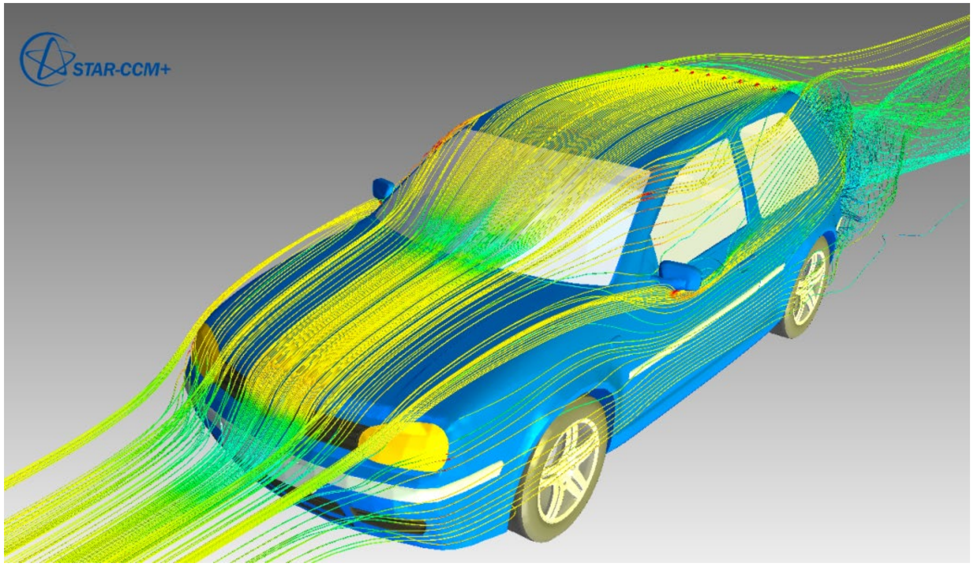
PREDICTION OF AERODYNAMIC PERFORMANCE OF AN AFTERMARKET VORTEX STABILIZER

PREPARED FOR:
CHESS & EVANS



PREPARED BY:
ANALYSIS & DESIGN APPLICATION CO., LTD.

Prepared By: ______________________
Takahiro Sonoda

Approved By: ______________________
Dan Spuckler

Approved By: ______________________
Robert Lotz

CD-adapco
adapco
60 Broadhollow Road
Melville, NY 11747, USA
www.cd-adapco.com

Contents

Disclaimer

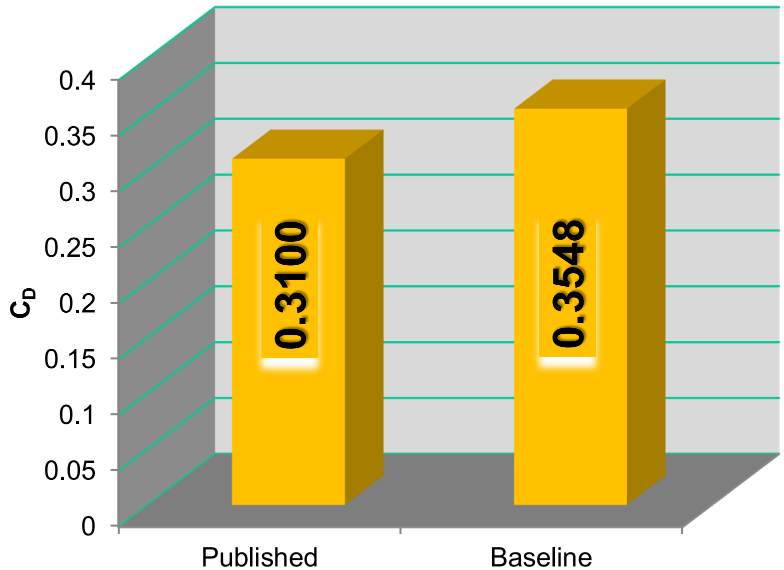
CD-adapco's engineering analysis Results provided herein are dependent on the accuracy of the geometry, boundary conditions, material properties, and other data provided by your Company and may deviate from real physical operation. These Results are not intended as a substitute for sound engineering judgment and CD-adapco is not liable for the outcome of the use of these Results.

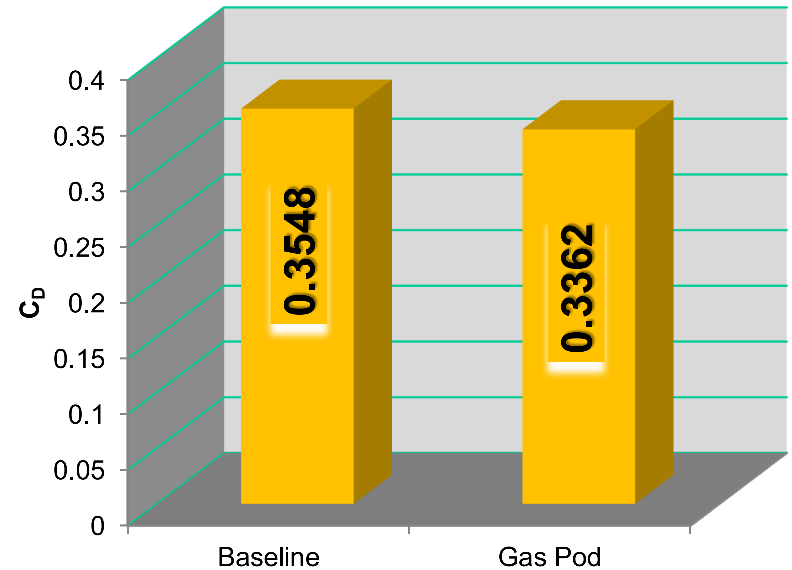
Model Overview

- The objective of this project was to investigate the effect of vortex stabilizers installed on the rear roof of a car for aerodynamic performance in terms of drag reduction.
- A 3D CAD geometry of a production personal vehicle (VW Golf IV two door) was selected and purchased form the website 3DCADBrowser.com, which provides three-dimensional geometries of common vehicles. These surfaces are not provided by the OEM, and as such do not necessarily conform with the actual dimension of the vehicle.
- Two CFD models were created by **CD-adapco** using **STAR-CCM+** version 5.04.008. These models represent a baseline geometry of the vehicle, as well as one geometry with the GasPods.
- Each GasPod was positioned 4 inches apart from each other in the lateral direction and 10 inches upstream from the trailing edge of the roof.
- A half symmetry model was used to simulate the flow field.
- The computational models were generated based on standard mesh settings for external aerodynamic analyses with appropriate refinement particular to this model.
- Analyses were performed for the vehicle speed of 65 miles/hour.

Executive Summary

Key Results: Drag Coefficient

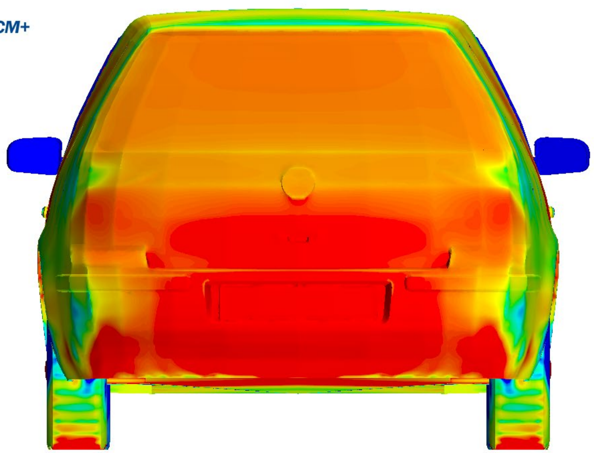


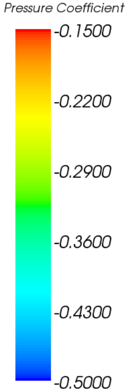


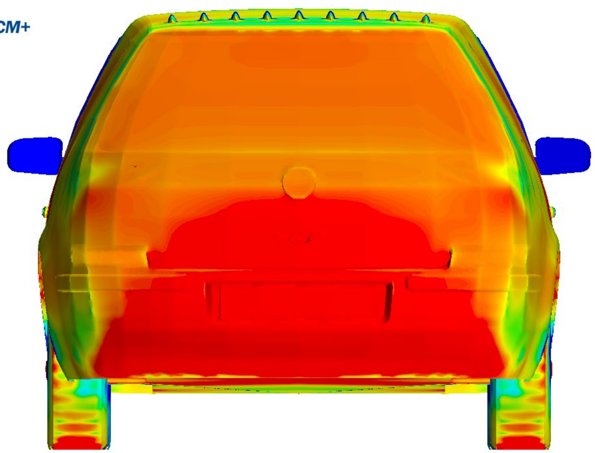
- The drag coefficient predicted from the baseline model was compared against the published data for the vehicle. (See for example http://www.new-cars.com/2003/volkswagen/volkswagen-golf-specs.html)
- The drag coefficient was predicted within about 13%. Considering that the CFD model is not based on actual CAD data, as well being considerably simplified (wheel wells, underbody, missing suspension and brake rotors), this is an acceptable result.
- The GasPods reduce the drag of the baseline car by about 5%.

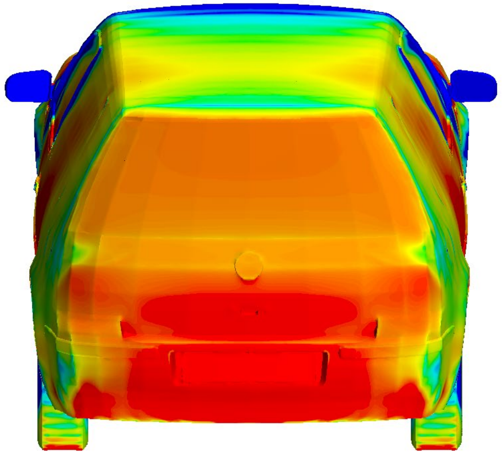
Key Results: Base Pressure Distribution

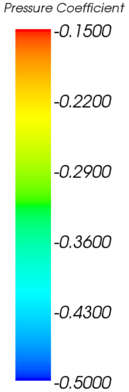
Baseline | Gas Pods Installed

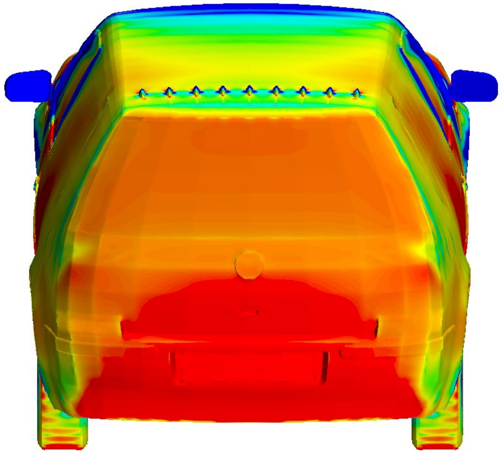








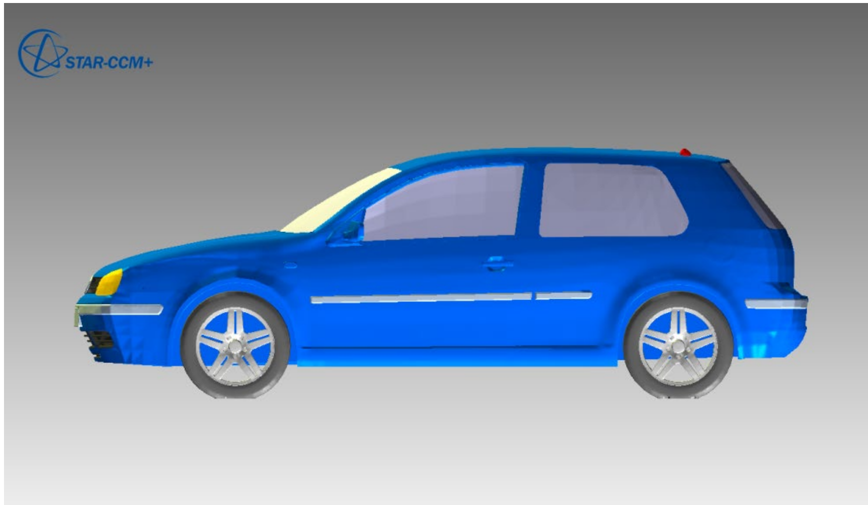


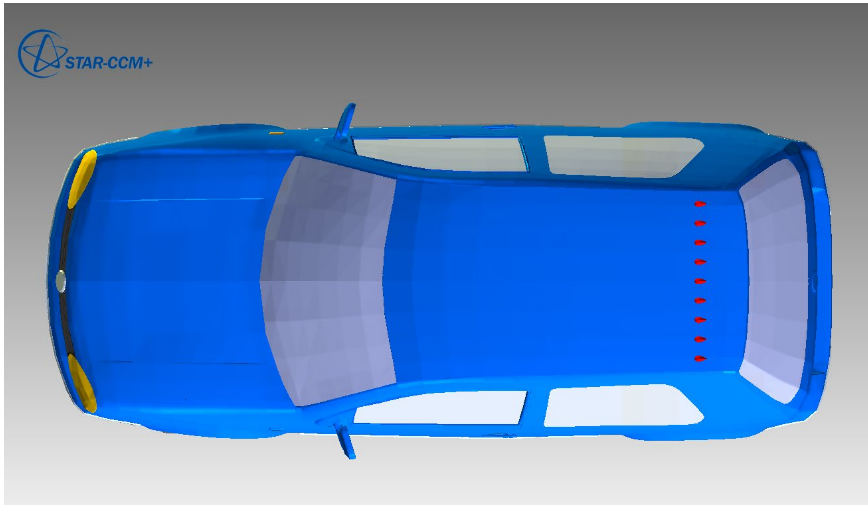


Summary and Discussions

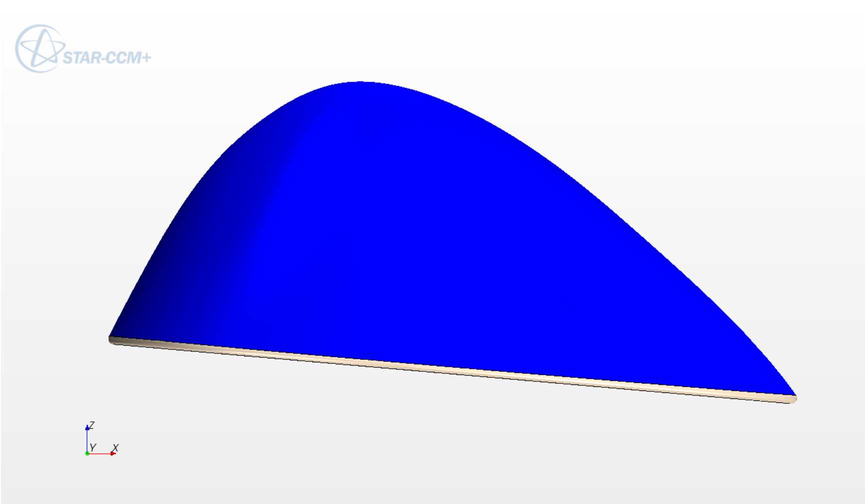
- CFD results show that installing GasPods reduces the drag coefficient by about 0.0186, which is equivalent to 5.24 % reduction in drag.
- Although Gas Pods increase the profile drag, overall reduction in drag by modifying flow structures in the wake layer turns out to be greater than the addition of the profile drag.
- Presence of GasPods in the growing boundary layer decelerates the downstream flow, which results in a pressure rise inside of the shear layer.
- This pressure rise propagates through the shear layer into the base region of the vehicle.
- Presence of GasPods on the roof causes increase in base pressure of the vehicle in the wake region, which results in reduction in drag.
- This analysis demonstrates that change in the flow field, especially pressure in the vicinity of the shear layer that lies between the free stream and the re-circulation region can decreases drag on the vehicle.

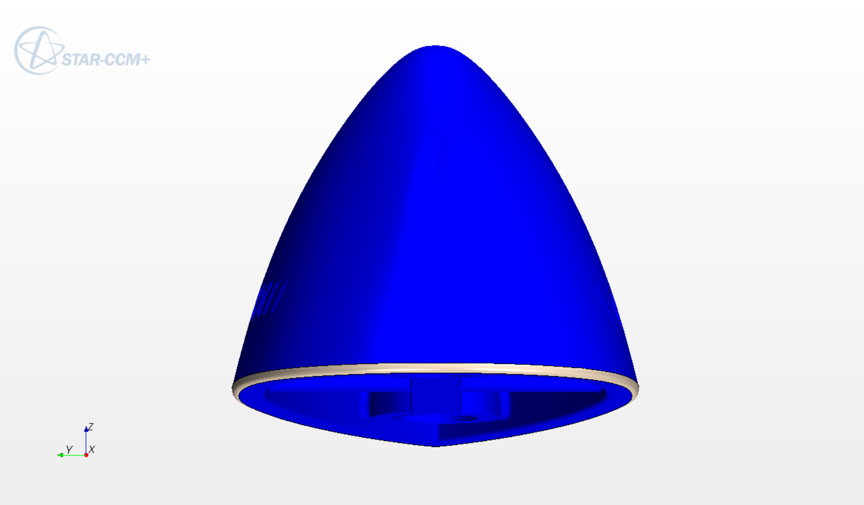
Model Overview: Selected Vehicle

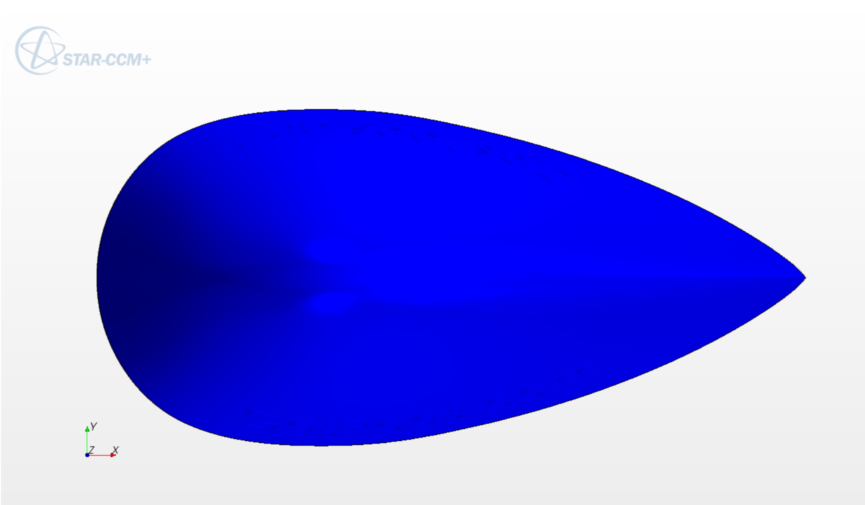




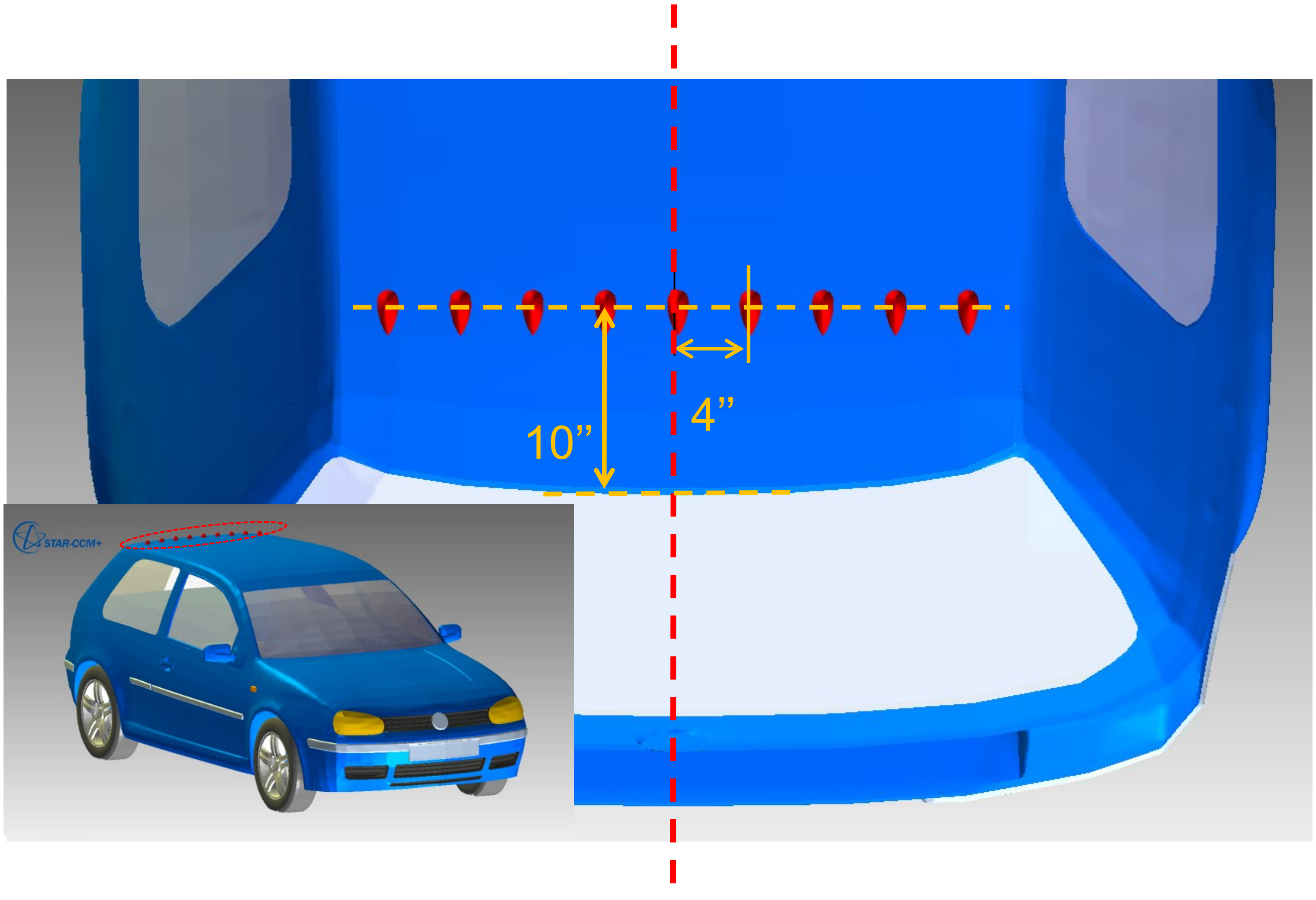
Model Overview: GasPod Geometry



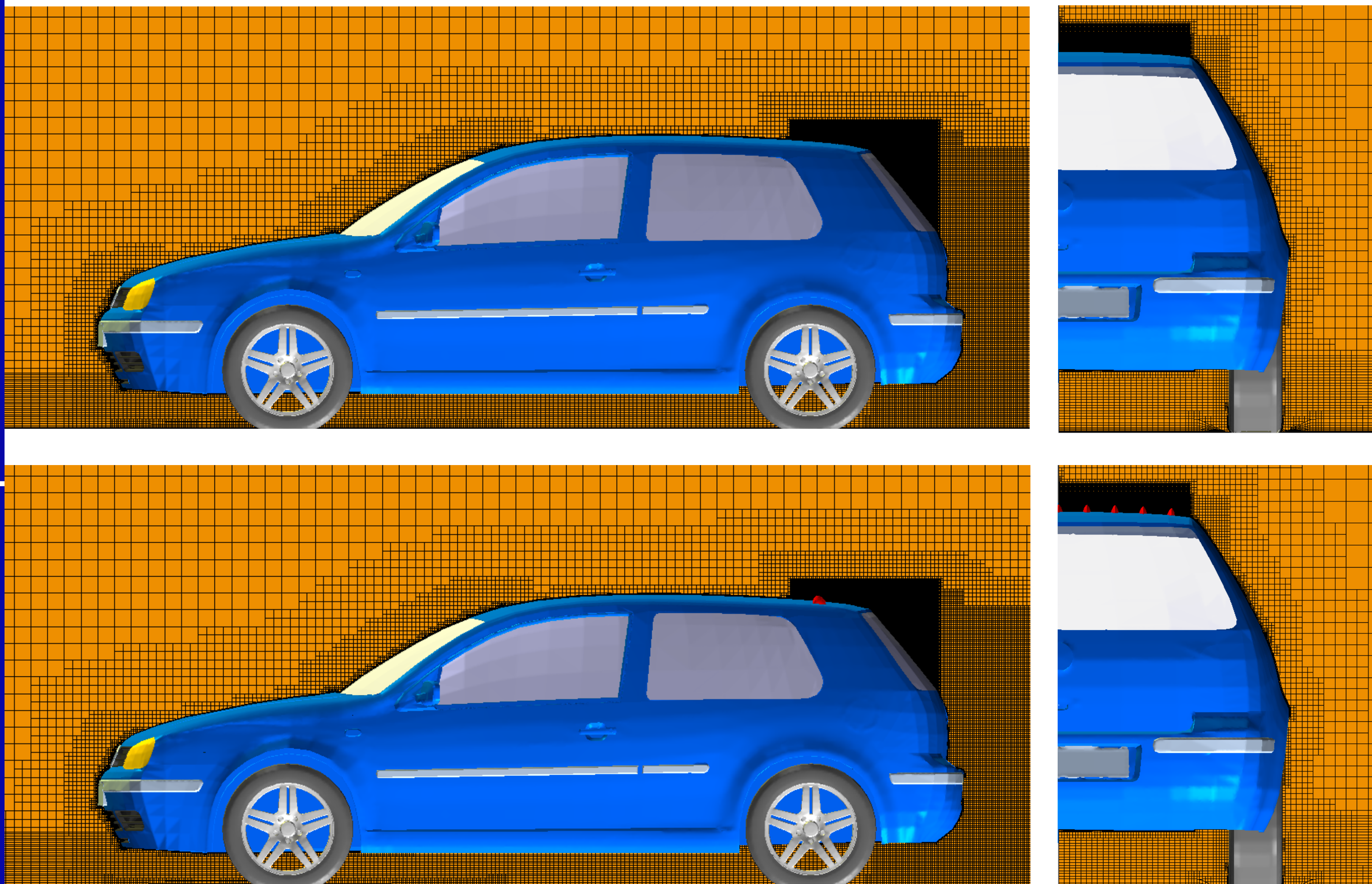




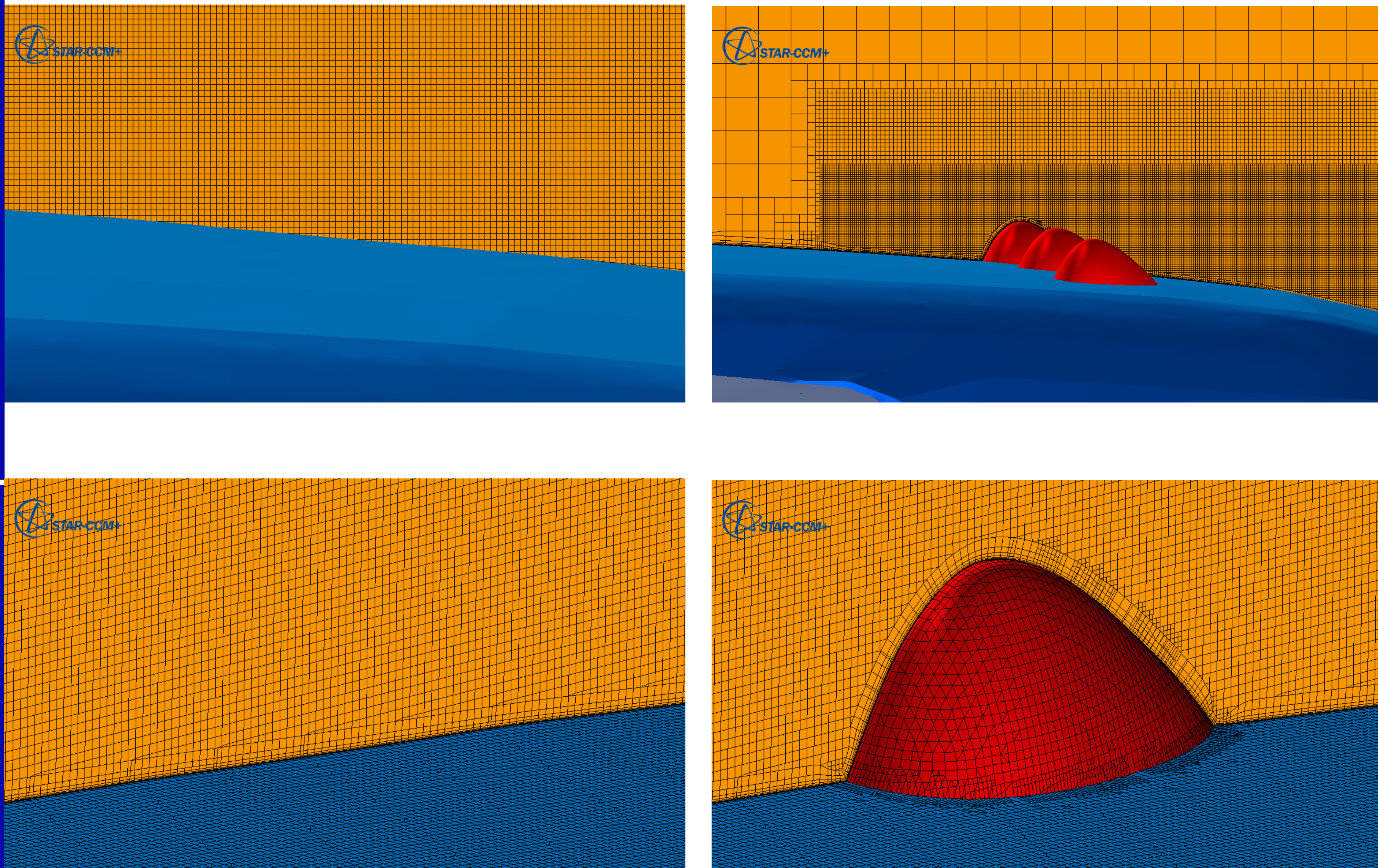
Model Overview: Position of Gas Pods



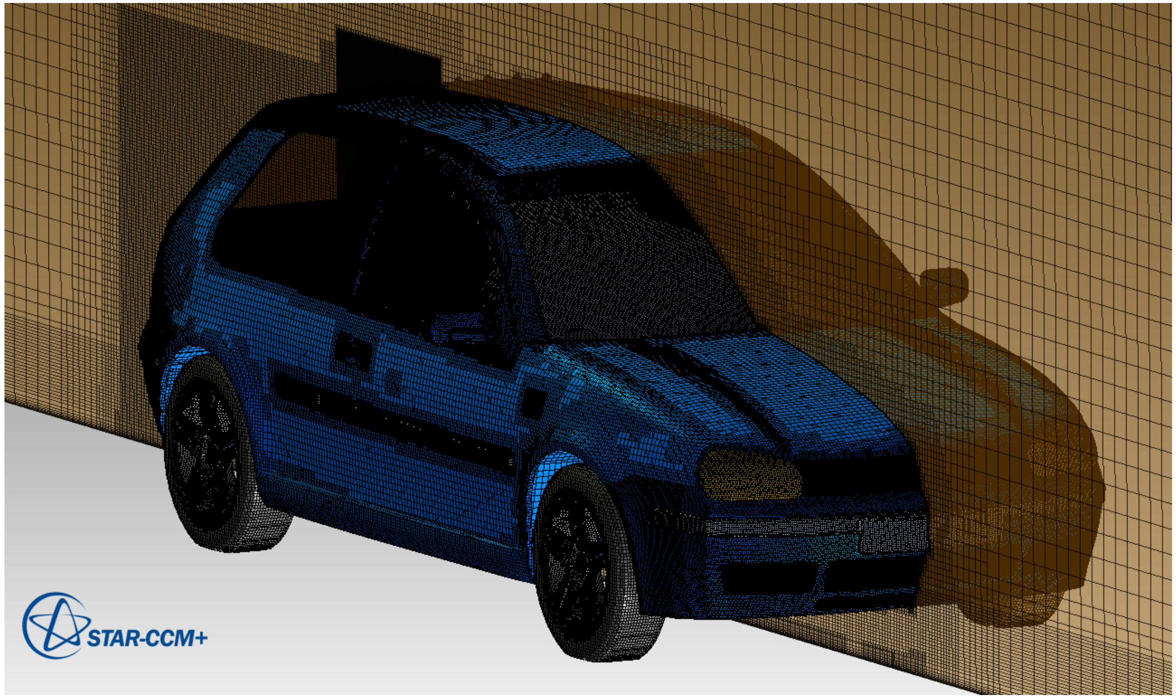
Computational Mesh



Computational Mesh



Computational Mesh

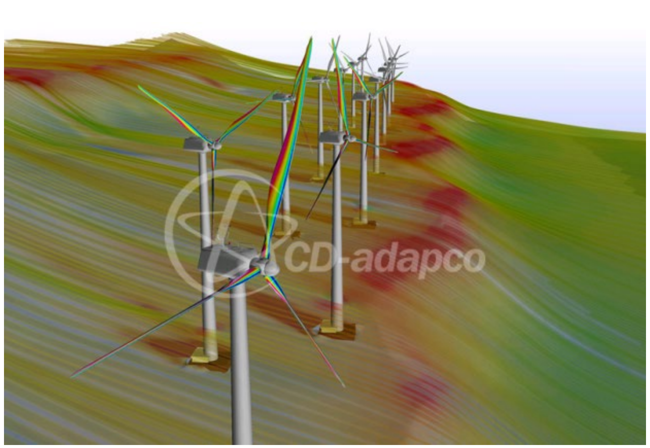


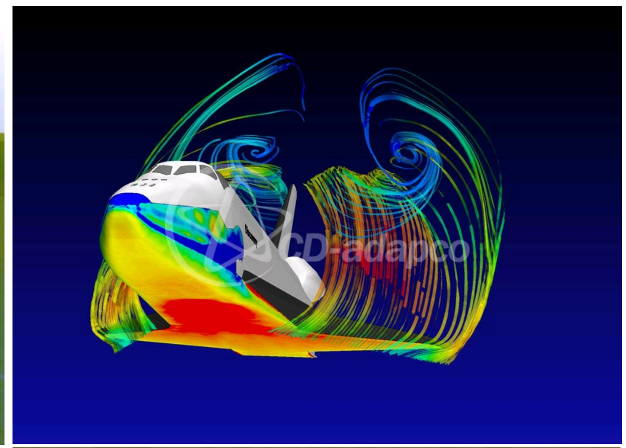
	Baseline	Gas Pods
Cell counts	23.67 millions	23.82 millions

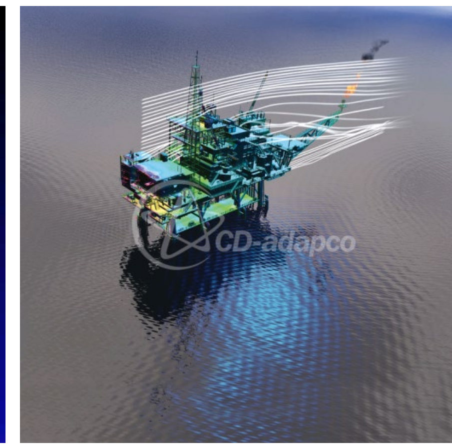
Base size		20 mm
Prism Layer Thickness		5.47 mm
Near wall thickness		0.002 mm
# of prism layers		12
Surface size	Minimum	10 mm
	Target	20 mm

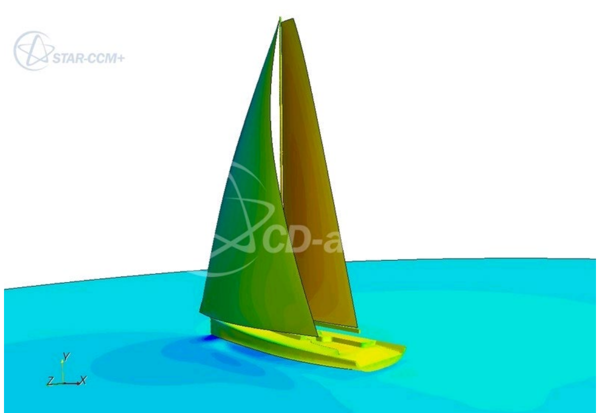
Computational Fluid Dynamics (CFD)

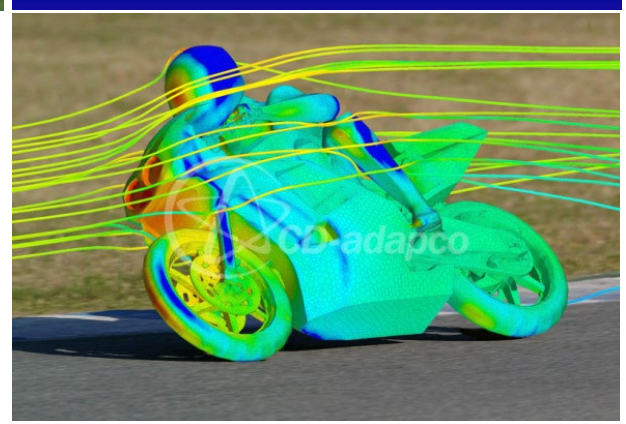
- Computational Fluid Dynamics is an analysis tool to predict physical fluid flow and heat transfer using computational methods
- CFD is widely used in engineering community to improve engineering design and to reduce the overall cost and time of product development

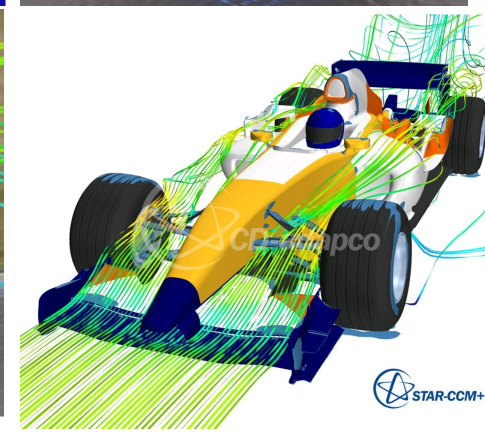












Drag and Drag Coefficient

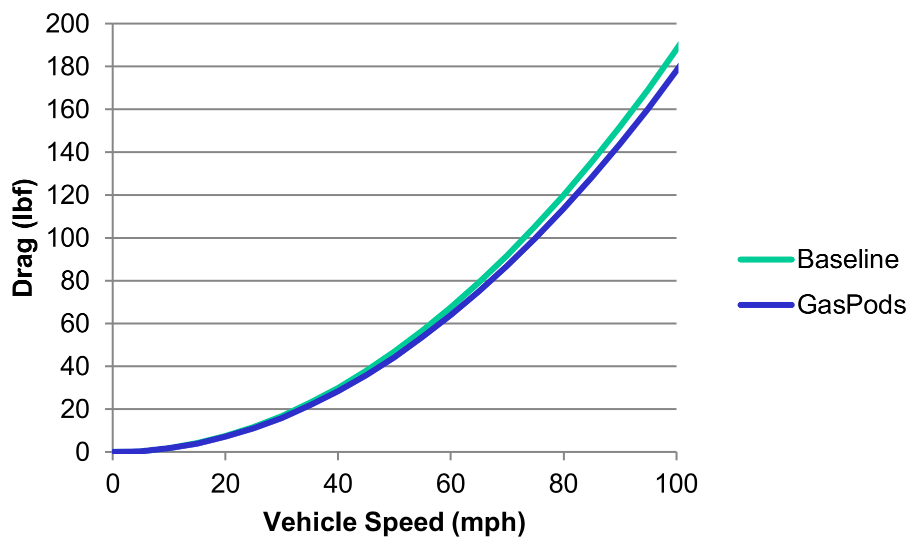
- A drag force is a component of the resultant force on an object that is parallel to the freestream.
- A drag force on a vehicle consists of two contributions: pressure (or form) drag and skin friction drag.
- Pressure drag is the stream-wise component of the pressure force integrated over the entire vehicle while skin friction drag is the stream-wise component of the shear force (friction between the air and vehicle surface) over the vehicle.
- Aerodynamic performance of various vehicles is often described by a drag coefficient which is independent of the vehicle speed and shape (area). The drag coefficient of a vehicle remains close to constant over the range of typical highway speeds (45 mph to more than 100 mph)
- A drag coefficient is defined by the following formulae

$$C_D = \frac{D}{\frac{1}{2}\rho_{ref} V_{ref}{}^2 A_{ref}}$$

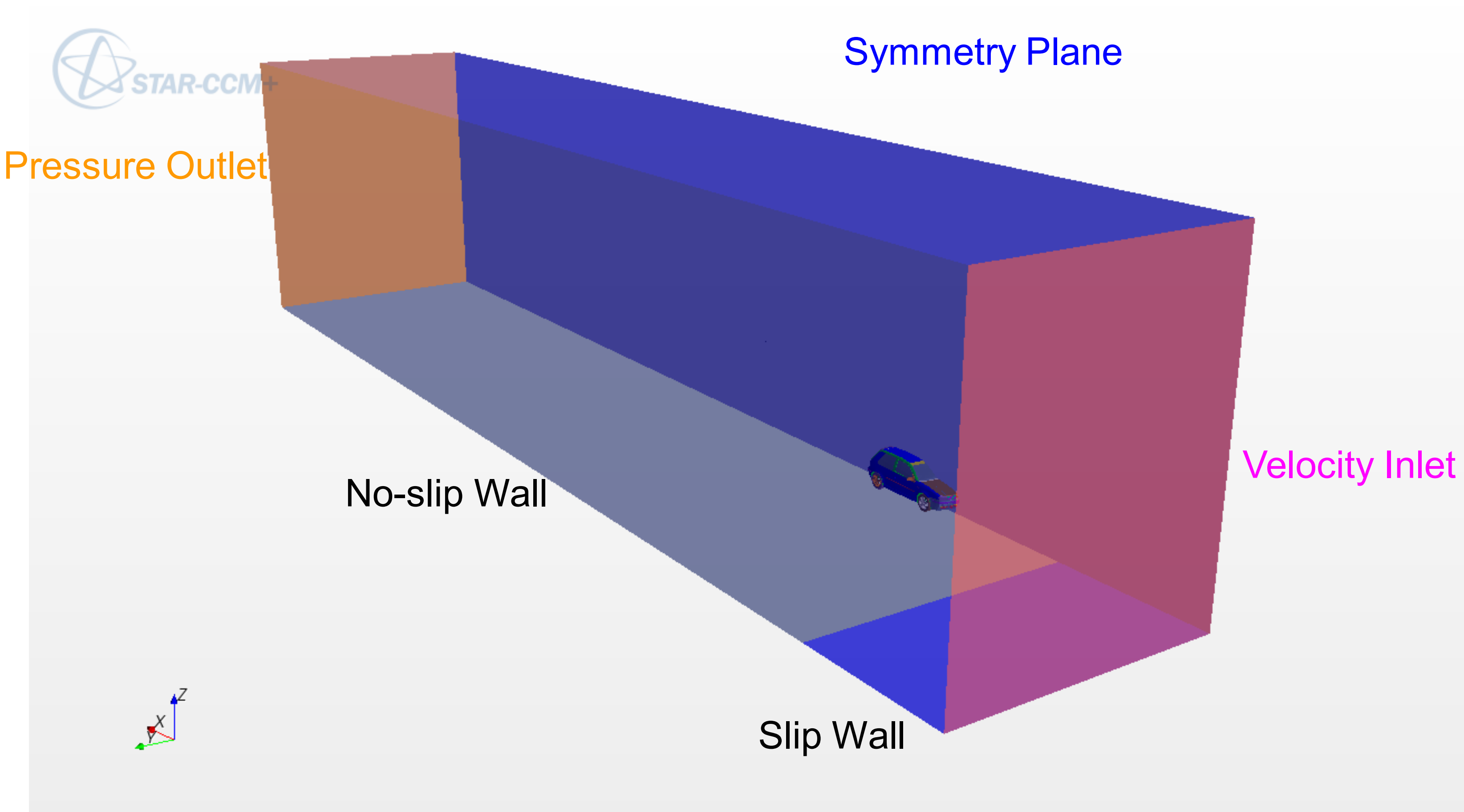
C_D	Drag coefficient
D	Drag force
ρ_{ref}	Reference density
V_{ref}	Reference velocity
A_{ref}	Reference Area (Frontal area of the vehicle)

Drag and Drag Coefficient

- As an advantage of expressing a drag force in terms of a drag coefficient, comparison of aerodynamic performance can be made regardless of the vehicle size, vehicle speed, and altitude at which the vehicle is going to be operated
- Since a drag coefficient remains relatively constant in a typical range of operations, a drag force varies with density (altitude), frontal area of the vehicle, and vehicle speed as shown below
- For a certain vehicle model at an certain altitude, a drag varies only with vehicle speed as shown below



Exterior Boundary Conditions



Description of Analysis

Exterior Boundary Conditions

Front Surface	Velocity Inlet	V = 65 mph P = 0.0 atm
Rear Surface	Pressure Outlet	Extrapolated P = 0.0 atm T = 308 K
Outer Surface	Symmetry	Extrapolated P = 0.0 atm T = 308 K
Ground	Wall	No-Slip
Front Tires (2x)	Moving Wall	No-Slip T = adiabatic ω_{wall} = 2126.28 rpm
Rear Tires (2x)	Moving Wall	No-Slip T = adiabatic ω_{wall} = 2126.28 rpm

Fluid Properties at Inlet

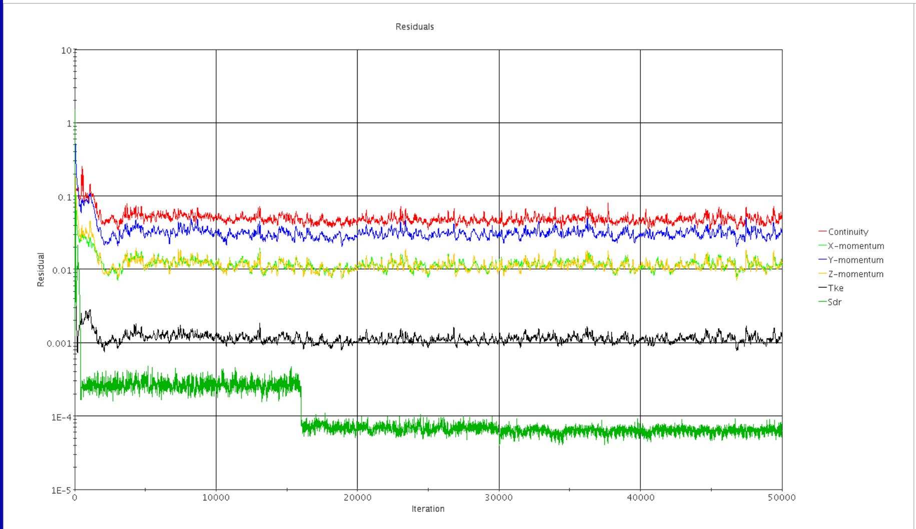
	Inlet Velocity	**65 mph**
AIR	Temperature	Iso-thermal
	Ambient Pressure	1.0 atm
	Dynamic Viscosity	1.86 E^{-5} Pa-s
	Density	1.18 kg/m^3 (4.28 lb/in^3)

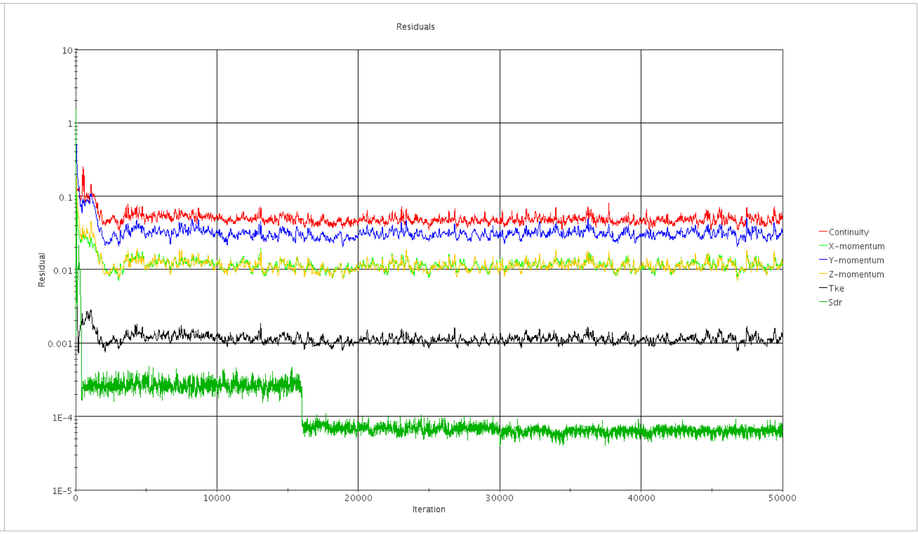
Analysis Setup, Run Statistics

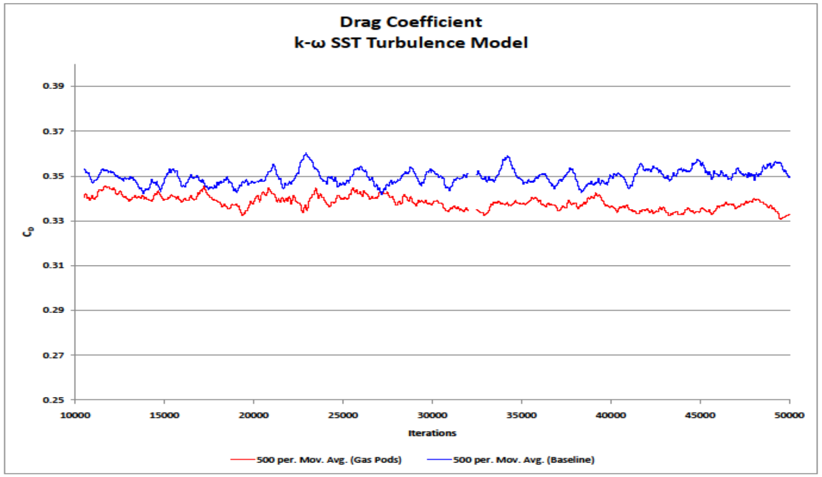
Solution Algorithm		SIMPLE
Convective Flux Differencing Scheme	Velocity	Upwind (2^{nd} order)
	Other Variables	Upwind (2^{nd} order)
Turbulence Model		k-ω SST
Wall Treatment		Two layer all Y+ wall function

STAR CCM+ Version	5.04.008
Computing Platform	LINUX Cluster 64-bit 3.2GHz Xeon Processors
Number of Processors	32
Total number of iterations	50,000
Average Speed	0.14 sec/iteration
Total Analysis Time	8 Days 4.5 hrs
Power Usage	1.00 kW

Convergence History

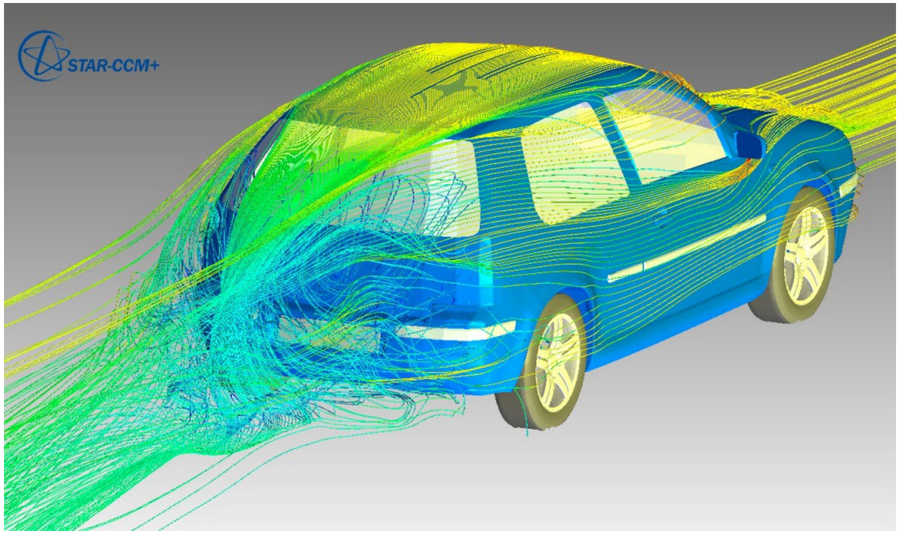


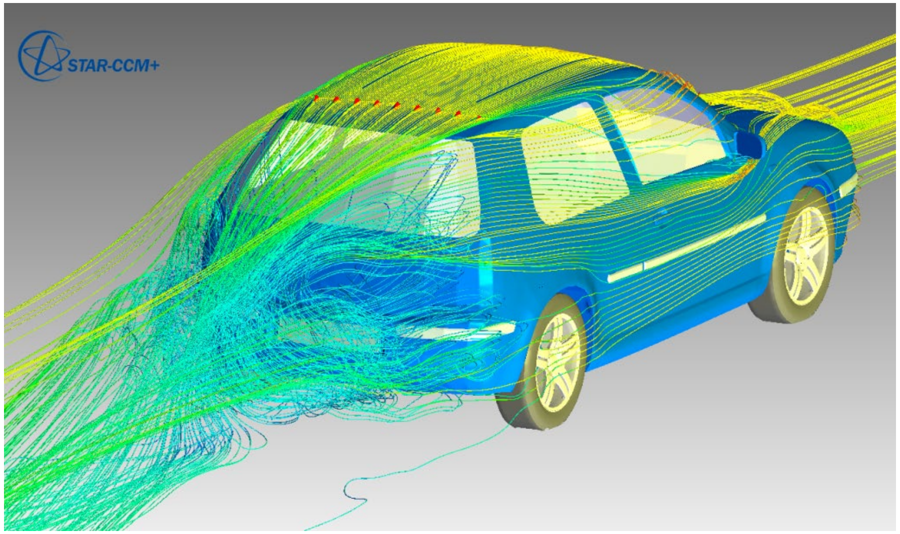


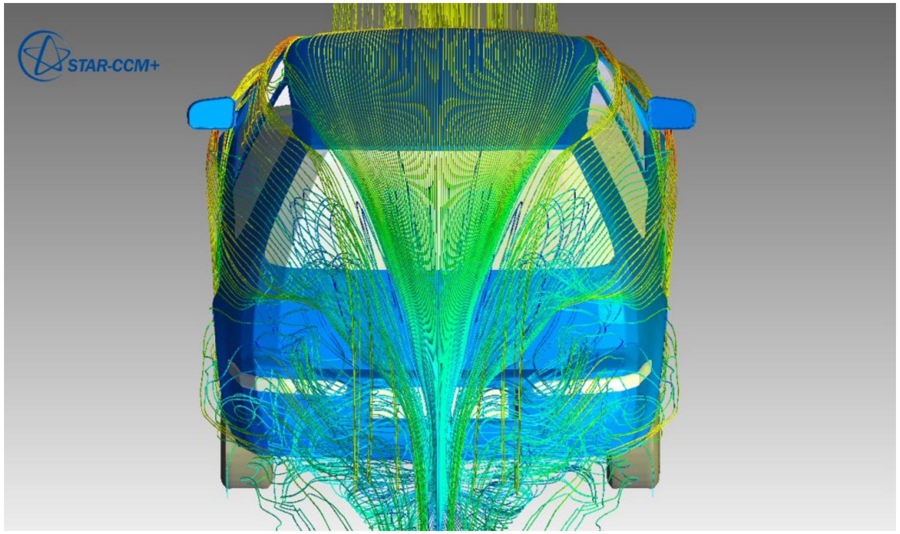


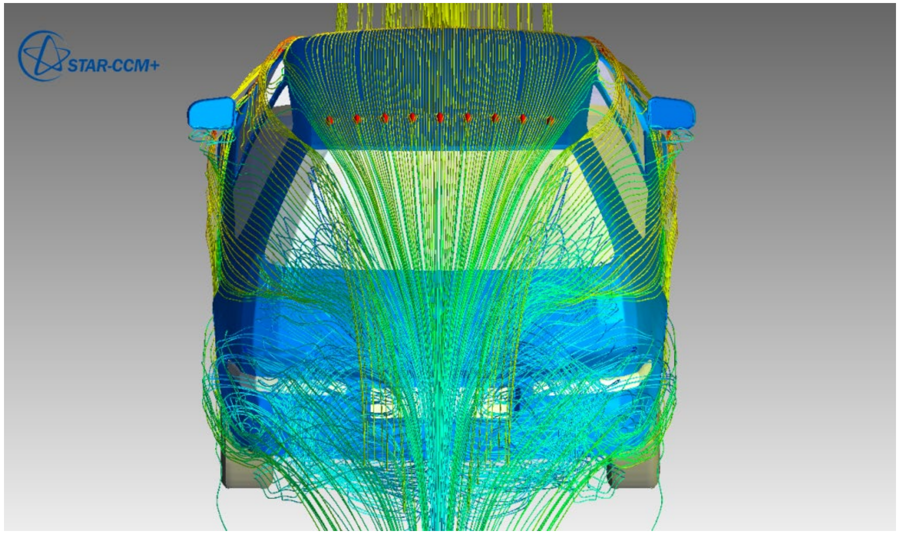
Results

Streamlines

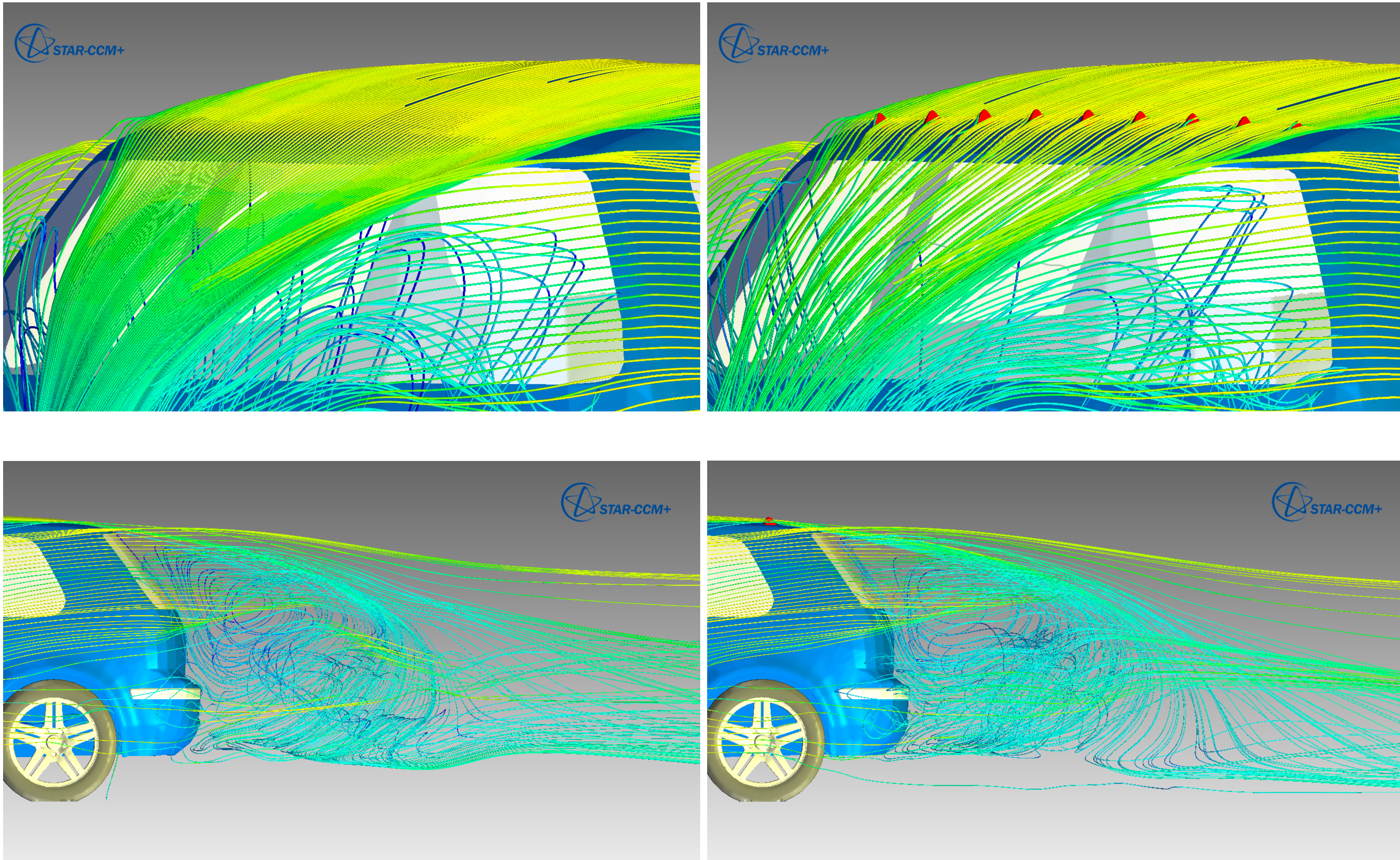




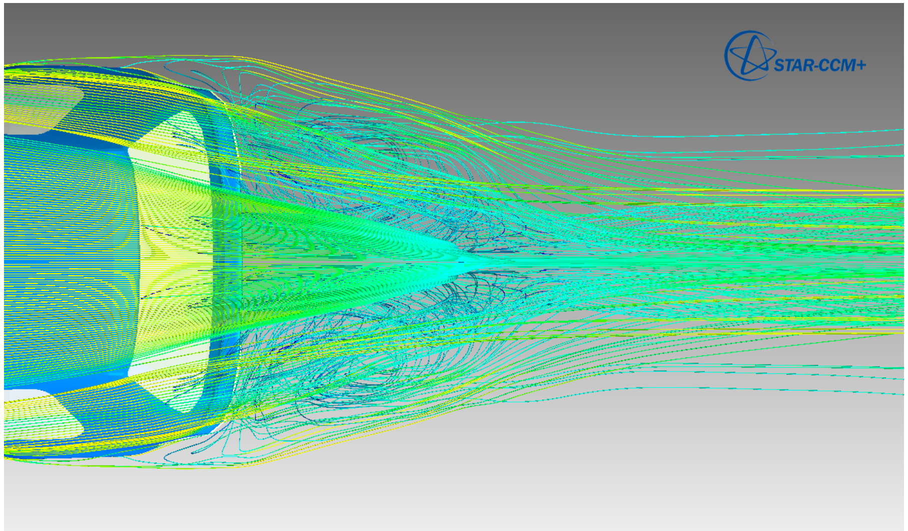


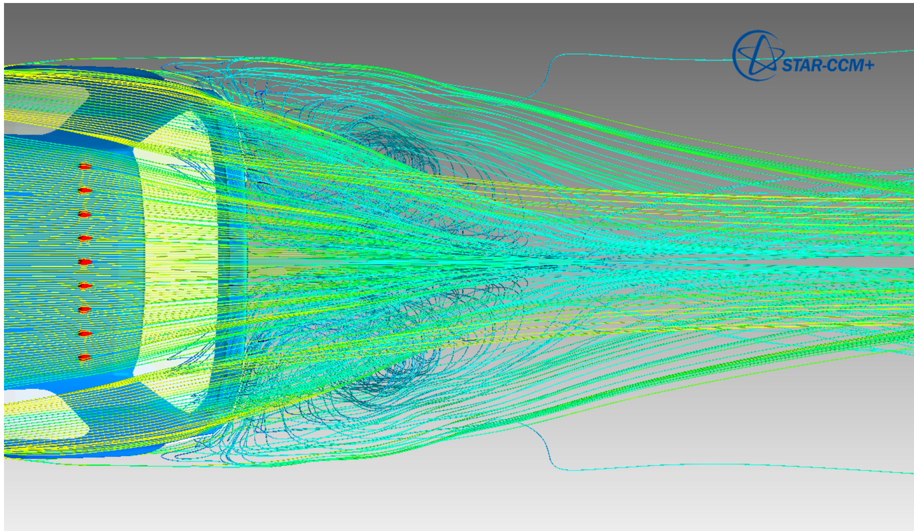


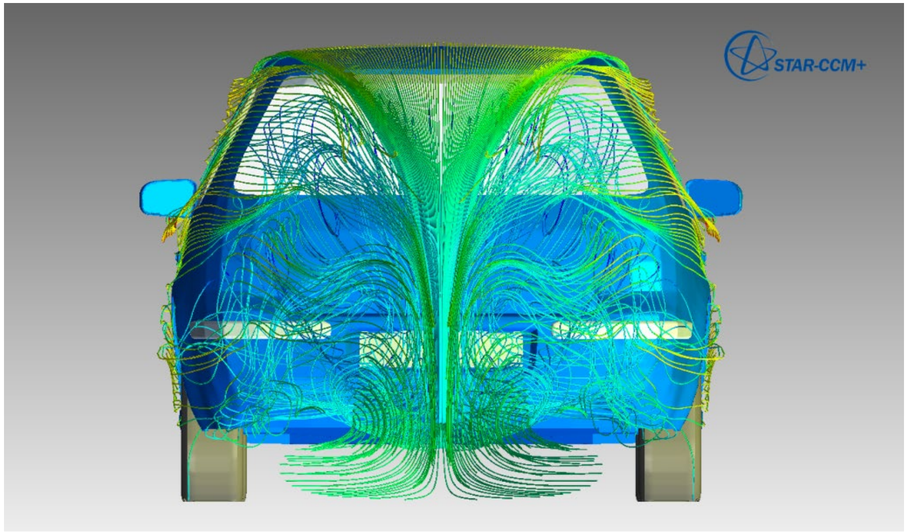
Streamlines

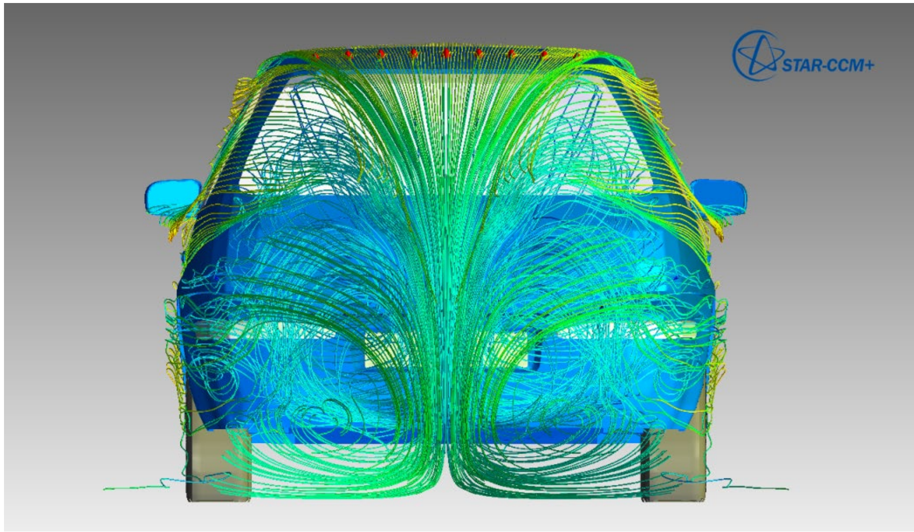


Streamlines

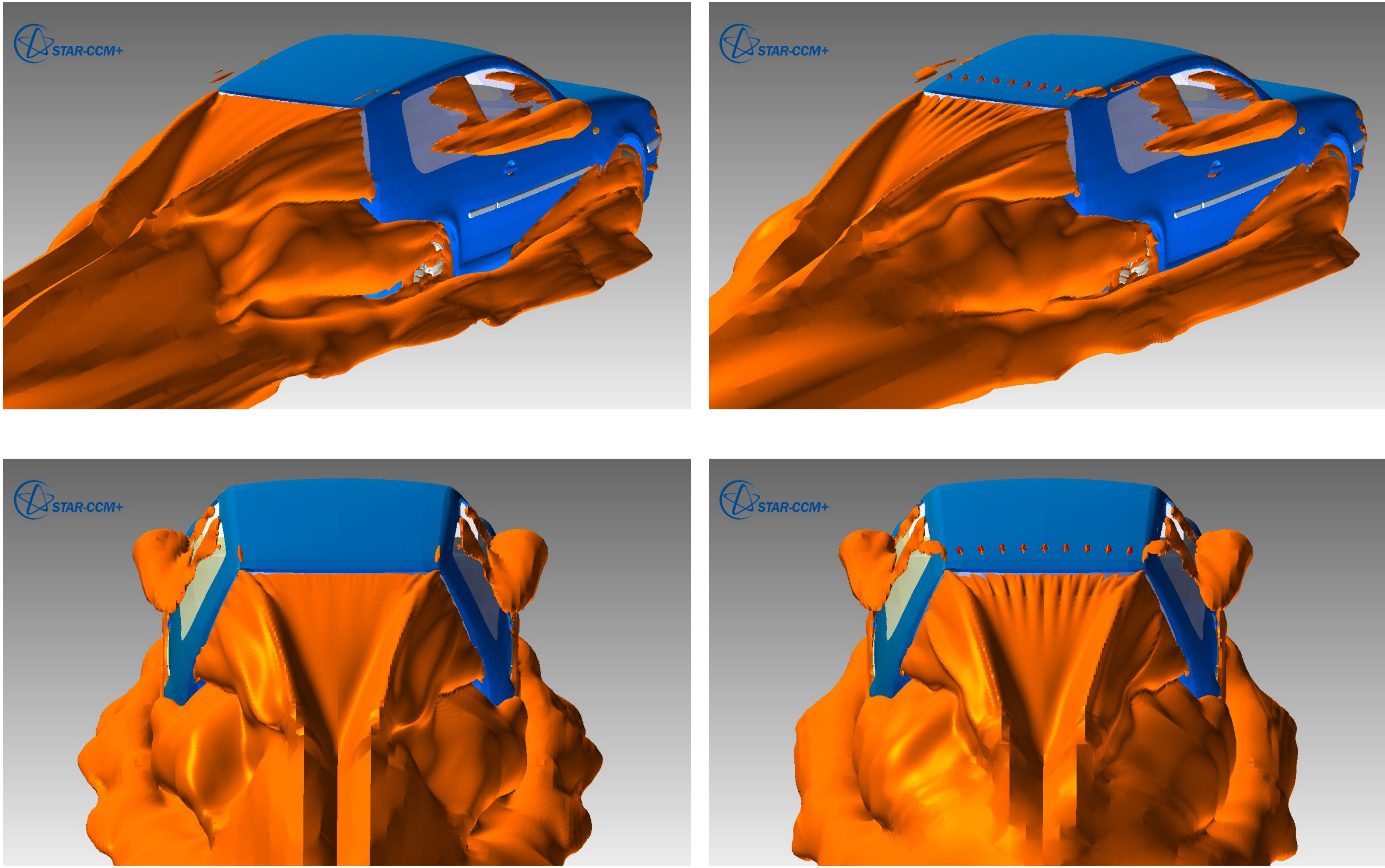




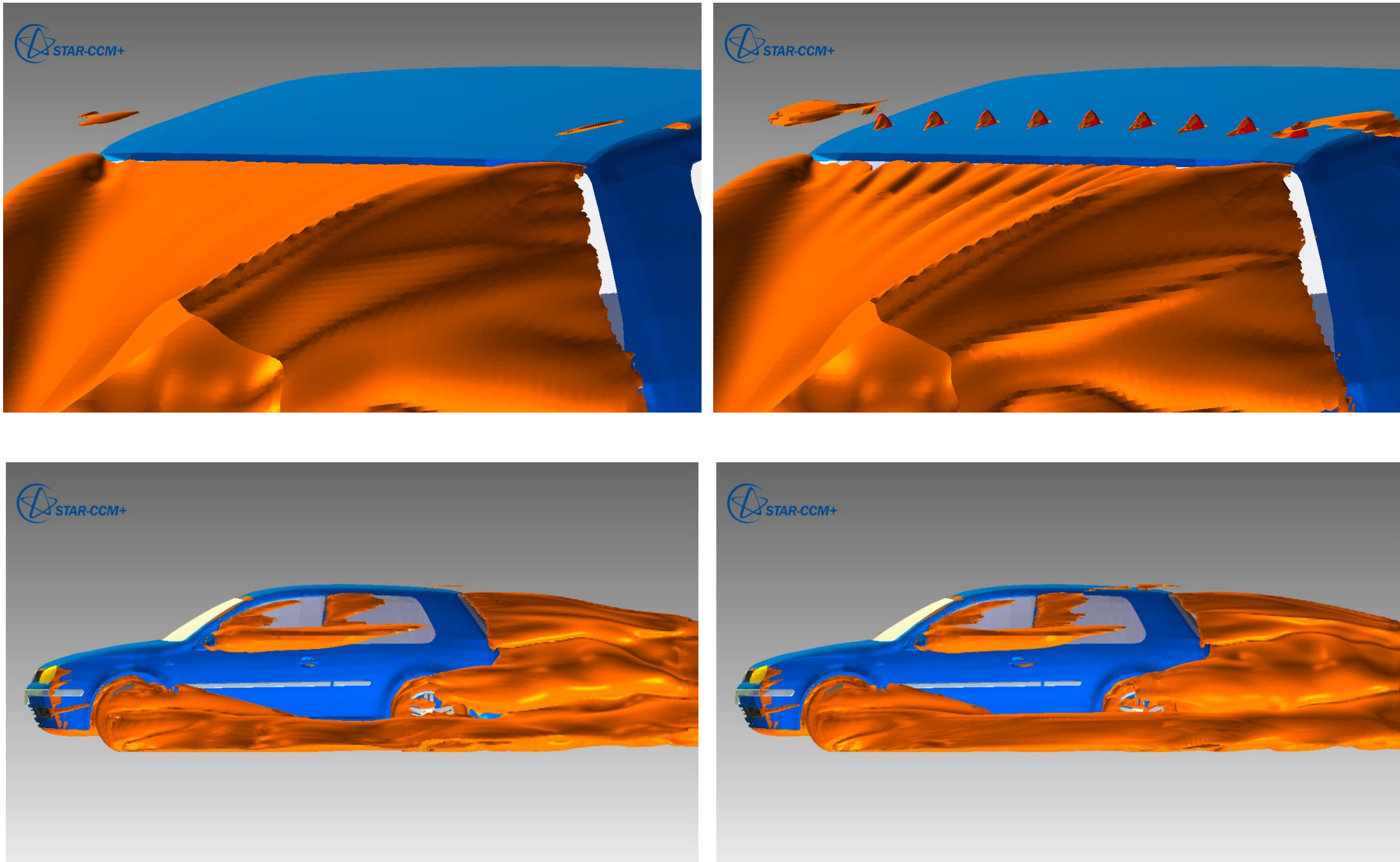




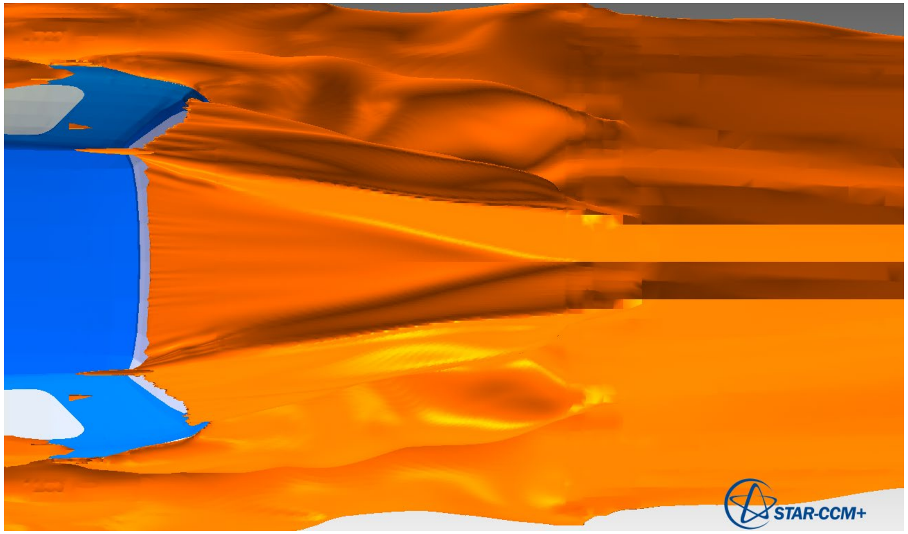
Iso-surface of Turbulence Kinetic Energy

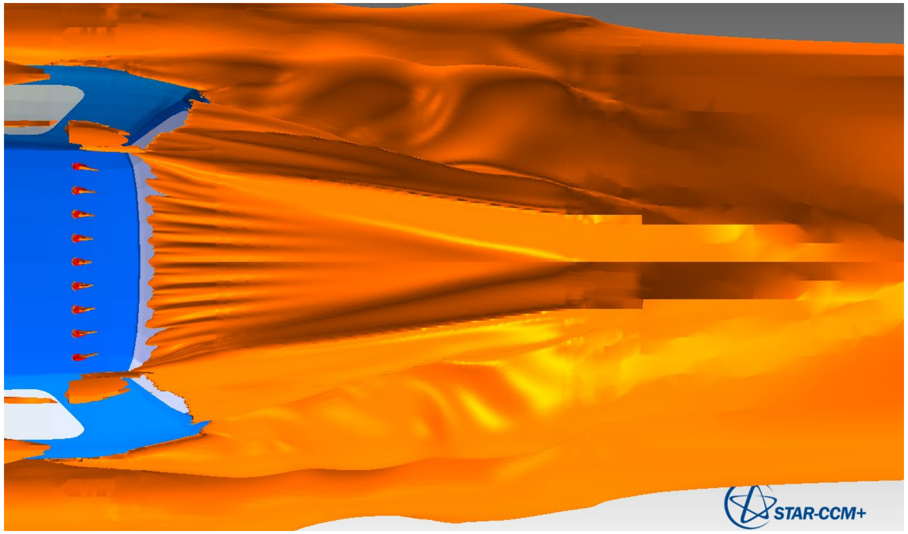


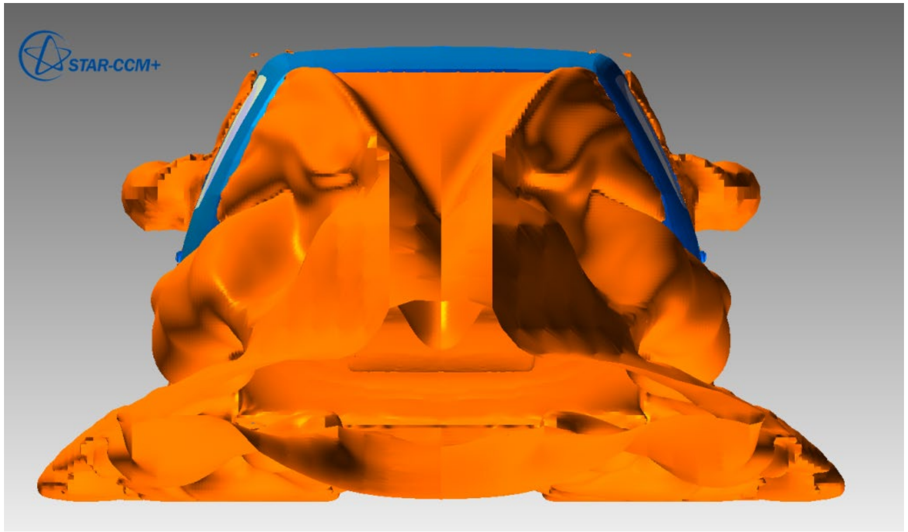
Iso-surface of Turbulence Kinetic Energy

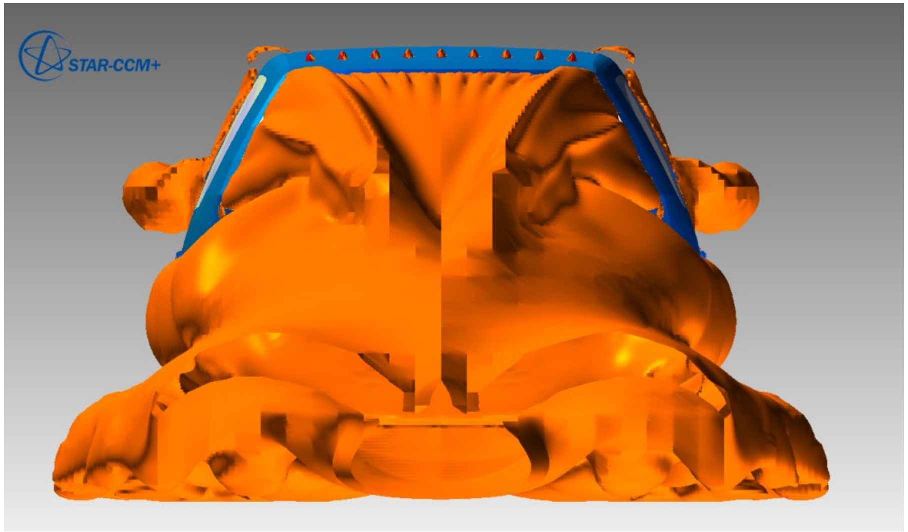


Iso-surface of Turbulence Kinetic Energy

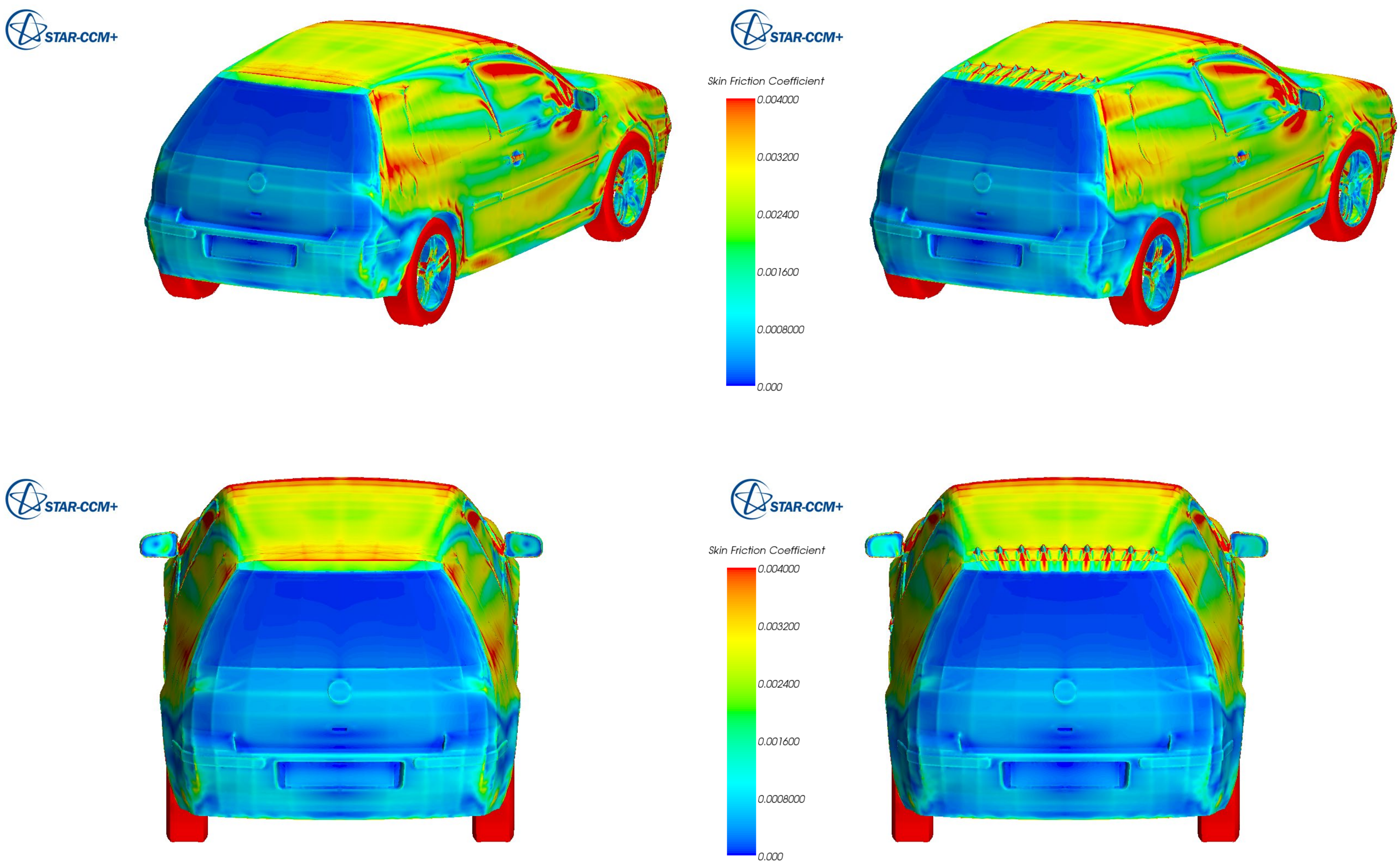






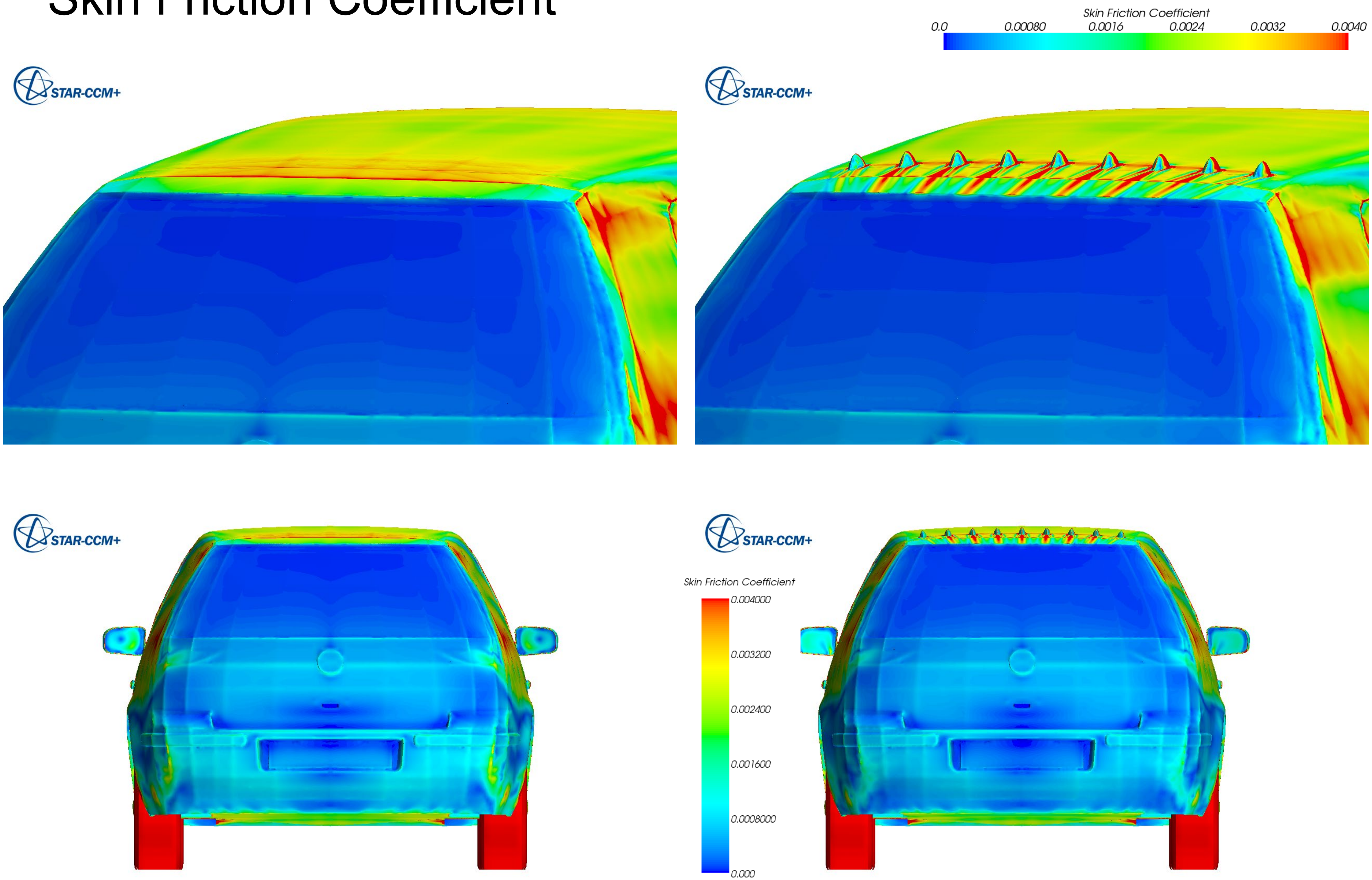


Skin Friction Coefficient



Results

Skin Friction Coefficient



Appendix

Definitions of Variables and Reference Values

- Pressure coefficient is another non-dimensional variable used to represent pressure (force per unit area).

$$C_P = \frac{P - P_{ref}}{\frac{1}{2}\rho_{ref} V_{ref}^{\;2}}$$

- Frontal area is the projected area of the vehicle onto the plane normal to the flow direction.
- The following values were used to convert dimensional variables such as drag, pressure, and shear stress to corresponding non-dimensional variables while reference values are arbitrary.

P_{ref}	Reference Pressure	1.0 atm
ρ_{ref}	Reference density	4.278 lb_m/in^3
V_{ref}	Reference velocity	65.0 mph
A_{ref}	Reference Area (Frontal area of the vehicle)	1515.4 in^2