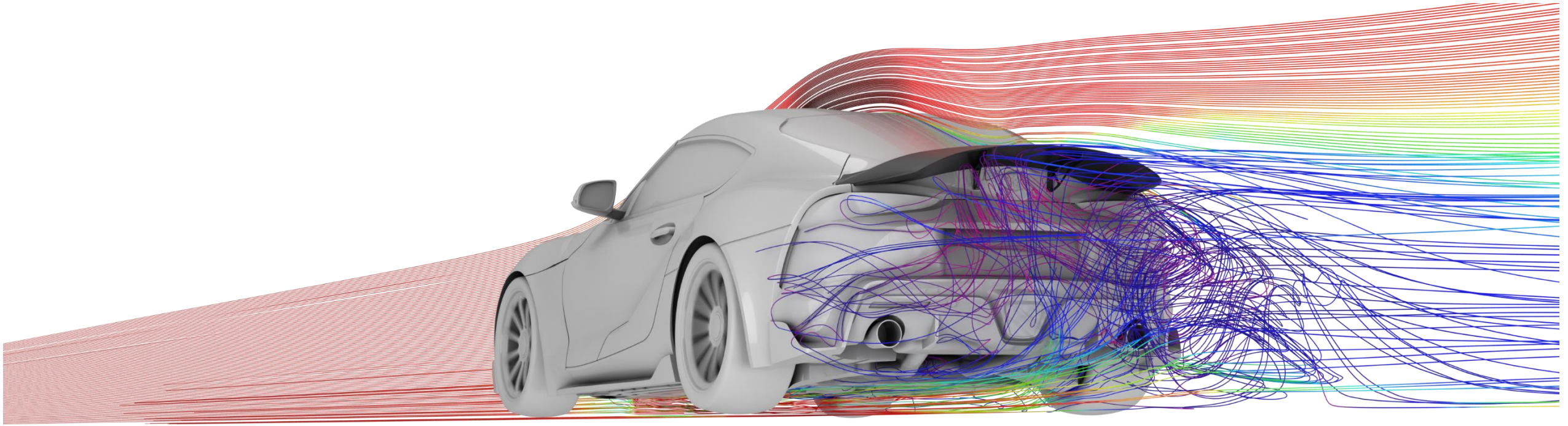


TOYOTA SUPRA (MK5) PLATFORM

PERFORMANCE OF VERUS ENGINEERING DRAG WING

OVERVIEW

DEFINITIONS

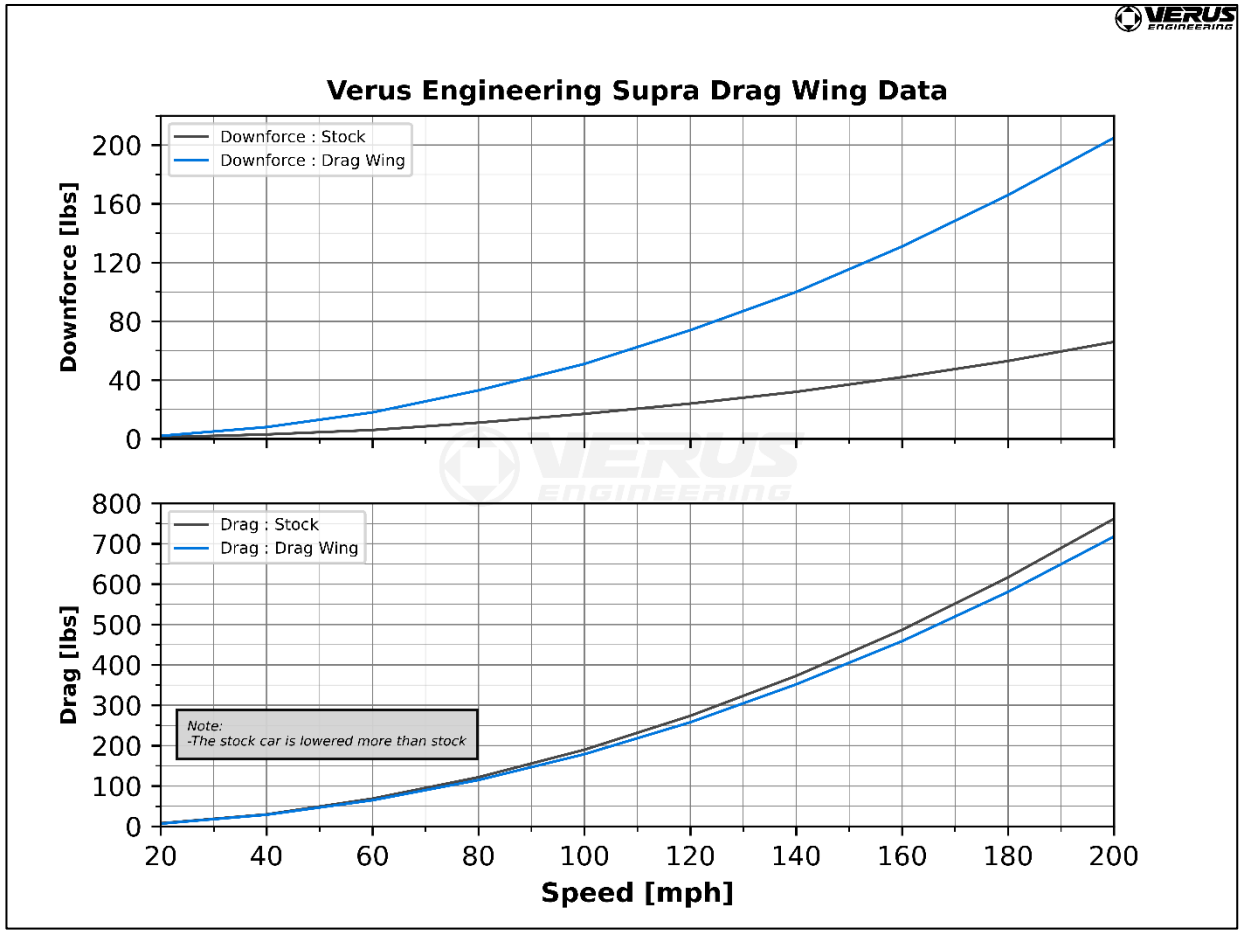
1. **Coefficient of Pressure (Cp)** = This is a dimensionless number which describes relative pressure to atmospheric pressure. A Cp of 0 equates to atmospheric pressure, while a number below 0 represents low pressure, and a number above 0 represents high pressure.
2. **CpX** = This is a dimensionless number which describes Cp normal to the x-direction. This helps us visualize locations which create drag. Red represents locations which are creating drag, while blue represents locations where thrust is created.
3. **CpZ** = This is a dimensionless number which describes Cp normal to the z-direction. This helps us visualize location which create downforce or lift. Red represents locations which are creating lift, while blue represents locations where downforce is created.
4. **Total Pressure Coefficient (CpT)** = This is a dimensionless number which describes total energy of the airstream. It is the sum of static pressure and dynamic pressure.
5. **Wall Shear** = This is a force per unit area due to fluid friction on the wall. This is used to visualize areas of separation and rapid changes on the surface.
6. **UNear** = Velocity near the surface, specifically 3mm from the surface.
7. **LIC Plot** = Line integral convolution (LIC) is used to visualize “oil” flow on the surface. It is a great way to correlate to flow vis testing and to study the flow on the surface of the vehicle.
8. **Streamline** = These are fluid tracers which are used to visualize where the air is going or coming from. These are normally colored as velocity where red is high-velocity and blue is low-velocity.
9. **Points** = One point is considered 0.001 of a coefficient. This is used in coefficient of drag (Cd) and coefficient of lift (Cl).
10. **CAD** = computer aided design

SUMMARY : AERODYNAMIC FORCES

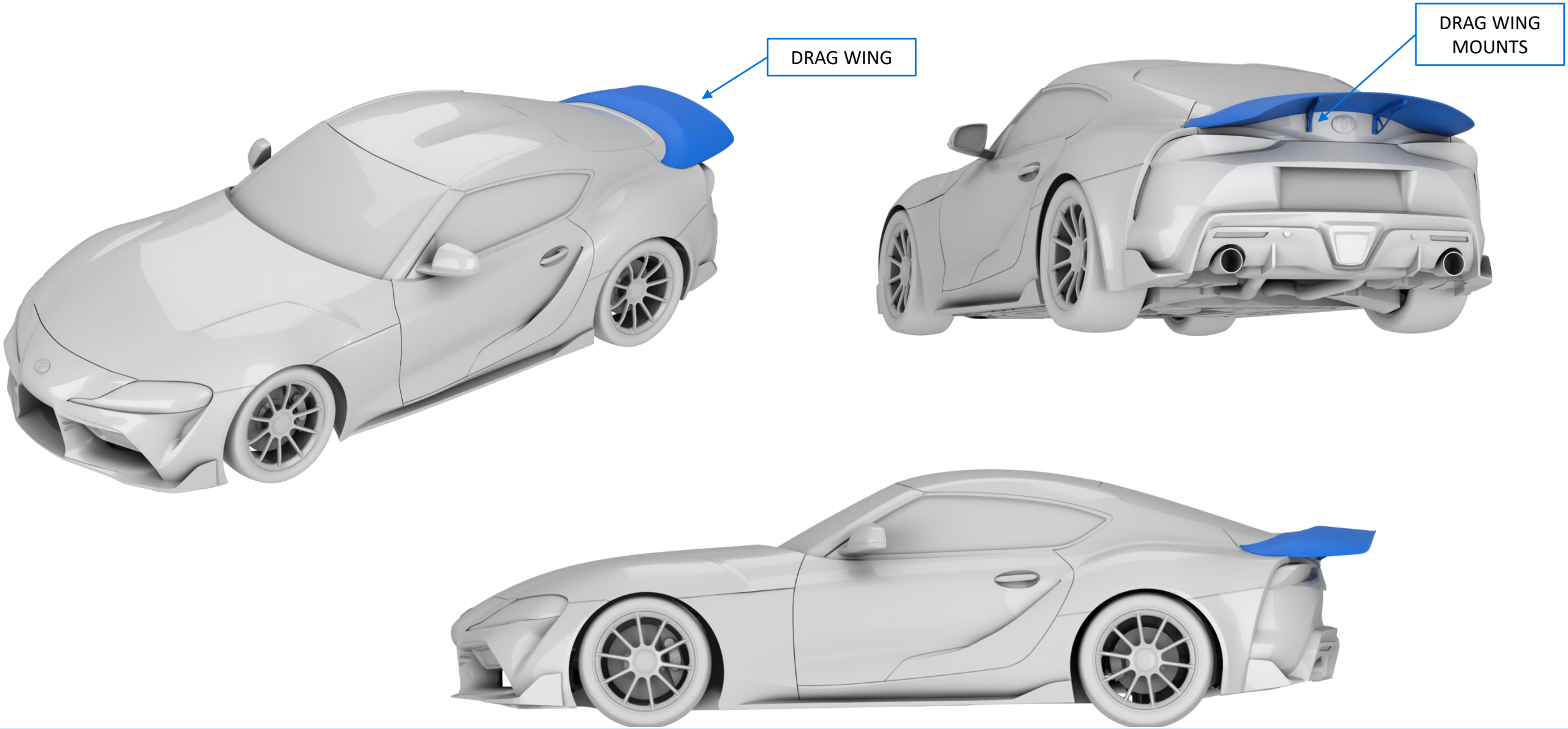
Aerodynamic forces change with the square of vehicle speed which is why we share graphs of data instead of listing a force.

When developing the drag wing, the goal was to decrease drag and increase rear downforce. The drag was reduced by 19 points! The downforce gain was even larger, with all the downforce gain being applied to the rear axle. The graph to the right displays total forces on the car and not just the drag wing itself.

The drag wing provides very efficient downforce as the downforce gain also decreases drag. The benefit of this downforce is more tractive force on the rear tires which will increase acceleration, inhibiting the tires from breaking loose. The drag decrease will improve the acceleration AND top speed.

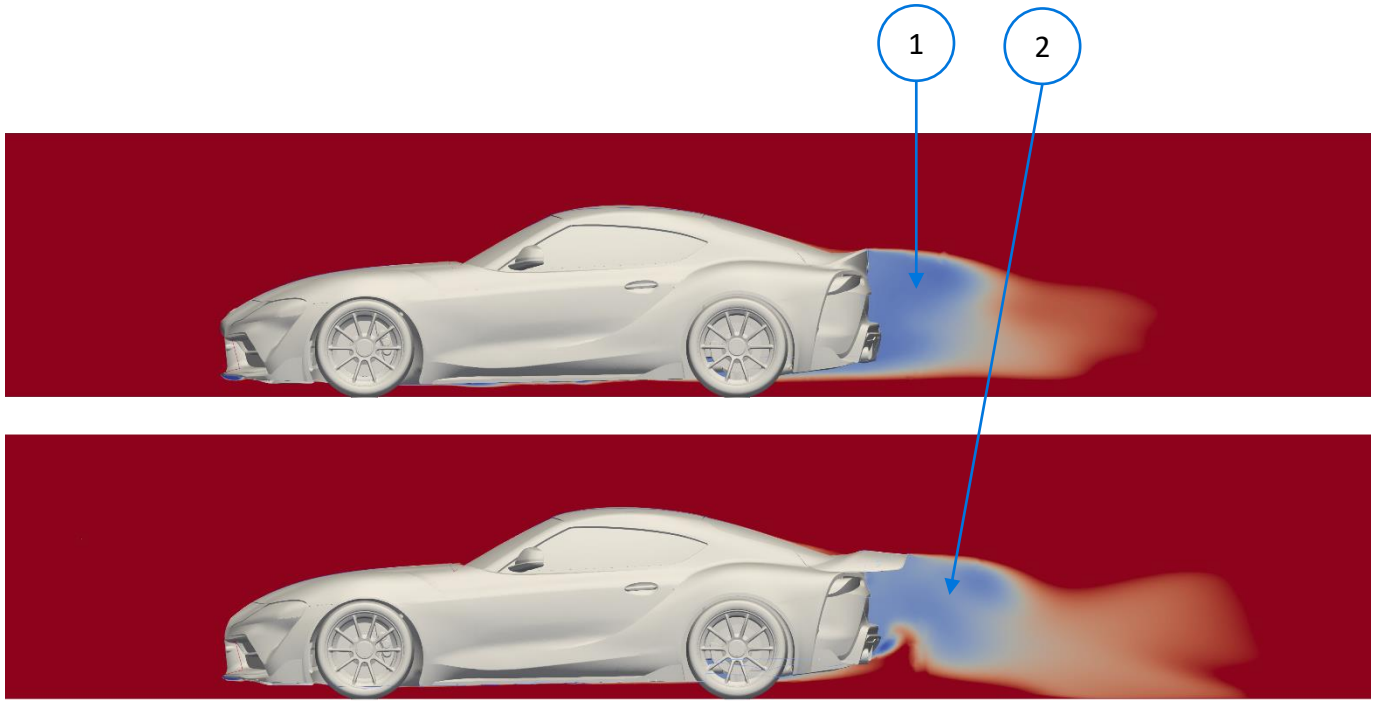


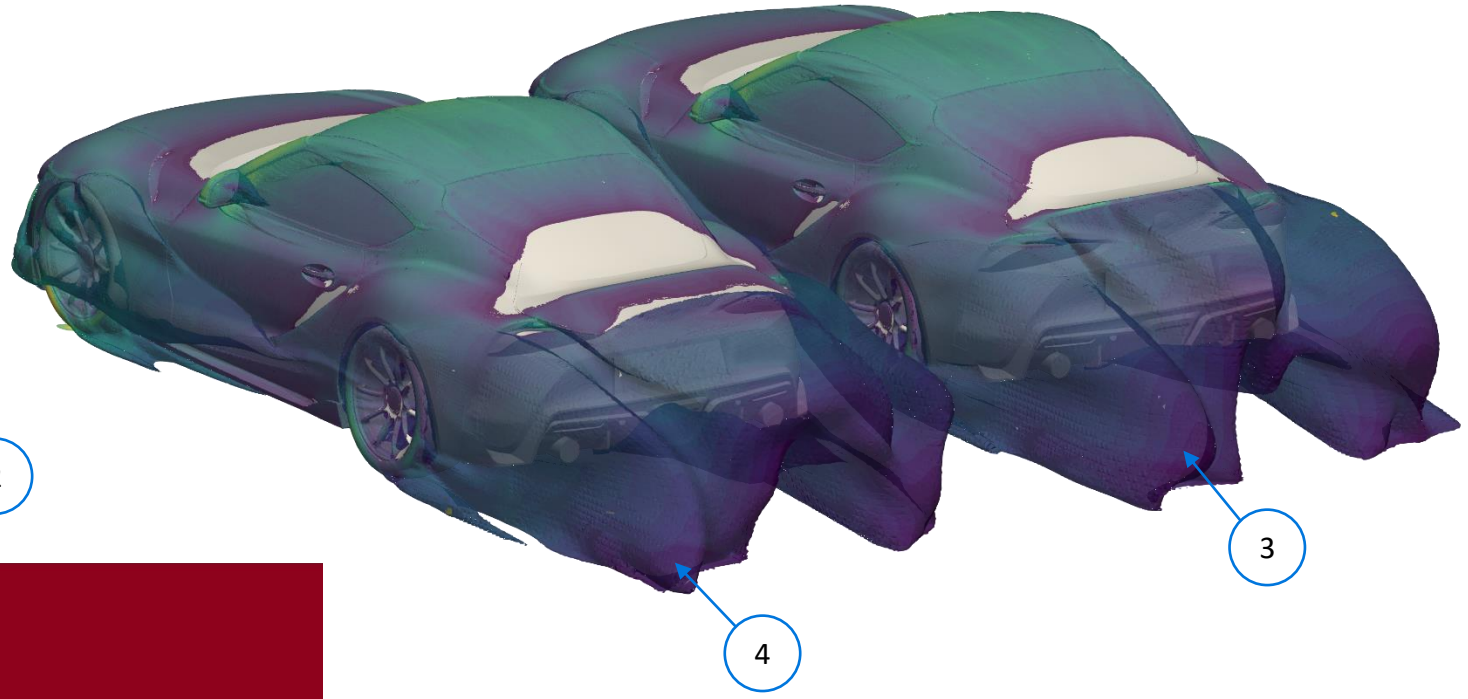
DRAG WING PACKAGE



WAKE

The drag wing reduces the wake and energy of the wake. Number 1 shows the larger and darker blue wake than number 2 (Drag Wing). The darker blue means lower pressure which is acting to pull the car rearward.

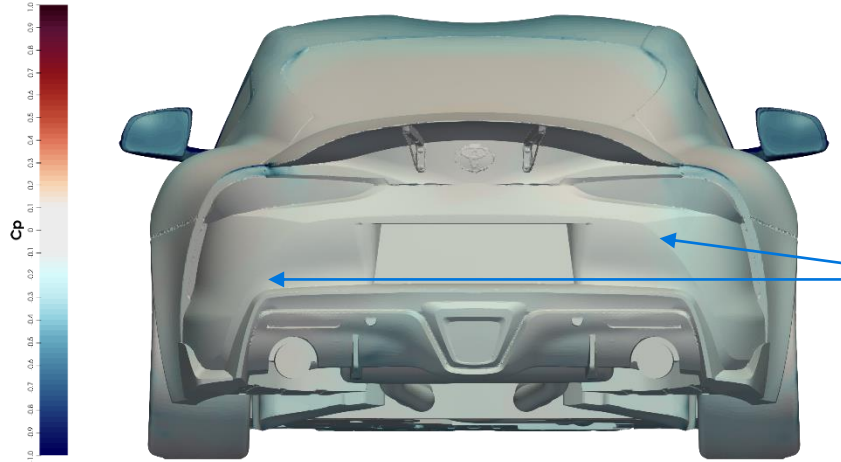




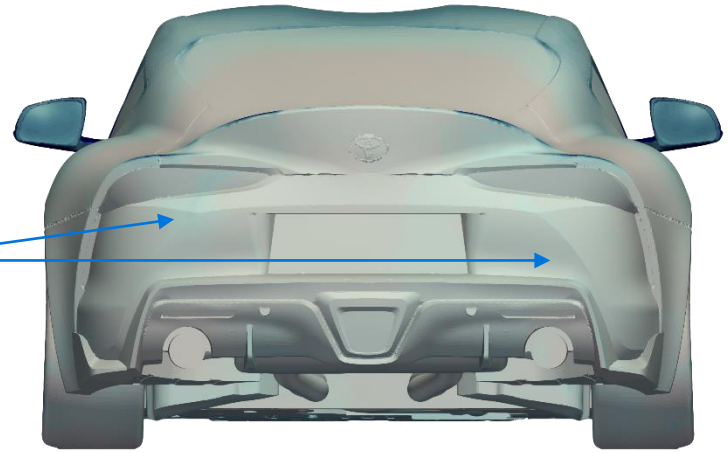
Another post processing image showing the reduced wake. This one is harder examine because of the 3D nature of it. Number 3 has a larger wake than number 4 (Drag Wing).

PRESSURE ON THE REAR

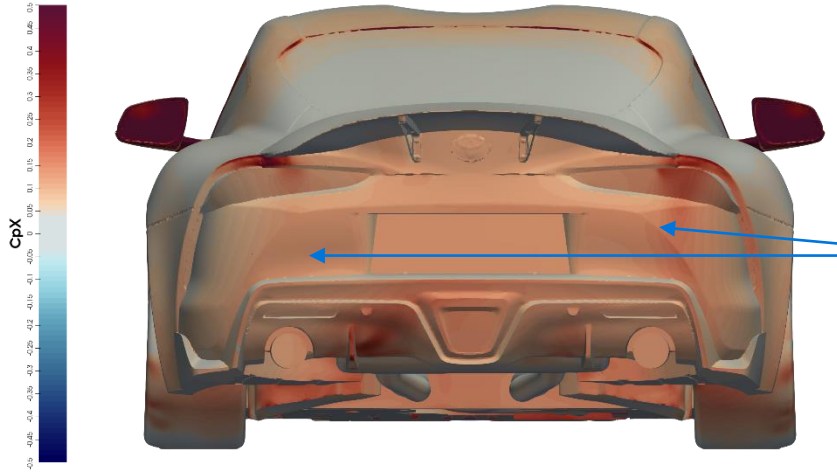
Drag Wing

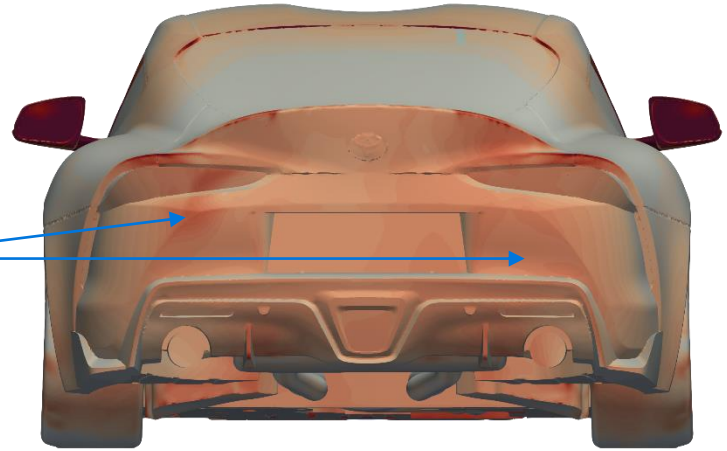


Stock



The pressure on the surface of the bumper is significantly lower on the bumper with the drag wing



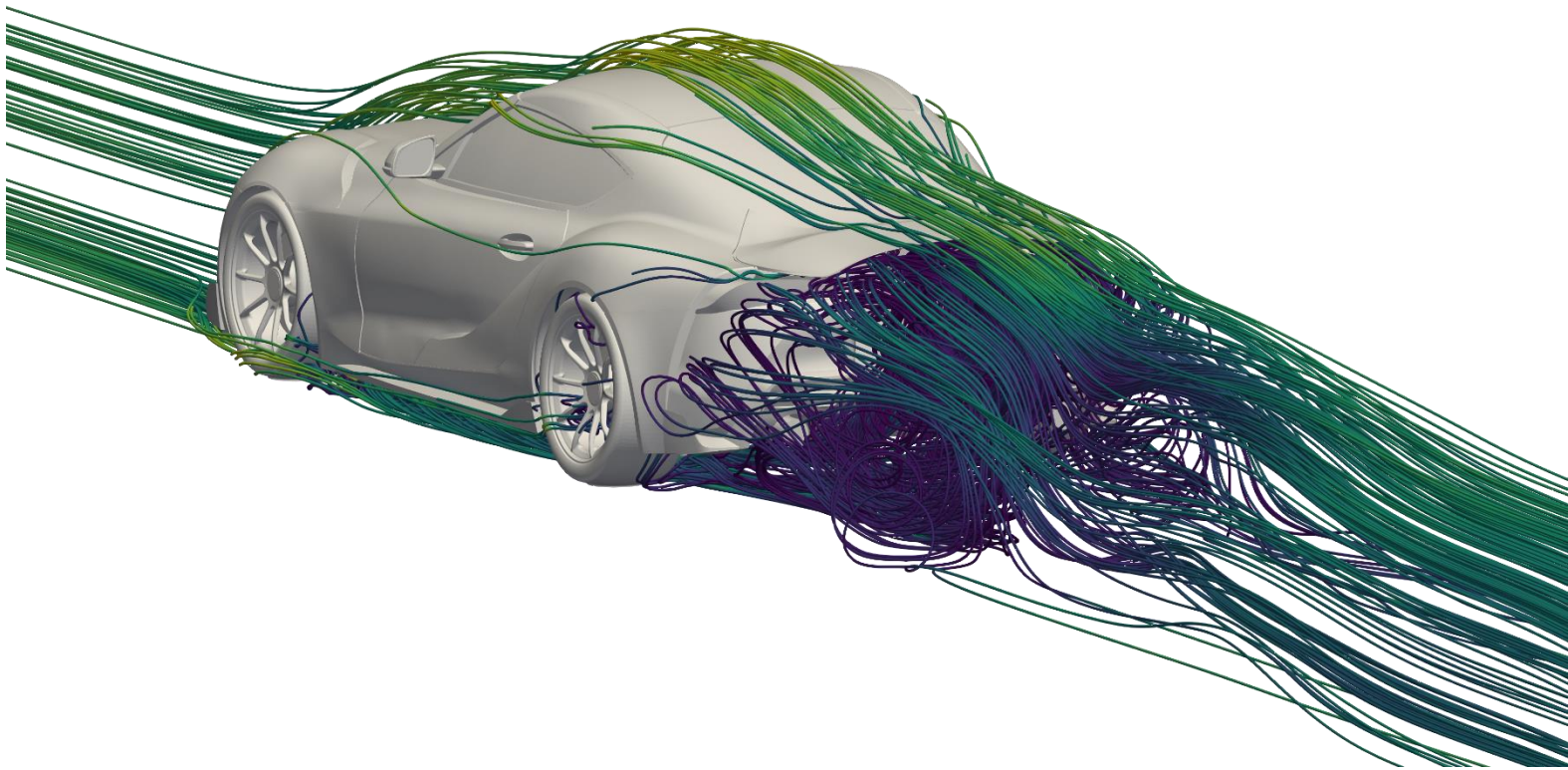


These CpX images show the pressure difference that causes drag and are more obvious to see

SUMMARY

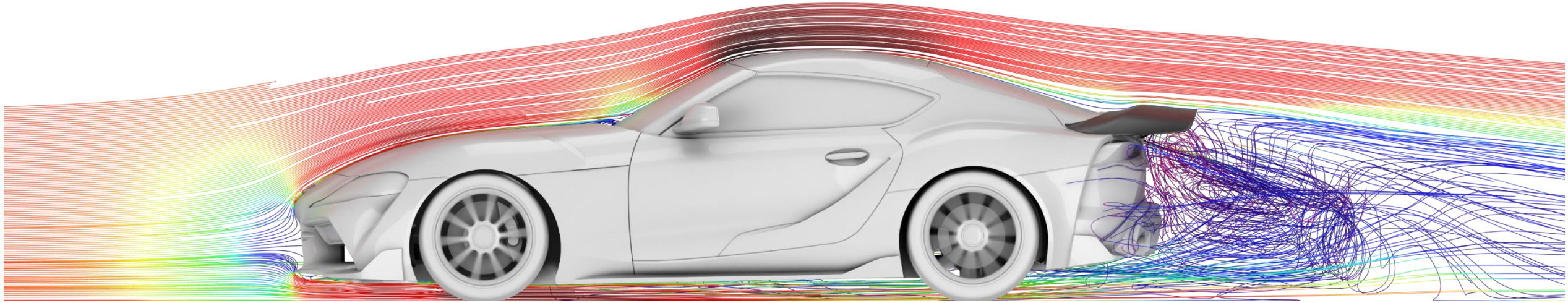
The Verus Engineering Drag Wing for the Mk5 Toyota Supra was designed to increase traction with downforce and increase acceleration with the increased traction and reduction of drag. The drag wing not only increased rear downforce, which is ideal for rear tractive forces to increase acceleration, it also reduced overall car drag to also increase acceleration and overall top speed.

The drag wing was specifically designed for drag racing and top speed events (such as ½ mile racing). Another benefit to the added rear downforce is stability. This can make the difference between a clean run down the track and an accident.

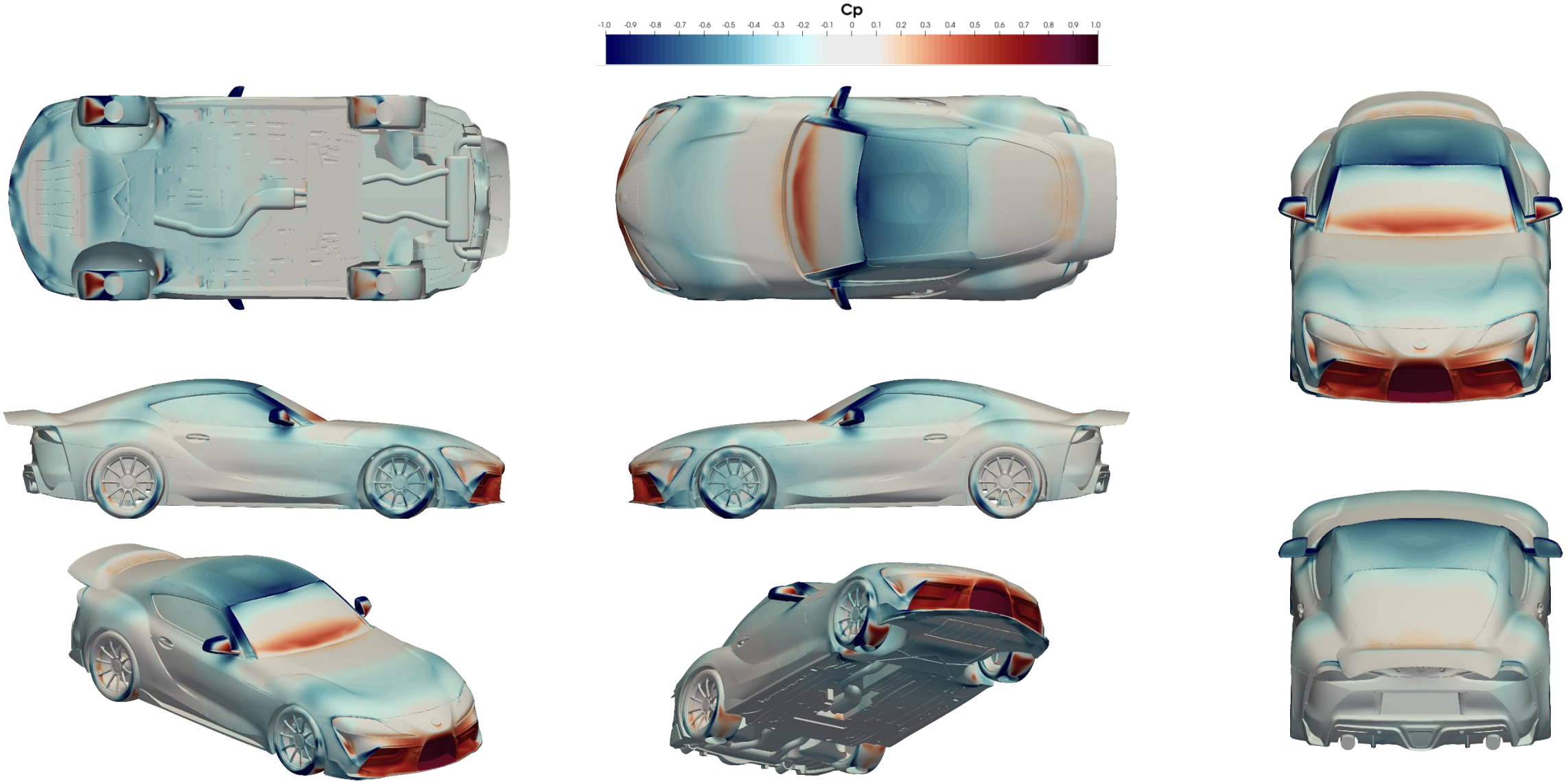


THE SCIENCE

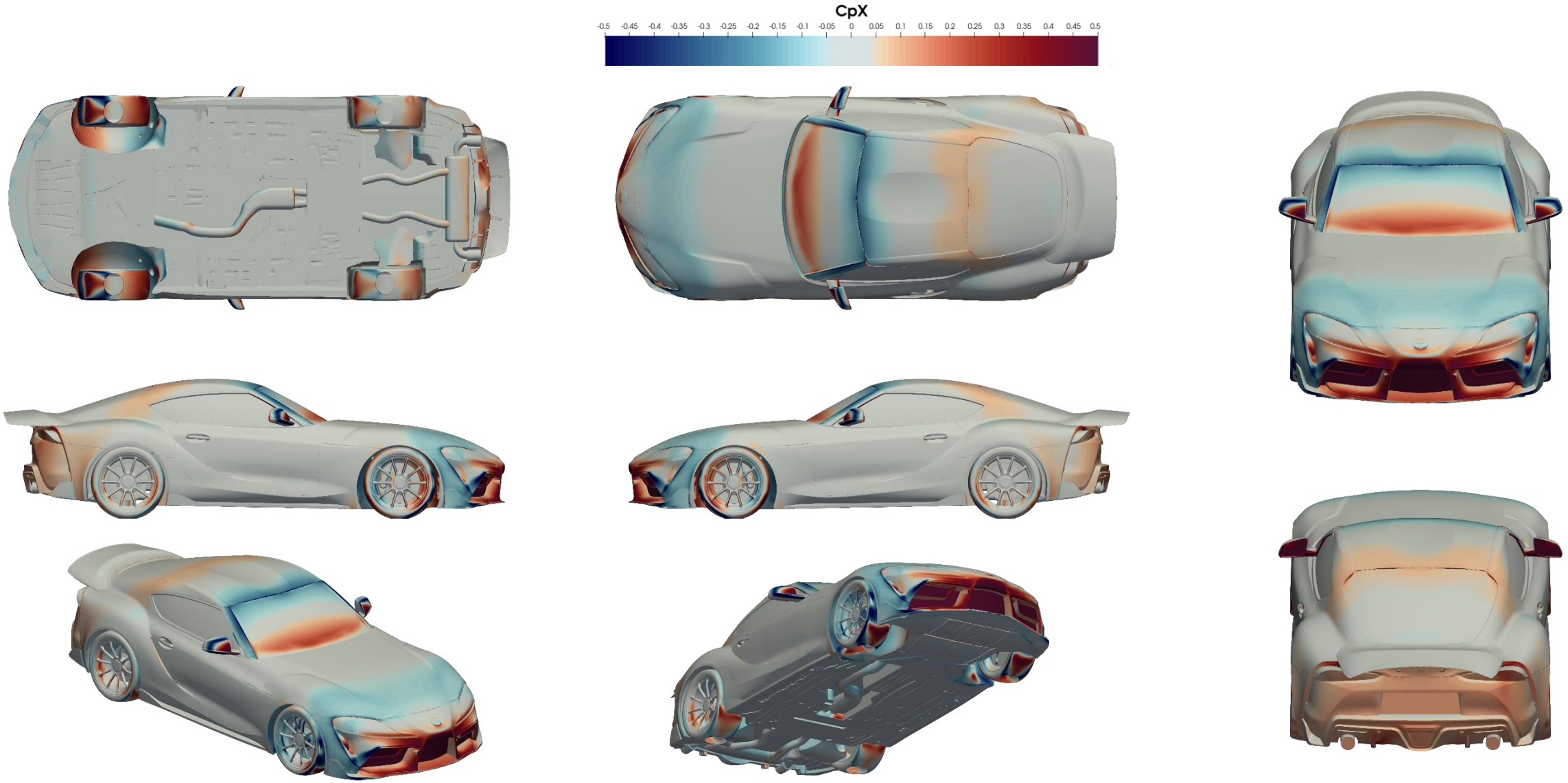
This analysis was done using OpenFOAM v2106 which is a finite volume CFD software. The solver was SIMPLE and the turbulence model was K-Omega SST using standard wall conditions. We use standard automotive arrangement when setting up boundary conditions and running a full-car. The case was simulated using slight yawed airflow of 0.5 degrees. This yawed airflow is to ensure we are not analyzing a condition the car will almost never see which is perfectly straight airflow down the length of the car.



Cp PLOTS : DRAG WING



CpX PLOTS : DRAG WING



CpZ PLOTS : DRAG WING