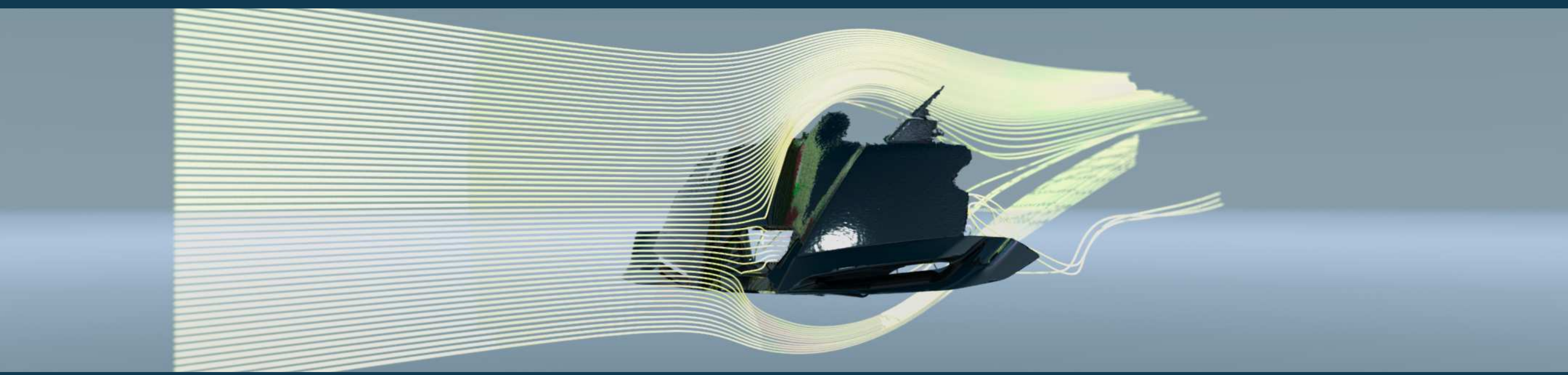
AEROCLOUD

by

NABLAFLOW

CFD report

Ver 4



Contents

Introduction

Thank you for using AeroCloud by NablaFlow. Our goal is to make advanced CFD-simulations fast and available to everyone. By keeping the workflow simple and efficient, we take care of the simulations while you can focus on design and development. The meshing and simulation processes are fully automated and tested on a wide range of validation cases to provide stable and reliable results.
This document summarizes the setup and results of a numerical aerodynamics study run on AeroCloud. The presented results are calculated from the time averaged flow field and shown as global parameters and detailed spatial velocity and pressure field presentations. We try to use common conventions and definitions from the aerodynamics community, but these may vary between specific applications and fields. All definitions used here are described in the report.
The simulation result webpage can be accessed by scanning the QR code below.

Disclaimer

The simulations executed with AeroCloud follow the best practices for CFD simulations. However, simulations will always involve modelling, discretization and convergence errors. Therefore, we highly recommend that final designs and safety critical dimensioning parameters are verified in wind tunnel experiments. We, at NablaFlow, collaborate with a large number of wind tunnel operators. To find a wind tunnel institute suited for your project, please contact us at info@nablaflow.io. NablaFlow is not responsible for the misuse of the information included in the present report.

AEROCLOUD

Simulation setup

Details

Name	Ver 4
Simulation date	May 12, 2026
Number of files	1
Number of solid parts	1
Number of porous parts	0
Frontal area	0.0138 m^2
Surface area	0.3076 m^2
Refined area	0.0000 m^2
Number of cells	5807273

Settings

Version	v7
Quality	Basic
Fluid	Air
Kinematic viscosity	14.6 cSt
Density	1.225 kg/m^3
Speed	66.66 m/s
Yaw angles	0.0°
Ground	slip
Ceiling	No
Rotation	No

3D model

The uploaded 3D model consists of 1 part. The complete model has a projected frontal area of 0.0138 m² and a total surface area of 0.3076 m². The maximum extent of the model along the x, y and z axes is 0.252 m, 0.655 m and 0.289 m.

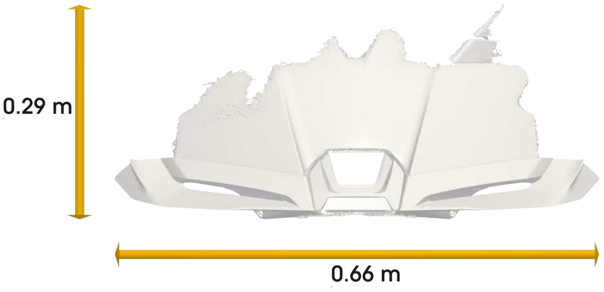


Figure 1: 3D model front view

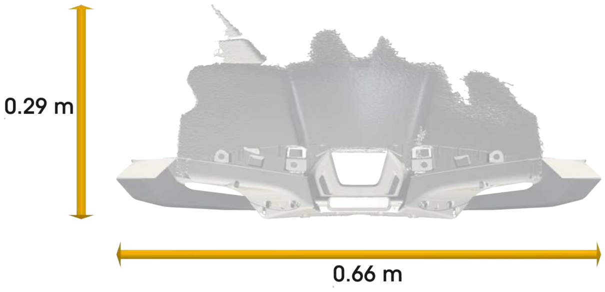


Figure 2: 3D model back view

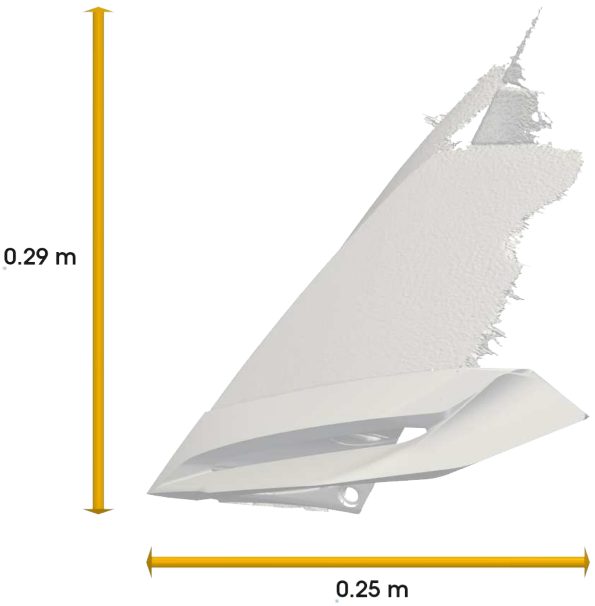


Figure 3: 3D model left view

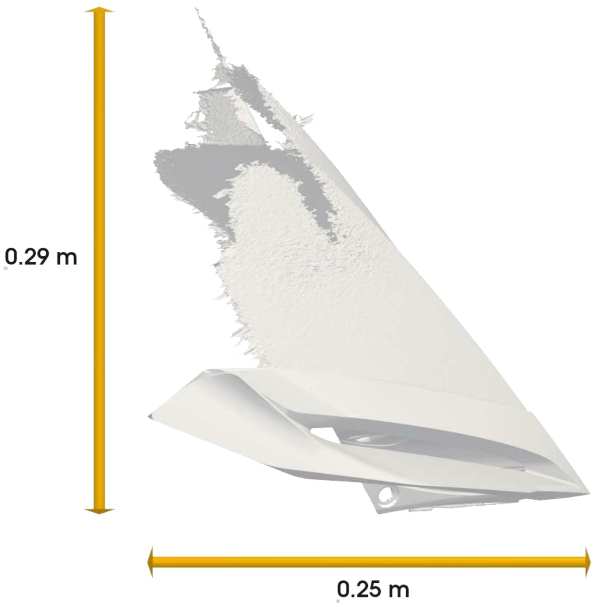


Figure 4: 3D model right view

Definitions

Coordinate system

Forces and moments are defined according to the right-hand coordinate system shown in figure 5. The original coordinate system of the 3D-model(s) is adopted, with positive drag force along the x-axis and positive lift force along the z- axis.
Roll, pitch and yaw moments are calculated around the corresponding coordinate axes.
The yaw angle is defined as the angle between the wind vector and the model x-axis and is positive counterclockwise.

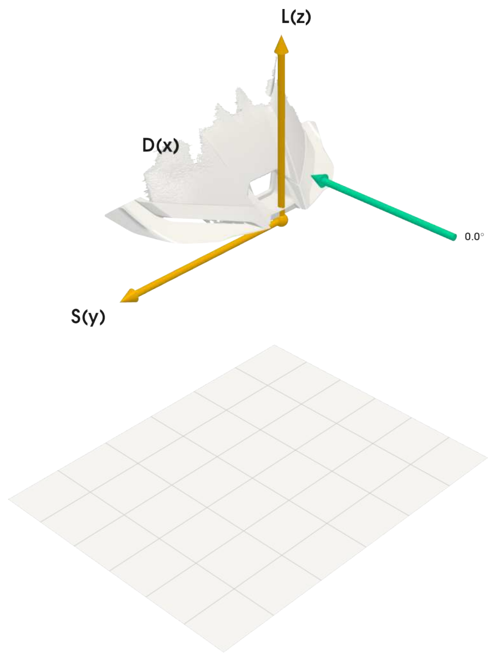


Figure 5: Coordinate system and wind direction

Nomenclature

Symbols		Subscripts	
A	Projected frontal area	d	Drag
C	Coefficient	l	Lift
F	Force	p	Pitch
L	Length scale	r	Roll
M	Moment	s	Side
P	Pressure	t	Total
U	Wind speed	w	Wall
μ	Dynamic viscosity	y	Yaw
ρ	Fluid density	∞	Freestream

Pressure

The total pressure field shows the energy in the flow around an object. The total pressure (P_t) is defined as the sum of the static pressure (Pstatic) and the dynamic pressure ($\frac{1}{2}\rho|u|^2$). Some of that energy will be dissipated into heat as an object is moving relative to a viscous fluid. When the boundary layer separates from an object, a low pressure, turbulent wake will form behind the object, and the resulting pressure difference around the object causes pressure drag. The velocity vector fields are visualized in the pressure plots in the report, by a technique called line integral convolution (LIC). This provides an intuitive visual impression of the flow direction of the decomposed velocity field on the projected plane.

The surface pressure shows the force per area normal to the surface of the body. The surface pressure decomposed along the three coordinate axes and integrated around the surface, gives drag, lift and side pressure force. The surface pressure is highest at the stagnation points, where the velocity is zero, and all the kinetic energy is converted to pressure energy.

Skin friction

Shear stress in a fluid flow is a viscous stress component that exerts a force parallel to the local flow direction due to velocity gradients. The shear stress on a solid boundary is called wall shear stress and can be interpreted as a frictional force which is directed along the local flow direction, tangential to the surface of the body. It is caused by the velocity gradient in the boundary layer next to the surface and fluid viscosity. This causes a drag force on the surface in the direction of the local flow. Areas of low wall shear stress has a low local flow velocity. Along separation lines the wall shear stress is zero. The wall shear stress is normalized by the dynamic free stream pressure to give the non-dimensional skin friction coefficient.

$$C_f = \frac{\tau_w}{\frac{1}{2}\rho U_\infty^2}$$

$$\tau = \frac{F}{A} \qquad \tau_w = \mu \frac{\delta u}{\delta y} y = 0$$

Forces

Fluid dynamic forces on an object are the sum of pressure forces and viscous forces. In the results section, these forces are shown both individually and combined. Pressure forces are dominating for high Reynolds number flows around bluff bodies, while viscous forces are dominating for streamlined bodies and low Reynolds numbers. Forces are calculated along each of the coordinate axes, fixed to the model.
Moments are calculated around each coordinate axes. The original coordinate system of the model is used to calculate the moments of all parts. Moments around each of the drag, lift and side force axes are denoted "roll" (r), "yaw" (y) and "pitch" (p) respectively.

Heat transfer

The convective heat transfer on the surfaces is estimated from the surface friction using the Reynolds analogy. This theory describes a similarity between momentum transfer and heat transfer between a surface and a fluid. It provides an approximation of the heat transfer from the velocity field without solving a thermal energy equation. As it is based on an analogy and not a law of physics the accuracy and applicability of the method is limited and should be used only for first approximations of heat transfer or comparison cases. The heat transfer coefficient (h) is calculated as

$$h = \rho U_\infty C_p \frac{C_f}{2} \qquad C_f = \frac{\tau_w}{\frac{1}{2}\rho U_\infty^2}$$

where Cf is the friction coefficient, τw is wall shear stress, U∞ is free stream velocity and C_p is specific heat capacity at constant pressure. The physical properties of the fluid are evaluated at standard atmospheric conditions. The averaged heat transfer coefficient values are based on the wetted surface area of the model or part. If parts or models are intersecting, this area will be smaller than the area of the original model or part.

Normalizations

Non-dimensional force- and moment coefficients are calculated by normalizing the aerodynamic force (F) and moment (M) components by the freestream dynamic pressure and predefined length scales. The reference area "A" is the projected frontal area and the reference length "L" is 1 meter.

$$C_F = \frac{F}{½\rho_\infty U_\infty^2 A} \qquad C_M = \frac{M}{½\rho_\infty U_\infty^2 AL}$$

Non-dimensional pressure coefficients are calculated by normalising static and total pressure by the freestream dynamic pressure (as shown in the equations below). Note that the ambient reference pressure in the simulations is 0 Pascal in all cases and hence the pressure coefficients may be negative when the local pressure is lower than the ambient pressure, especially in the wake regions.

$$C_{P_t} = \frac{P_{static} + ½\rho|u|^2}{½\rho_\infty U_\infty{}^2} \qquad C_P = \frac{P_{static}}{½\rho_\infty U_\infty{}^2}$$

A standard fluid density ρ = 1.2250 is used for all normalizations. The freestream wind speed (U = 66.6600 m/s) is used for normalization for all yaw angles, implying that the model aligned wind will decrease for increasing yaw angle.

Drag pressure

The drag pressure visualizes the local drag force density on the surface of a body. It is composed of the sum of the drag axis (x-axis) component of the local surface pressure and the wall shear stress vector and hence includes both the pressure drag and friction drag contributions. The purpose of this field is to identify regions of the body where drag force is generated. The drag pressure is normalized by the dynamic free stream pressure to give the non-dimensional drag pressure coefficient:

$$C_{P_x} = \frac{\tau_{w_x} + P_x}{\frac{1}{2}\rho U_\infty^2}$$

Streamlines

Streamlines show the trajectory of a fluid particle as it moves downstream in a fluid flow. They can be used to show the overall structure of the flow field around a body and the extent of the wake. To visualize the streamlines, we add a seeding array of weightless particles into the flow upstream of the object, like smoke traces are used to visualize airflow in a wind tunnel. In areas of accelerated flow, the streamlines come closer to each other and in decelerated regions they will separate. The colours of the streamlines correspond to the total flow velocity.

Recirculation

As flow separation occurs, regions of low pressure are formed downstream. In this region recirculating vortices are formed. These low-pressure zones on the leeward side of a body causes a suction contributing to the drag force. The coloured regions in the recirculation figures indicate volumes of a negative x-axis velocity component.

Results

The overall results are summarized in tables 1, 2, 3, 4 and 5. Details of the flow field and forces per part for each individual CFD-simulation are presented in the following subchapters.

Forces and moments

Table 1: Aerodynamic forces and moments on the model coordinate system

Yaw [°]	Fd [N]	Fs [N]	Fl [N]	Mr [Nm]	Mp [Nm]	My [Nm]
0	238.07	8.09	-258.93	-0.51	40.58	2.79

Force and moment coefficients

Table 2: Aerodynamic force- and moment coefficients on the model coordinate system

Yaw [°]	Cd [-]	Cs [-]	Cl [-]	Cr [-]	Cp [-]	Cy [-]
0	6.350	0.216	-6.906	-0.014	1.082	0.074

Force area

Table 3: Aerodynamic force coefficient x frontal area along model coordinate axes

Yaw [°]	CdA [m²]	CsA [m²]	ClA [m²]
0	0.087	0.003	-0.095

Force breakdown

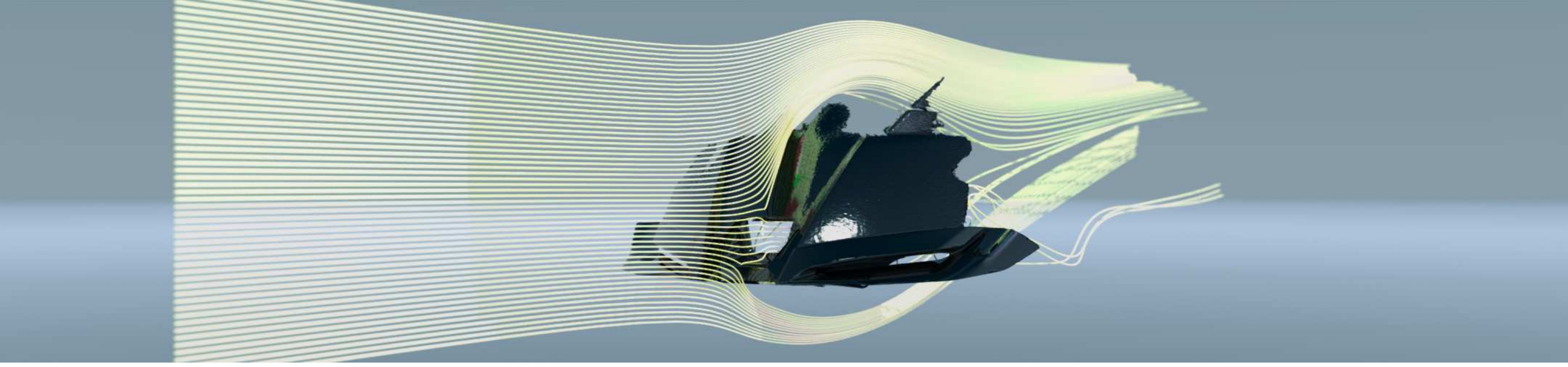
Table 4: Pressure forces, viscous forces and porous forces on model coordinate system

	Pressure force			Viscous force			Porous force		
Yaw [°]	Fd [N]	Fs [N]	Fl [N]	Fd [N]	Fs [N]	Fl [N]	Fd [N]	Fs [N]	Fl [N]
0	236.83	8.01	-259.42	1.23	0.09	0.49	0.00	0.00	0.00

Heat transfer

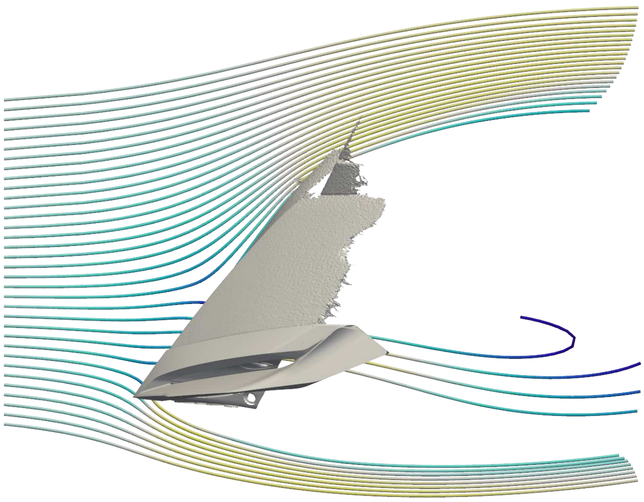
Table 5: Average convective heat transfer

Yaw [°]	Heat transfer coefficient [W/m²K]	Total heat transfer [W/K]	Surface area [m²]
0	73.604	40.569	0.551

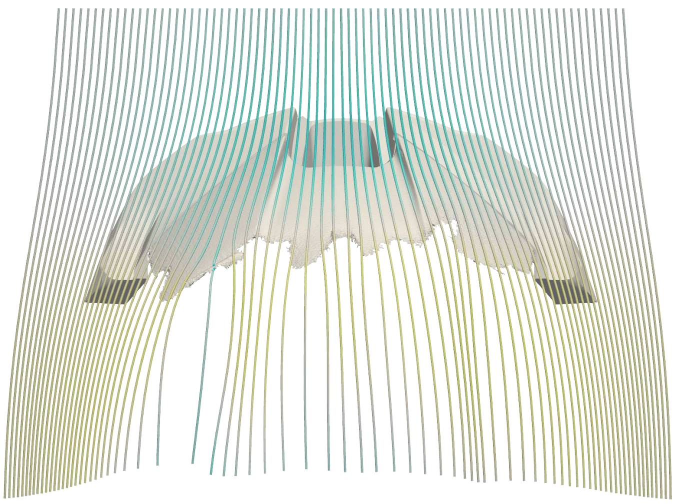


Simulation details: Yaw angle = 0°

Streamlines



(a) left view



(a) top view

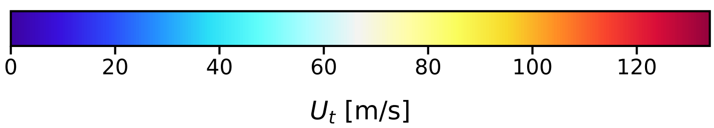
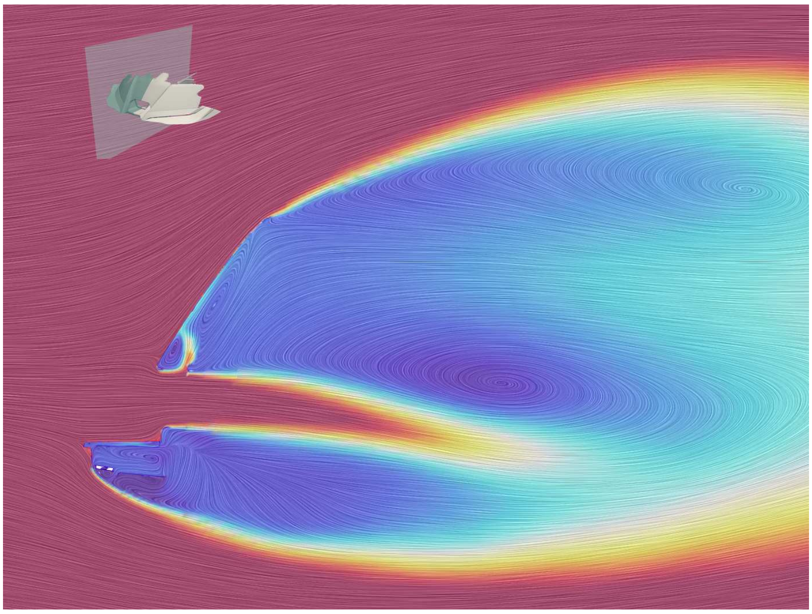
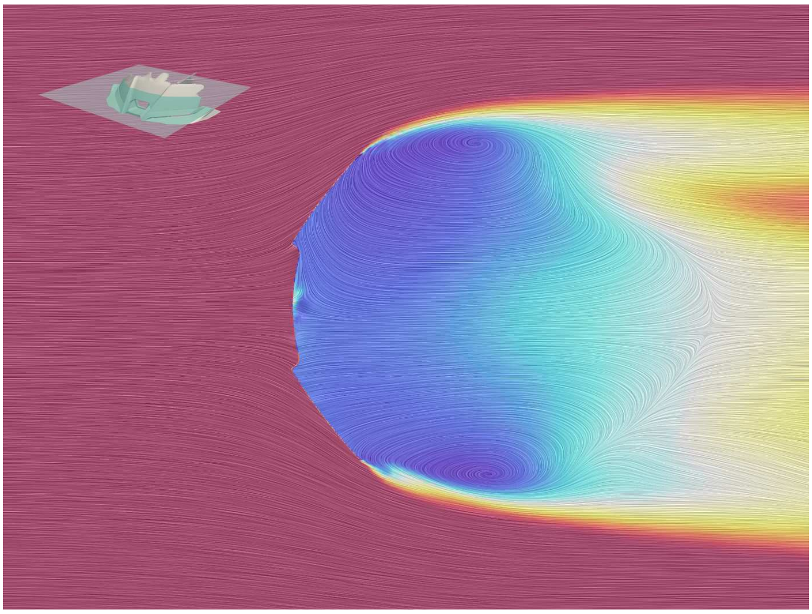


Figure 6: Streamlines

Pressure field



(a) centre XZ-plane



(b) centre XY-plane

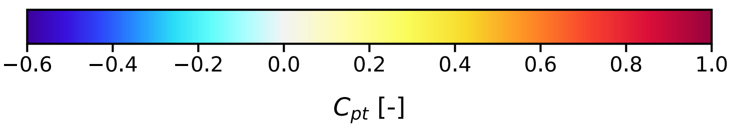
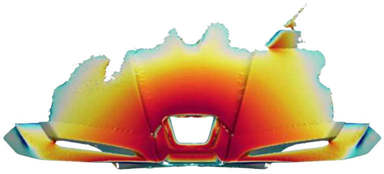
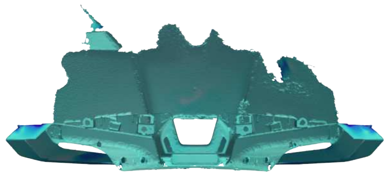


Figure 7: Total pressure coefficient (Cpt) LIC-plot

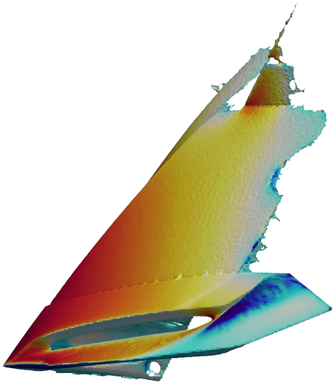
Surface pressure



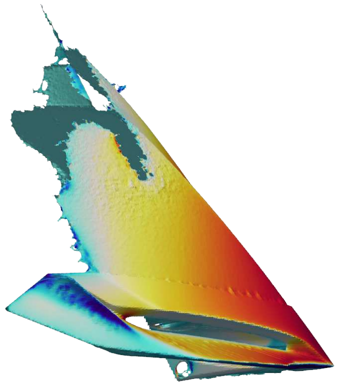
(a) front view



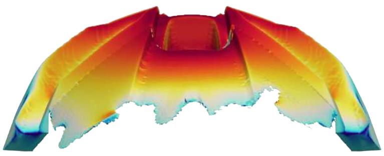
(b) back view



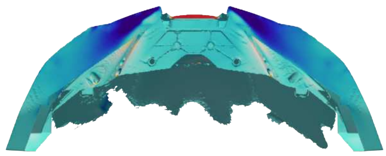
(c) left view



(d) right view



(e) top view



(f) bottom view

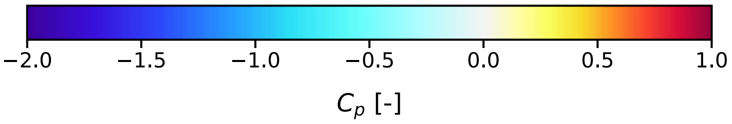
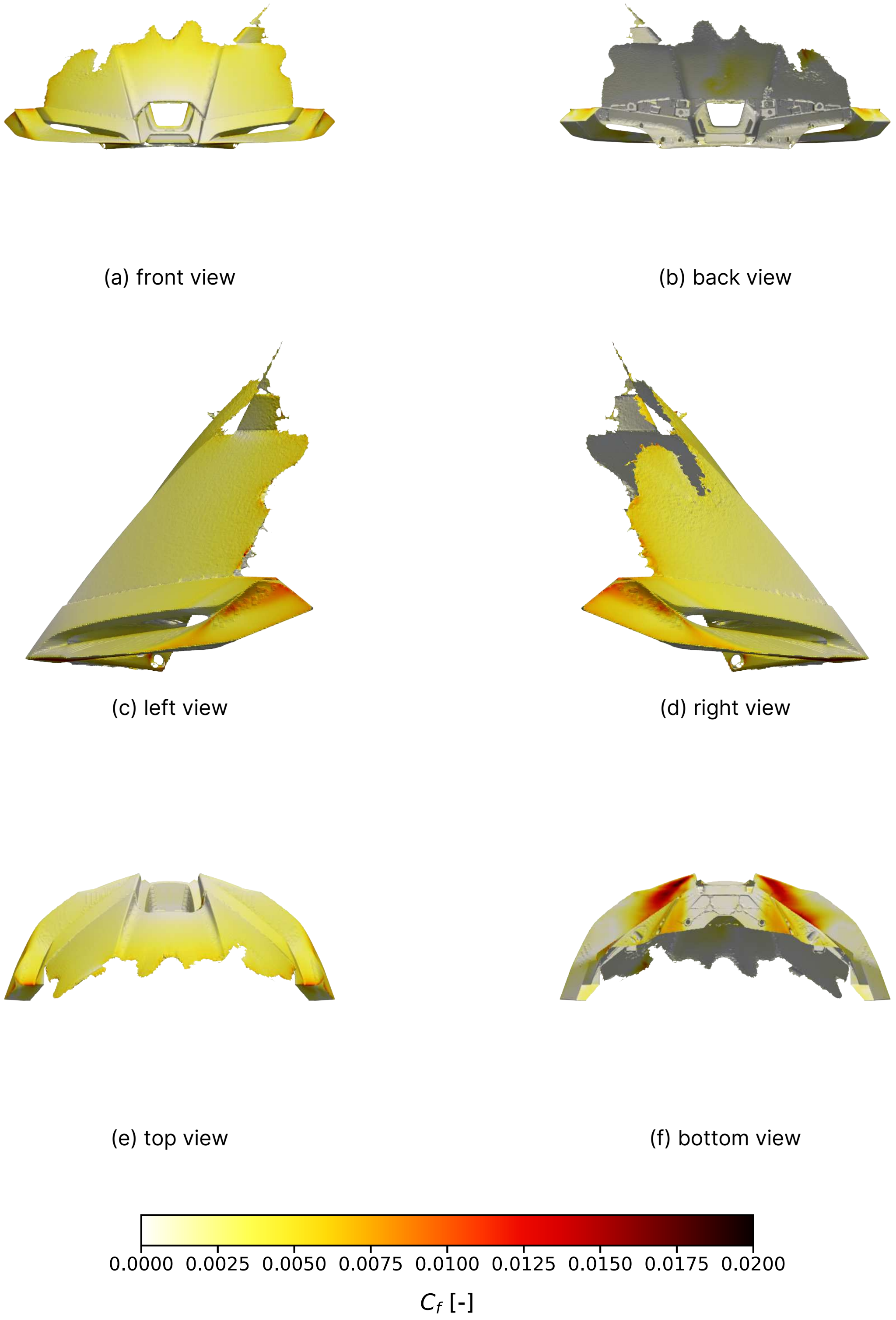


Figure 8: Surface pressure coefficient

Surface friction



(a) front view

(b) back view

(c) left view

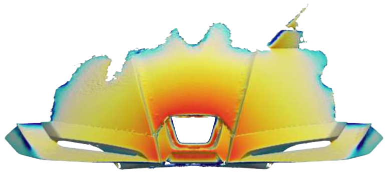
(d) right view

(e) top view

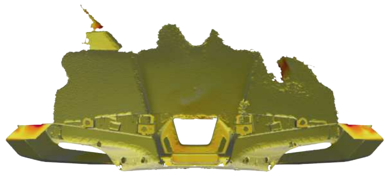
(f) bottom view

Figure 9: Skin friction coefficient

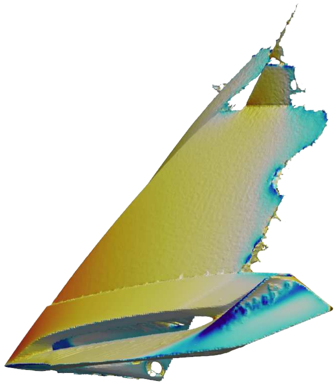
Drag pressure



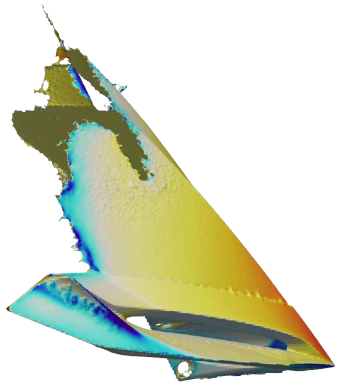
(a) front view



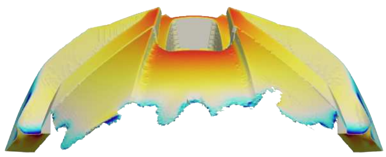
(b) back view



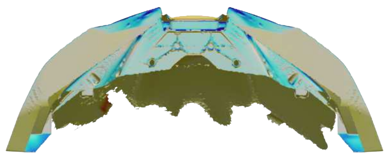
(c) left view



(d) right view



(e) top view



(f) bottom view

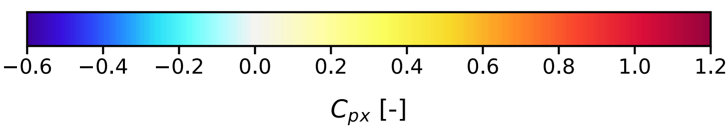
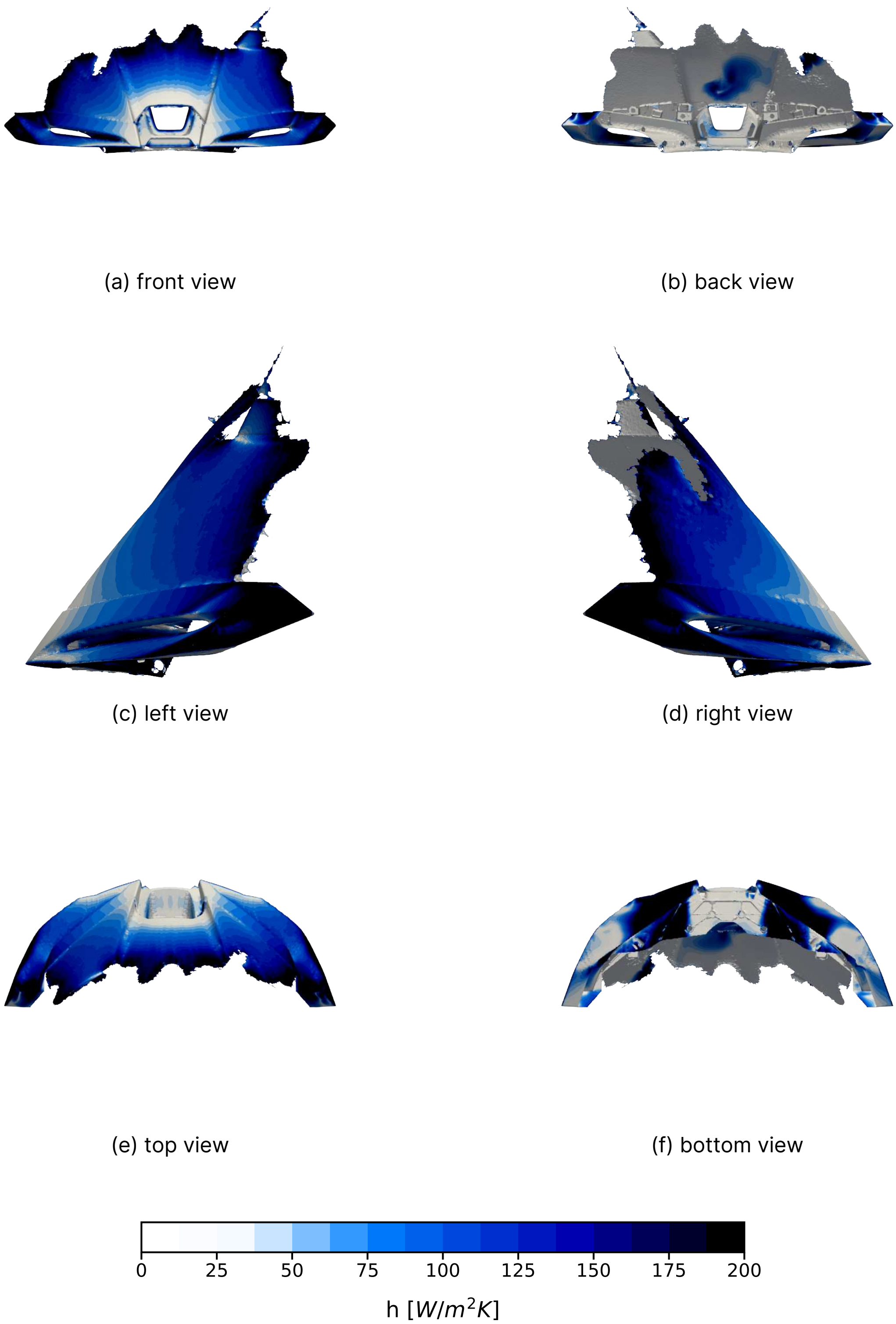


Figure 10: X-axis pressure coefficient

Heat transfer



(a) front view (b) back view

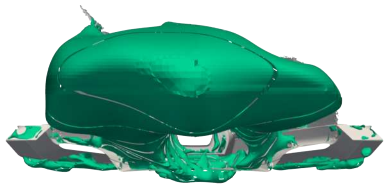
(c) left view (d) right view

(e) top view (f) bottom view

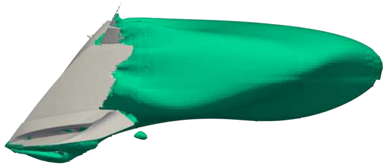
Figure 11: Convective heat transfer coefficient

Recirculation zones

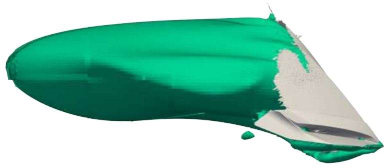
(a) front view



(b) back view



(c) left view



(d) right view

(e) top view

(f) bottom view

Figure 12: Recirculation zones

Wake development

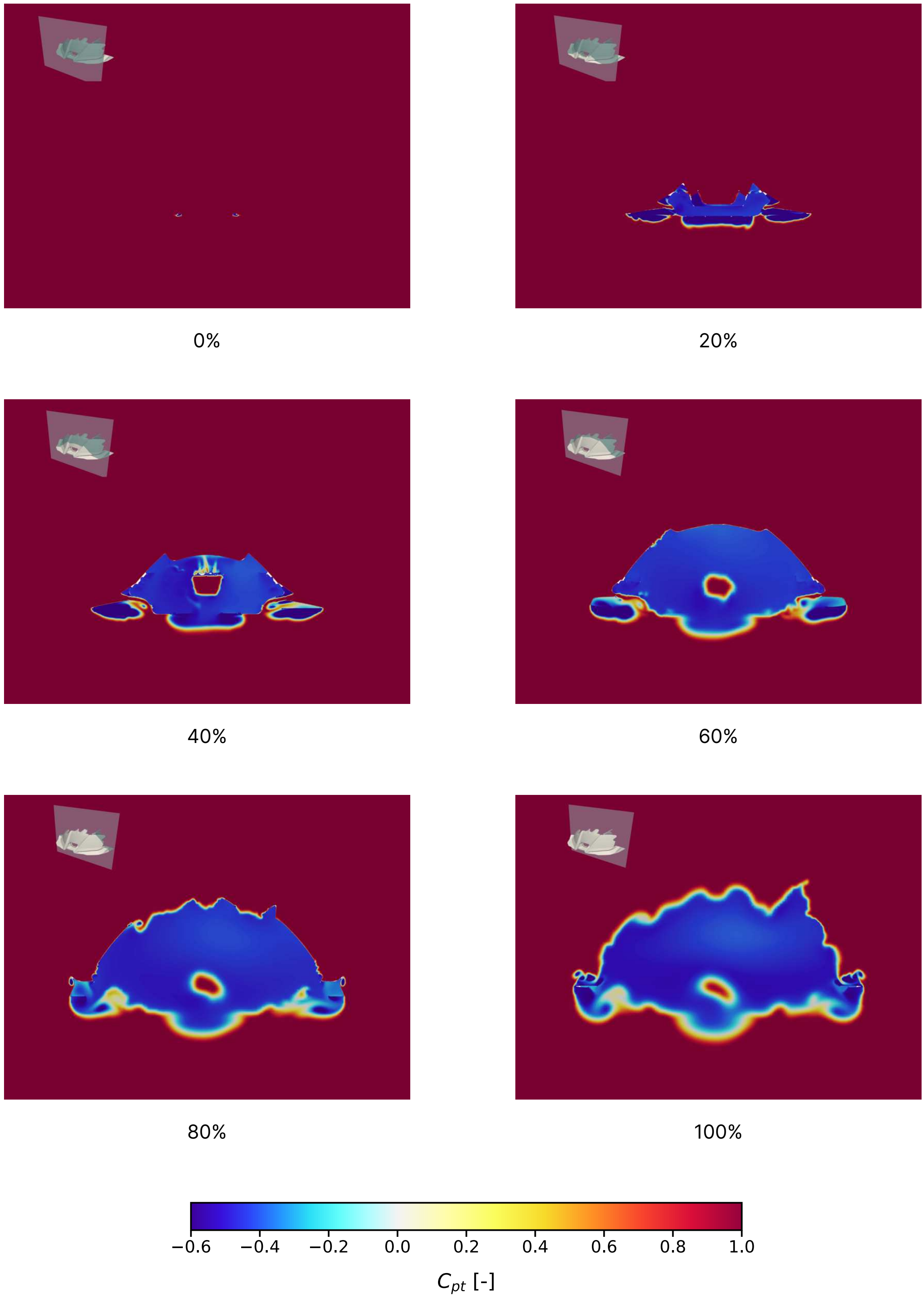


Figure 13: Wake development of total pressure coefficient in the x-direction

Force distribution x-axis

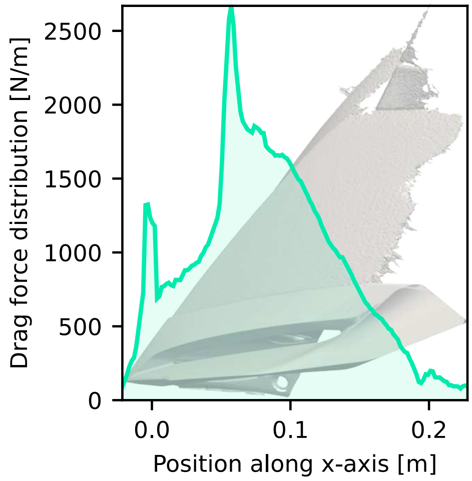


Figure 14: Drag force along x-axis

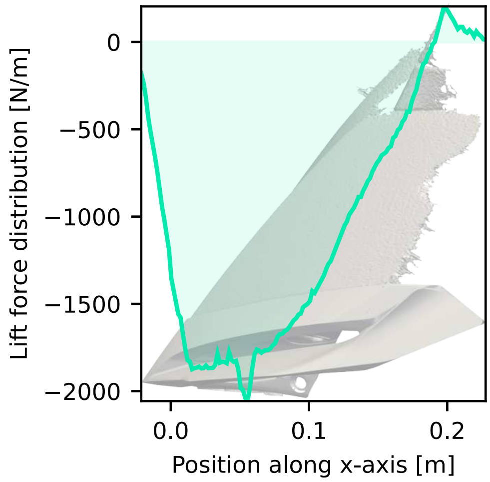


Figure 15: Lift force along x-axis

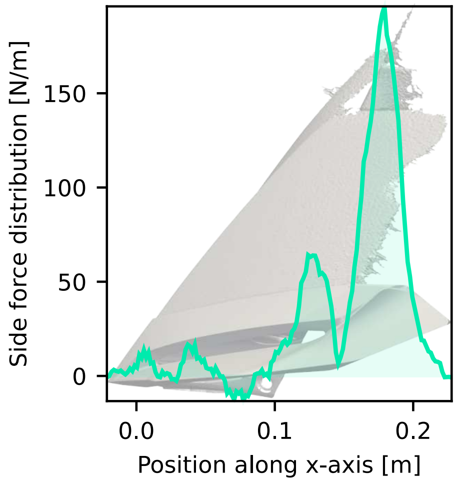


Figure 16: Side force along x-axis

Cumulative force distribution x-axis

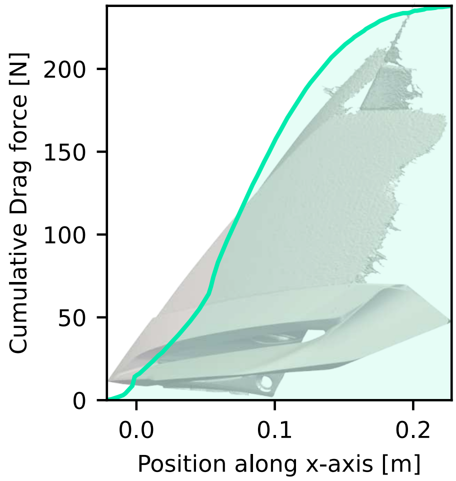


Figure 17: Cumulative drag force along x-axis

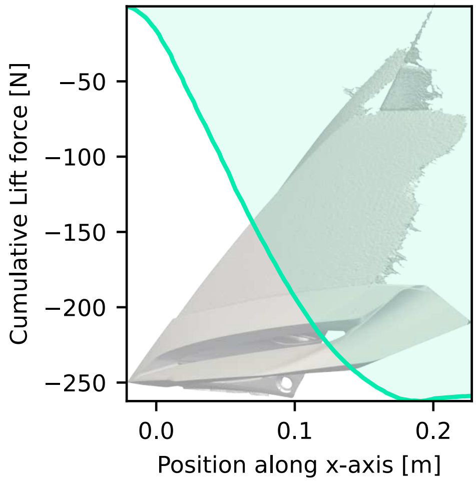


Figure 18: Cumulative lift force along x-axis

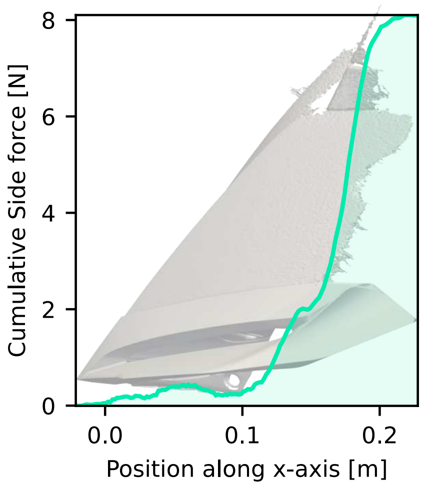


Figure 19: Cumulative side force along x-axis

Force distribution y-axis

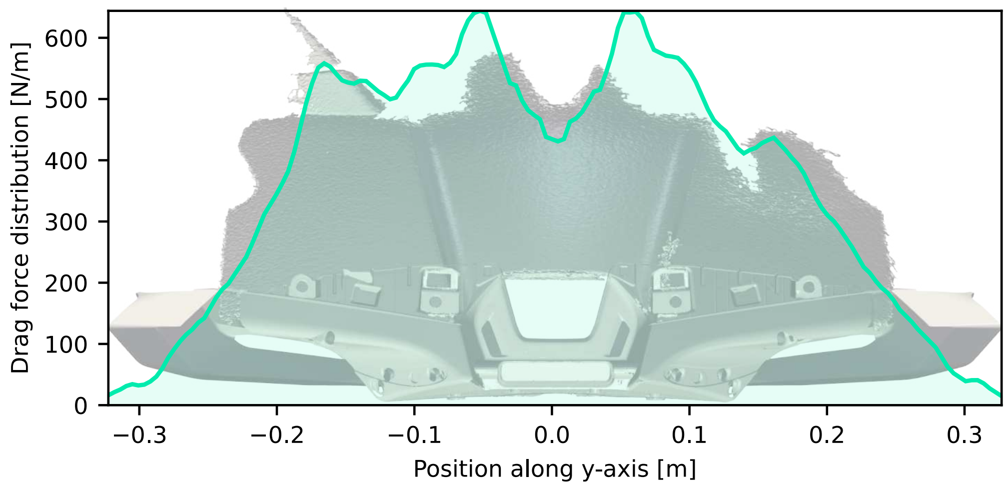


Figure 20: Drag force along y-axis

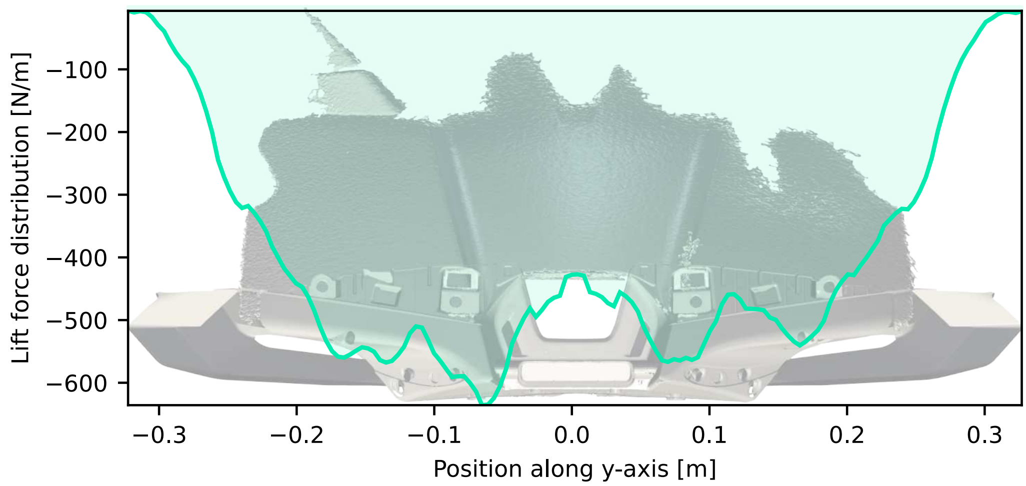


Figure 21: Lift force along y-axis

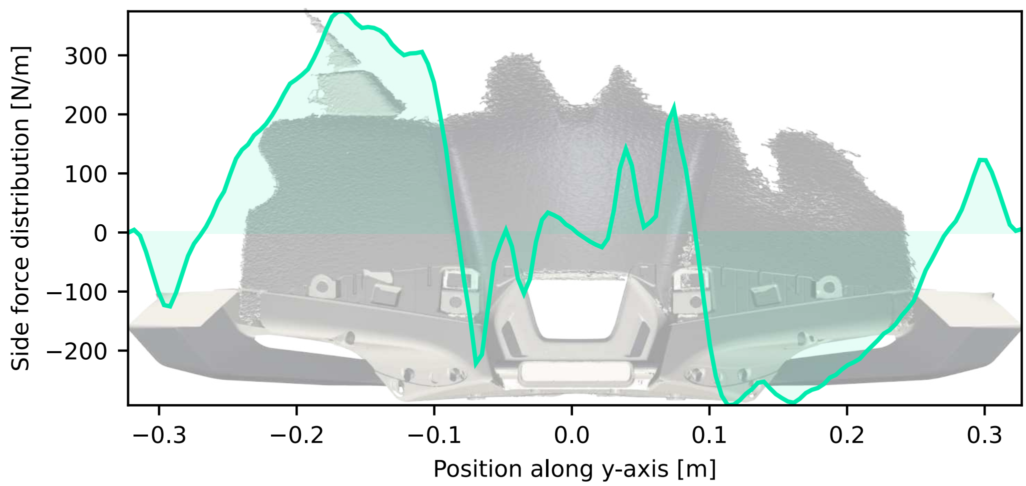


Figure 22: Side force along y-axis

Force distribution z-axis

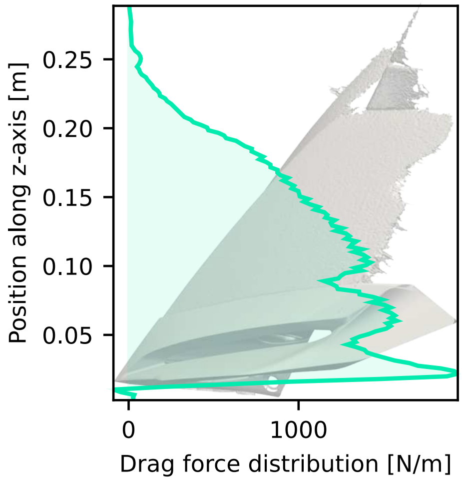


Figure 23: Drag force along z-axis

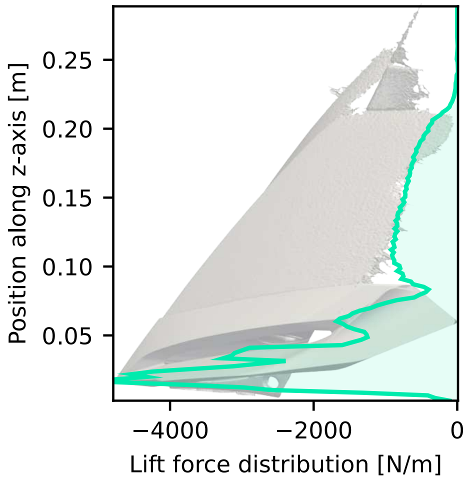


Figure 24: Lift force along z-axis

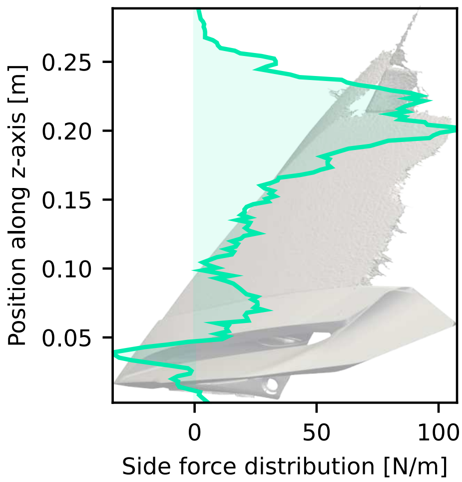


Figure 25: Side force along z-axis

Flow monitor surfaces

Table 6: Flow monitor functions and values

Name	Function	Value	Surface area

NablaFlow AS

Company address	Sverdrups gate 27 4007 Stavanger
E-mail	info@nablaflow.io
Website	https://nablaflow.io

Appendix A: Model settings

Part name	File name	File unit	Surface area [m^2]	Refined	Rotating	Porous	Porosity coefficients
Aerodinamica_ver4	Aerodinamica_ver4.stl	mm	0.1538				