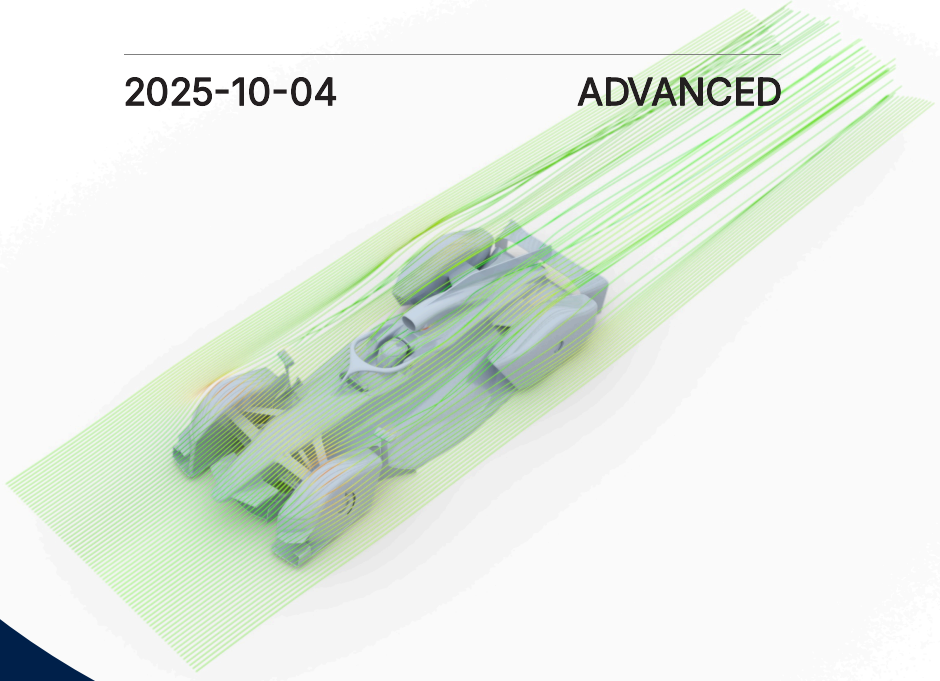
# FORMULA NG CFD RUN03 PITCH CORRECTION NEW OCC

## AERODYNAMIC SIMULATION REPORT

| DATE: | ACCURACY: |
| --- | --- |
| 2025-10-04 | ADVANCED |



AirShaper

# TABLE OF CONTENT


INTRODUCTION 3

PROJECT DETAILS 3

DISCLAIMER 3

3D MODEL 4

BOUNDING BOX DIMENSIONS 4

VIEWS 5

PERFORMANCE ANALYSIS 7

FORCES 7

MOMENTS 8

CONVERGENCE 8

DRAG COEFFICIENT 9

FORCE AND POWER CURVES 9

AERODYNAMIC COEFFICIENTS 10

DRAG ANALYSIS 11

3D PRESSURE CLOUDS 11

SURFACE PRESSURE 14

SURFACE FRICTION 17

FLOW ANALYSIS 20

STREAMLINES 21

Vertical 21

Horizontal 23

2D SECTIONS 25

Velocity 25

Pressure 26

NOISE ANALYSIS 27

# INTRODUCTION

Thank you for choosing airshaper to analyze & optimize the aerodynamics of your design. We hope that this report will provide you with the necessary insight. Should you require further assistance, feel free to get in touch with our experts via info@airshaper.com

# PROJECT DETAILS

| | |
|---|---|
| Project name | FORMULA NG CFD RUN03 pitch correction new OCC |
| Date | 2025-10-04 |
| Location | On the ground |
| Motion | Moving |
| Fluid | Air |
| Rotating Wheels | Yes |
| Rotation - X Y Z [°] | 0.0, 0.0, 0.0 |
| Units | mm |
| Scale | 1.0 |
| wind speed [m/s] | 50.0 |
| Temperature [°C] | 15 |
| Density [kg/m³] | 1.225 |
| Atmospheric pressure [Pa] | 101300 |
| Blockage factor [%] | 1.00 |
| Calculation algorithm | v2406 |
| Solver type | RANS - steady state |
| Turbulence model | k-omega SST |
| Number of cells | 110.022.490 |

# DISCLAIMER

Calculation results provided by airshaper are based on a “virtual wind tunnel”. The size of this virtual wind tunnel is much larger than the flow phenomena you see on the following images. So don’t worry, your model was not “clamped” between virtual walls.

No matter how accurate, simulations are always a simplification of reality, containing modeling, discretization and iteration errors. Therefore, especially in safety critical applications, we always recommend to perform a physical wind tunnel test. To find a wind tunnel institute suited for your project, please contact us at info@airshaper.com

# 3D MODEL

These are the dimensions of the bounding box around the object after rotation and scaling - according to the dimensions of the virtual wind tunnel (X = flow direction, Y = sideways, Z = vertical)

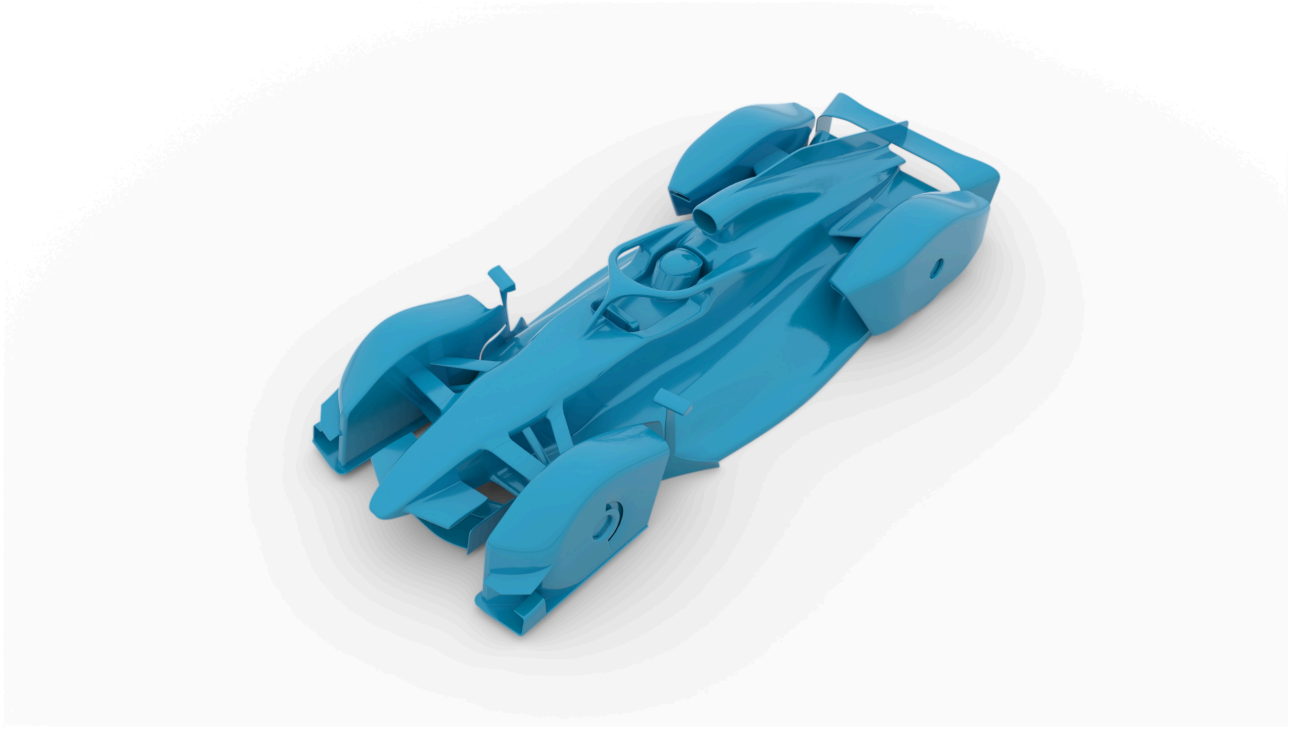
## BOUNDING BOX DIMENSIONS

| | |
|---|---|
| X [m] | 4.84 |
| Y [m] | 1.92 |
| Z [m] | 1.02 |
| Frontal area * [m²] | 1.27 |
| Planform area [m²] | 7.92 |

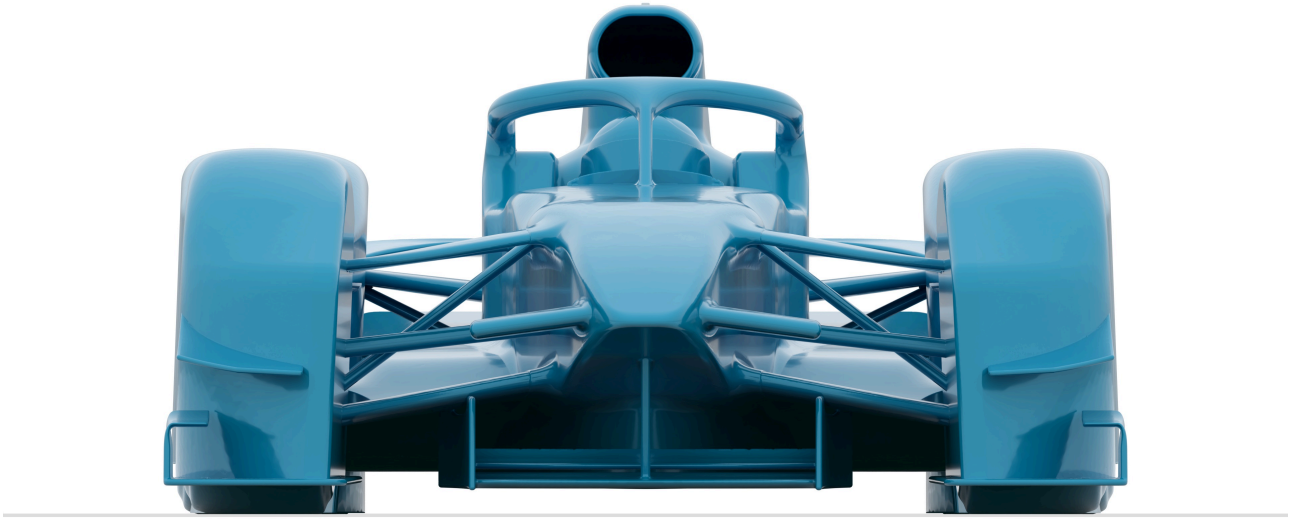
* Used for lift & drag coefficient calculation

# VIEWS

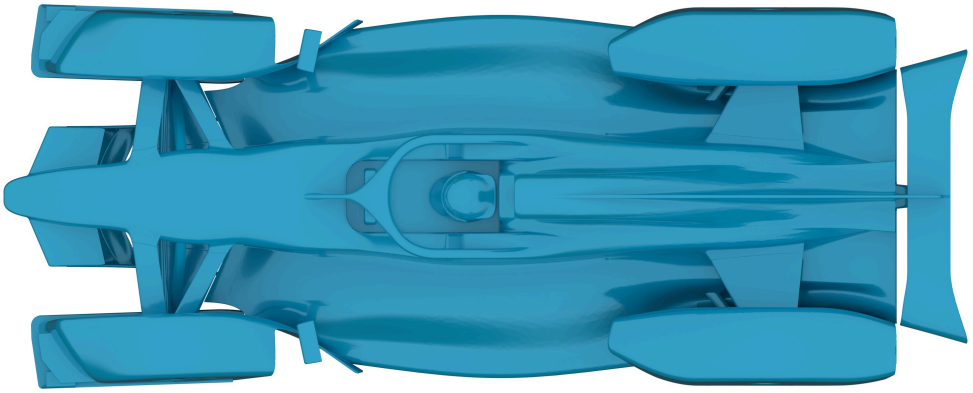
3D view



Front view



## Top view



## Side view



# PERFORMANCE ANALYSIS

## FORCES

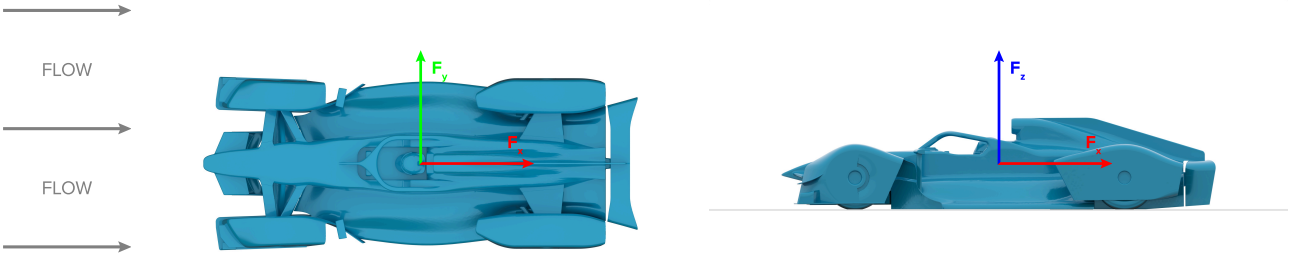
The force generated on the object by the wind has three components:

- $F_x$ - Drag force: Along the direction of the wind
- $F_y$ - Lateral force: Perpendicular to the direction of the wind - horizontal
- $F_z$ - Lift force: Perpendicular to the direction of the wind - vertical

There are two ways in which the wind generates force:

- Pressure force: arises from the pressure difference between two opposite sides of an object. It is the sum of all the local forces pushing or pulling perpendicular to the surface.
- Friction force: arises from the wind sliding across the surface of the object. It is the sum of all the local friction forces parallel to the surface.

**Note:**
Negative values indicate a force acting in the direction opposite to the arrows shown below.
Original coordinate system: origin location & axis orientation as provided with the original 3D file
Wind tunnel coordinate system (shown on images): origin set to the center of the object and the axes aligned with the air vector




WIND TUNNEL COORDINATE SYSTEM

| | $F_x$ | $F_y$ | $F_z$ |
|---|---|---|---|
| **Pressure** | 768.43 N (78.33 kg) | -41.90 N (-4.27 kg) | -5053.20 N (-515.11 kg) |
| **Friction** | 142.03 N (14.48 kg) | 0.56 N (0.06 kg) | 4.31 N (0.44 kg) |
| **Porous** | 17.28 N (1.76 kg) | -0.12 N (-0.01 kg) | -125.78 N (-12.82 kg) |
| **Total** | 927.73 N (94.57 kg) | -41.46 N (-4.23 kg) | -5174.67 N (-527.49 kg) |

ORIGINAL COORDINATE SYSTEM

| | $F_x$ | $F_y$ | $F_z$ |
|---|---|---|---|
| **Pressure** | 768.43 N (78.33 kg) | -41.90 N (-4.27 kg) | -5053.20 N (-515.11 kg) |
| **Friction** | 142.03 N (14.48 kg) | 0.56 N (0.06 kg) | 4.31 N (0.44 kg) |
| **Porous** | 17.28 N (1.76 kg) | -0.12 N (-0.01 kg) | -125.78 N (-12.82 kg) |
| **Total** | 927.73 N (94.57 kg) | -41.46 N (-4.23 kg) | -5174.67 N (-527.49 kg) |

## MOMENTS

The moments have been calculated around the global coordinate system as defined in the uploaded 3D model. Moments also have a friction & pressure component and have been calculated around the X, Y and Z axis.

WIND TUNNEL COORDINATE SYSTEM

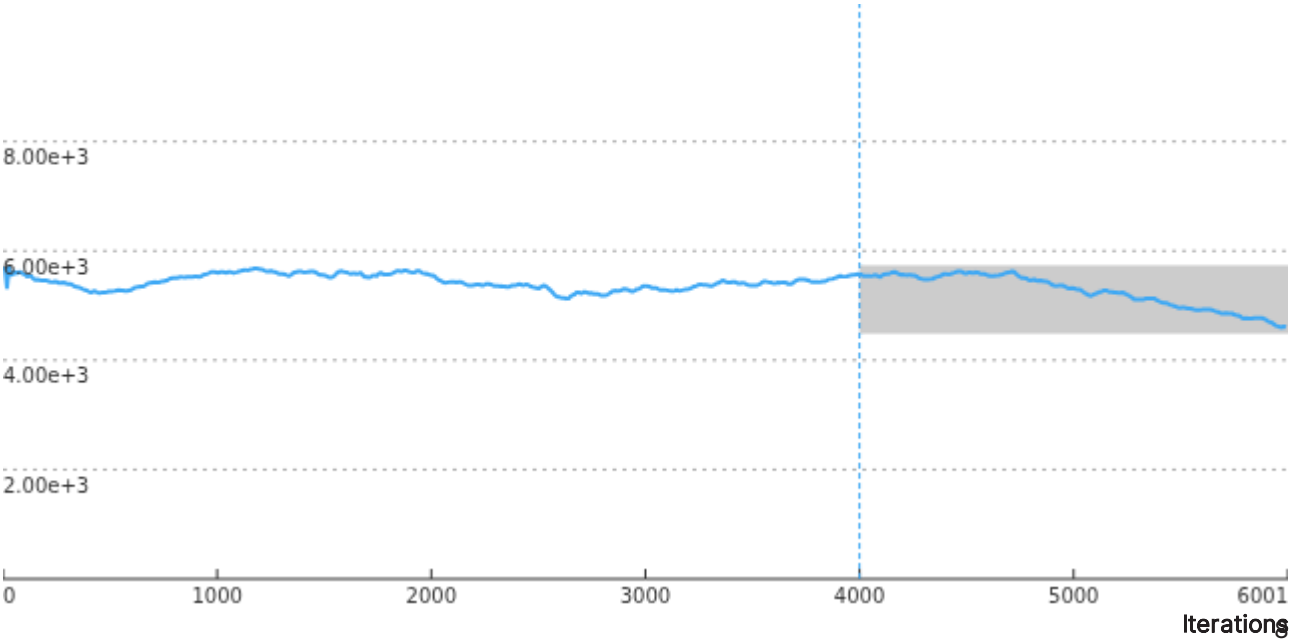
| | $M_x$ | $M_y$ | $M_z$ |
|---|---|---|---|
| **Pressure** | -127.89 Nm (-13.04 kg·m) | 1642.88 Nm (167.47 kg·m) | -36.71 Nm (-3.74 kg·m) |
| **Friction** | 0.25 Nm (0.03 kg·m) | -9.16 Nm (-0.93 kg·m) | 0.13 Nm (0.01 kg·m) |
| **Porous** | 0.02 Nm (0.00 kg·m) | 124.78 Nm (12.72 kg·m) | -0.11 Nm (-0.01 kg·m) |
| **Total** | -127.62 Nm (-13.01 kg·m) | 1758.50 Nm (179.25 kg·m) | -36.69 Nm (-3.74 kg·m) |

ORIGINAL COORDINATE SYSTEM

| | $M_x$ | $M_y$ | $M_z$ |
|---|---|---|---|
| **Pressure** | -106.61 Nm (-10.87 kg·m) | 10193.34 Nm (1039.08 kg·m) | -104.37 Nm (-10.64 kg·m) |
| **Friction** | -0.03 Nm (-0.00 kg·m) | 56.00 Nm (5.71 kg·m) | 1.04 Nm (0.11 kg·m) |
| **Porous** | 0.08 Nm (0.01 kg·m) | 336.67 Nm (34.32 kg·m) | -0.31 Nm (-0.03 kg·m) |
| **Total** | -106.56 Nm (-10.86 kg·m) | 10586.01 Nm (1079.10 kg·m) | -103.64 Nm (-10.56 kg·m) |

## CONVERGENCE

**Total Force** (N)




Convergence detected after 4000 iterations

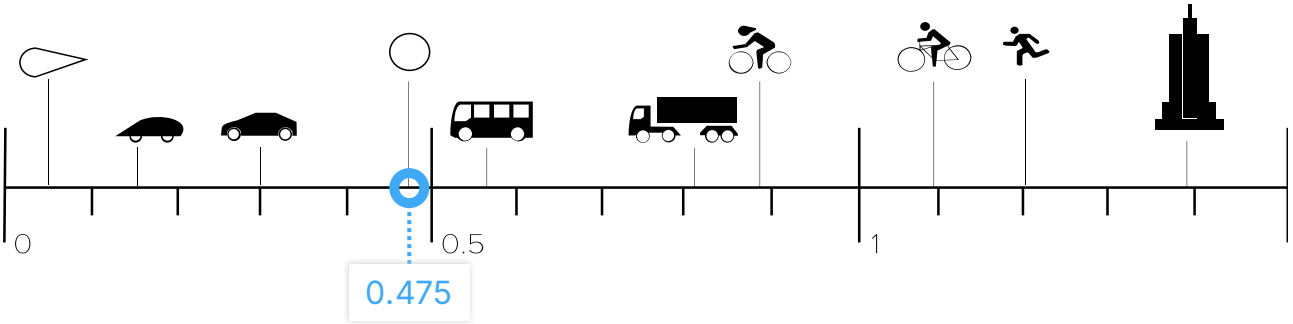
Force averaging window (2001 iterations)

## DRAG COEFFICIENT

Drag, or air resistance, is the force of the wind on an object in the direction of the wind. It is composed of pressure drag (pushing/pulling normal the surface) and friction drag (sliding over the surface). In most cases the pressure drag is dominant.
The drag coefficient is a dimensionless quantity that indicates the aerodynamic resistance of an object moving through its medium. It is defined as follows (note: the Cd value in this report has been corrected for blockage):

$$C_d = \frac{2F_d}{\rho v^2 A}$$

The scale below illustrates typical Cd values (NASA and Wikipedia). More streamlined objects will have a low Cd, less streamlines objects will have a high Cd. The Cd of your project has been indicated as well. Please note that this is an indicative figure, mainly suited for comparing different concepts. For a highly accurate value, contact us at info@airshaper.com.



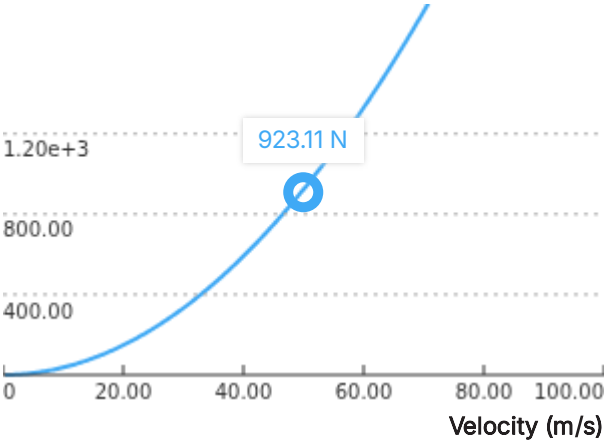

## FORCE AND POWER CURVES

The Cd provides a measure for the aerodynamic efficiency of an object. By approximation, the actual drag force Fd on the object as well as the power required to propel it will vary in function of the wind speed according the following formula:

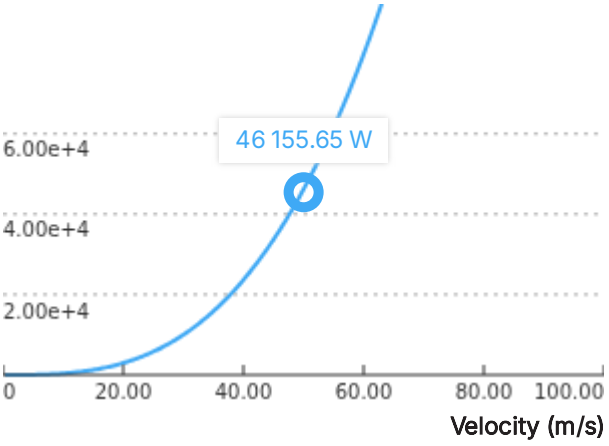
$$F_d = \frac{\rho v^2 C_d A}{2} \qquad P = F_d v$$

For your project, that leads to the force & power curves shown below. Please keep in mind that this curve is an estimation, based on extrapolation from the simulation wind speed. For more accurate forces at a given velocity, please perform a simulation for that velocity.

**Drag Force Fd(N)**




**Required Power P(W)**



## AERODYNAMIC COEFFICIENTS

The coefficients below indicate the aerodynamic performance of your vehicle (all of them have been corrected for in terms of blockage factor).

- Drag coefficient: the efficiency at which the vehicle moves through the air (lower drag means less fuel / energy consumption).
- Lift coefficient: the effectiveness of the vehicle to generate lift (positive value) or down force (negative value). The front & rear lift coefficient denote the split of the lift coefficient between the front & rear wheels.

| | Value |
|---|---|
| Drag coefficient Cd | 0.475 |
| Lift coefficient Cl | -2.647 |
| Front lift coefficient Cl(f) | -0.857 |
| Rear lift coefficient Cl(r) | -1.790 |

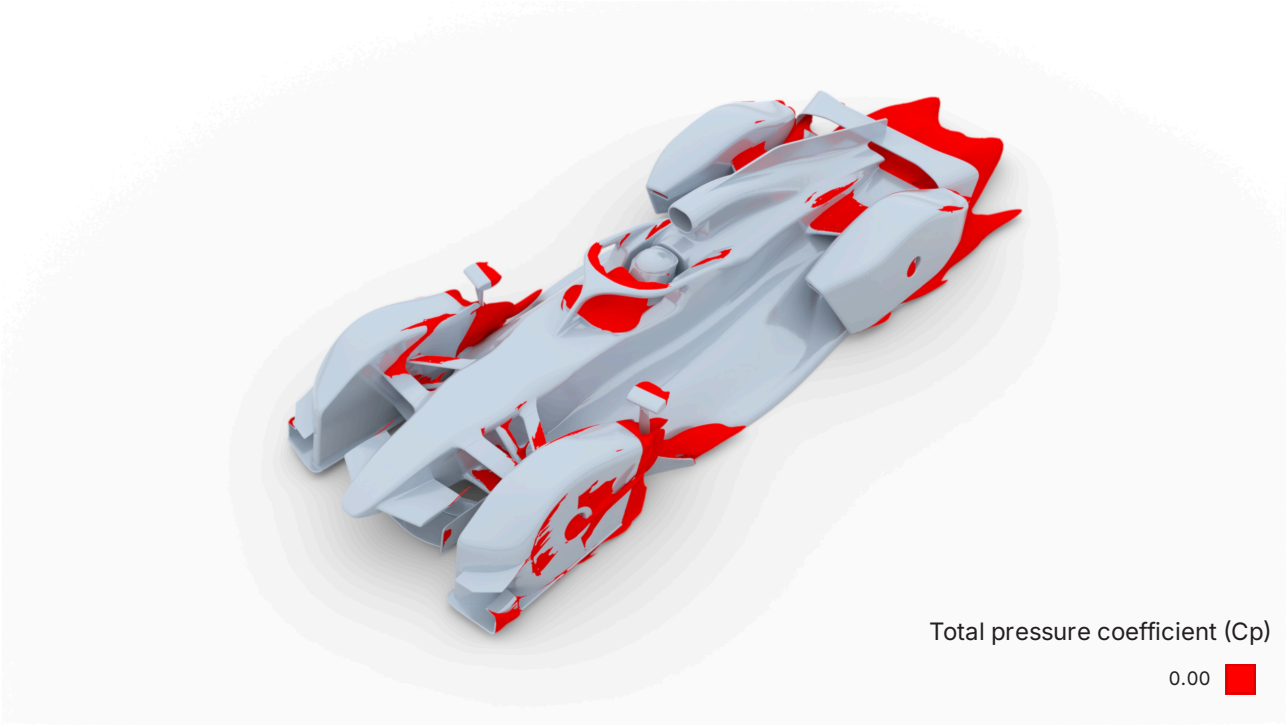
# DRAG ANALYSIS

## 3D PRESSURE CLOUDS

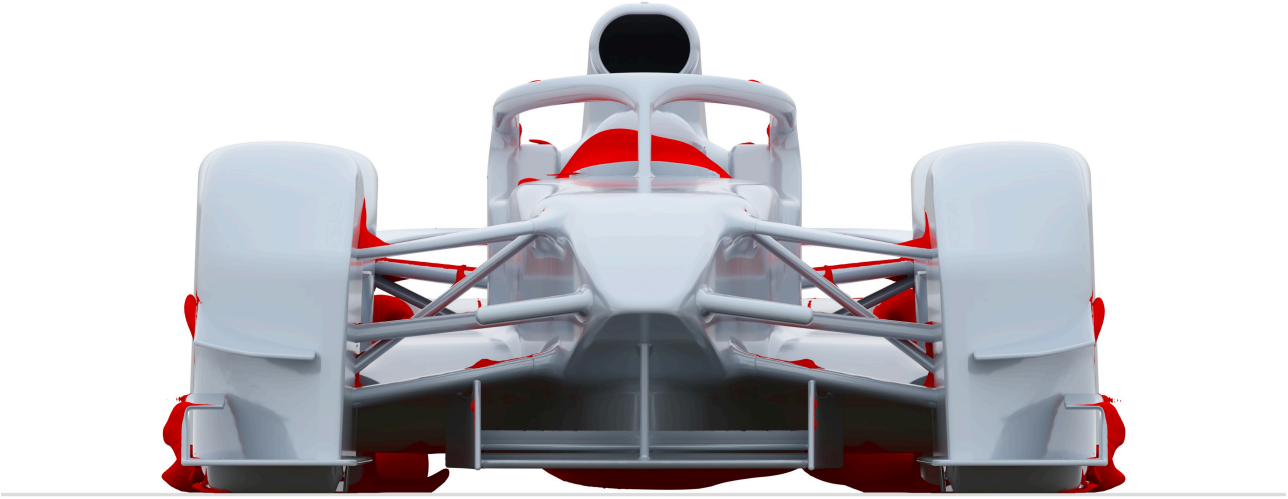
Underpressure zones are areas with a pressure lower than that of the local surrounding air. Often these zones are caused by abrupt changes in shape, most pronounced by abrupt changes in cross section or surface geometry. Smooth these out to lower drag.

The following images provide insight into which features of your object are causing drag by showing “clouds” of low pressure.

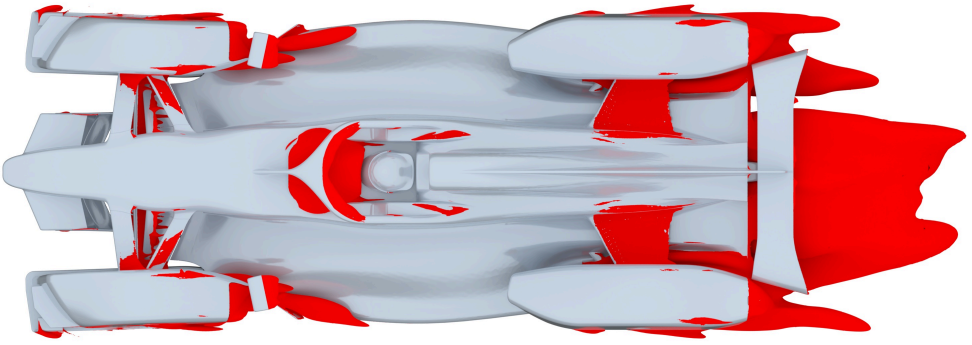
## 3D view



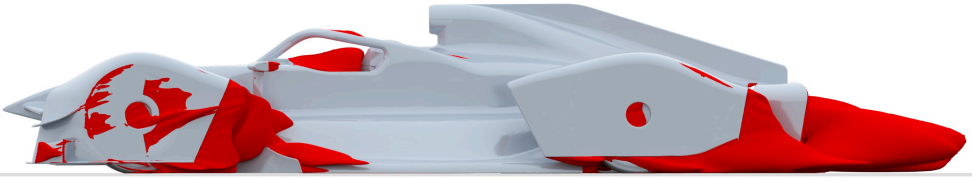

## Front view



Top view



Side view

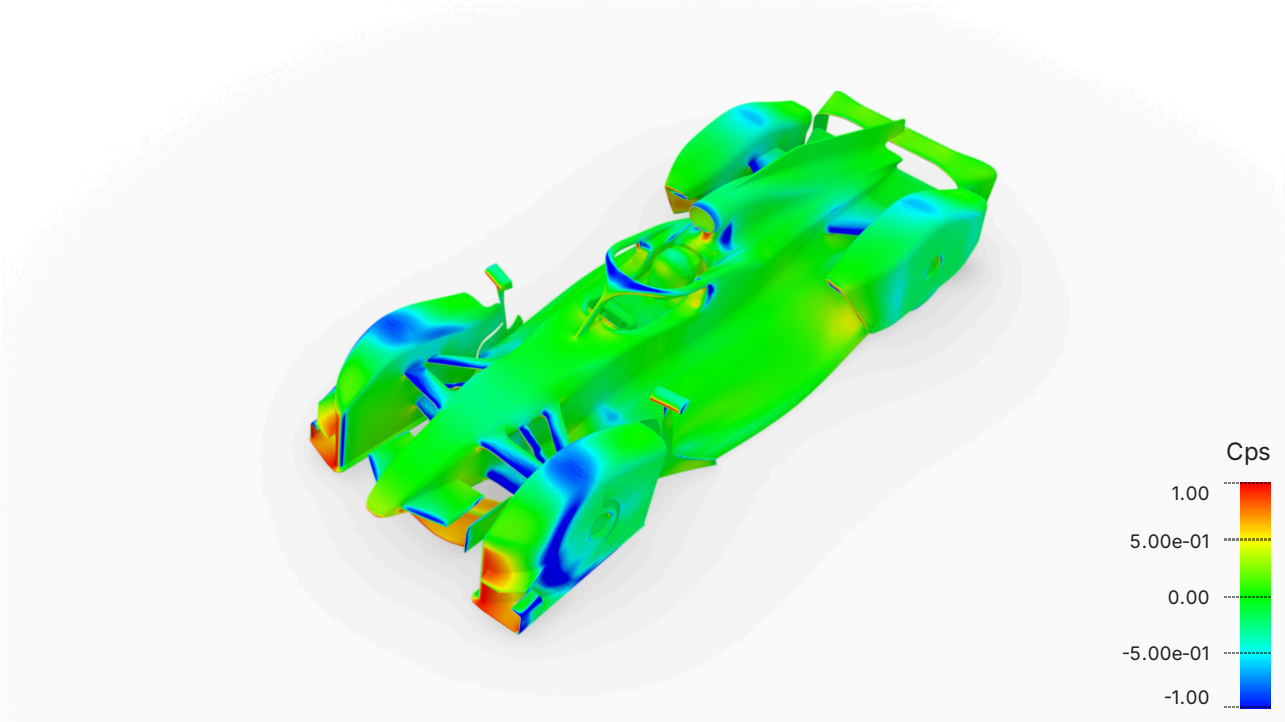


## SURFACE PRESSURE

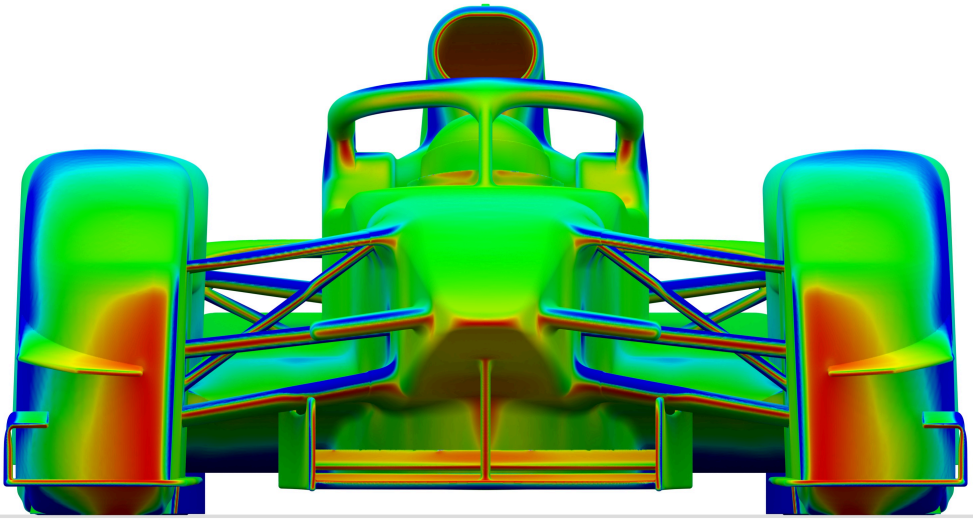
Air impacts the object and this creates overpressure. The highest pressure is reached when the air comes to a complete standstill (with respect to the object), for example at the nose of a car. It's called the stagnation point and the pressure at that location is the stagnation pressure (corresponding to a static pressure coefficient, or Cps, of +1)

The more perpendicular a forward-facing surface is to the flow, the more likely it will create overpressure and thus drag. Likewise, a backward-facing surface experiencing underpressure (a negative pressure coefficient) will pull the object backwards, again creating drag. Try to reduce peaks in overpressure and underpressure by smoothing the surface at peak locations, critical transitions etc. to recude drag.

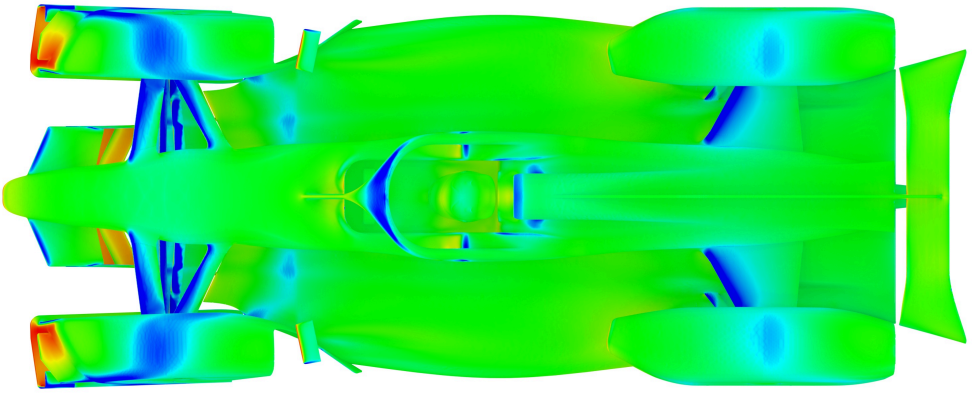
## 3D view



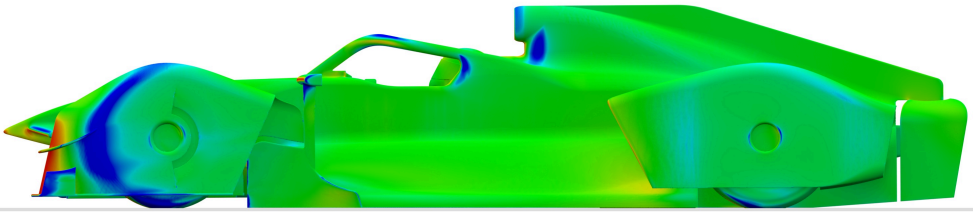

## Front view



Top view



Side view



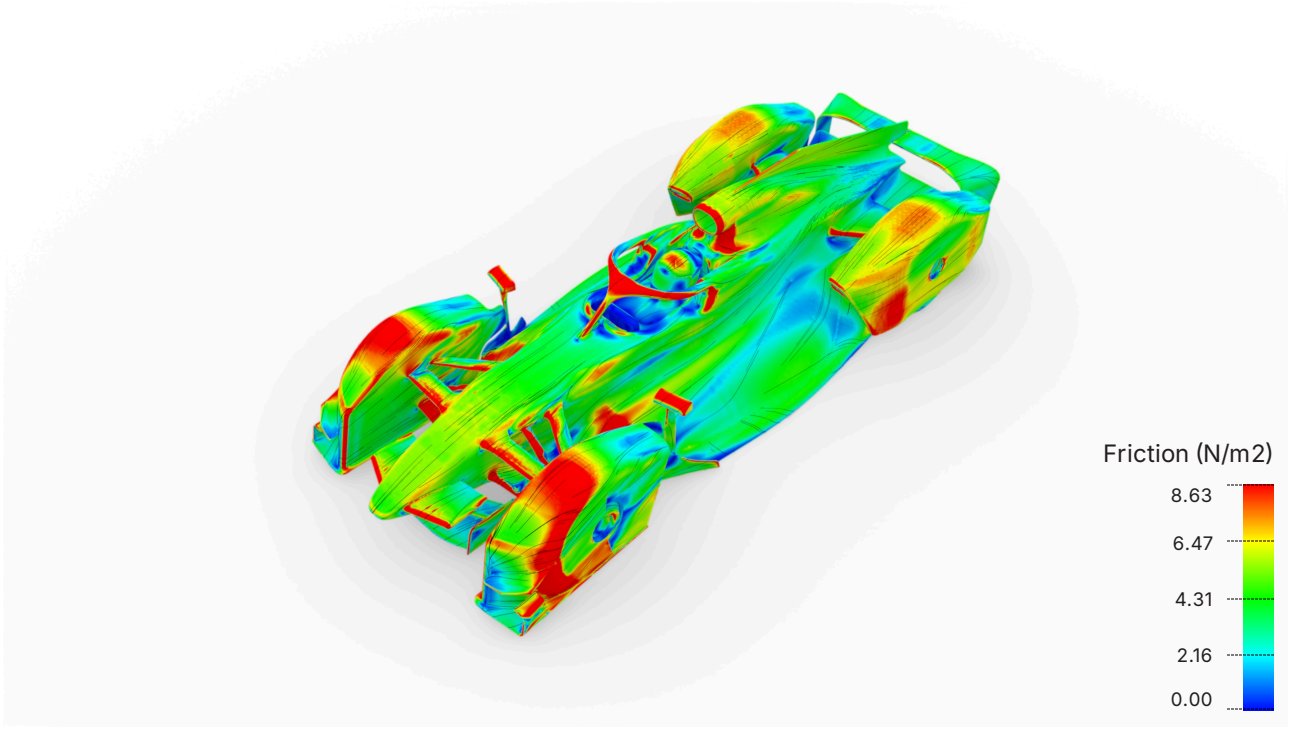
## SURFACE FRICTION

Friction drag is caused by air sliding across the surface, generating friction in the process. The faster the air moves along a surface, the higher the friction force. A smooth surface (a coating with low roughness for example) can reduce friction drag.

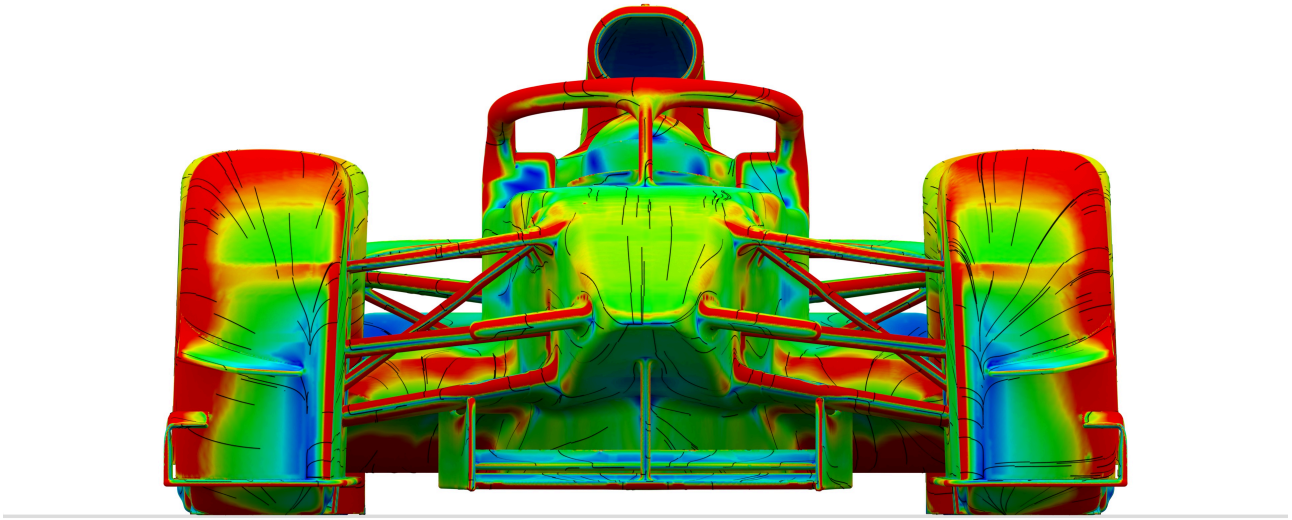
The following images show the friction drag, mapped in color on the surface of your object. Although friction drag typically contributes only a small amount to the total drag, it might be worth to try and re-route some of the air to reduce it.

Also shown are streamlines, showing the pattern followed when air moves across the surface. This allows you to find locations where detachment occurs: as long as the flow is able to follow the surface (attached flow), you will see rather straight streamlines and at least some shear (color green to red) force associated with it. Once the flow detaches, it will start to swirl (curly streamlines) and move slower with respect to the surface (lower shear forces – color towards blue). To optimize airflow, look at these detachment spots and try to make transitions & angles smoother.

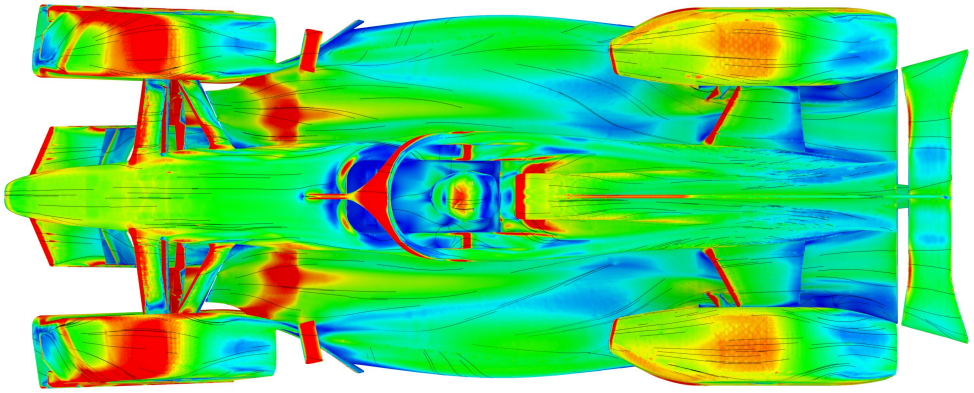
## 3D view



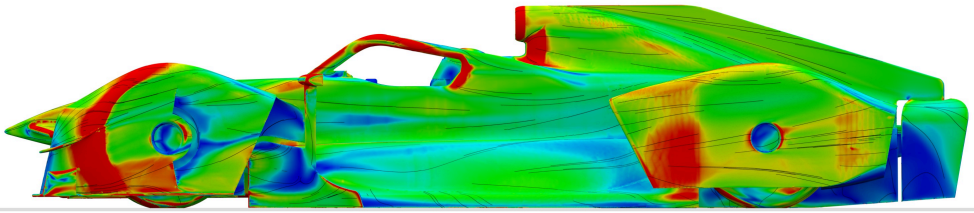

## Front view



Top view

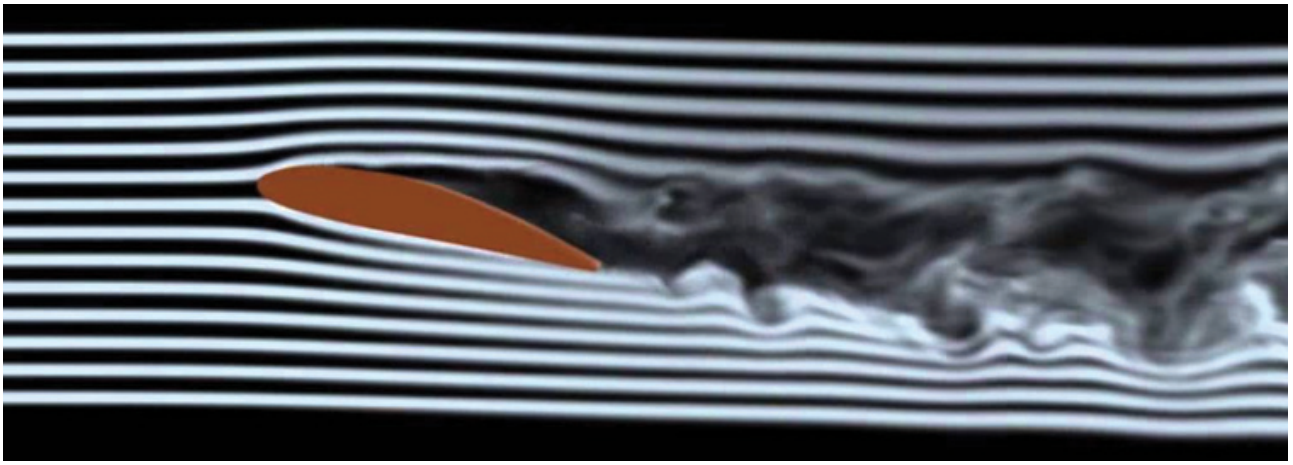


Side view



# FLOW ANALYSIS

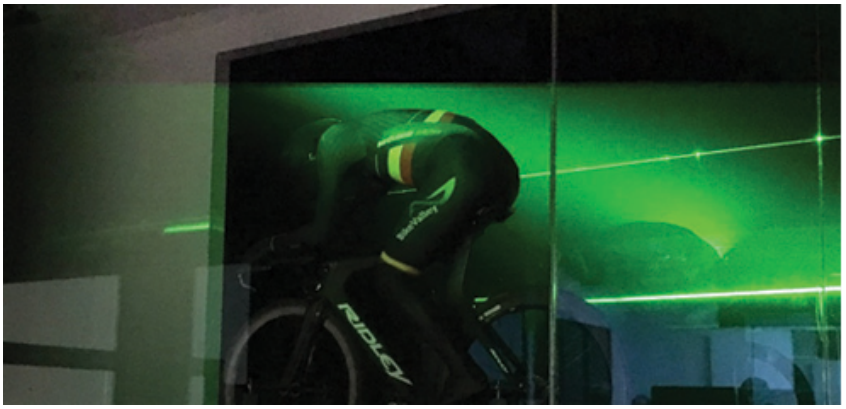
Understanding the way the wind flows past your object is the basis for optimizing your design. In physical wind tunnels, this is often done by releasing smoke (small oil droplets) in the air, upstream of the object, as illustrated below (image courtesy of Paul Selhi).

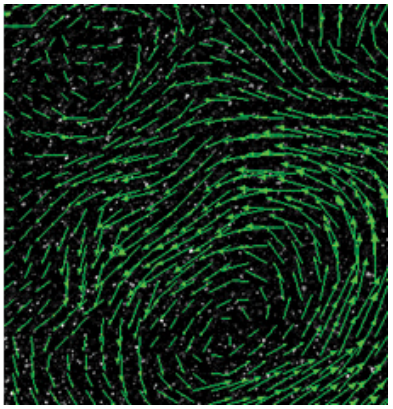


These streamlines can provide multiple insights. First of all, it allows to spot laminar flow zones (steady streamlines) and turbulent zones (swirling streamlines). The transition from laminar to turbulent often occurs when the airflow is unable to follow the surface of the object, because the “negative angle” of the surface is too large or because of a geometrical disturbance in the surface. This is called separation which, in most cases, increases drag.

Another observation is the compression and expansion of these streamlines (the density of the streamlines). As the air can be considered as incompressible, the air needs to speed up when the available cross section narrows. Speeding up and slowing down air, by changes in geometry / cross section, can again be a source of drag. Reducing these changes in velocity, by smoothing these geometry / cross section changes, can reduce drag.

In the virtual world, the streamlines can be colored by the velocity, allowing to detect increases/decreases in velocity directly and not just via streamline density. The images below show the streamlines (the trajectory a weightless particle would follow when released in the air) both for a vertical and horizontal array of “smoke” sources.



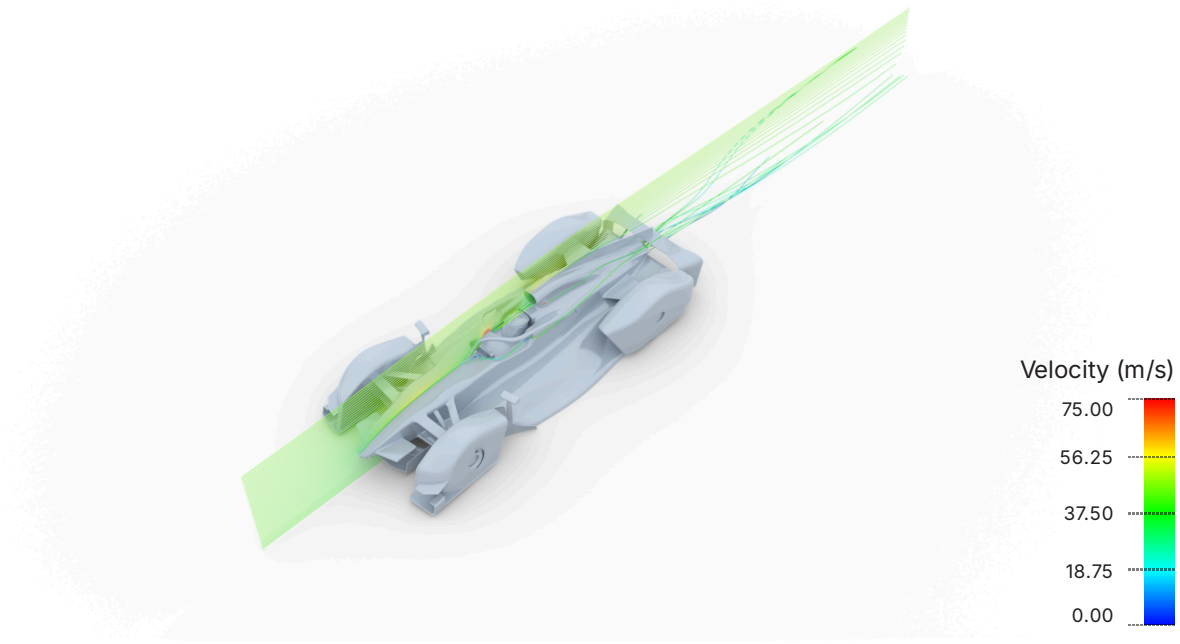



A second, more advanced visualization technique in physical wind tunnels is to light up a 2D plane with a laser and film it from a direction perpendicular to this plane (see images above – courtesy of Flanders Bike Valley and formula1-dictionary.net). This makes it possible to detect in-plane movement patterns of “particles”. This is shown in the second series of images, 2D flow patterns, colored by velocity and by pressure.

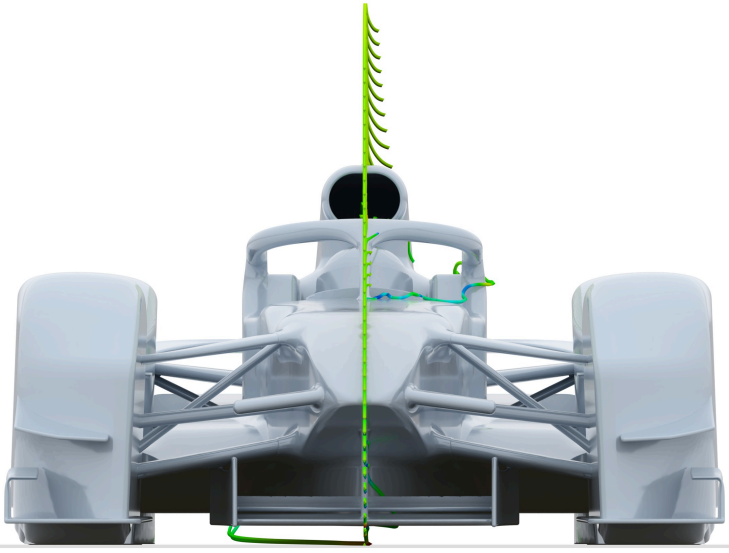
# STREAMLINES

## VERTICAL

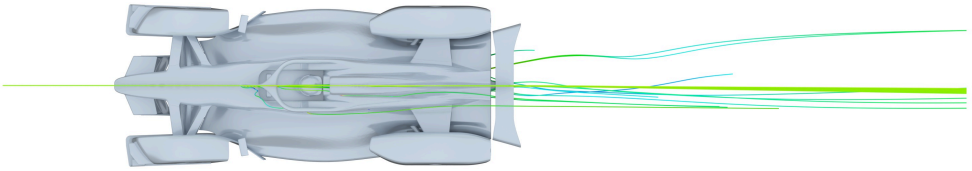
3D view



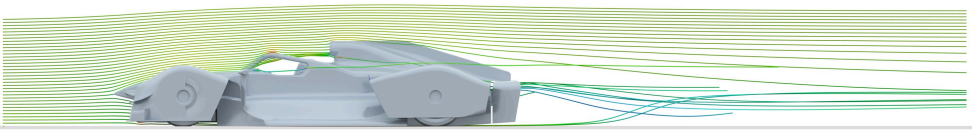

Front view



Top view

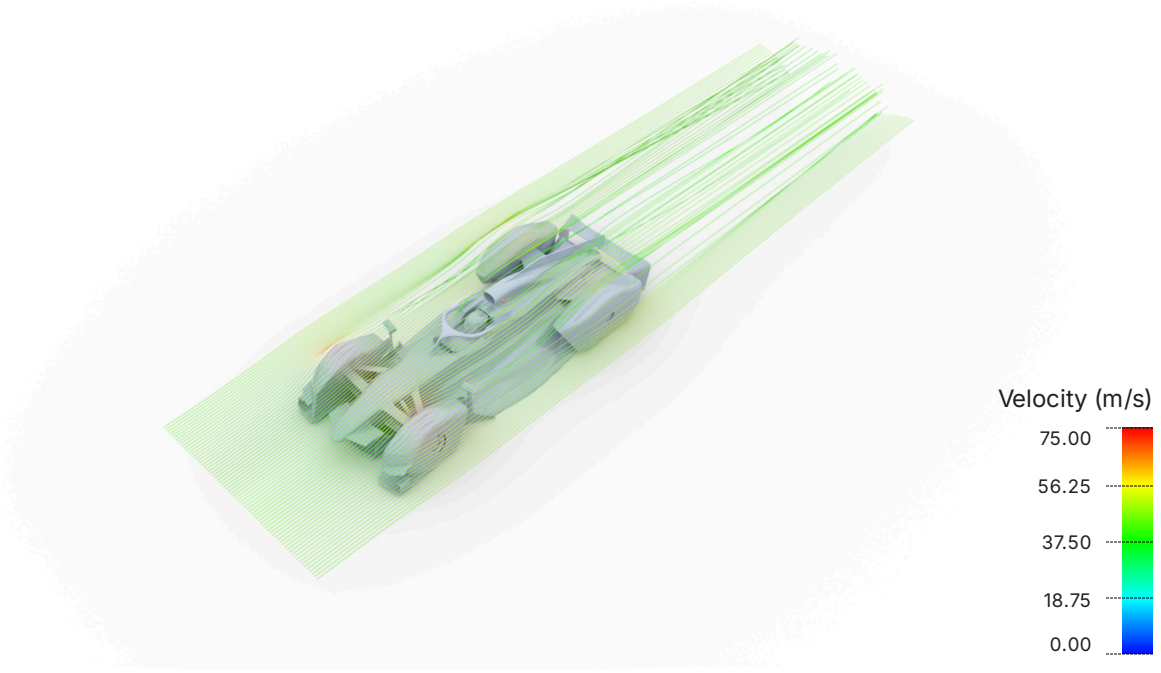


Side view

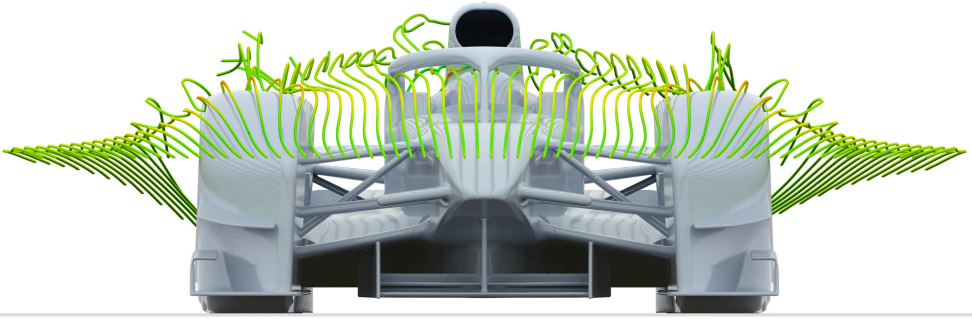


HORIZONTAL

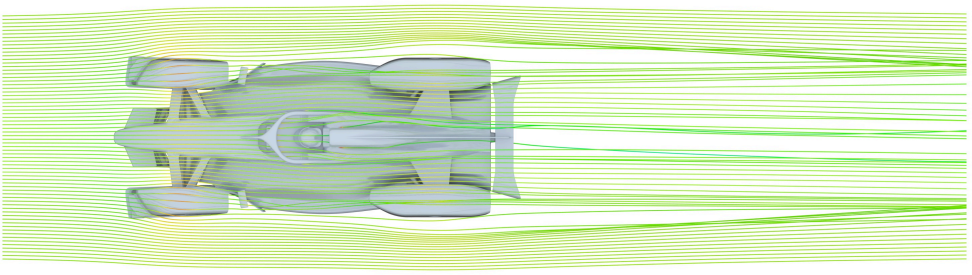
3D view



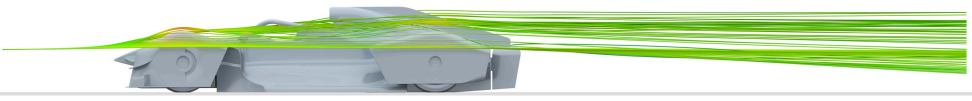

Front view



## Top view



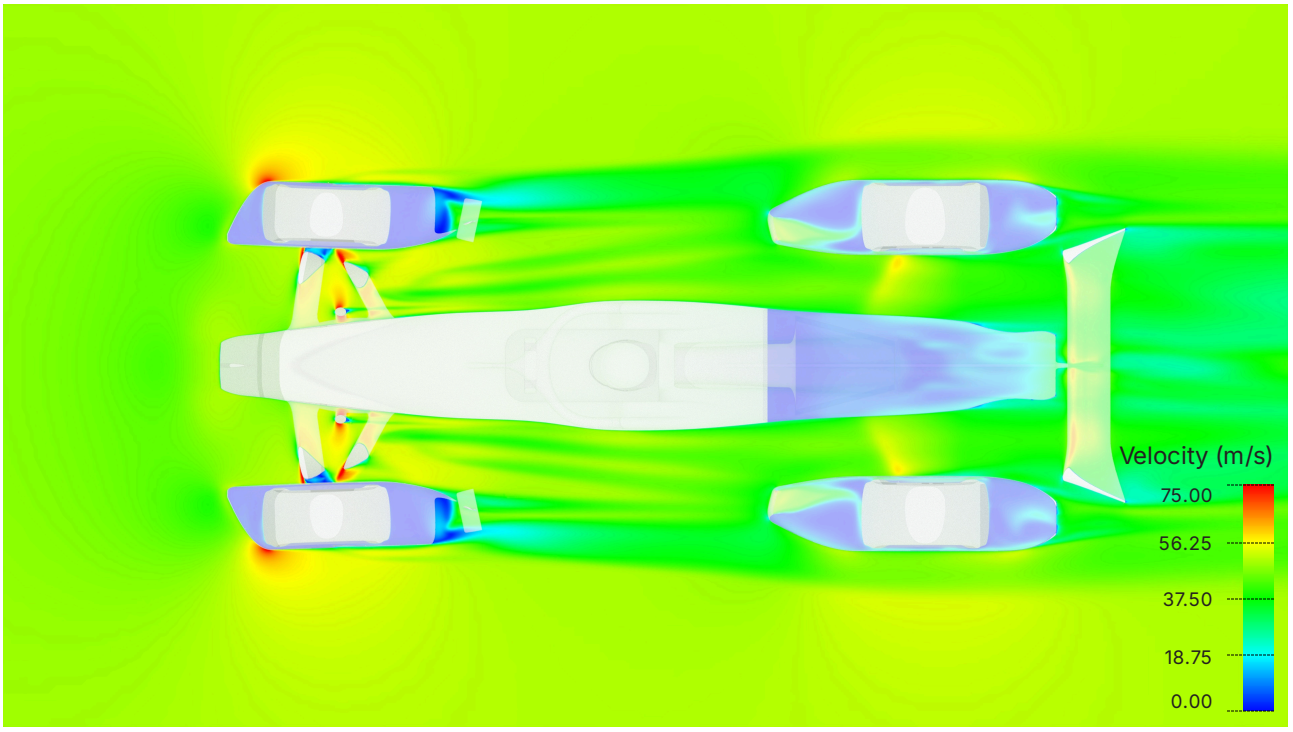
## Side view



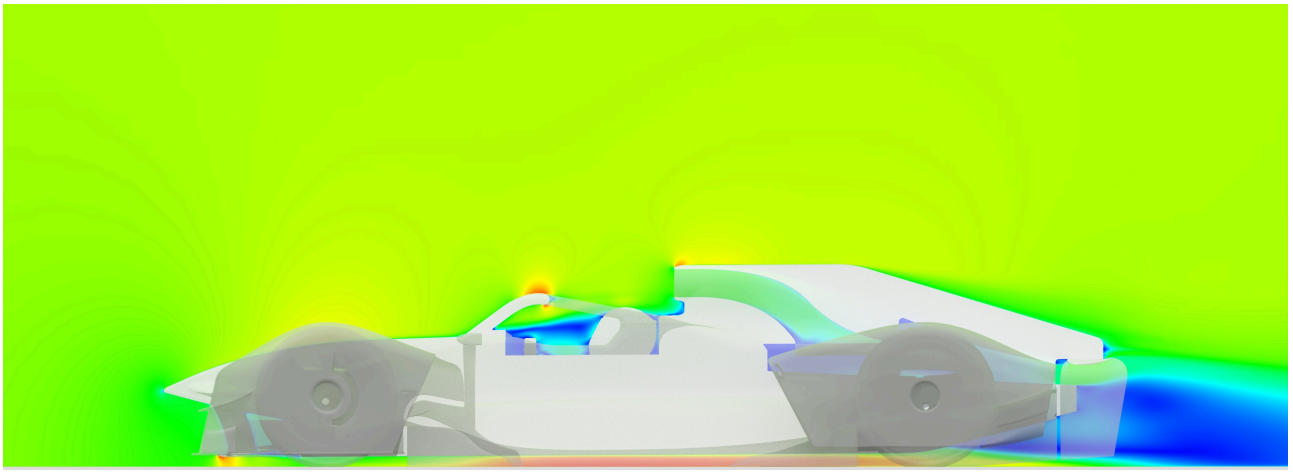
# 2D SECTIONS

## VELOCITY

Top view

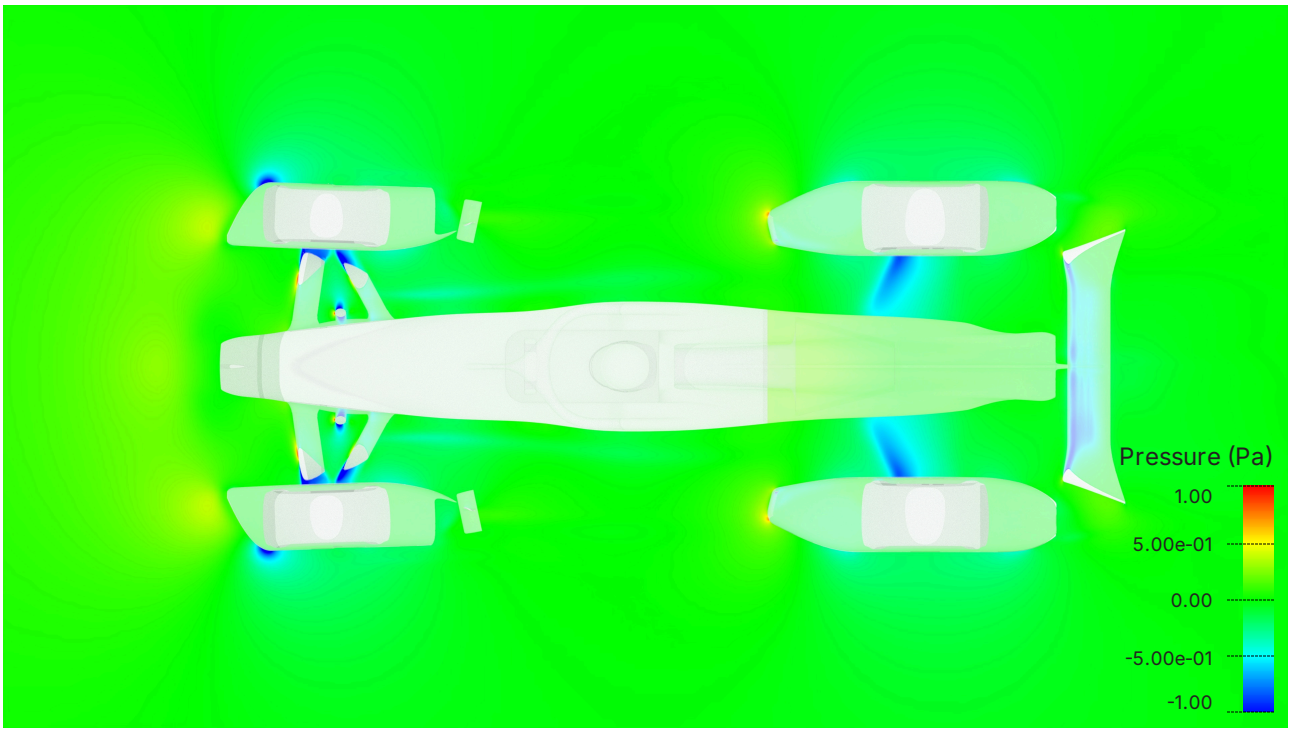



Side view

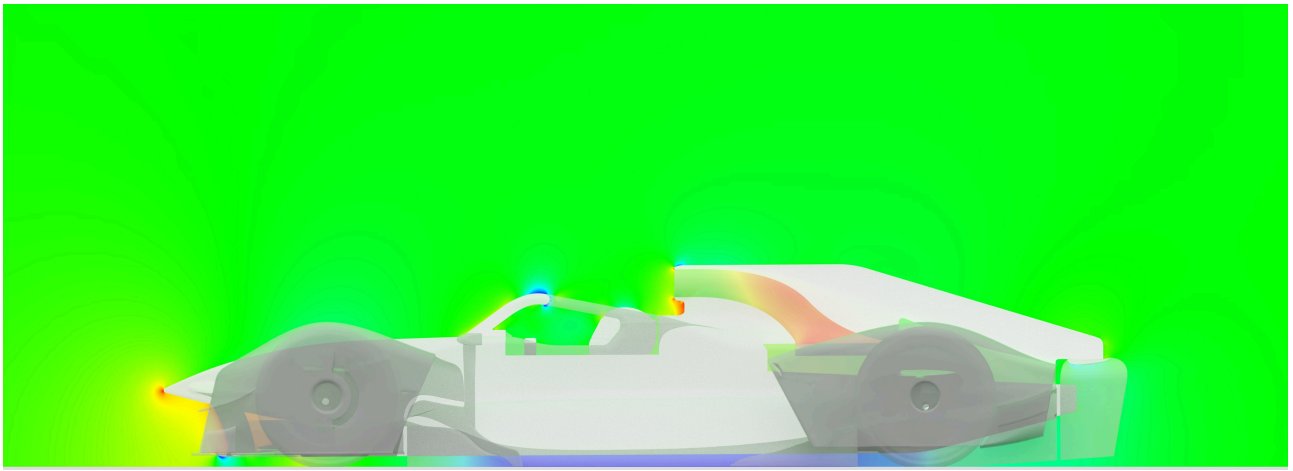


PRESSURE

Top view




Side view

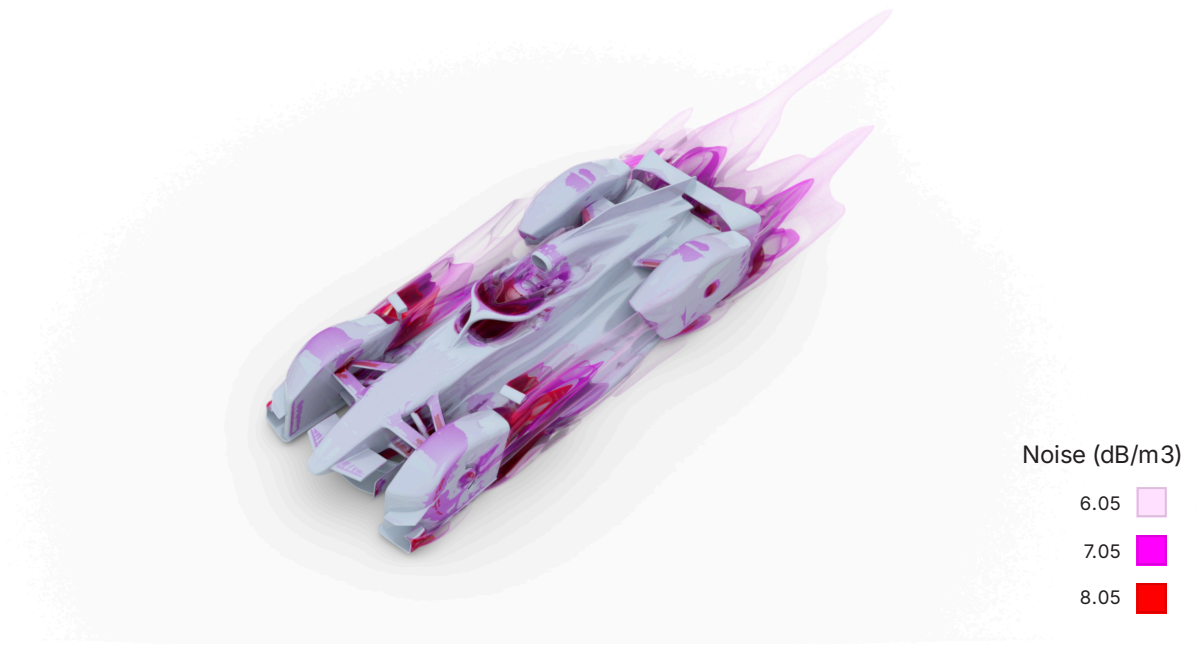


# NOISE ANALYSIS

Noise estimation by means of simulations is an advanced field of engineering. Therefore, more simplified models ('acoustic analogies') have been introduced to obtain a rough estimation of the local noise generation without going through prohibitively expensive simulations.

To reduce the noise generation in your design, look for the origin of these “noise clouds” in the following images. Typically, noise is generated slightly downstream of the location where the flow is disturbed. Smoothen the source and you can reduce noise generation. Avoid cavities and external parts.

## 3D view



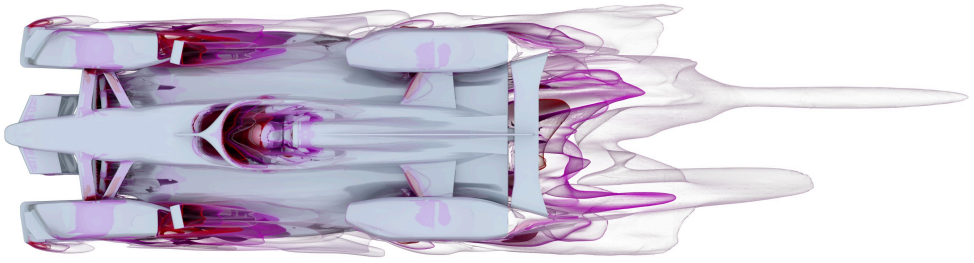

## Side view



Front view



Top view



Back view



