



COMPUTATIONAL FLUID DYNAMICS

Project: CFD Simulation around pickup truck

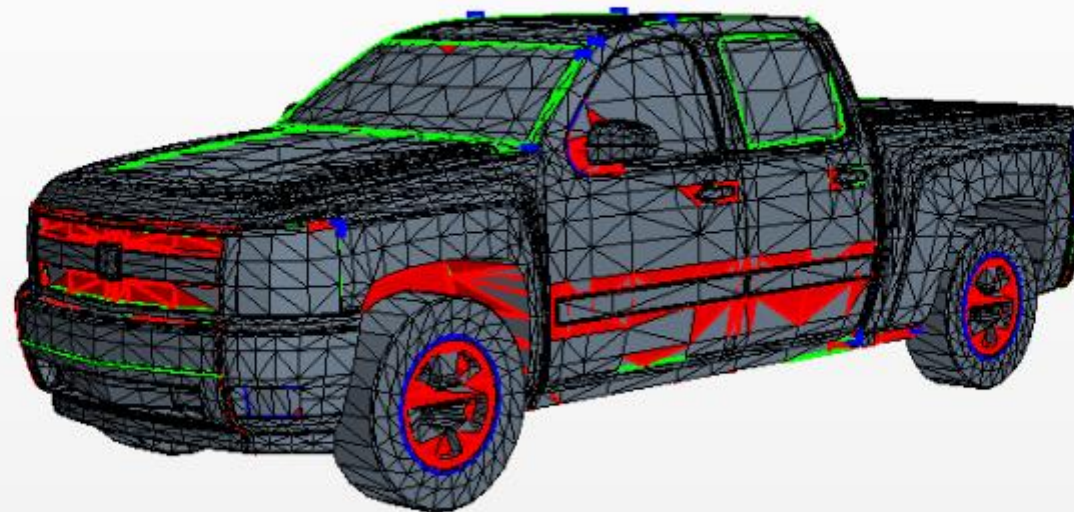
Name : Nikhilesh Sandela

Matriculation number : 224677

Under the supervision of

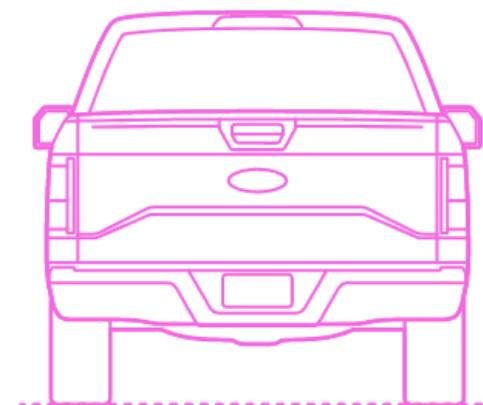
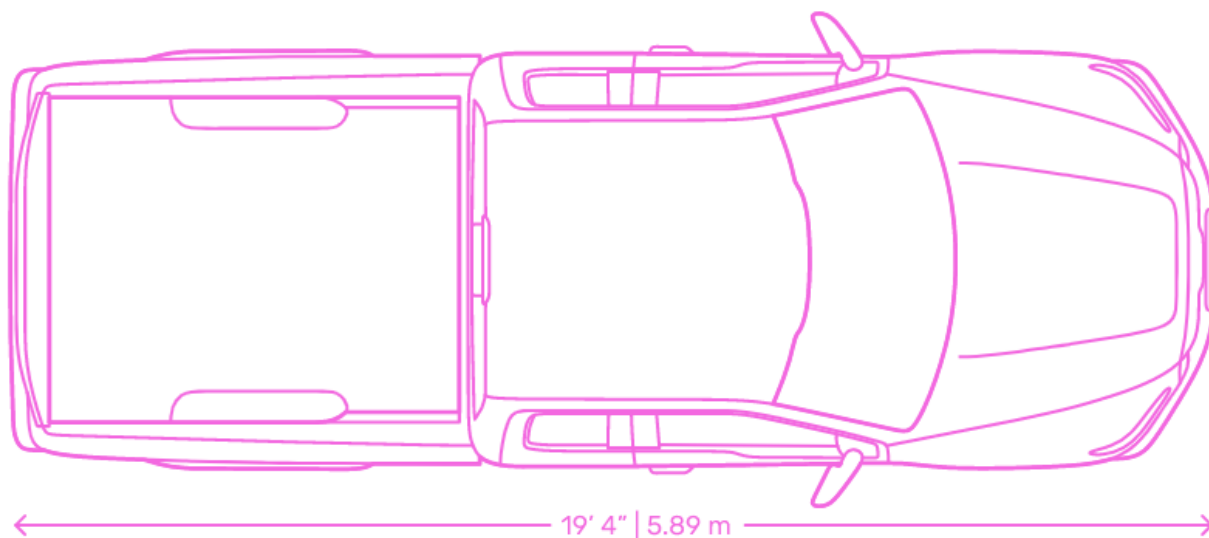
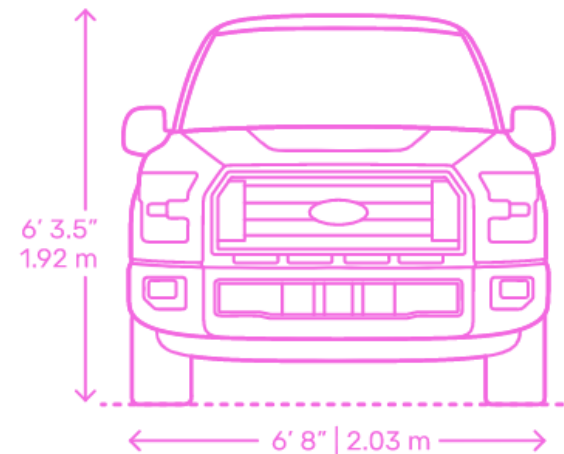
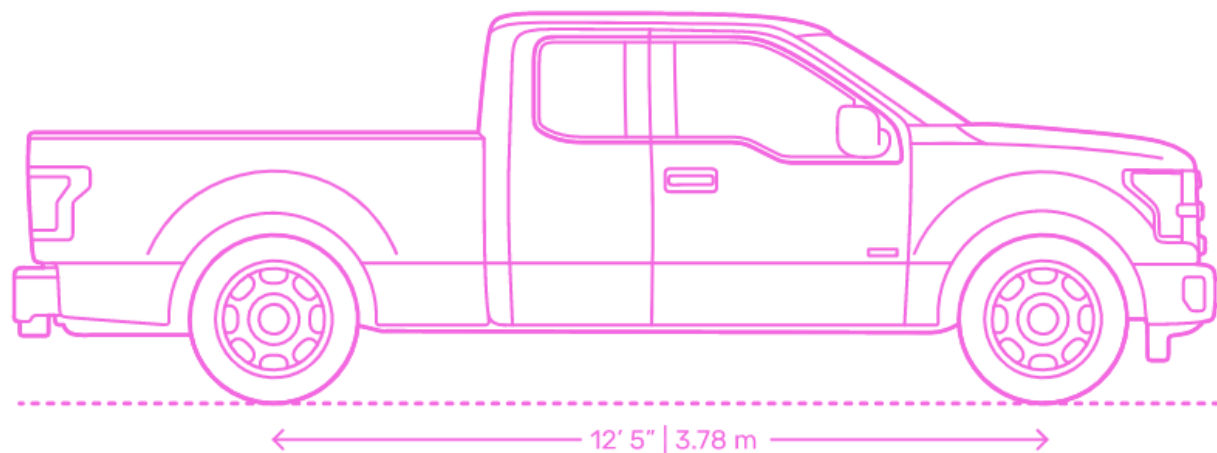
Dr. Gabor Janiga

Pickup Truck



Task

1. Import the provided geometry (adjust the dimension of the geometry if necessary).
2. Create a computational domain with a sufficient size.
3. Generate a computational mesh of sufficient quality and define the boundary conditions.
4. Simulate a steady state three-dimensional flow with inflow-velocities of 75 km/h and 150 km/h.
5. Understand and discuss the relevant flow characteristics, such as flow velocity and pressure, drag and lift coefficients such as drag and lift forces.



For the truck dimensions I have taken the following reference

<https://www.dimensions.com/collection/pickup-trucks>

Domain and Model selection

Physics 1 Model Selection

Transition

☐ Gamma Transition

☐ Gamma-ReTheta Transition

☐ Turbulence Suppression

<Optional>

Optional Models

☐ Adaptive Mesh

☐ Aeroacoustics

☐ Boussinesq Model

☐ Cell Quality Remediation

☐ Co-Simulation

☐ Dispersed Multiphase

☒ Electrochemistry

☐ Electromagnetism

☐ Fluid Film

☐ Gravity

☐ Lagrangian Multiphase

☐ Mesh Deformation

☐ Passive Scalar

☐ Porous Media

☐ Radiation

☐ Segregated Fluid Enthalpy

☐ Segregated Fluid Isothermal

☐ Segregated Fluid Temperature

☐ Turbulent Viscosity User Scaling

☐ Virtual Disk

☐ Vorticity Confinement Model

<Optional>

Enabled Models

☒ Solution Interpolation

<Not required by other models>

☒ All y+ Wall Treatment

☒ Wall Distance

☒ SST (Menter) K-Omega

☒ K-Omega Turbulence

☒ Reynolds-Averaged Navier-Stokes

☒ Turbulent

☒ Steady

☒ Constant Density

☒ Gradients

☒ Segregated Flow

☒ Gas

☒ Three Dimensional

☒ Auto-select recommended models

Close Help

domain

Expand/Contract Tree Expand/Contract Values

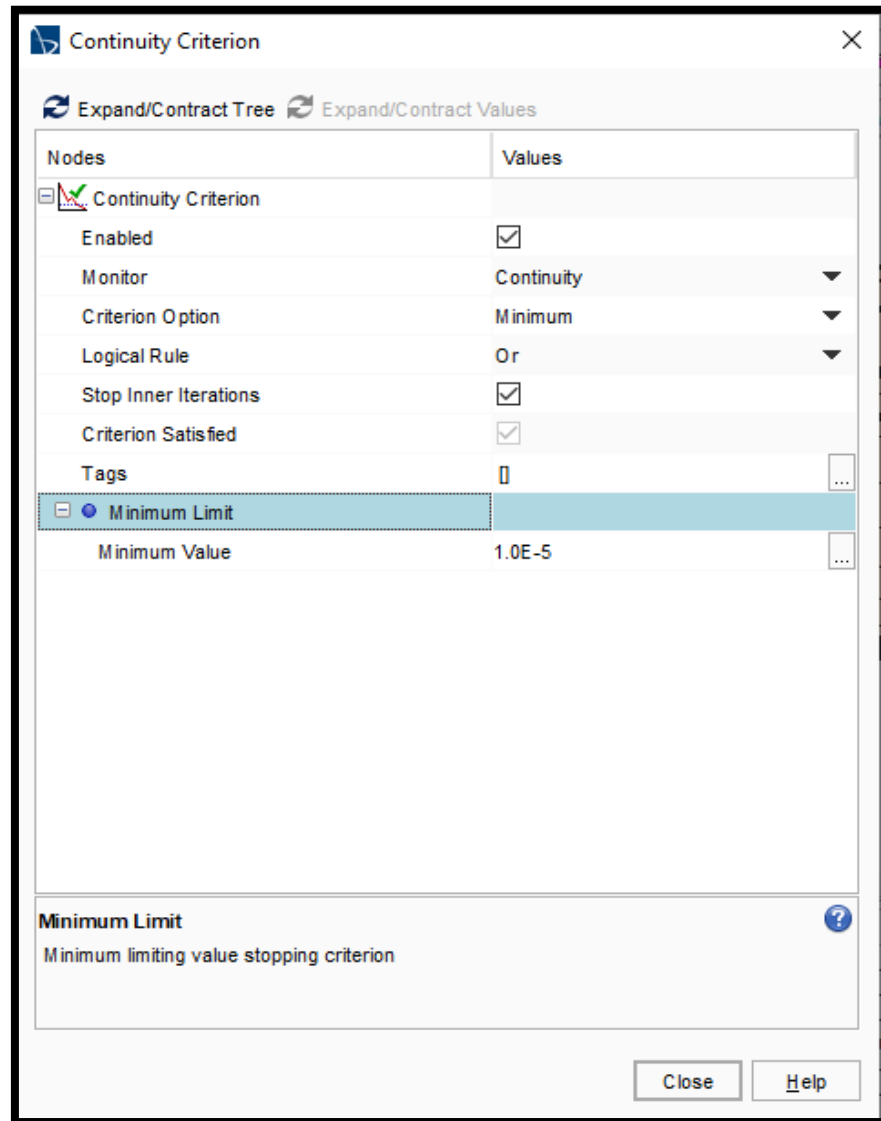
Nodes	Values
domain	
Region	
Coordinate System	Laboratory
Corner 1	[-7.2620..., 5.62835..., 0.10476...] m, m
Corner 2	[36.0886..., 19.2801..., 10.6625...] m, m
Tags	
Metadata	{}
Index	2
Contacts	
Descriptions	[Root]
Face Count	12
Surfaces	
Curves	

domain

A Leaf-level Block Part

Close Help

Continuity Criteria



Model selection:

Steady, Liquid,

Segregated Flow, Gradients,

Constant Density, Turbulent,

Reynolds-Averaged

Navier Stokes, K-Omega

Turbulence, SST (Menter)

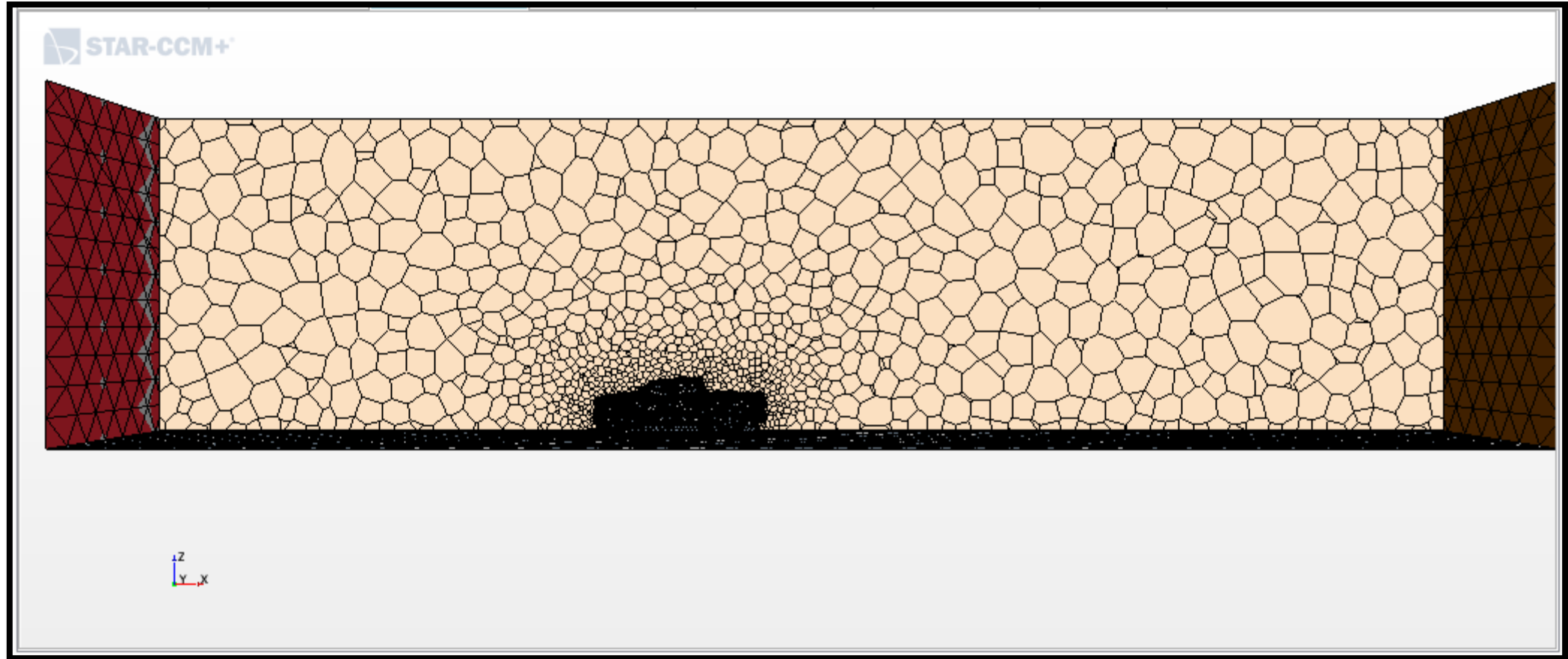
K-Omega,

All y+ Wall Treatment

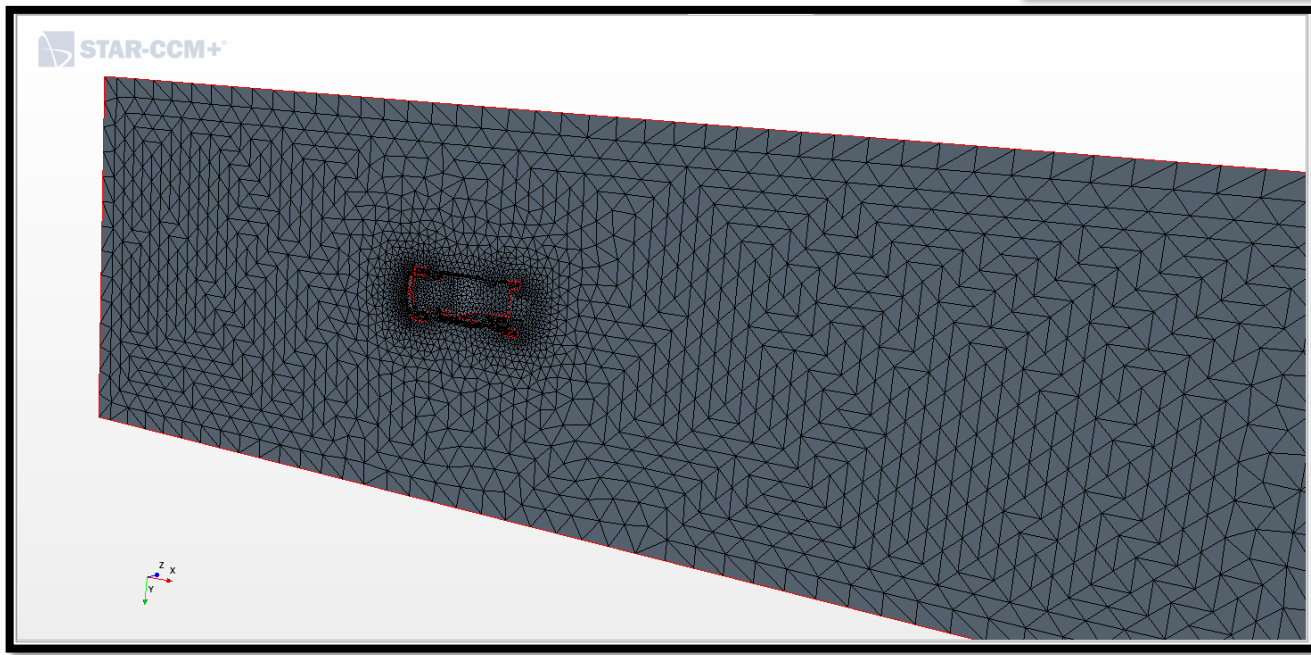
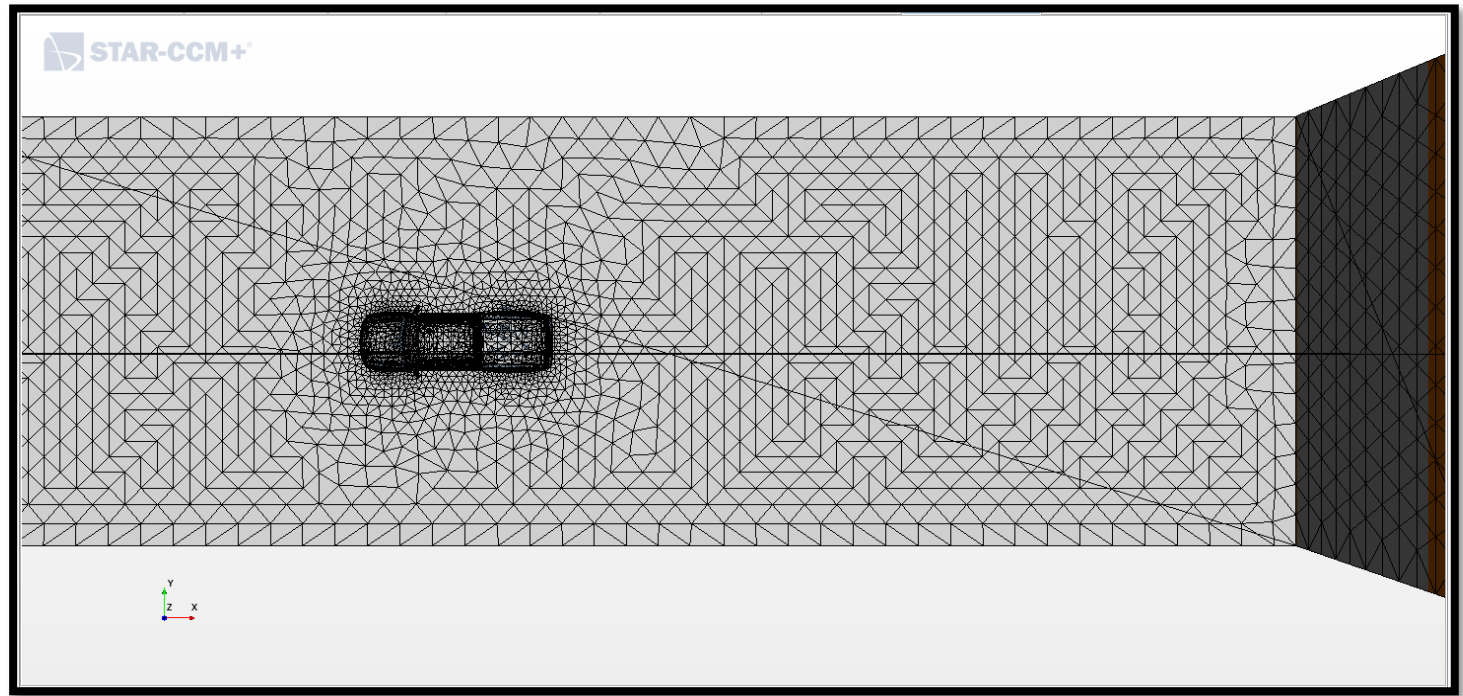
models are enabled.

Gas Model is selected.

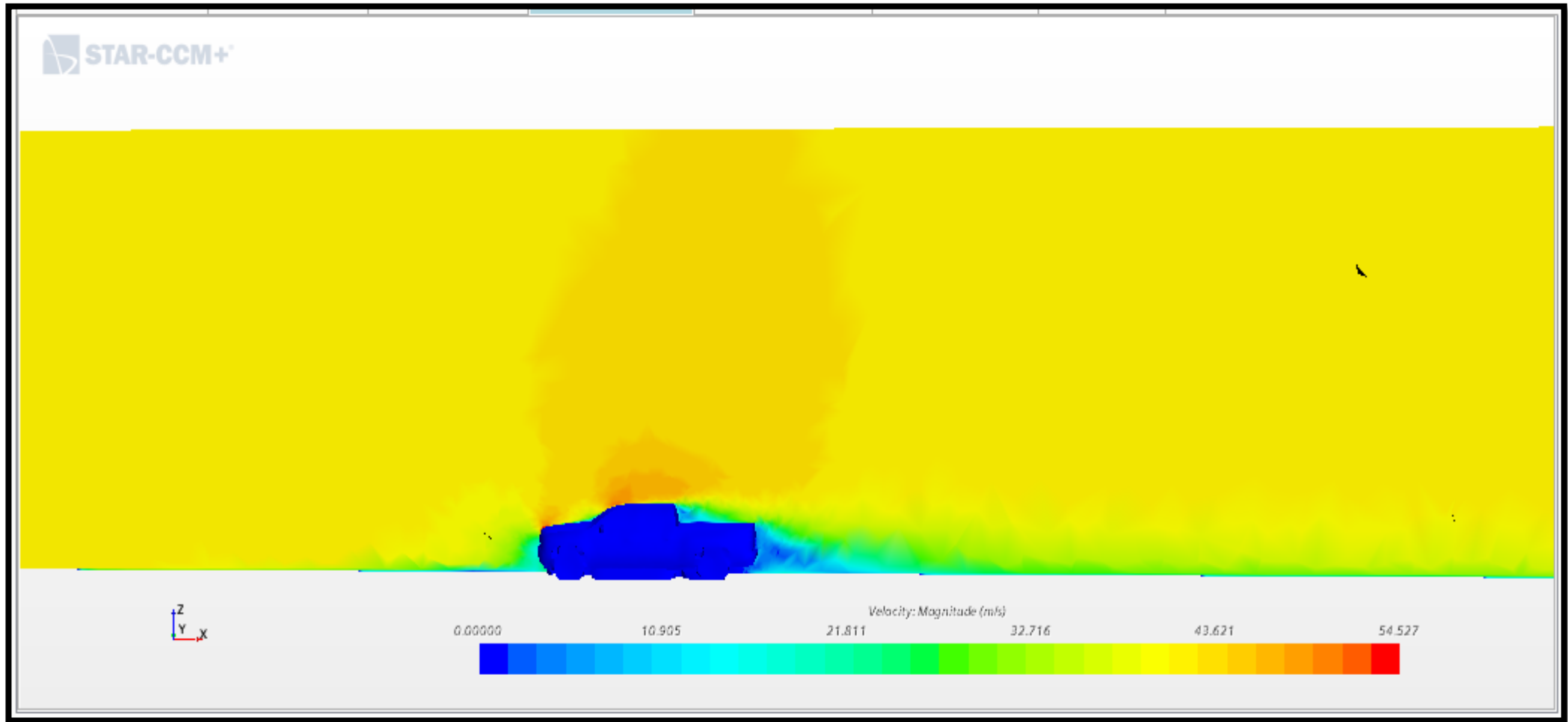
Geometry and mesh scenes



Geometry and mesh scenes

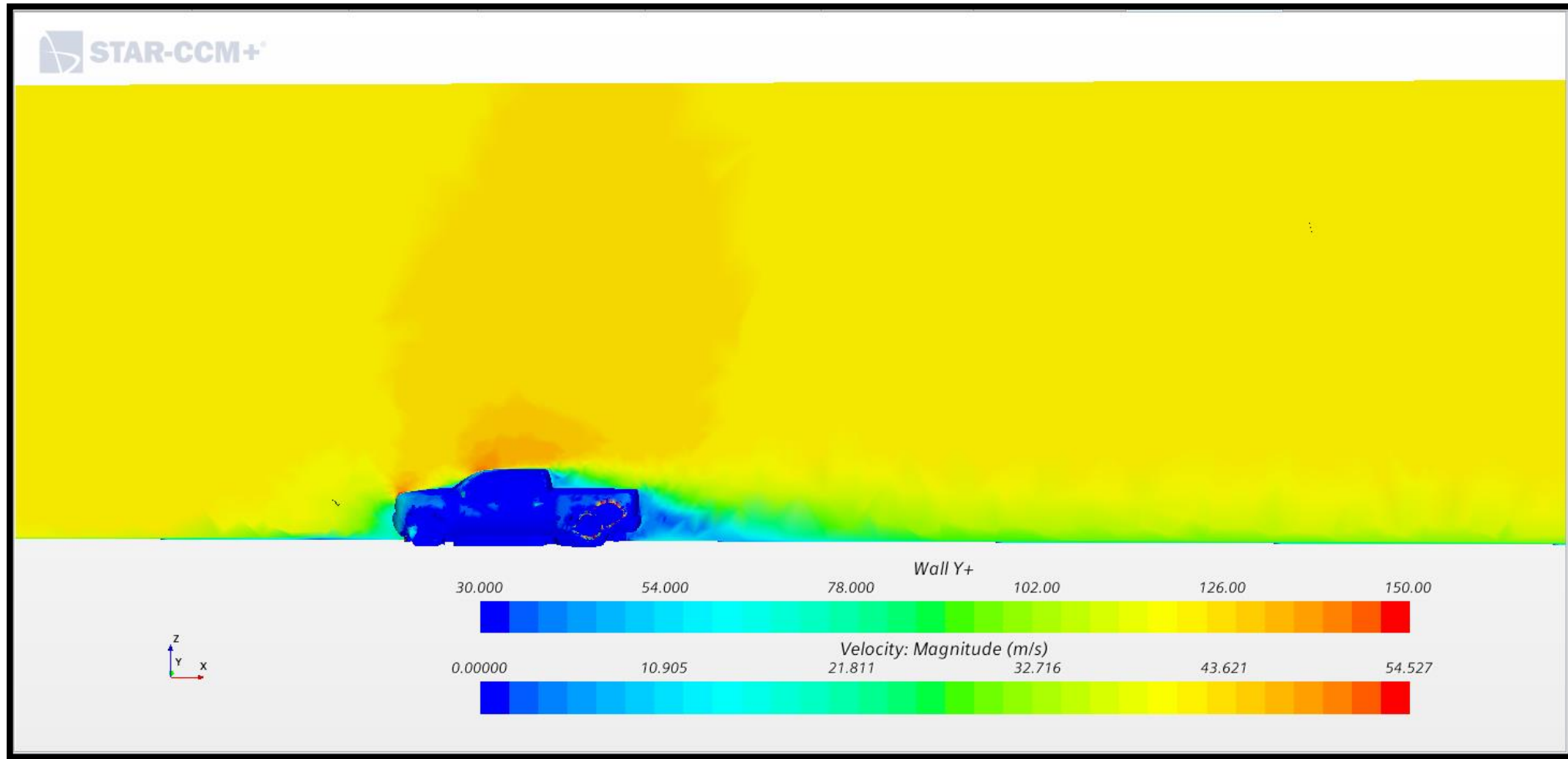


Scalar Scene



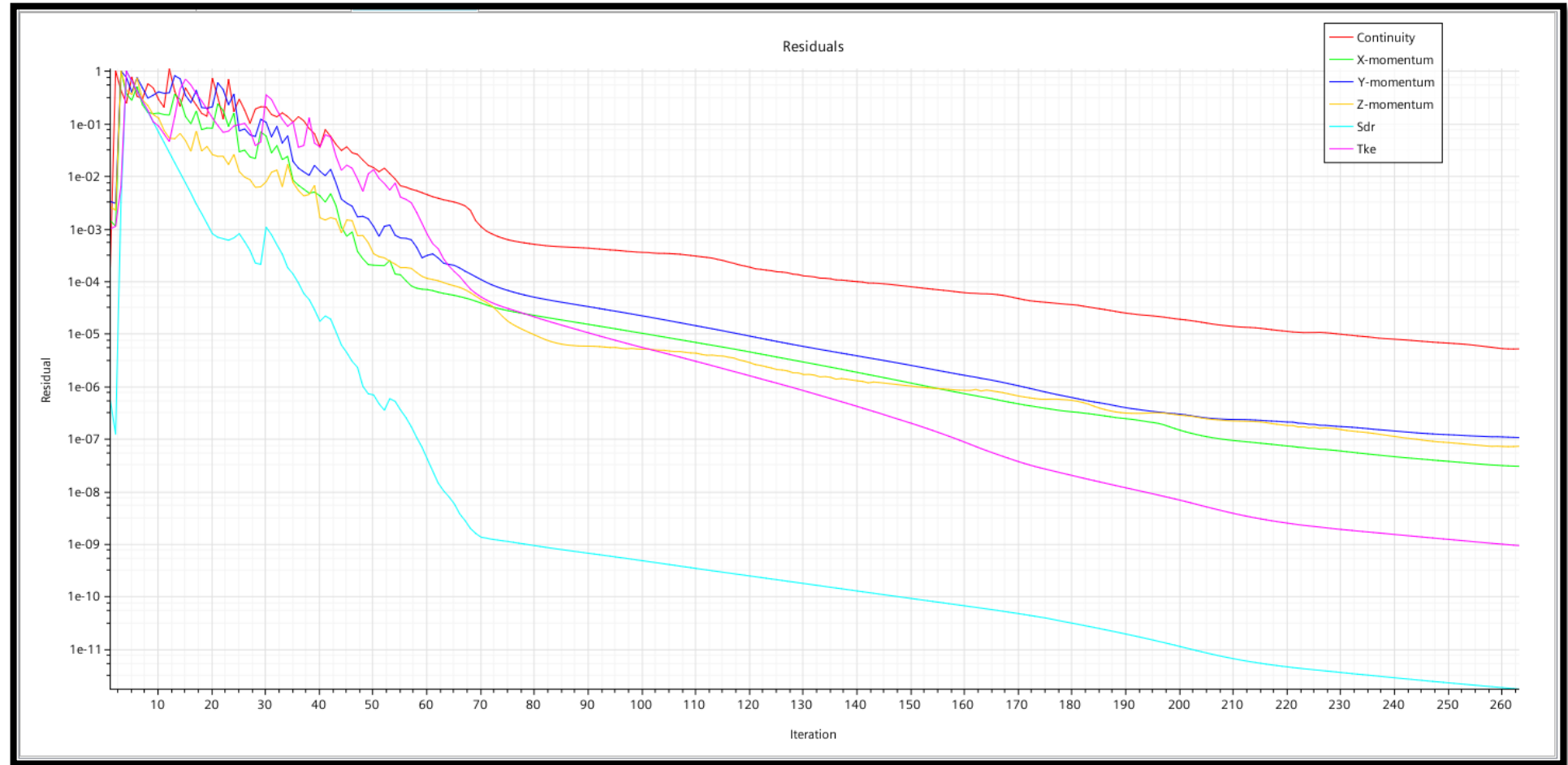
The model was studied for two wind speeds (75 km/h and 150 km/h) that simulate city and highway driving.

Results and discussion

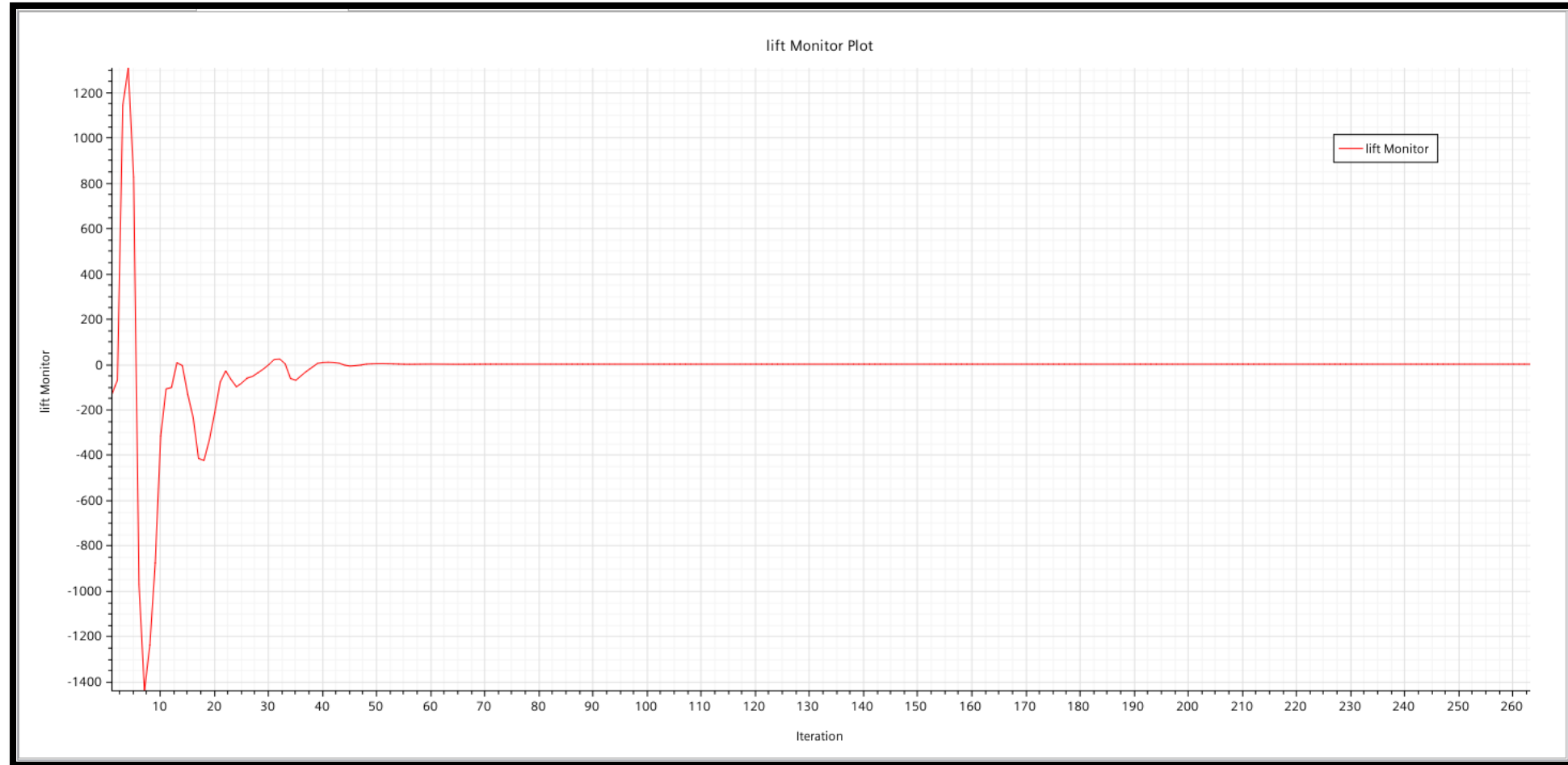


For a maximum velocity – 150 km/h

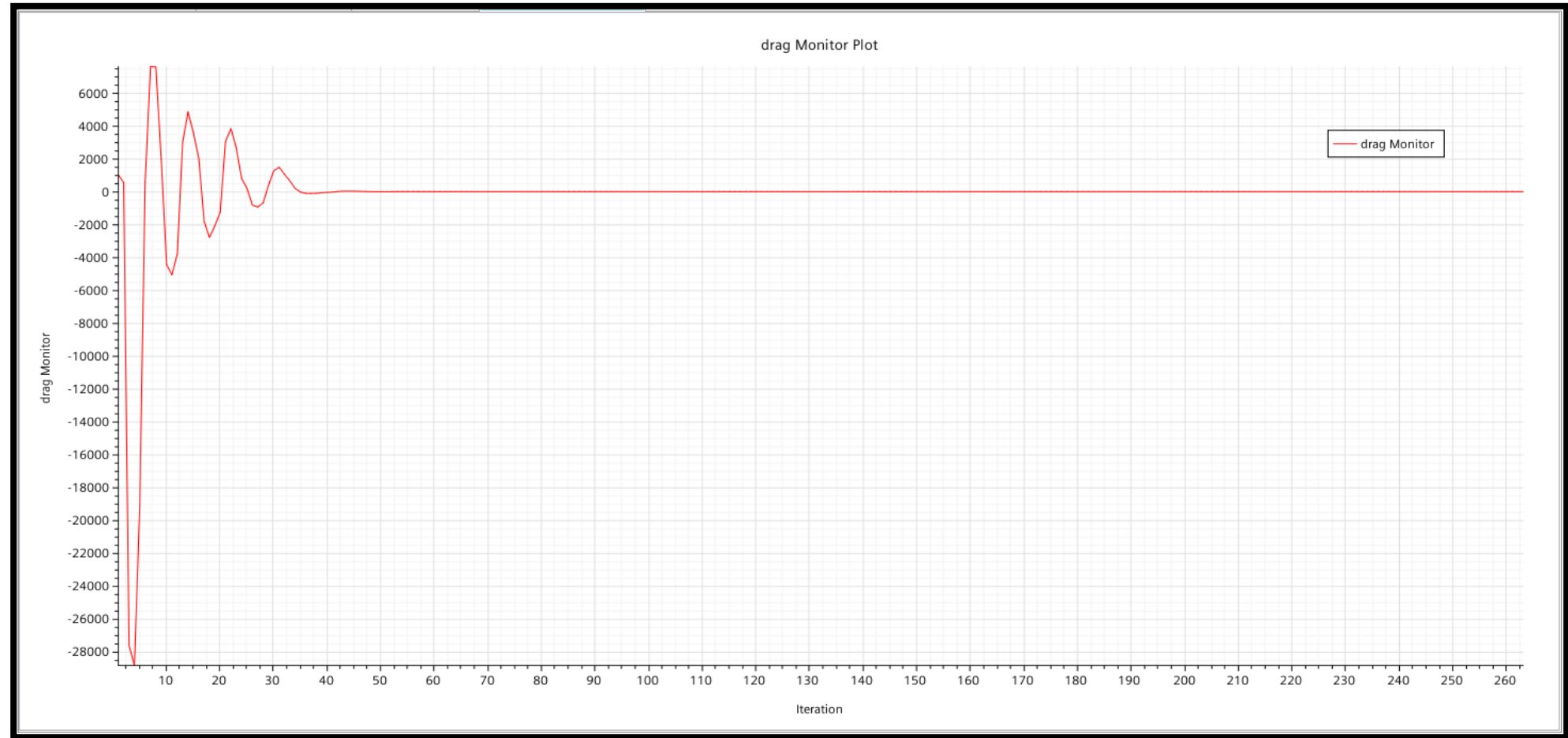
Residuals



Lift monitor plot

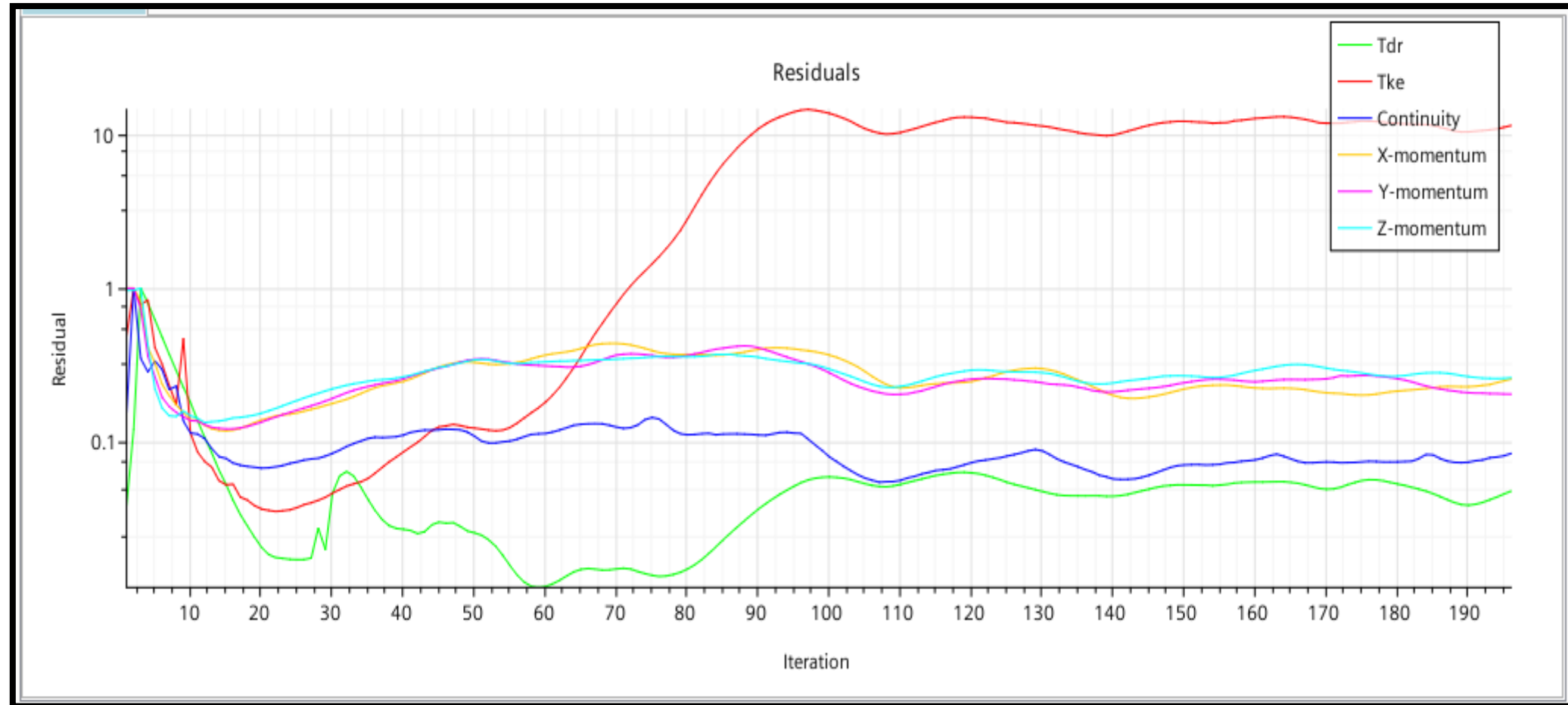


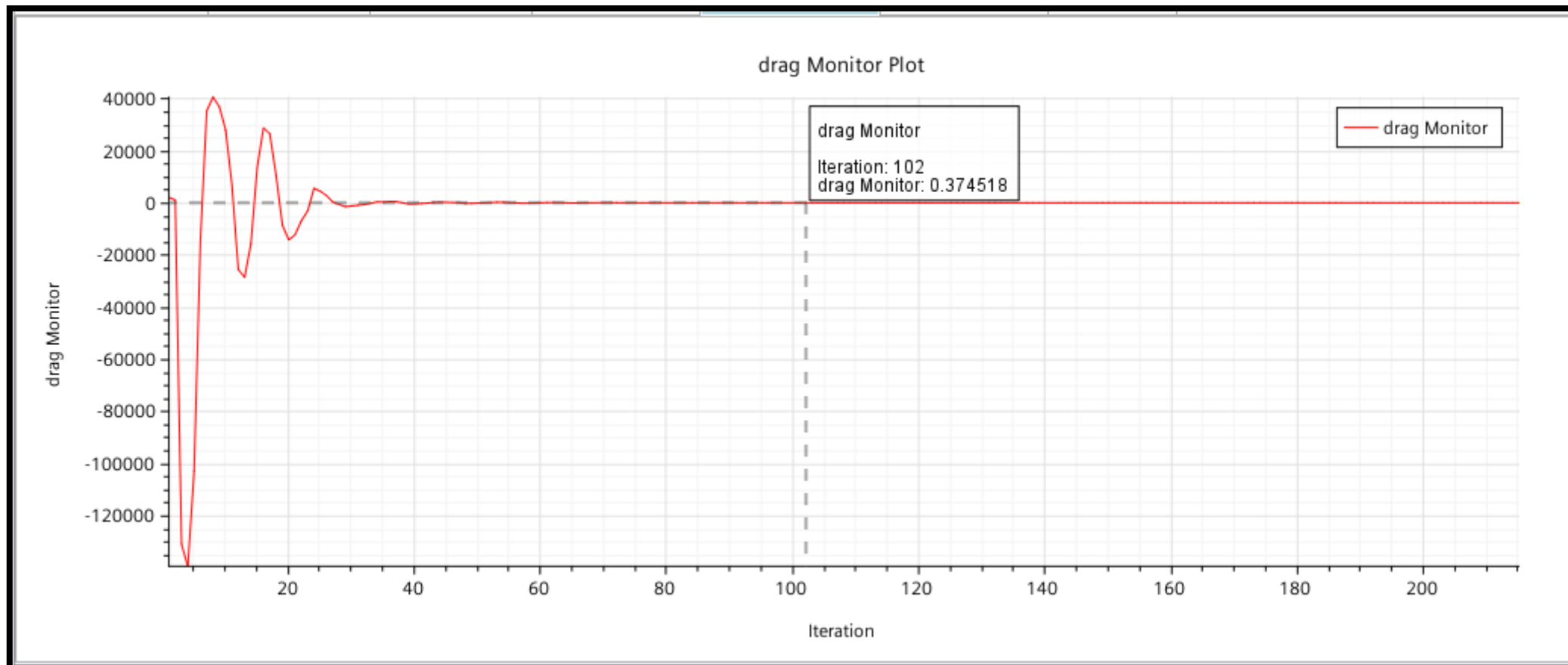
Drag monitor plot



The drag coefficient is computed as $C_D = \frac{F_x}{\frac{1}{2}\rho v^2 A}$

Residuals





Star 1 ×
star2 ×
drag Report ×

Totals: [1.232677e-01, -7.722705e-05, 1.961014e-01] [3.500414e-03, 1.311149e-04, 5.335054e-04] [1.267681e-01, 5.388785e-05, 1.966349e-01]

Component in direction: [1.000000e+00, 0.000000e+00, 0.000000e+00] in Laboratory coordinate system

Part	Pressure()	Shear()	Net()
Surface Wrapper.put.Surface	1.232677e-01	3.500414e-03	1.267681e-01
Totals:	1.232677e-01	3.500414e-03	1.267681e-01

Monitor value: 0.12676807643645835

Lessons Learned

- This project exposed me to CFD simulation principles currently used in the industry.
- Now I have a better understanding of fluid dynamics and heat transfer.
- Able to conceptualize a CFD problem.
- Learnt the process of mesh generation.
- Able to define and solve a fluid flow CFD problem.
- Use the results for making design decisions.

Thank You

