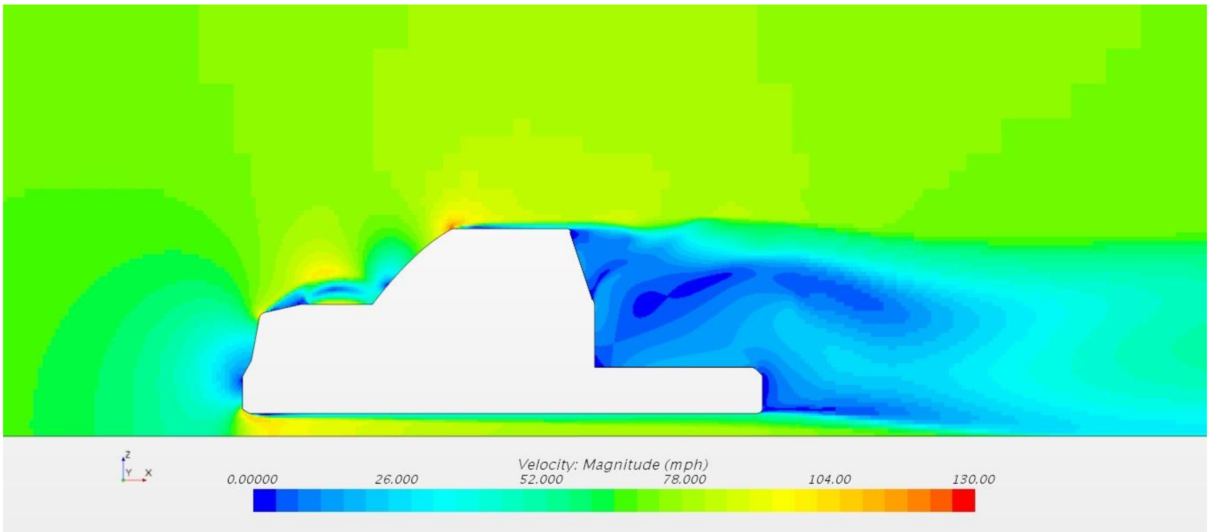

Fig. 6.9. Velocity magnitude in optimized tailgate down case.

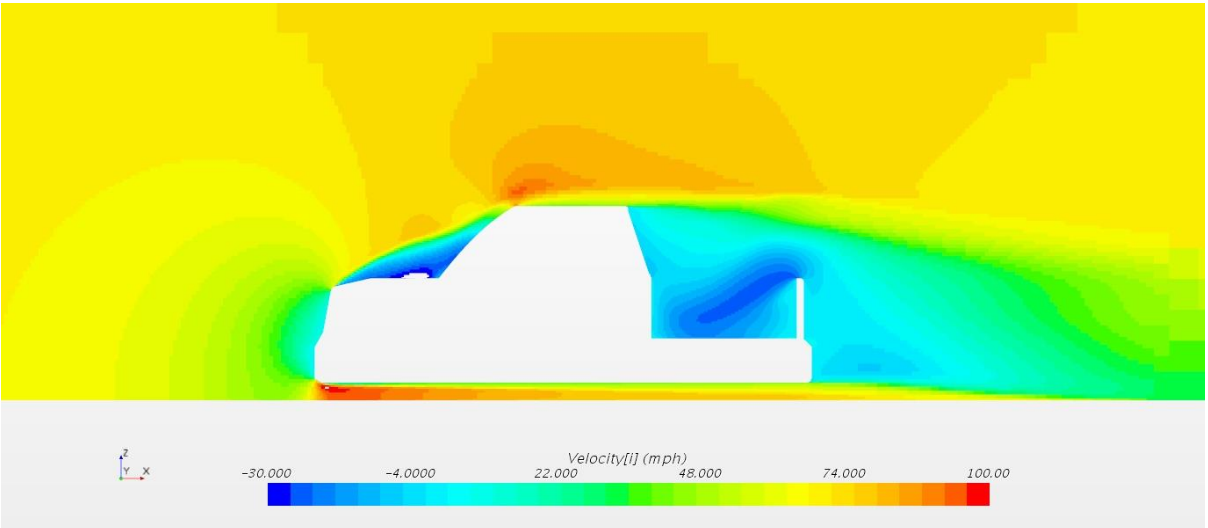

Fig. 6.10. Velocity in X direction for baseline case.

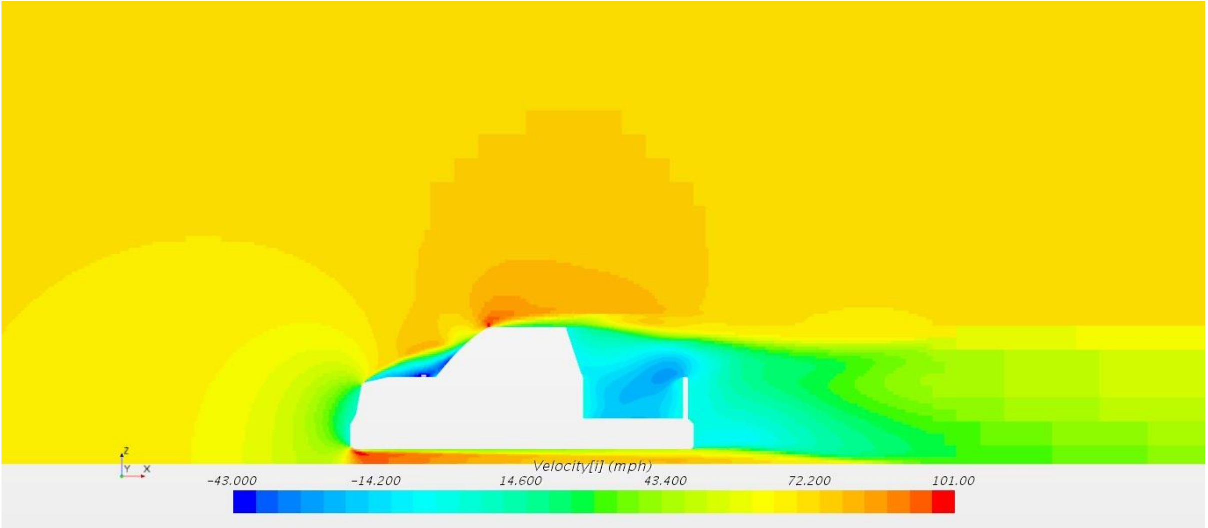

Fig. 6.11. Velocity in X direction for optimized tailgate up case.

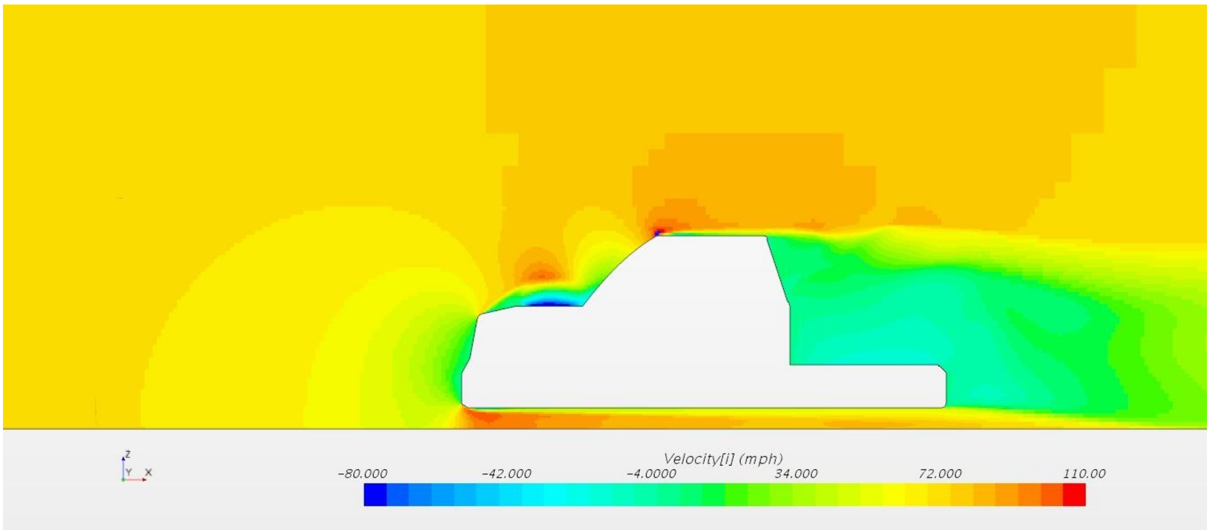

Fig. 6.12. Velocity in X direction for optimized tailgate down case.

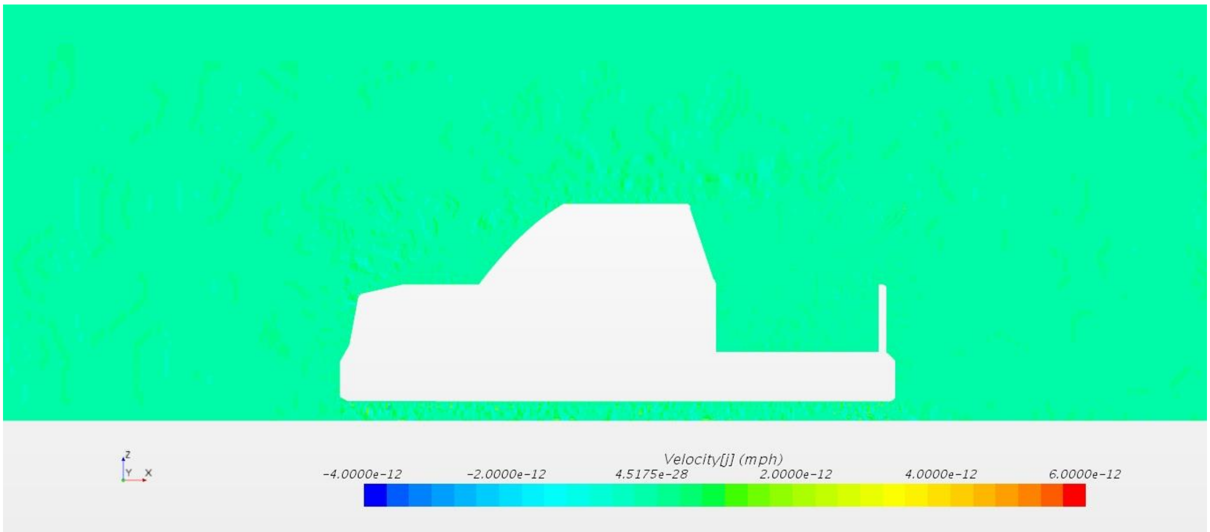

Fig. 6.13. Velocity in Y direction for baseline case.

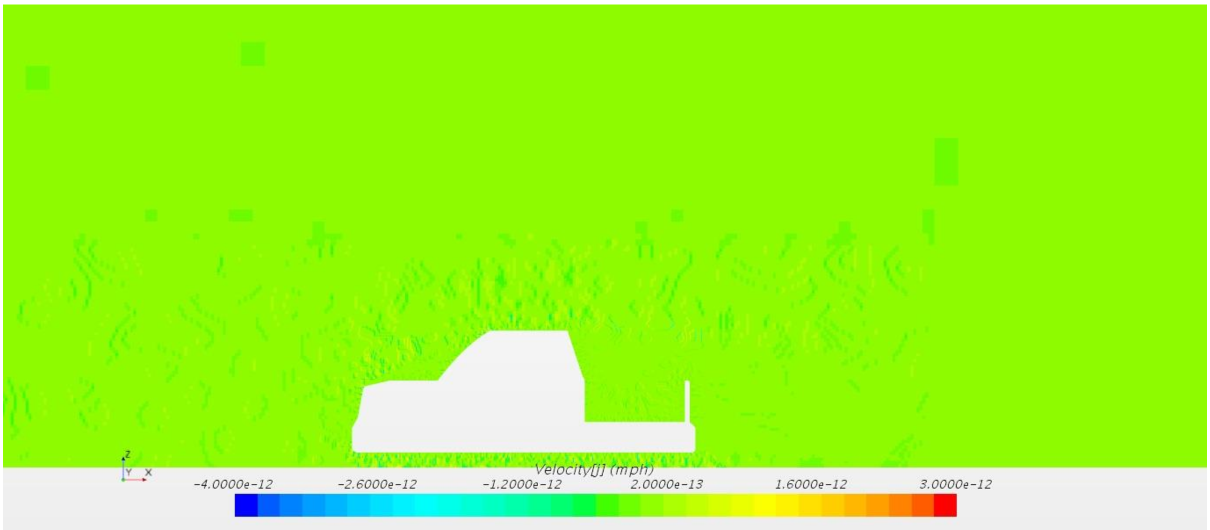

Fig.6.14. Velocity in Y direction for optimized tailgate up case.

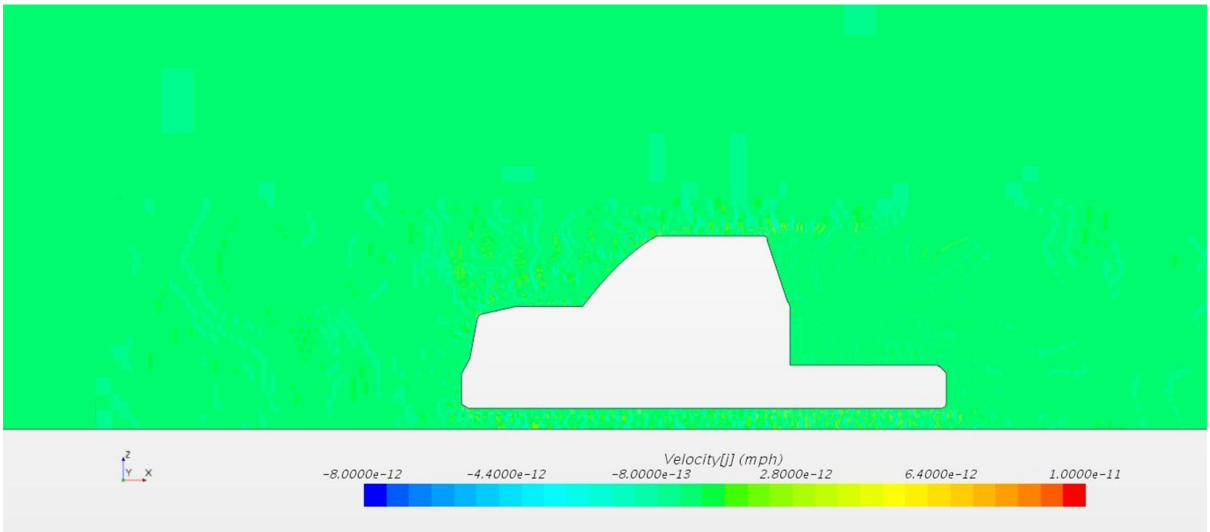

Fig. 6.15. Velocity in Y direction for optimized tailgate down case.

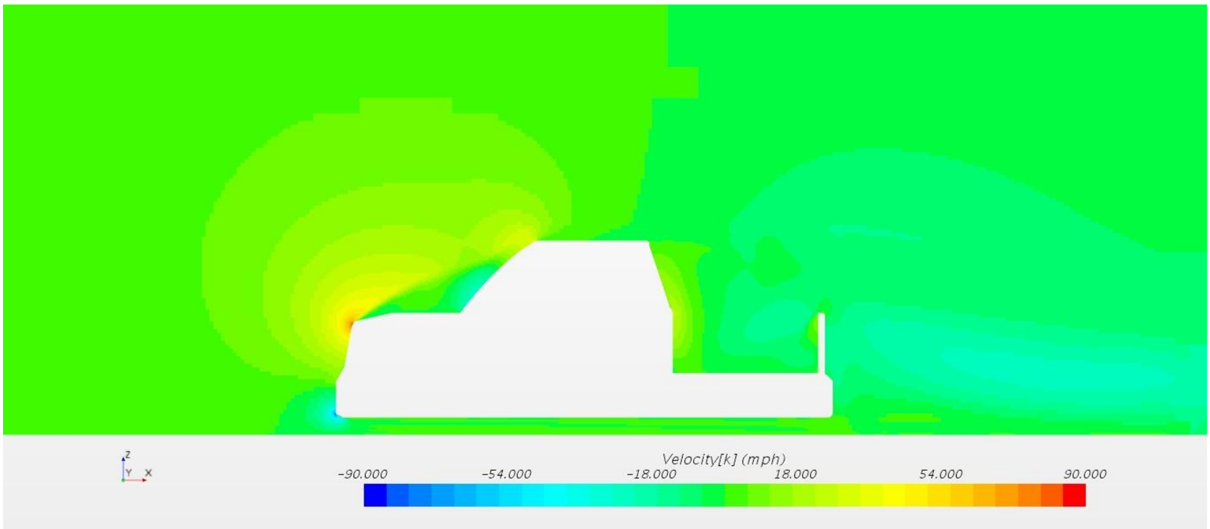

Fig.6.16. Velocity in Z direction for baseline case.

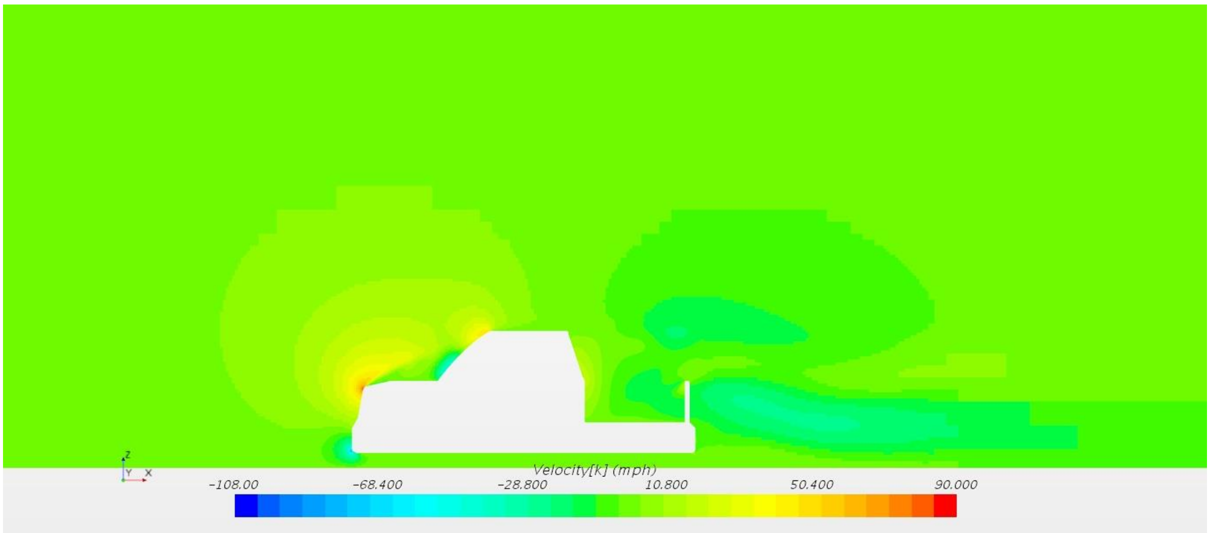

Fig. 6.17. Velocity in Z direction for optimized tailgate up case.

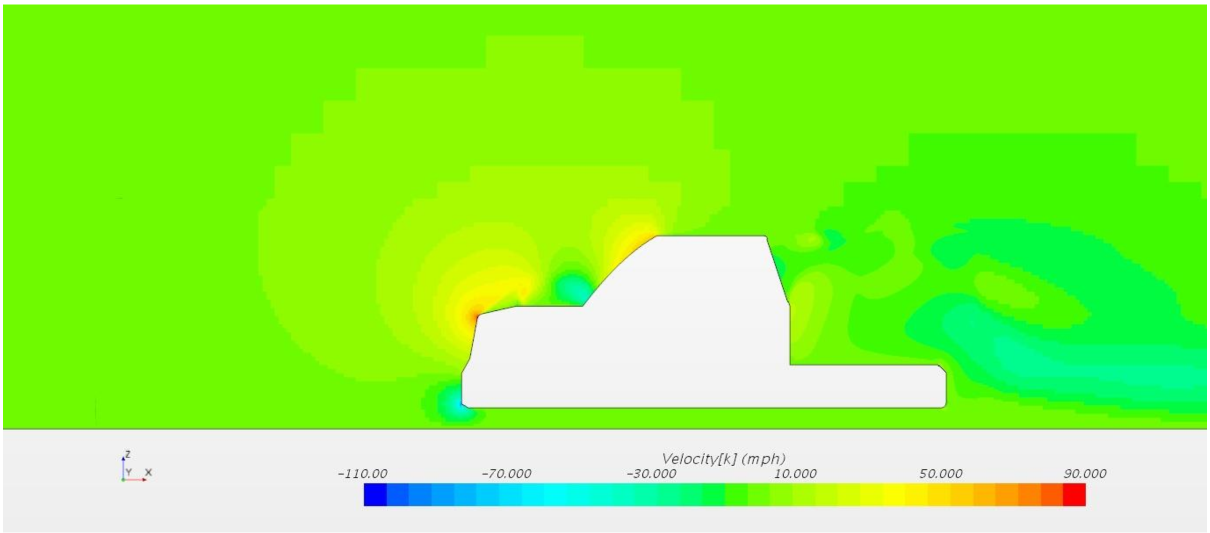

Fig. 6.18. Velocity in Z direction for optimized tailgate down case.

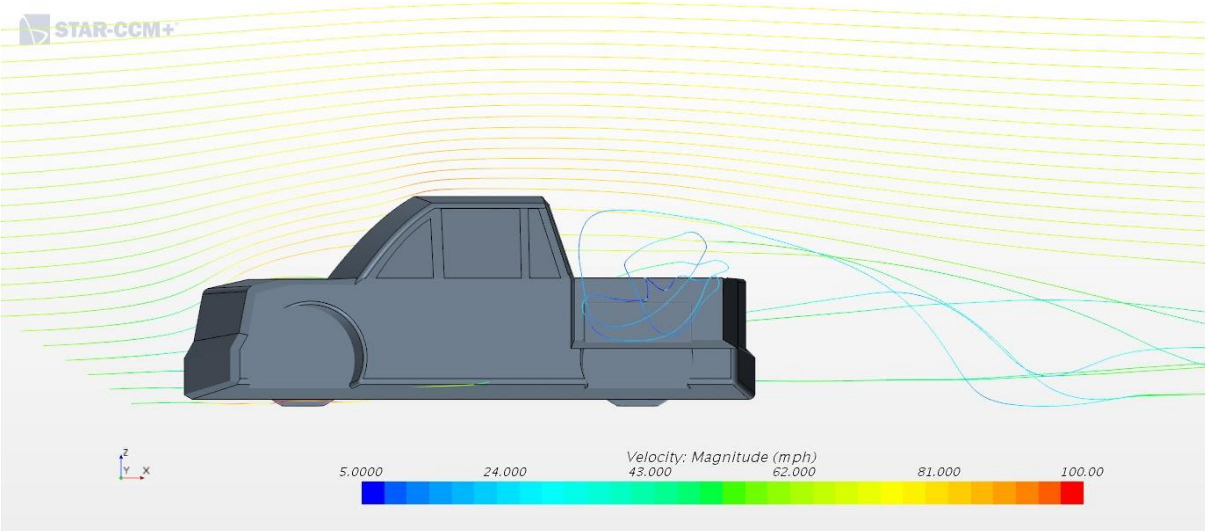

Fig. 6.19. Streamlines on symmetry plane for baseline case.

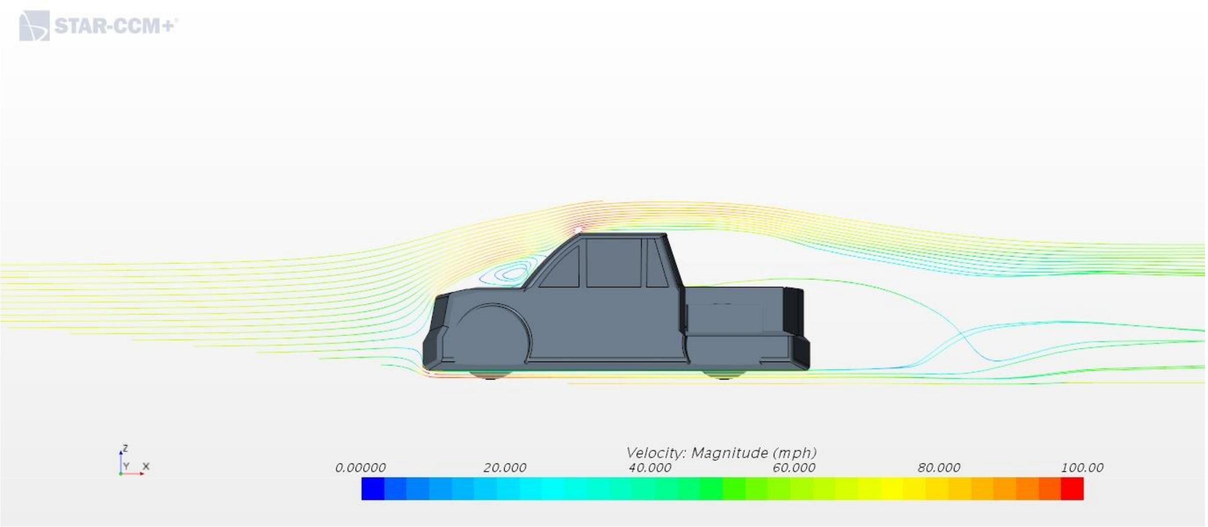

Fig. 6.20. Streamlines on symmetry plane in optimized tailgate up case.

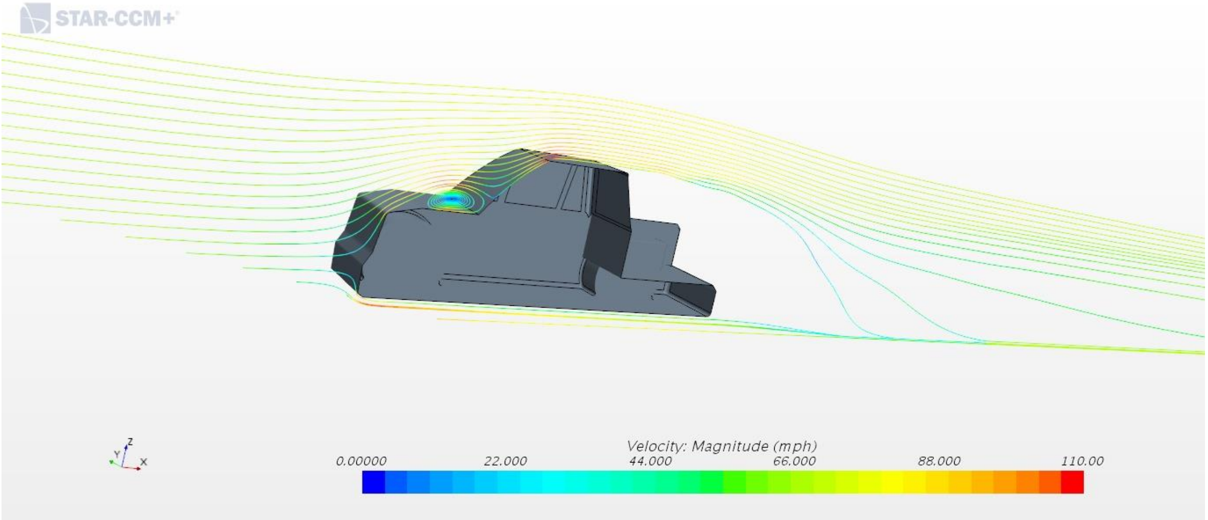

Fig. 9. 21. Streamlines on symmetry plane in optimized tailgate down case.

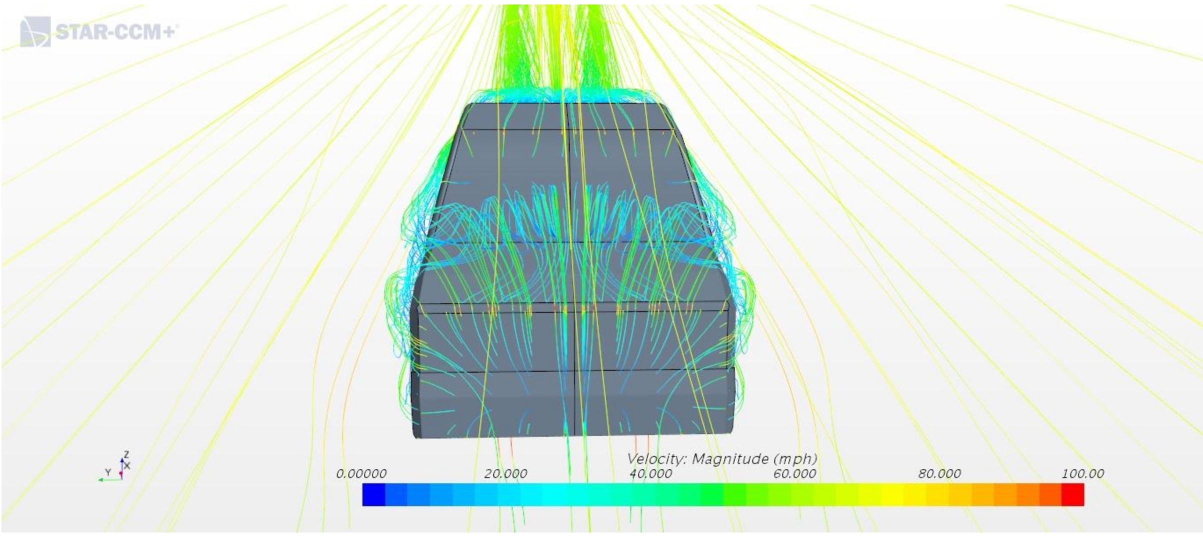

Fig. 9.22. 3D Streamlines in baseline simulation (front view).

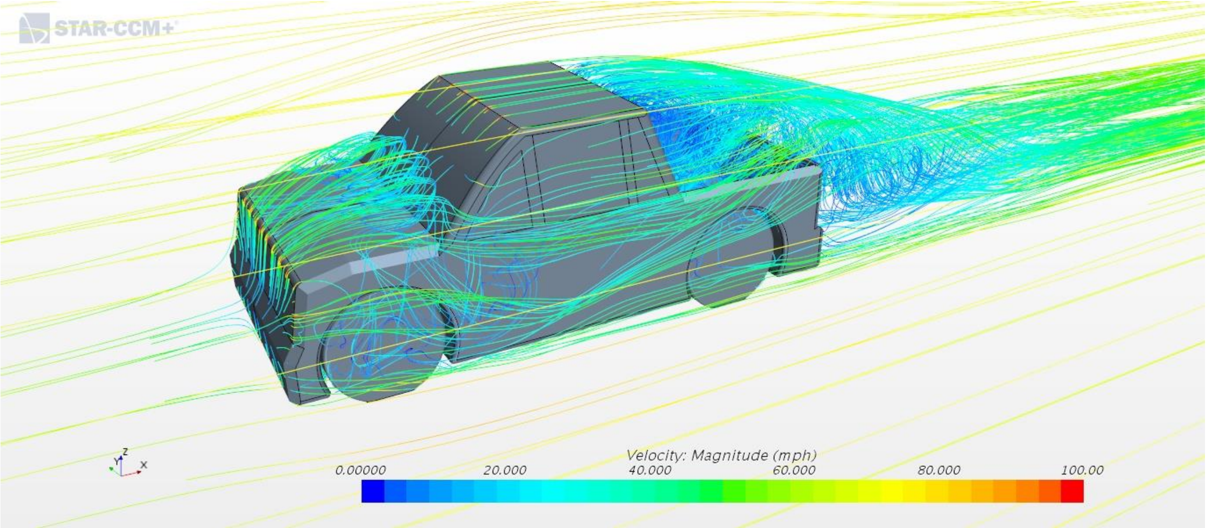

Fig. 9.23. 3D streamlines in baseline simulation.

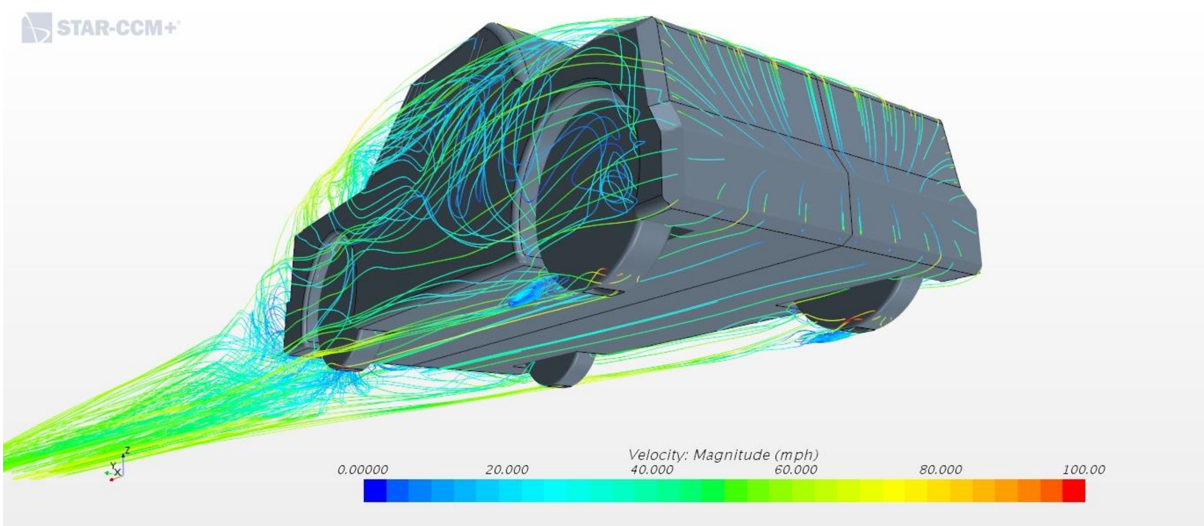

Fig. 9.24. 3D streamline under the truck in optimized tailgate up case.

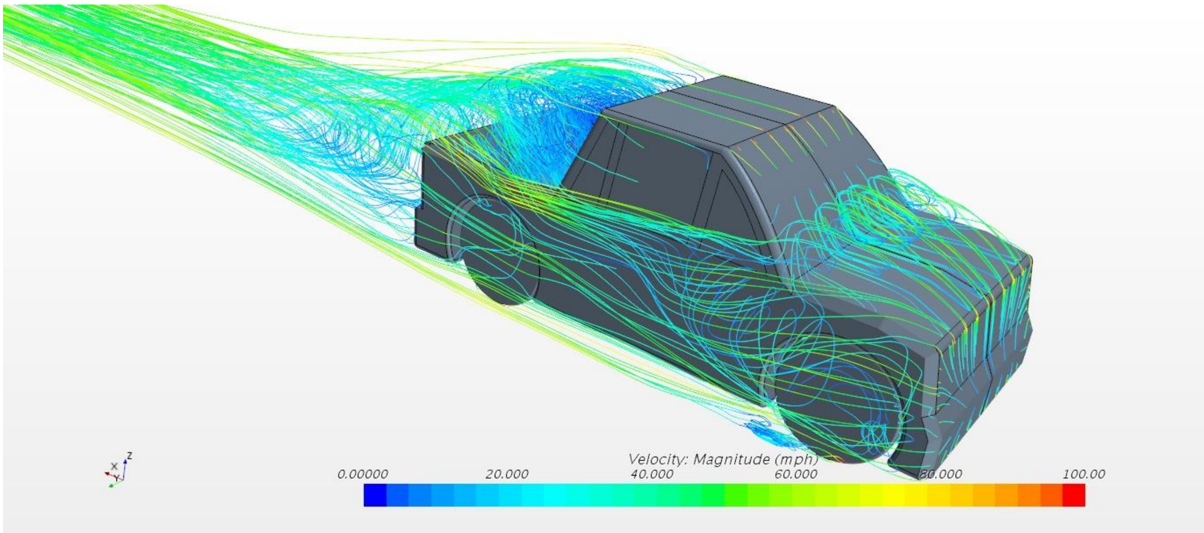

Fig. 9.25. 3D streamlines over the truck in optimized tailgate up case.

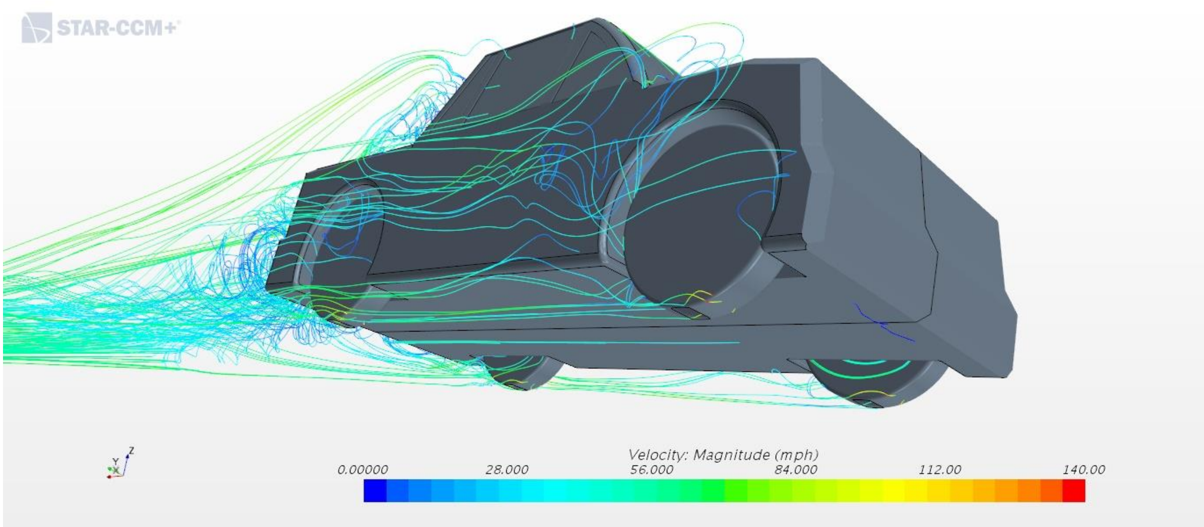

Fig. 6.26. 3d streamlines under the truck in optimized tailgate down case.

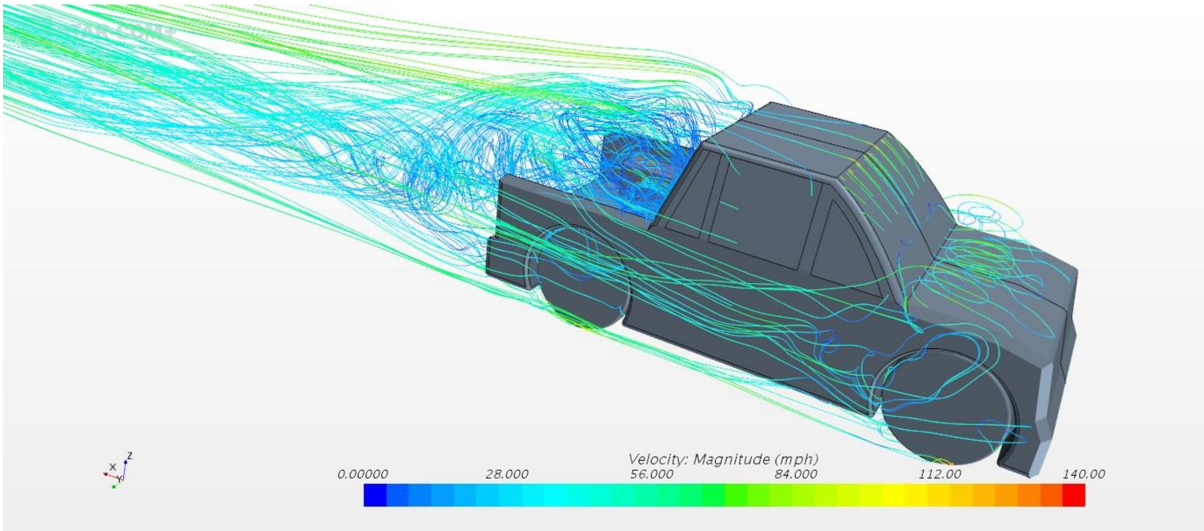

Fig. 6.27. 3D streamlines over the truck in optimized tailgate down case.

7. *Analysis*

- Checking the graphs plotted for residuals for all the cases the fluctuations are more for the optimized case. In the first 1000 iterations the fluctuations in the residuals were very high and later they started to converge after 1300 iterations and there was no much difference in the 1300 to 1600 iterations, implying the solution is converged.
- For checking the drag values for each simulation we have averaged out the values for each case from 500 iterations to 1600 iterations. The reason for averaging the values in between these iteration is that both the drag and lift in all three cases are not having much variation in their values from 500 iterations as we see the graphs are not fluctuating.

	Baseline	**Optimized tailgate up**	**Optimized tailgate down**
Lift (N)	131.9634	-43.0343	-5.72387
Drag (N)	746.8648	765.4174	923.0287

- With these Average values we can calculate the C_{df}*A and C_d*A.
  The formula for calculation is $C*A= [(force)/(0.5*\varrho*v^2)]$
  We have density as 1.2 kg/m^3 and velocity as 70 mph that is 31.2928 m/s.
  Calculating the values from above formula we get :
  For Optimized tailgate down case: $C_{df}*A = 9.742e\text{-}3$ and $C_d*A = 1.571$
  For Optimized tailgate up case: $C_{df}*A = 0.0732$ and $C_d*A = 1.303$

- Now checking the pressure coefficient in the three cases we see that the high pressure area is the stagnation point for the truck in all cases, there is formation of local air bubbles in the hood area and the roof area causing increase in pressure at those regions. Also in the under tray region we can see the pressure is low at front wheel region and later moderate not to high through out the underbody.
- Looking at the velocity magnitude and velocities in x, y, z directions we see low velocity in stagnation point of vehicle, in front of windshield, and in the wake region as well. In the rest of the area the velocity is moderate and highest in the under tray ahead of front wheel and later on it decreases.
- Looking individually in the x, y, z direction velocities we see that in y direction velocity the there is no high or low velocity region as it the rotational axis and perpendicular to flow direction.
- In the x and z directions the velocities regions are changing. The x direction velocity scene is similar to the velocity magnitude scene. The z direction velocity scene shows highest velocity in the front of hood region and we can also see the wake region as well in here.
- The streamlines generated gives us the idea of how and where the wake region is generated in all the three cases of the simulation as mentioned earlier there is formation of local wake bubbles over the hood region and then major wake behind the truck.
- The wake region for tailgate up case as compared to the tail gate down case.

8. *Conclusion*

- In this simulation we modeled a pickup truck in solidworks CAD software. Dimensions of the truck are mentioned and required changes were made in the case setup for the simulation. Simulation was run in STAR CCM+ software.
- 3 simulations were run, a baseline simulation for tailgate up case and optimized simulations for tailgate up and tailgate down case. Each simulation was run for 1600 iterations.
- Necessary modifications were made in optimized solution cases for the improvement of mesh quality in particular regions, like curves the under body, near tires, etc. New sections were added in the geometry to check the mesh in these areas.

- The cell count was increased in optimized cases due to improved meshing parameters which was, 6643185 cells for tailgate down case and 7044385 cells in tailgate up case.
- Plots for lift, drag were plotted to analyze the simulation. Also the pressure coefficient, velocity magnitude and velocities in x, y, z directions were plotted.
- The main case for the simulation was to check for which case the truck would give more gas mileage, for tailgate up case or for tailgate down case.
- In general it is pretty obvious that the drag for the tailgate up case are higher than the drag values for the tailgate down case. There is a little difference but this is what is considered.
- But when we run the simulation we see that the drag for the tailgate up case is less than the tailgate down case. This simulation acts as a myth buster.
- Considering the lift conditions as well there is no much variation both cases, basically the judging parameter for the better mileage performance is the drag itself.
- Finally after comparing the optimized cases the debate can be settled in favor of tailgate up case, as this condition provides less drag and so that fuel efficiency for this case is better as compared to the other case.