Chassis and Spoiler Aerodynamics in Legends Car Racing

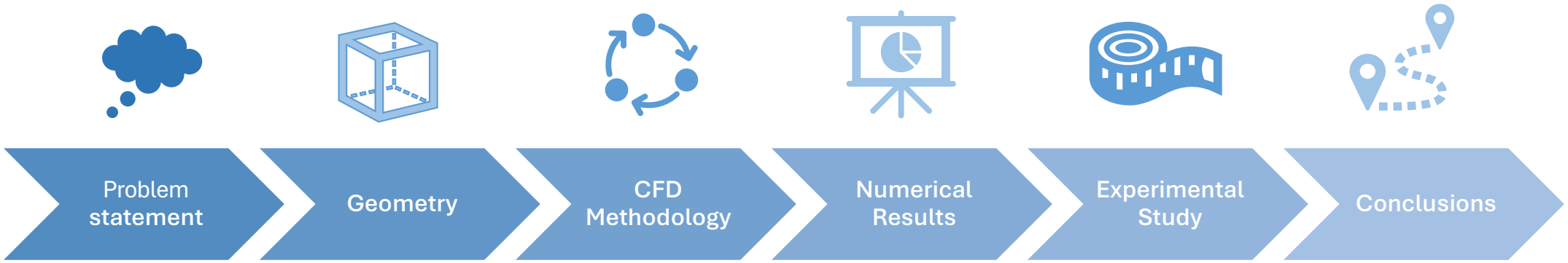
Matteo Baio
Gaia Lapucci
Laura Livoti
Thomas Magni

Aerodynamics of Transport Vehicles

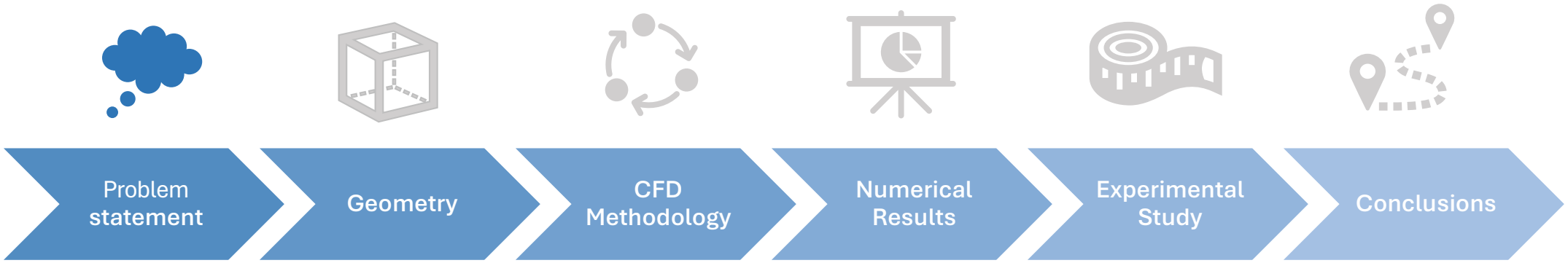
Academic year 2024-2025

Prof. Alex Zanotti
Prof. Paolo Schito

Presentation outline



Presentation outline



Problem Statement

Legends Cars :

- 5/8-scale race cars;
- styled after American automobiles from the 1930s and 1940s;
- built on a tubular chassis;
- powered by a motorcycle engine.

Legends Car rising popularity led them to compete on the most famous italian tracks.

With higher velocities, the aerodynamic loads impact is no longer negligible.

Legends car race at Monza Circuit

Legends car at Castelletto Circuit

Problem Statement

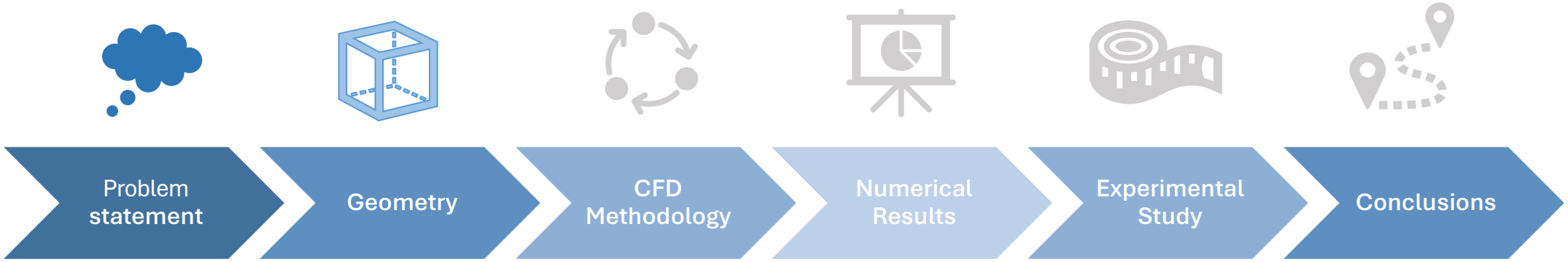
How much is the impact of the two different chassis on aerodynamic loads ?

- CFD simulations to study aerodynamic differences between the configurations;
- Aerodynamic effect of a roof spoiler;
- Experimental study conducted through tuft testing.

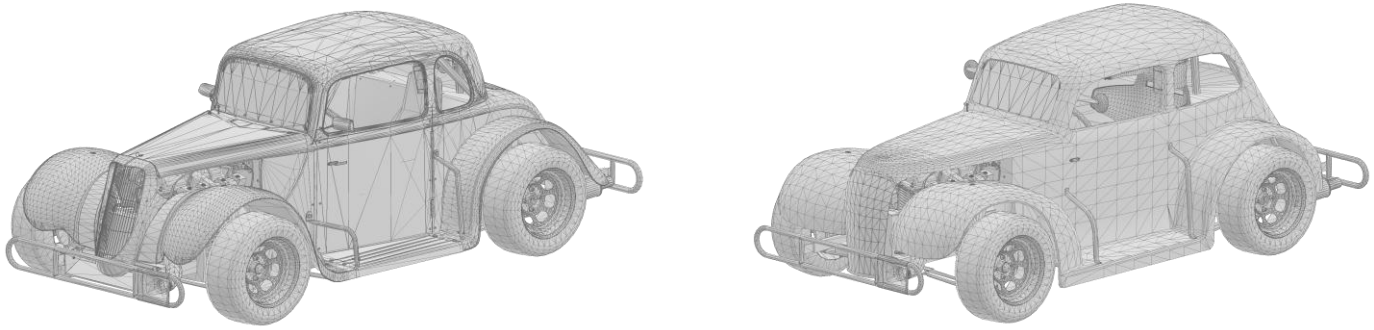
Legends Ford 34 Coupé

Legends Chevrolet 37 Sedan

Presentation outline



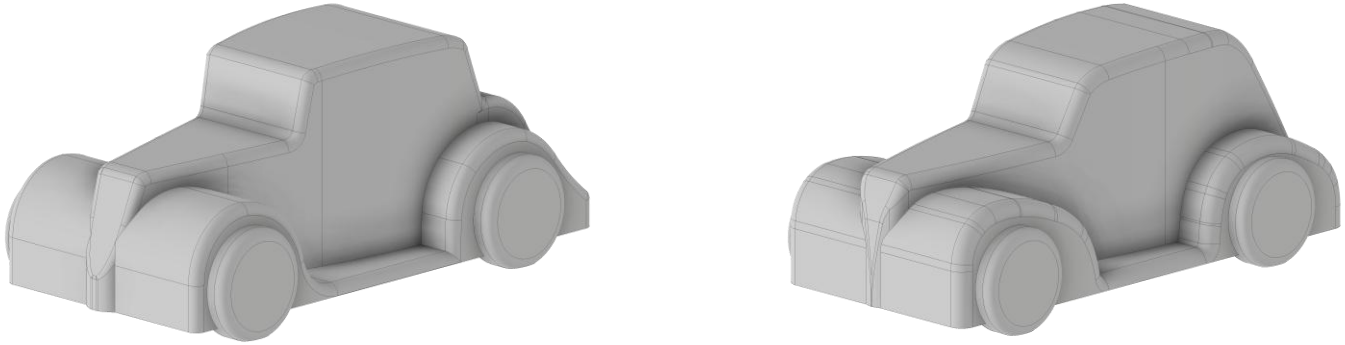
Geometry



Original CAD models from Assetto Corsa: Coupé (left) and Sedan (right)

CAD simplifications:

- Closure of cavities and air intakes;
- Flattening of the car underbody;
- Removal of external features.

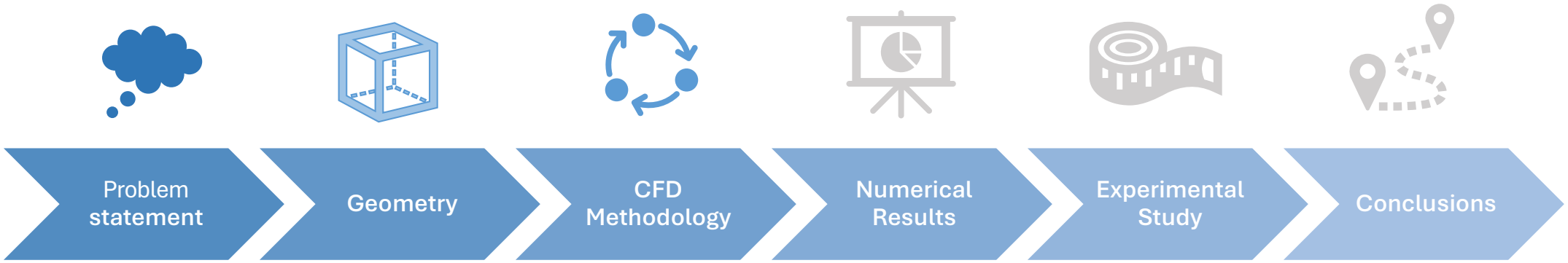


Simplified CAD models: Coupé (left) and Sedan (right)

Assetto Corsa Simulator

Inventor

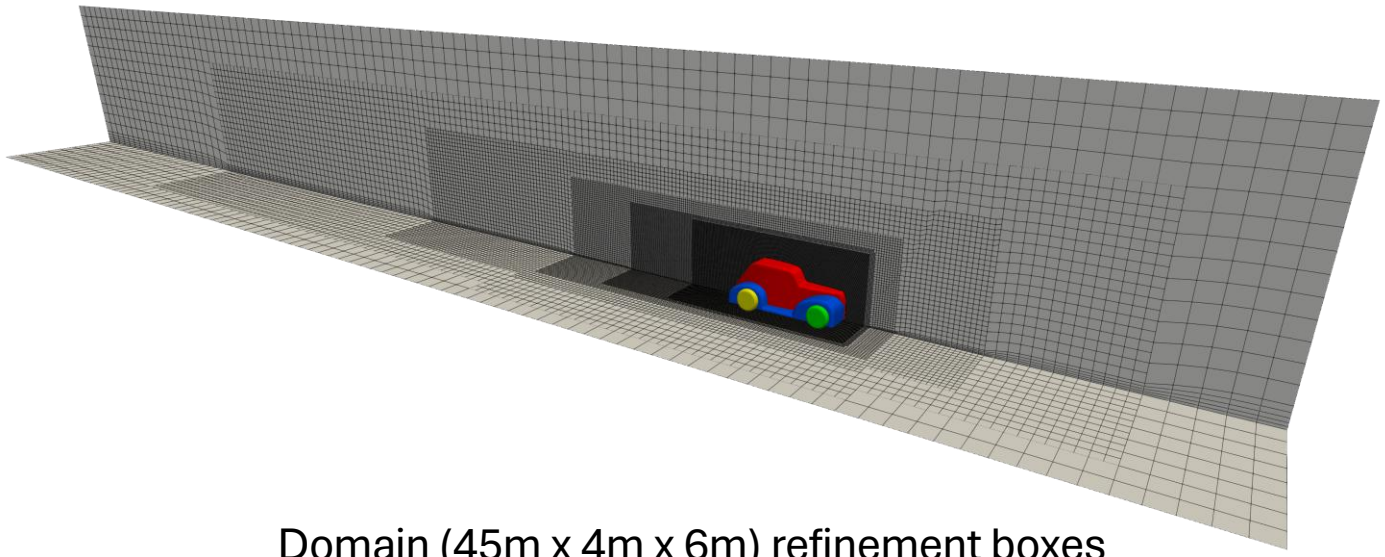
Presentation outline



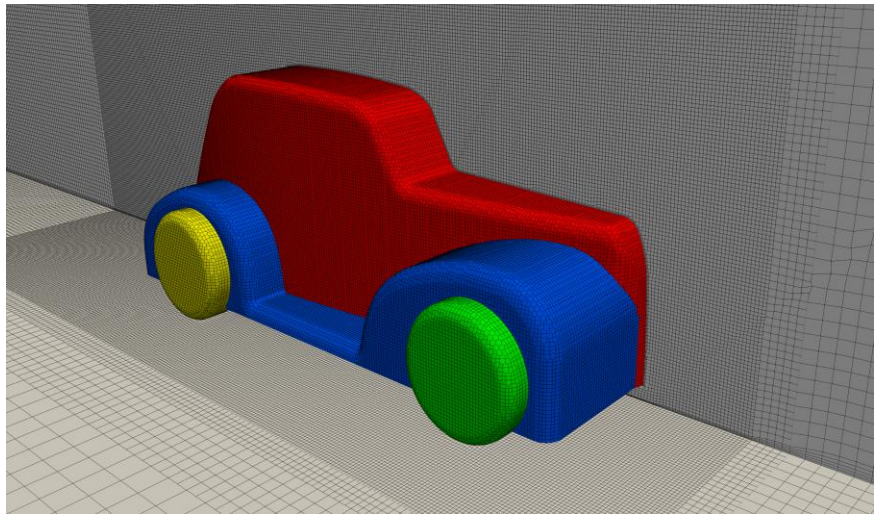
CFD Methodology

Mesh Generation:

- *SnappyHexMesh* tool;
- High-quality tetrahedral-dominant 3D elements;
- Octree-based mesh refinement approach;
- Five refinement boxes.



Domain (45m x 4m x 6m) refinement boxes



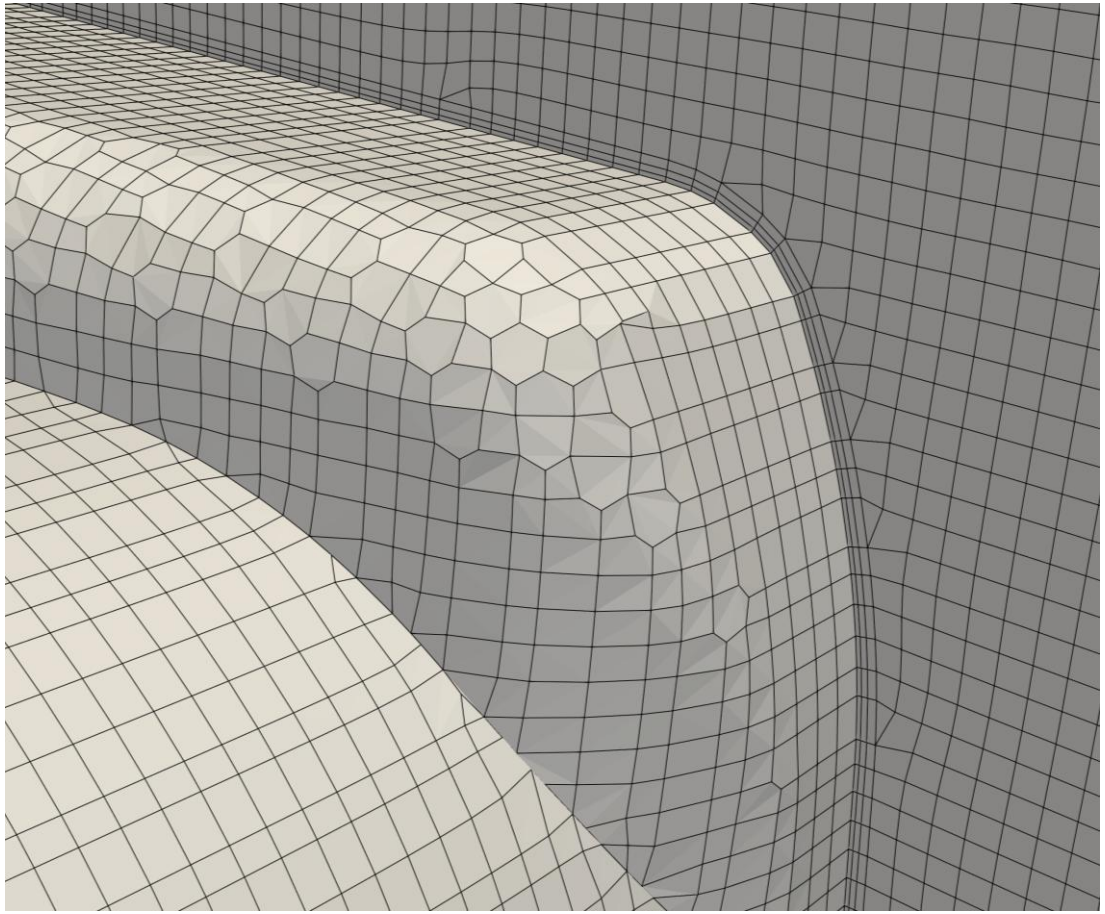
Multiple *patches* vehicle with surface mesh and inflation layer

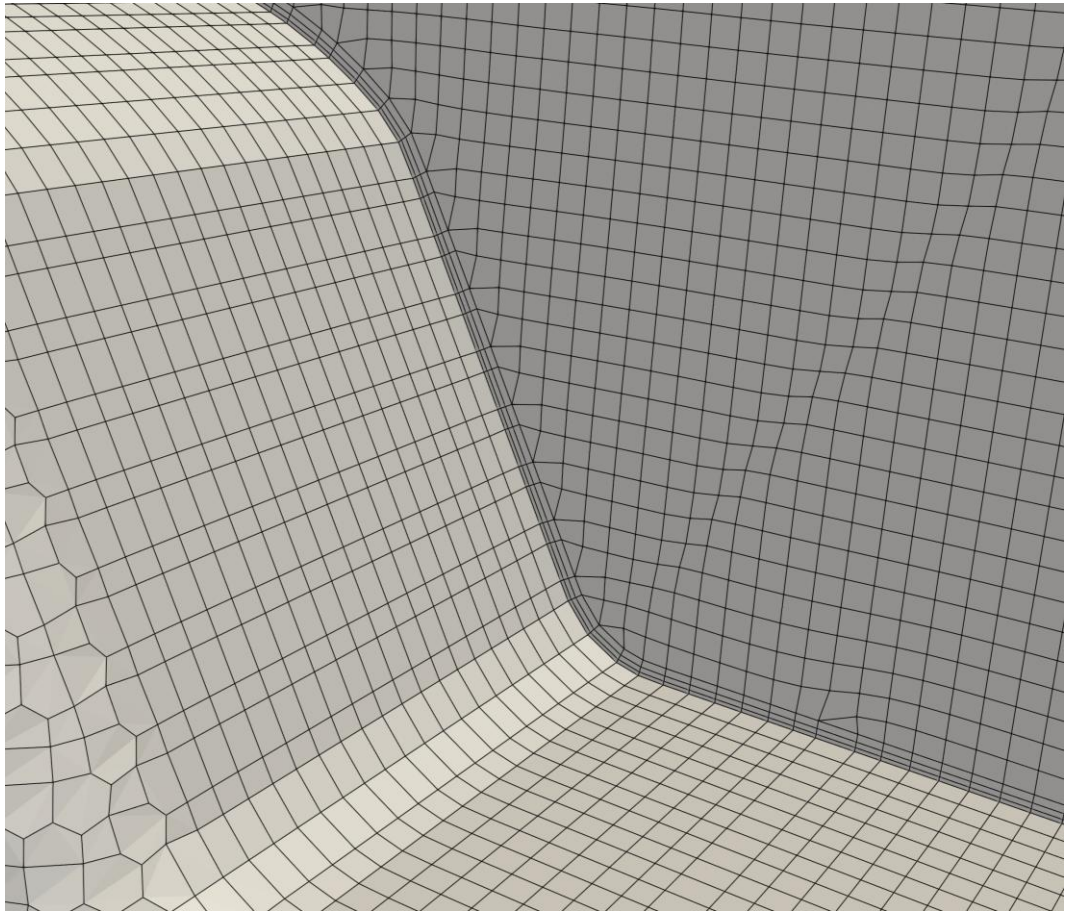
OpenFOAM

https://doc.openfoam.com/2306/tools/processing/Solvers/rtm/incompressible/simpleFoam/

CFD Methodology

Details of surface mesh and *inflation layer*

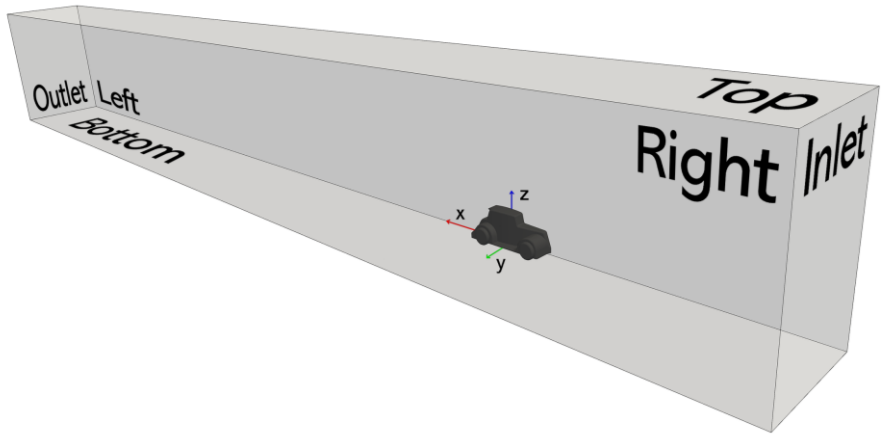




CFD Methodology

Numerical Setup:

- Second-order schemes;
- *Rotating wall* boundary condition for the wheels;
- Symmetry condition;
- Turbulence model: *k-w* SST;
- Application of *wall functions*.



Domain boundary box

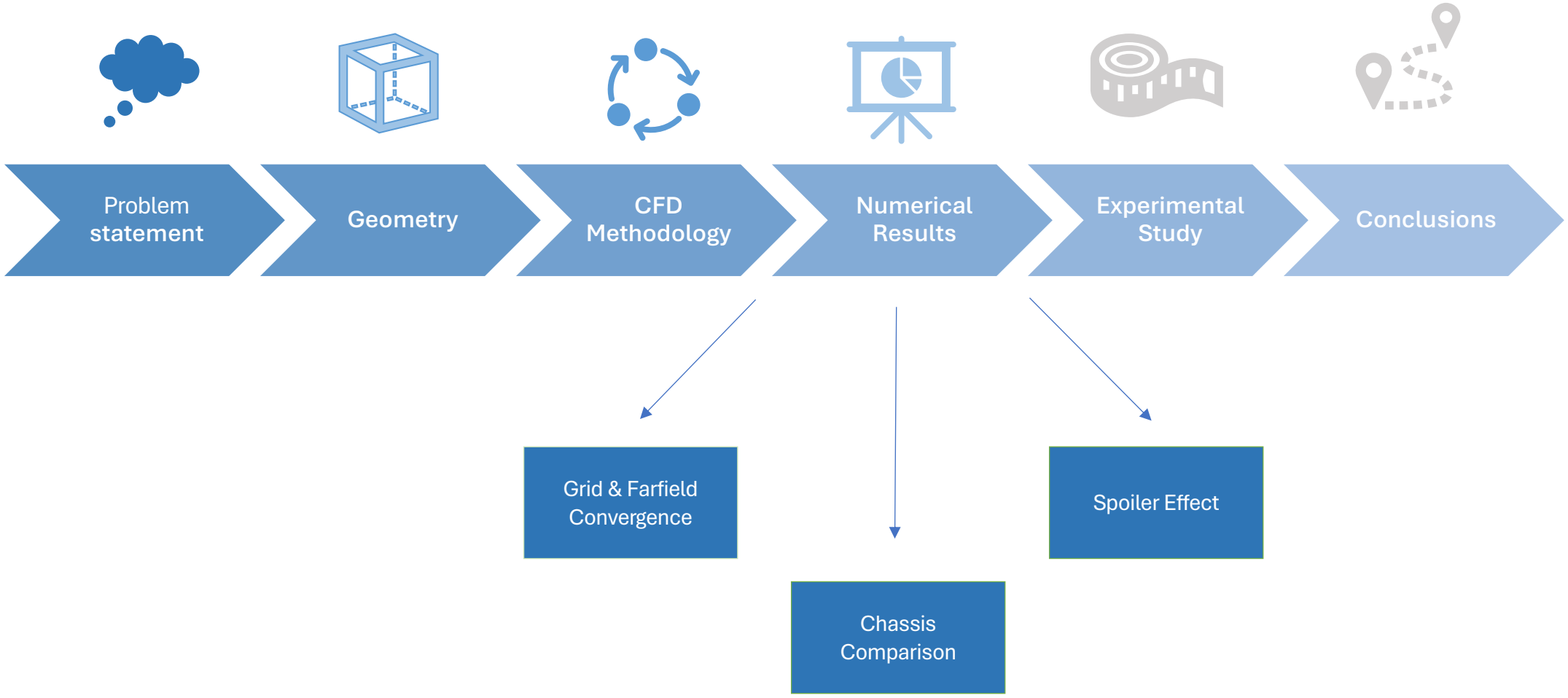
Region	Condition
Freestream velocity, U_∞	$17.5\,\mathrm{m/s}$
Tyres angular speed, ω_∞	$60.34\,\mathrm{rad/s}$
Relative pressure, P_{rel}	$0\,\mathrm{Pa}$
Reynolds number, Re	2.6497×10^6
Air density, ρ	$1.225\,\mathrm{kg/m^3}$
Turbulent energy, k	$0.0459\,\mathrm{m^2/s^2}$
Turulent frequency, ω	$0.6329\,\mathrm{rad/s}$
Kinematic viscosity, μ	$1.5 \times 10^{-5}\,\mathrm{N\,s/m}$
Wheelbase length, L_c	$1.854\,\mathrm{m}$
Sedan frontal area, A^S_{ref}	$0.665\,\mathrm{m^2}$
Coupé frontal area, A^C_{ref}	$0.645\,\mathrm{m^2}$

Main parameters used in CFD simulations

Region	Condition
Inlet	Fixed speed U_∞
Top, Right, Outlet	Freestream
Left	Symmetry
Bottom	Moving wall U_∞
Chassis	No slip
Tyres	Rotating wall ω_∞

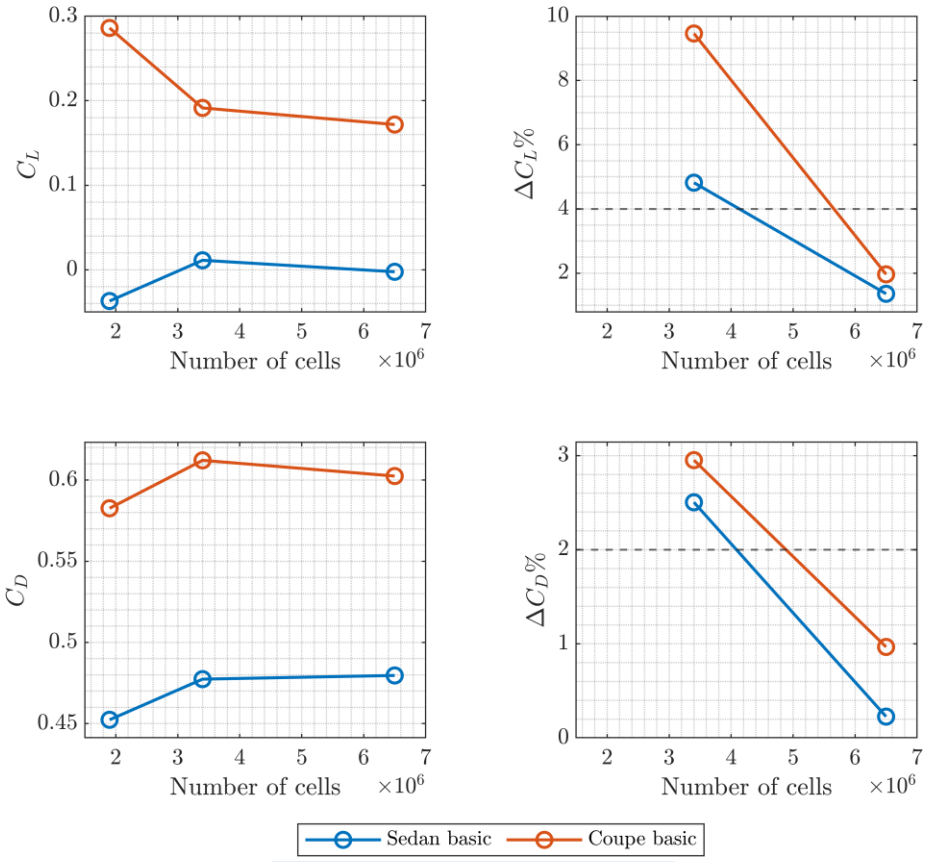
Boundary conditions applied to the domain

Presentation outline



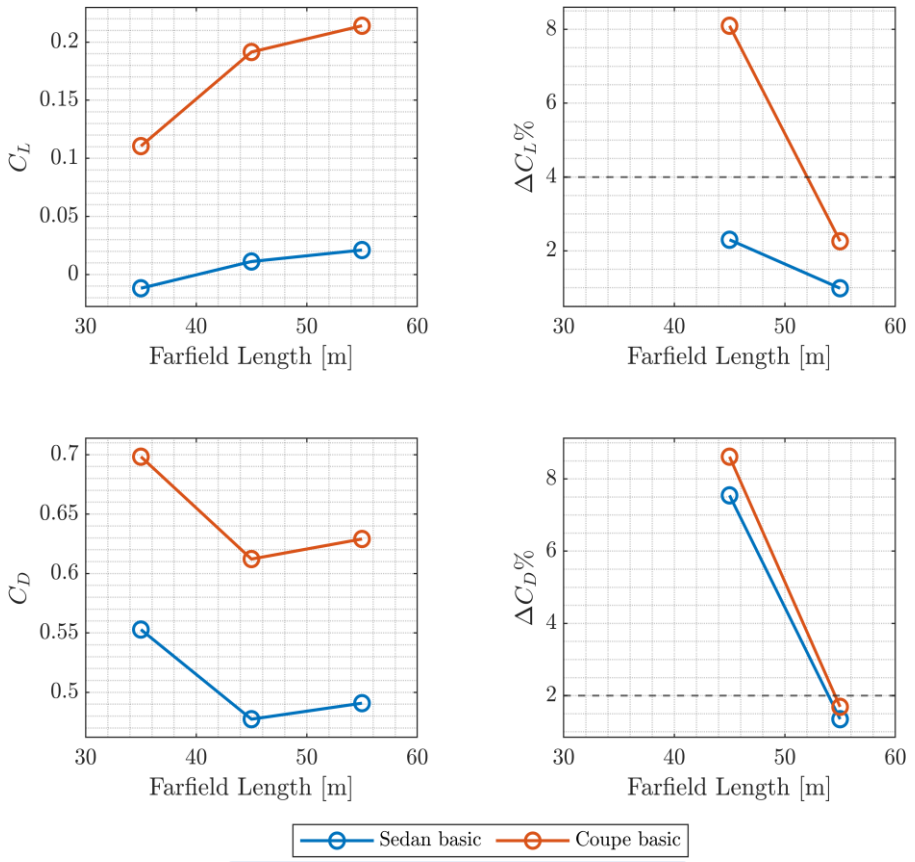
Numerical Results

Grid Convergence



N. of cells: 3.4 M

Farfield Convergence



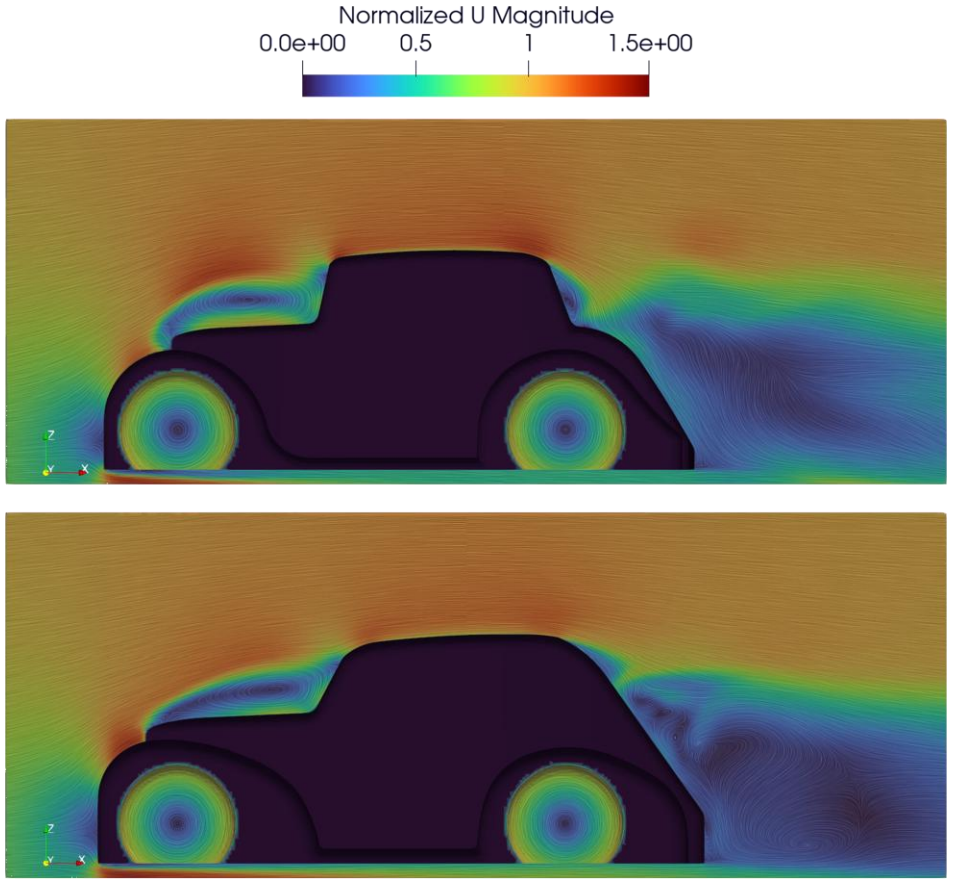
Farfield Length: 45 m

Numerical Results

Chassis Comparison

Coupé

Sedan



LIC visualization with the normalized velocity field (Y = 0.2 m)

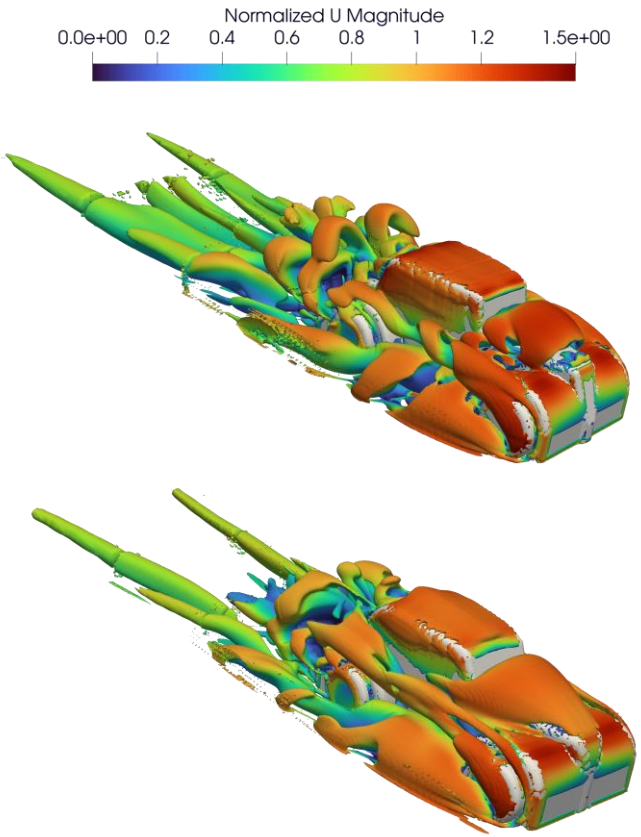
Patch	C_L	C_D	% of C_D
Upper Body	0.163	0.180	29.36%
Lower Body	0.018	0.379	61.98%
Front Tyre	0.012	0.022	3.66%
Rear Tyre	-0.002	0.031	5.00%
TOTAL	0.191	0.612	100%

Patch	C_L	C_D	% of C_D
Upper Body	0.014	0.102	21.44%
Lower Body	-0.011	0.323	67.69%
Front Tyre	0.010	0.021	4.31%
Rear Tyre	-0.001	0.031	6.56%
TOTAL	0.011	0.477	100%

Basic configuration drag breakdown

Numerical Results

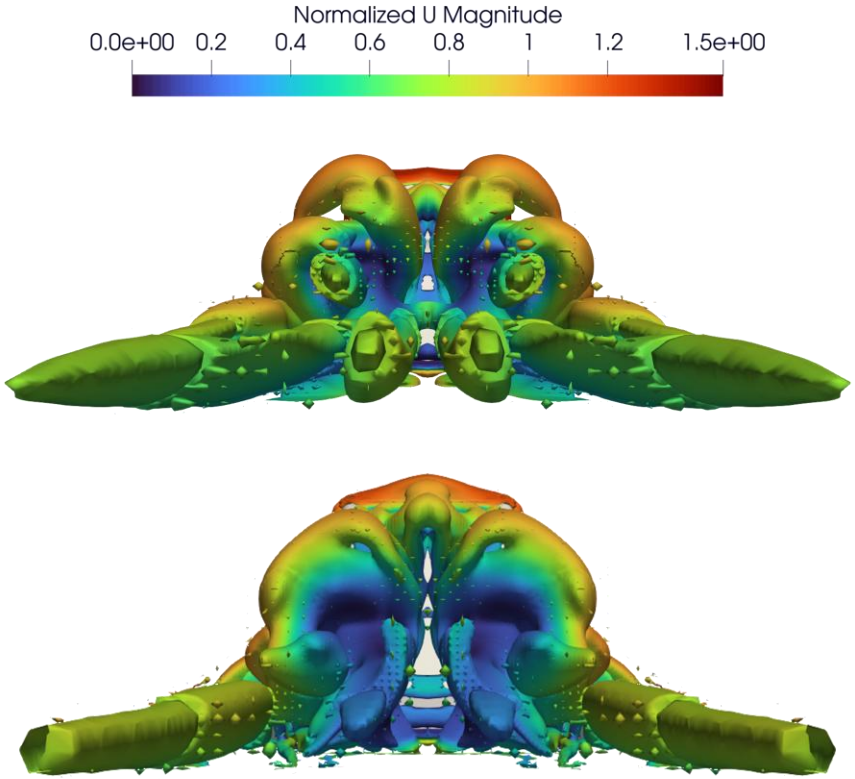
Chassis Comparison



Q-criterion isosurfaces with the normalized velocity field

Coupé

Sedan

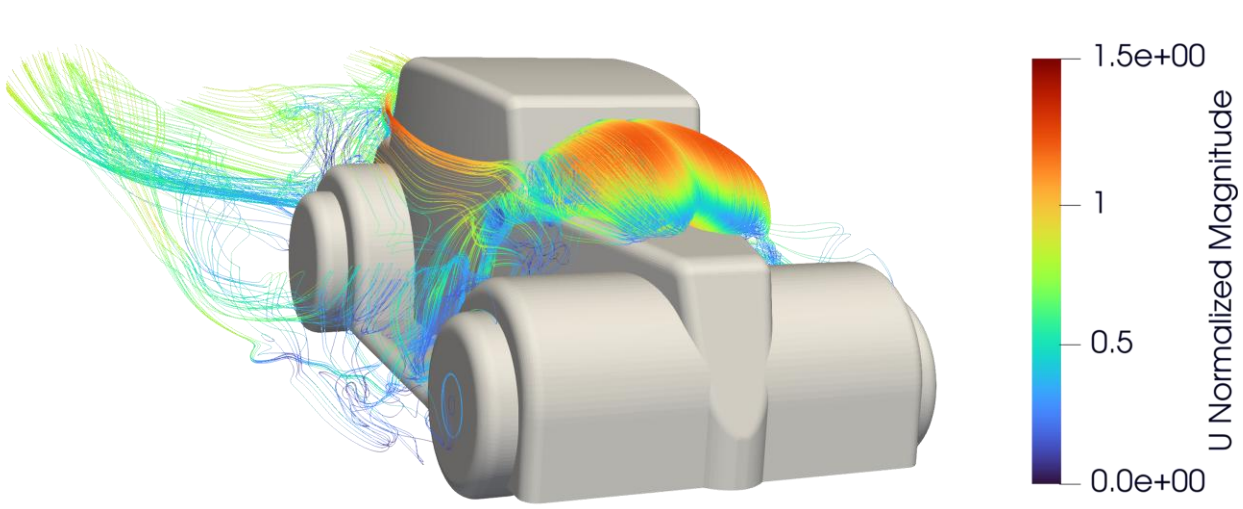


Q-criterion isosurfaces behind the vehicles with the normalized velocity field

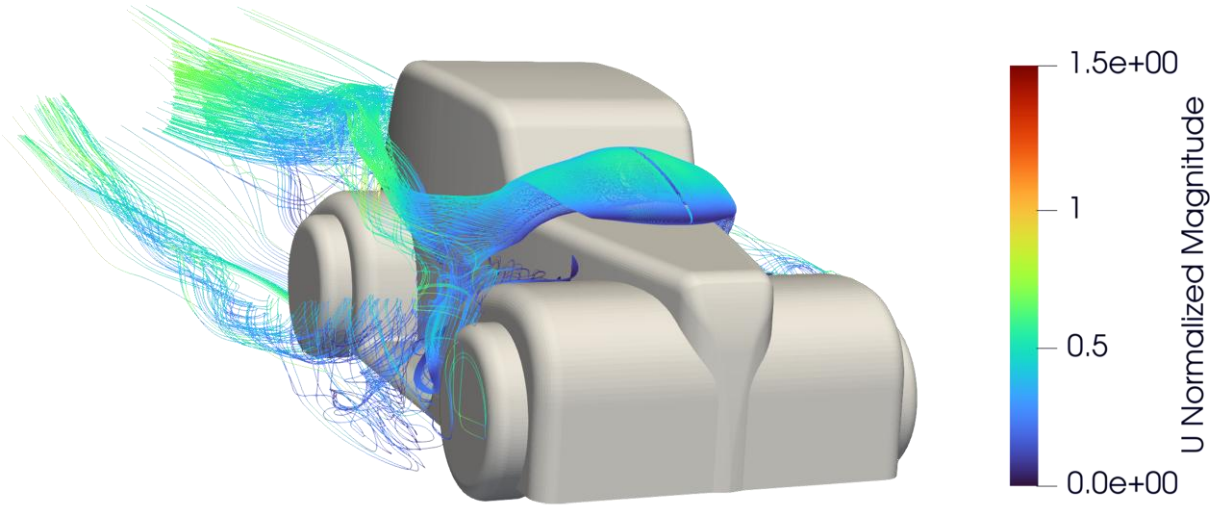
Numerical Results

Chassis Comparison

Coupé



Sedan

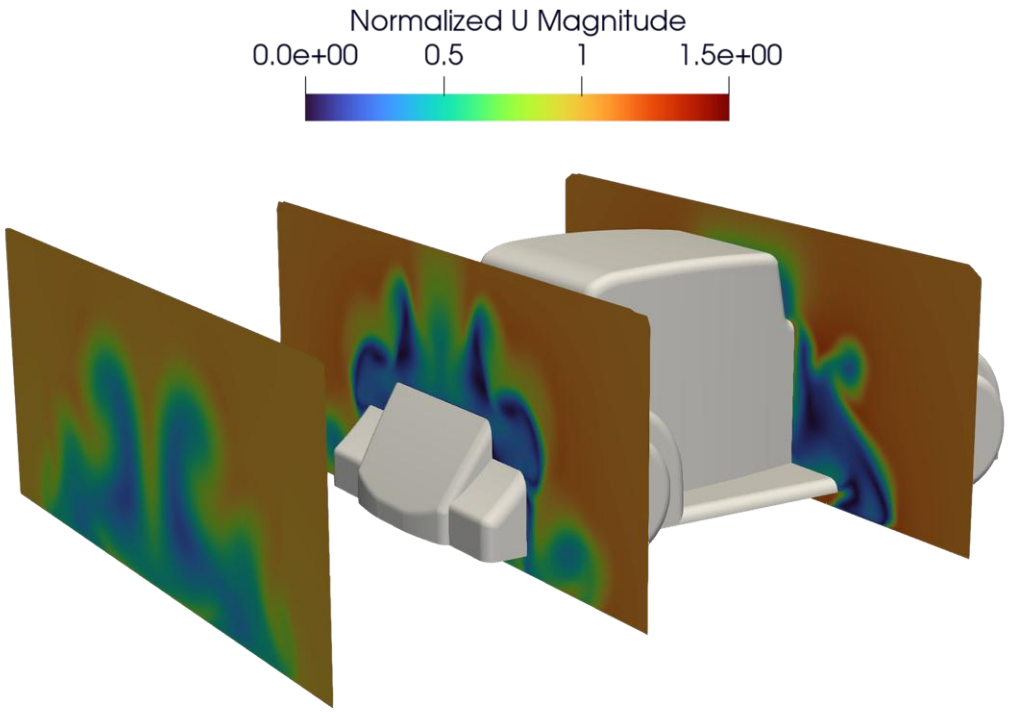


Streamlines visualization with the normalized velocity field

Numerical Results

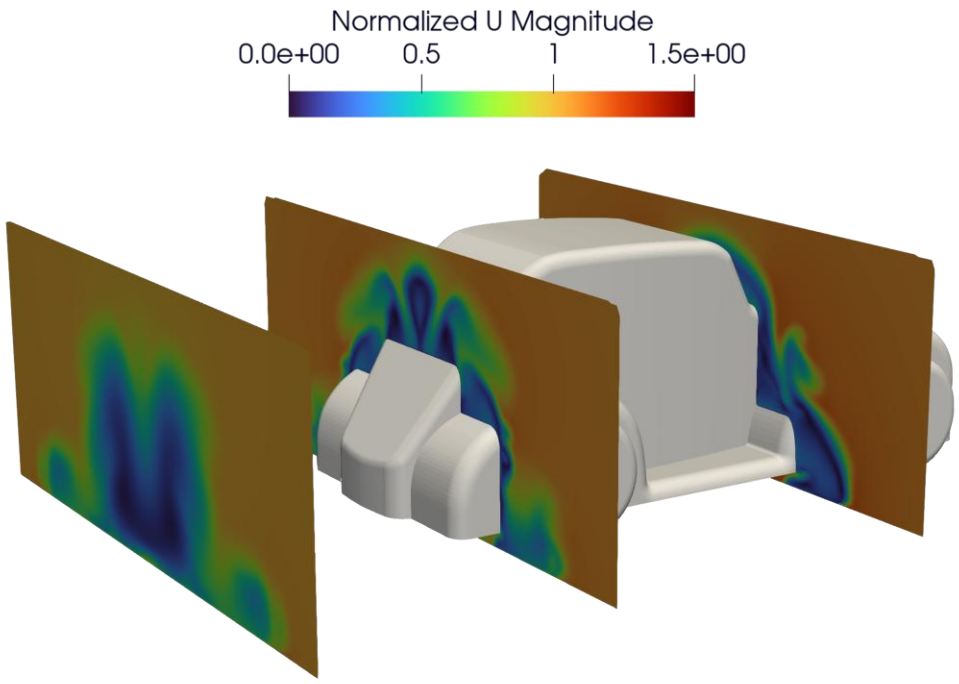
Chassis Comparison

Coupé



Coupé slices with the normalized velocity field

Sedan

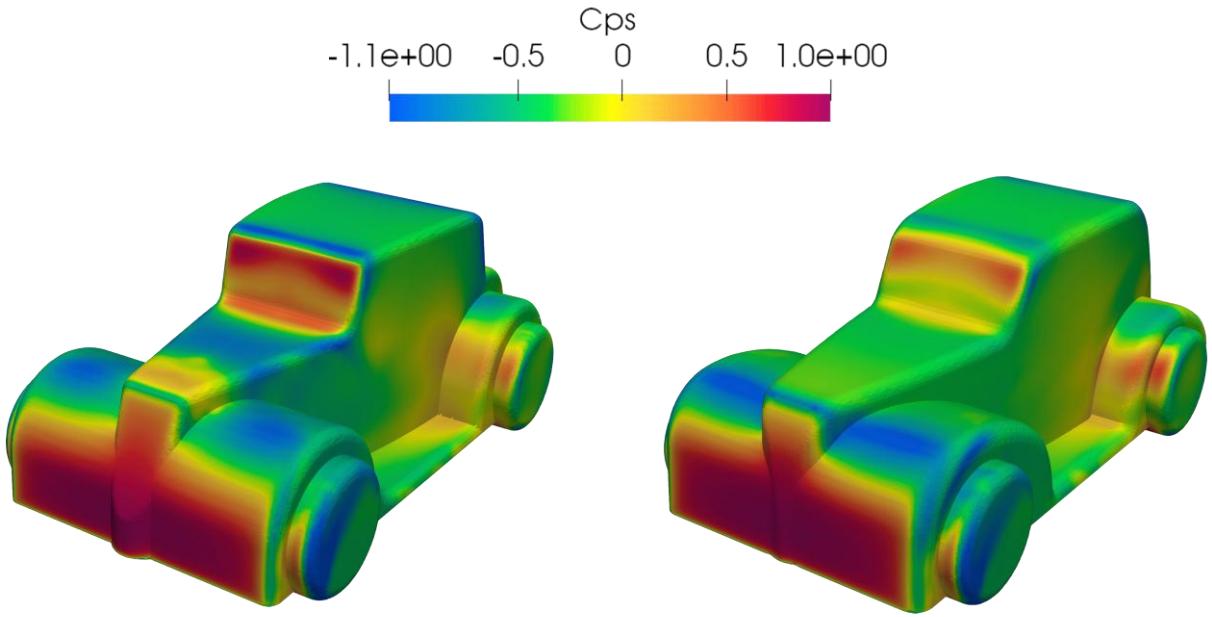


Sedan slices with the normalized velocity field

Numerical Results

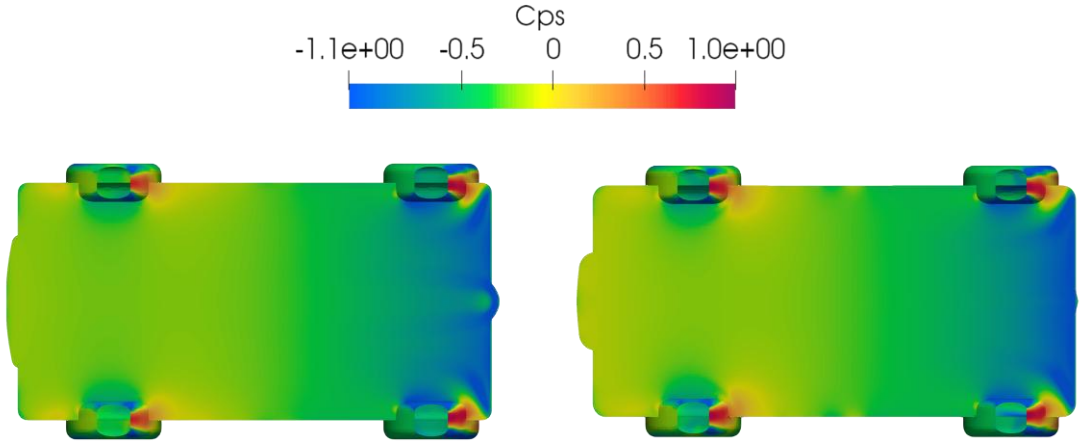
Chassis Comparison

Coupé **Sedan**



C_{ps} surface distribution

Coupé **Sedan**

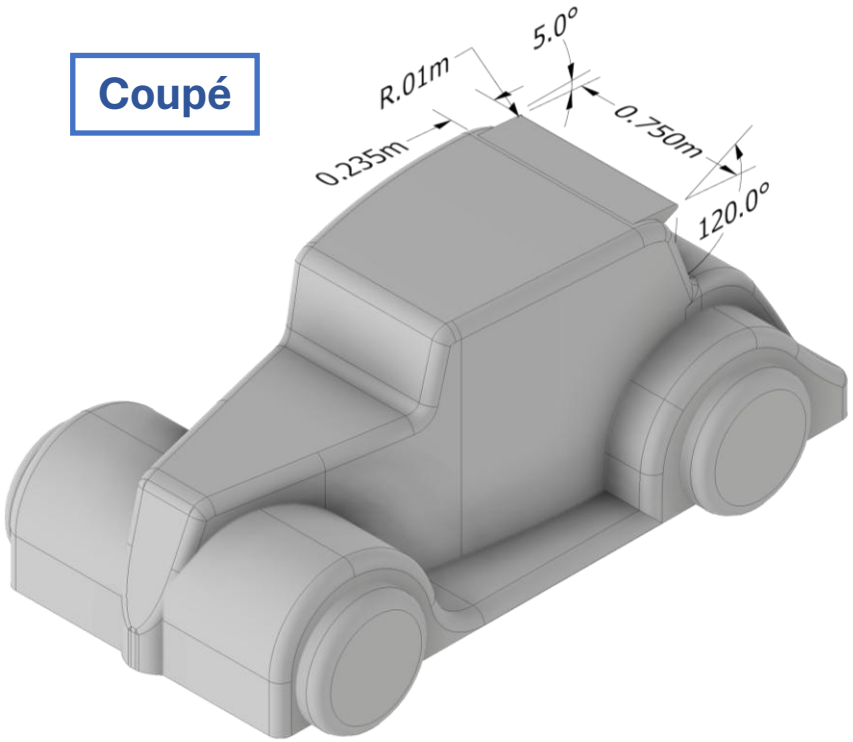


C_{ps} surface beneath the car distribution

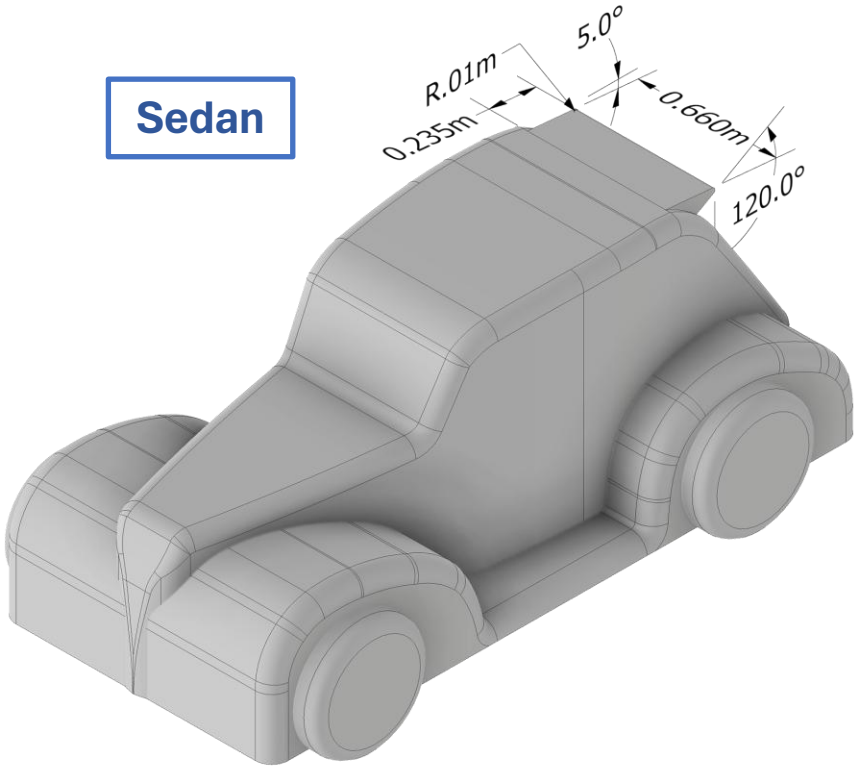
Numerical Results

Roof Spoiler Contribution

Coupé



Sedan



Simplified CAD models with spoiler

Numerical Results

Roof Spoiler Contribution

Spoiler configuration:

- Pitch angle is selected to continue the roofline geometry;
- Delayed separation point;
- Drag reduction is prioritized over increased downforce.

Chassis	ΔC_L	ΔC_D	$\%dC_D$
Coupé	-0.208	-0.019	3.08%
Sedan	-0.101	-0.012	2.53%

Patch	C_L	C_D	% of C_D
Upper Body	-0.071	0.139	23.40%
Lower Body	0.016	0.391	65.84%
Front Tyre	0.015	0.024	4.01%
Rear Tyre	-0.009	0.029	4.92%
Spoiler	0.032	0.011	1.82%
TOTAL	-0.016	0.593	100%

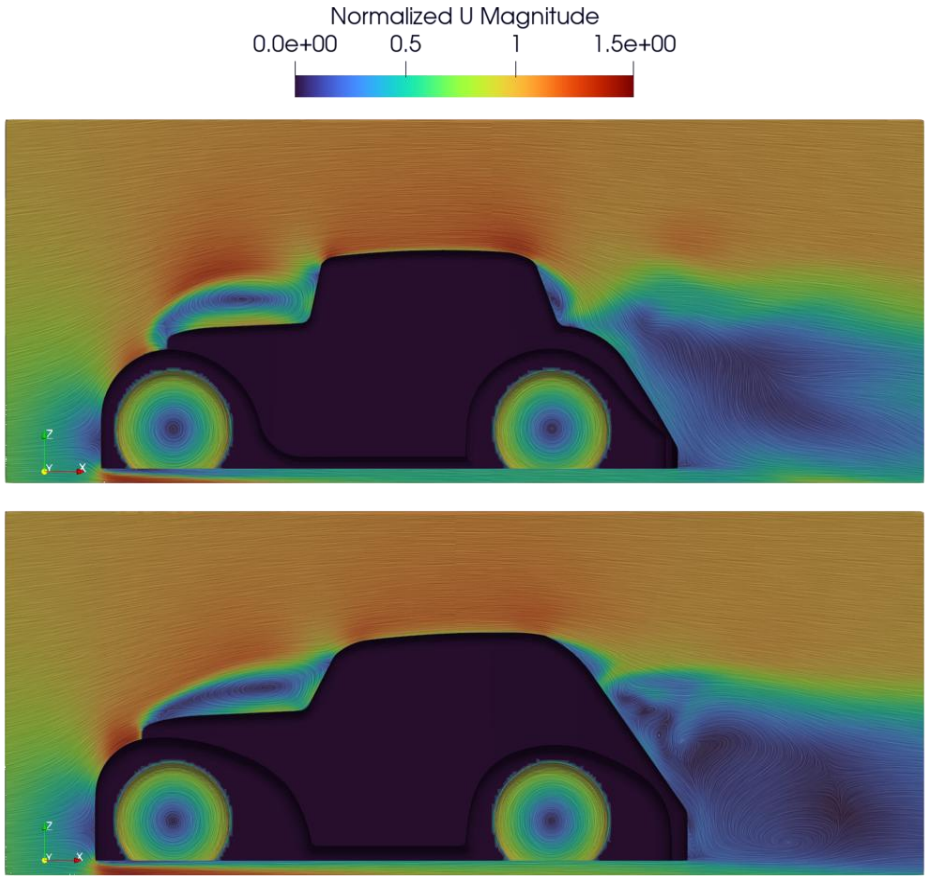
Coupé spoiler drag breakdown

Patch	C_L	C_D	% of C_D
Upper Body	-0.086	0.079	16.97%
Lower Body	-0.033	0.324	69.73%
Front Tyre	0.010	0.020	4.35%
Rear Tyre	-0.008	0.030	6.46%
Spoiler	0.027	0.012	2.49%
TOTAL	-0.090	0.465	100%

Sedan spoiler drag breakdown

Numerical Results

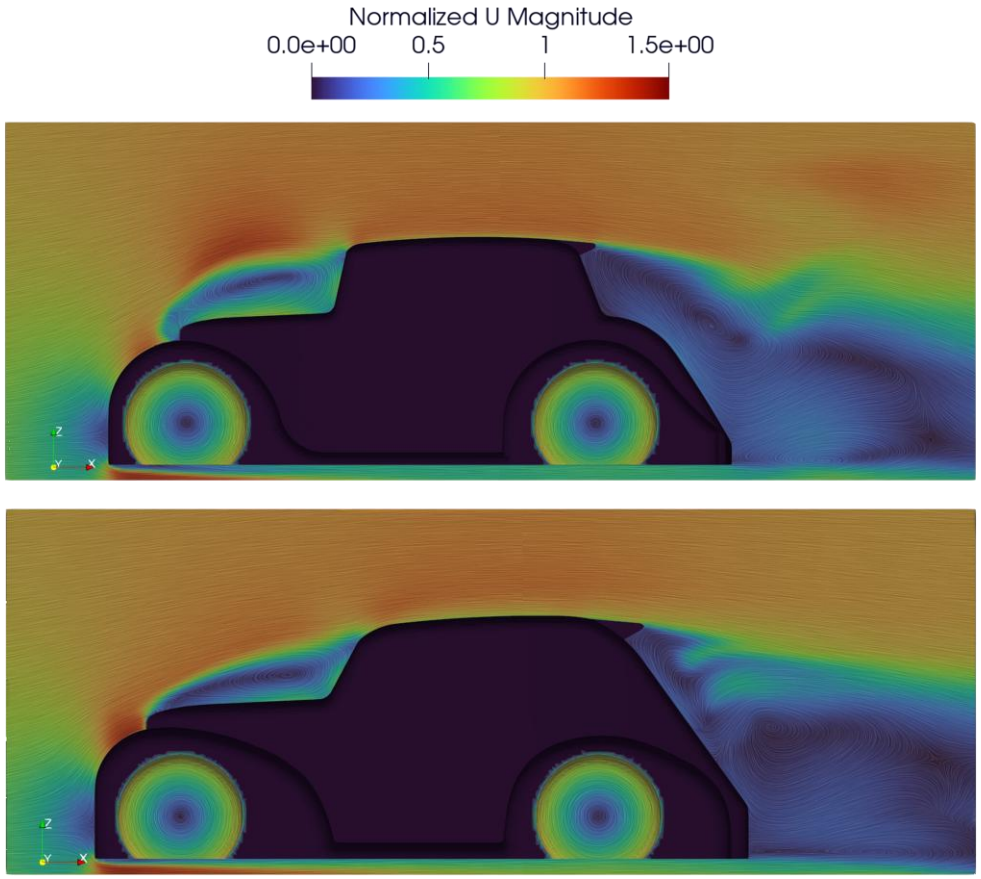
Roof Spoiler Contribution



Baseline configuration LIC visualization with the normalized velocity field (Y = 0.2 m)

Coupé

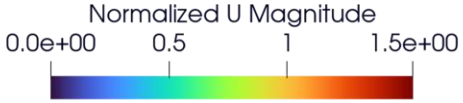
Sedan

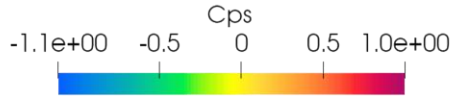


Spoiler configuration LIC visualization with the normalized velocity field (Y = 0.2 m)

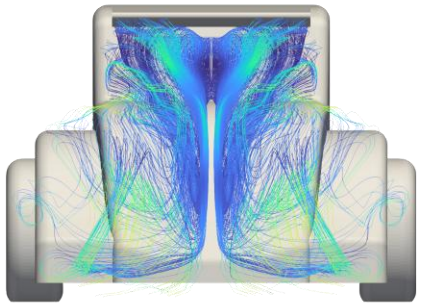
Numerical Results

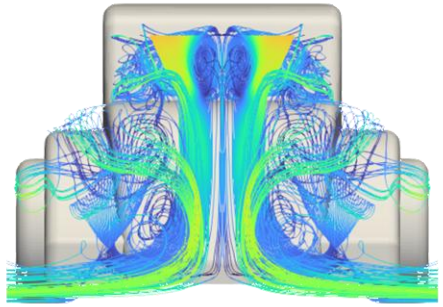
Roof Spoiler Contribution

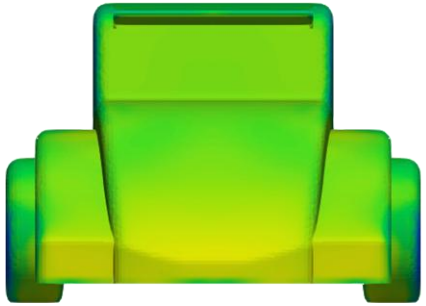


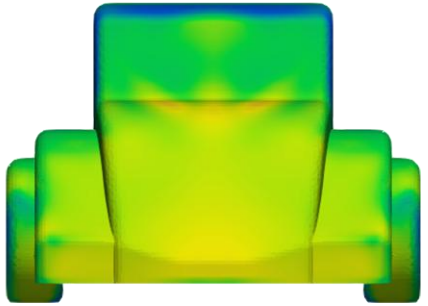


Coupé

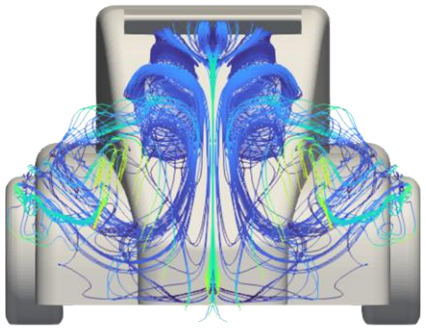


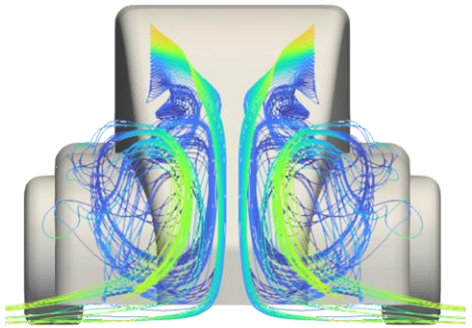


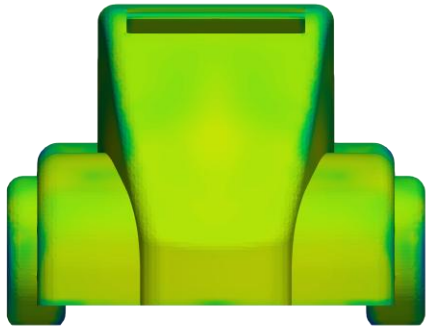


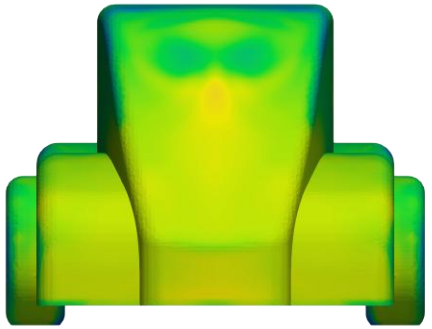


Sedan





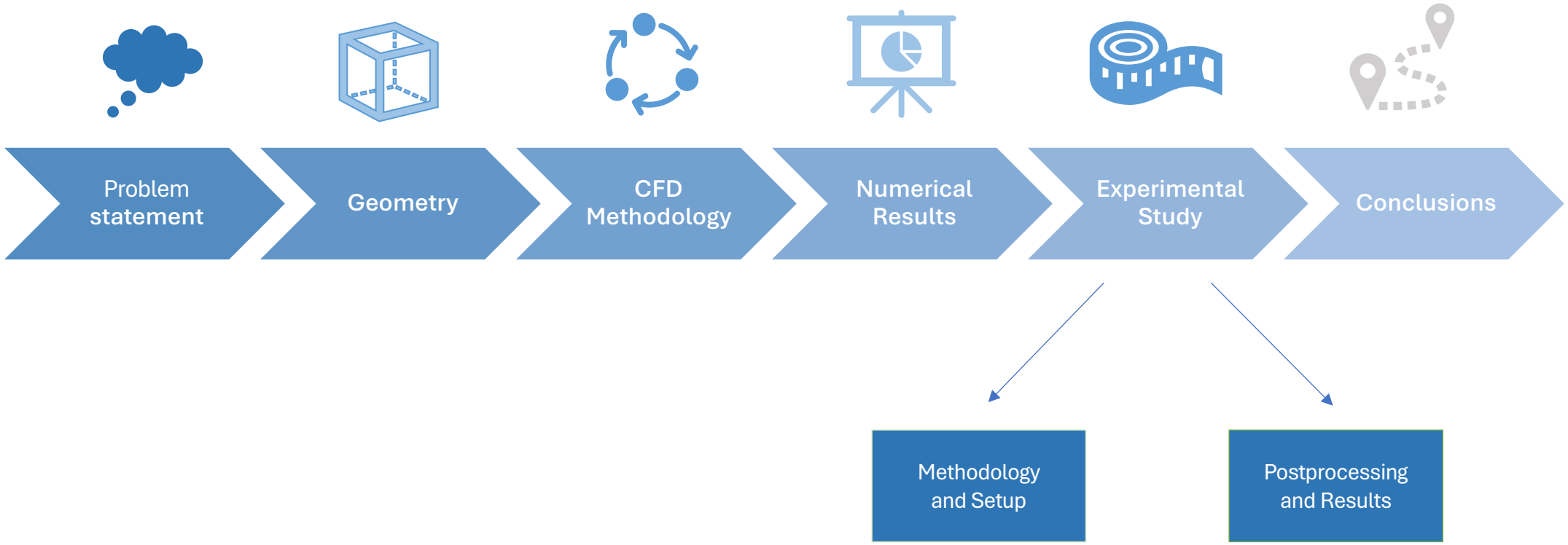




Rear streamlines visualization with the normalized velocity field

Rear C_{ps} surface distribution

Presentation outline



Experimental Study

Introduction

- Tuft test performed to validate CFD predictions on real car flow behavior
- Chosen for low cost and ease of execution
- Tufts applied on body surface, car driven straight at constant speed

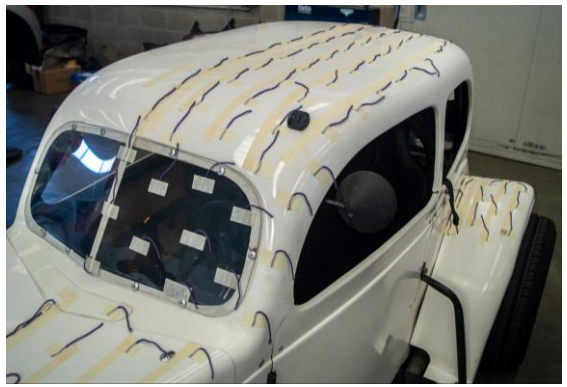
  $\Longrightarrow$ Replicates CFD simulation flow condition

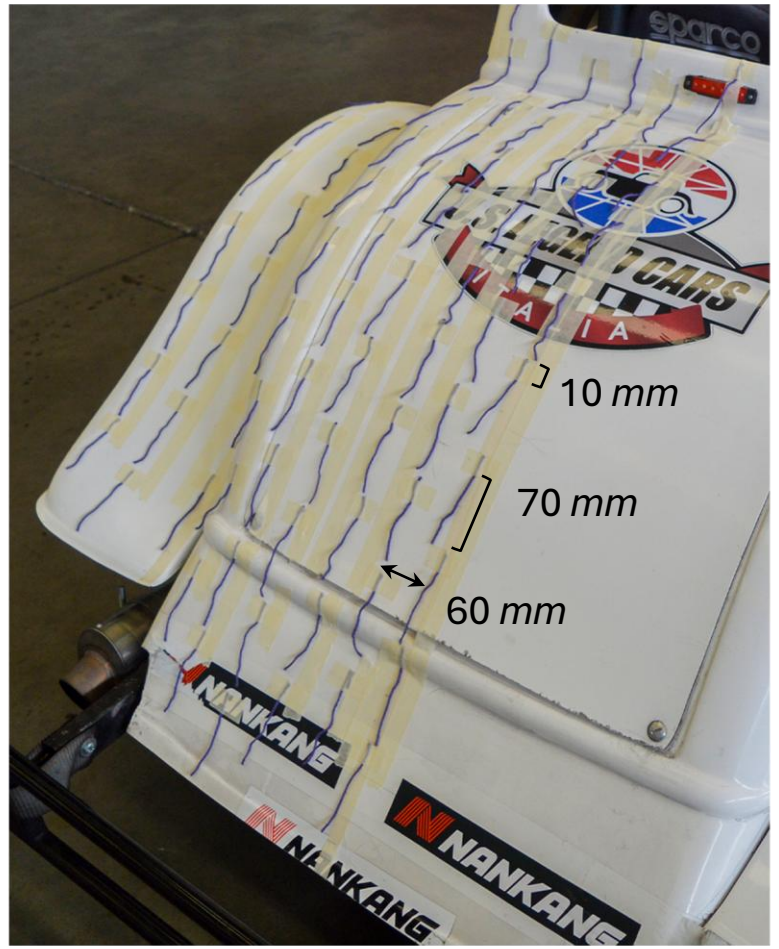
Goal: Visualize recirculation, separation, and laminar flow regions

Experimental Study

Tuft application setup

- Tufts applied on left side of both chassis, reasons:
  - *Easier to see from the external camera spot*
  - *Symmetric flow, no cross wind and no other cars wake*
- Focused on key aerodynamic regions, excluding CAD-simplified areas
- Grid dimensions selected for visibility and relevance to flow structures
- Spacing designed to avoid tuft tangling





Experimental Study

Measurement instruments setup

External camera:

- Nikon D3200 with 18–55 mm lens
- Photo bursts preferred over video for higher clarity

Onboard camera:

- GoPro Hero 5 on roof for bonnet/front wheel arch view
- Interference noted → tests done with and without camera

Telemetry:

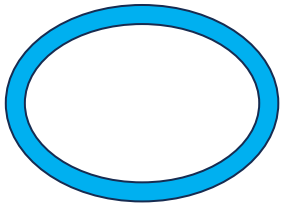
- AIM MXM system with GPS-based speed measurement
- Used by driver to maintain constant velocity

Experimental Study

Postprocessing

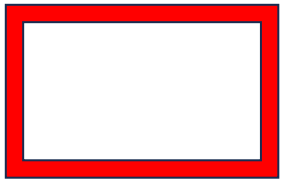
- Multiple frames analyzed to verify flow coherence
- Outliers identified (e.g., stuck or tangled tufts)
- External shots timed for consistency across runs/chassis
- Key flow features annotated in selected frames using these sybmols:

: **Main flow direction**



: **Recirculation zone**

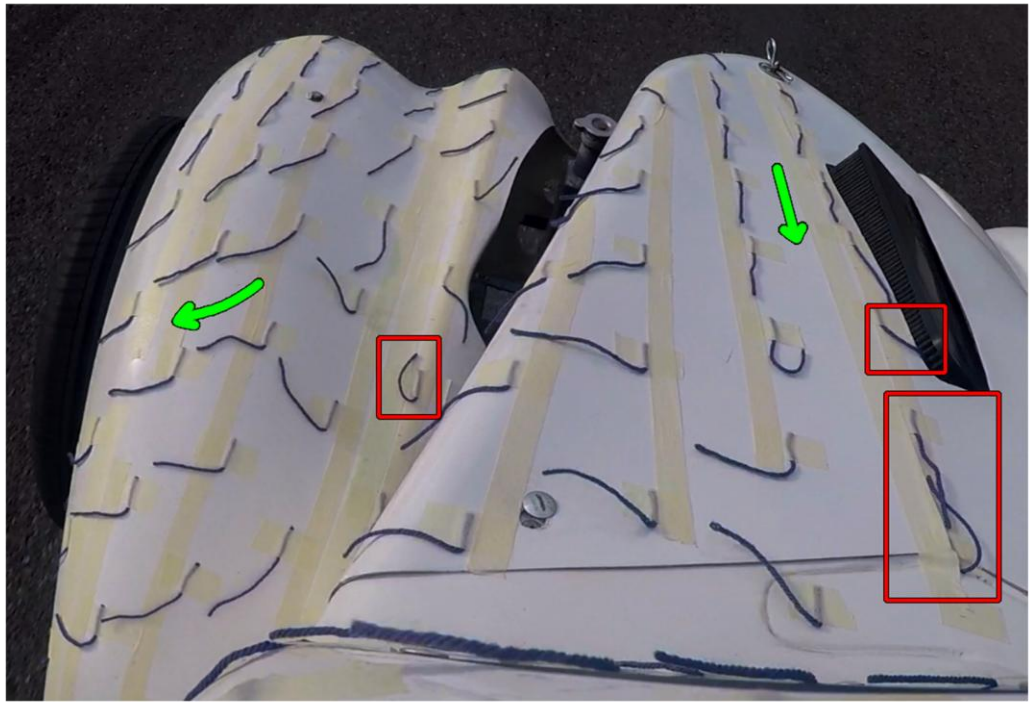
: **Separation line**

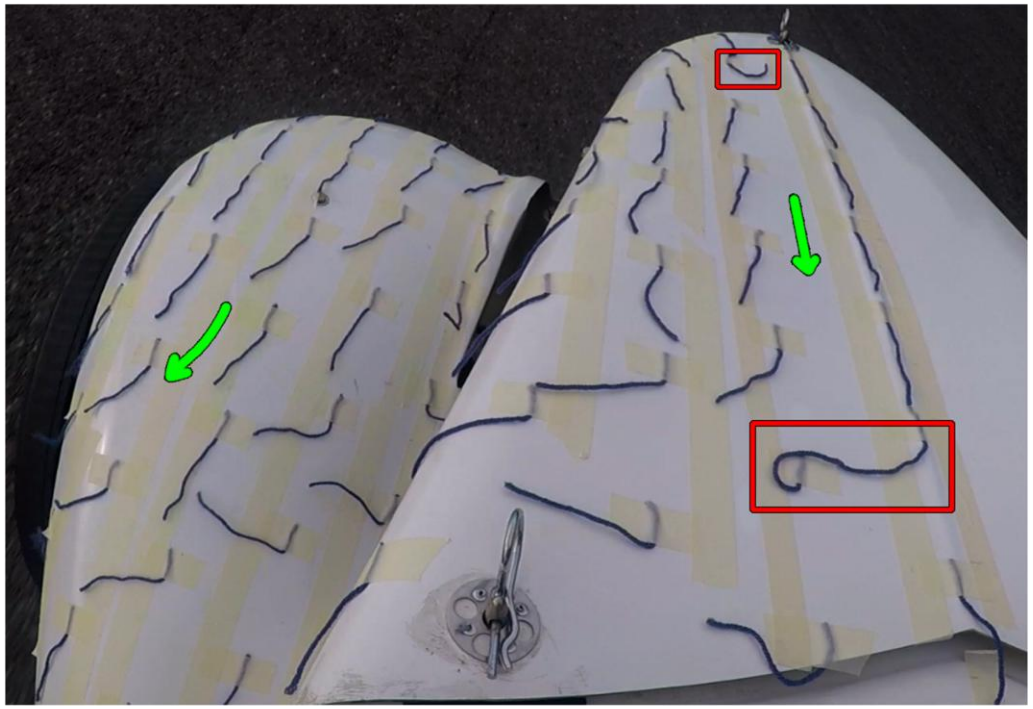


: **Outliers tufts**

Experimental Study

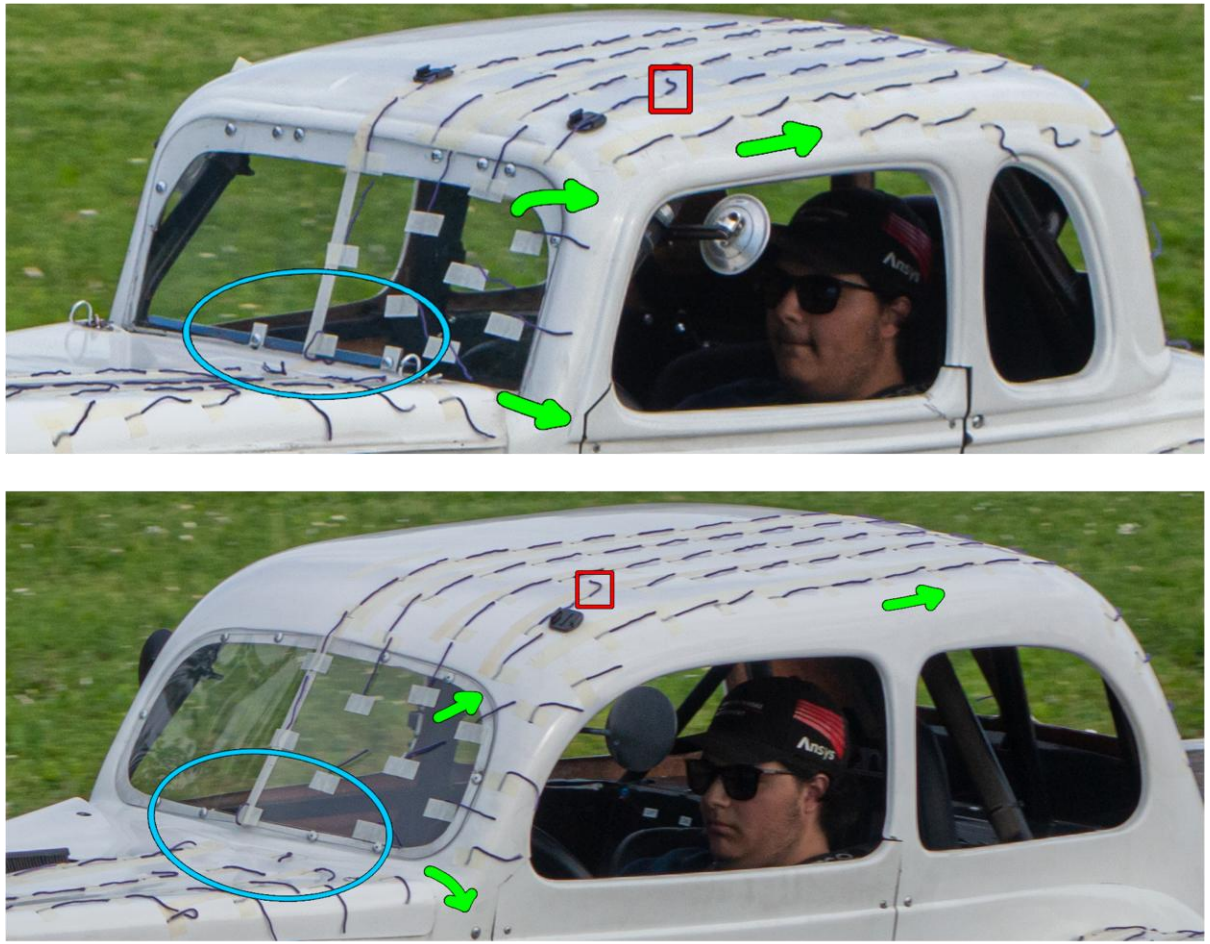
Roof-mounted camera





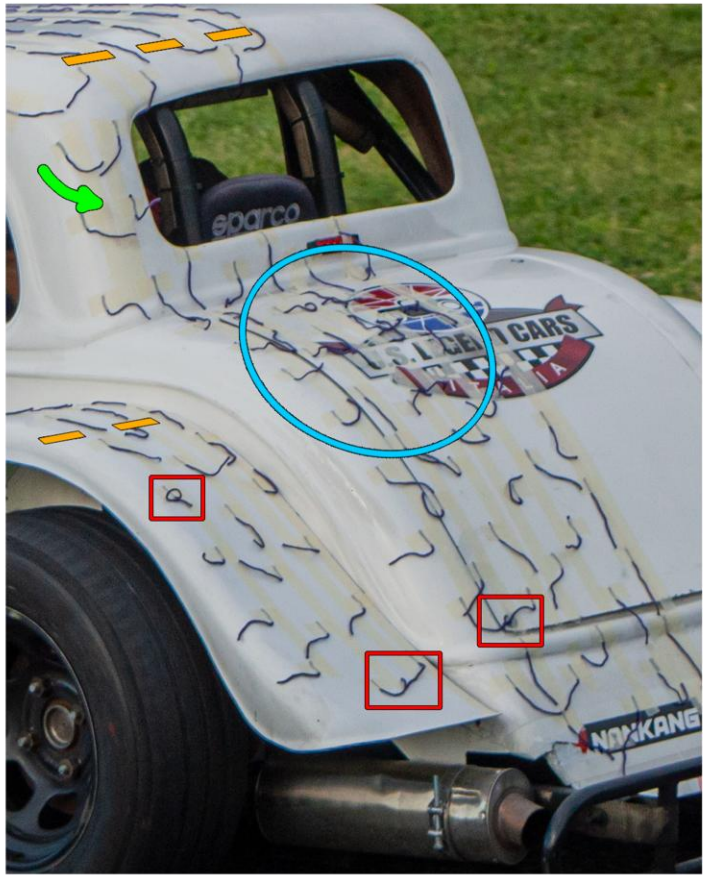
Experimental Study

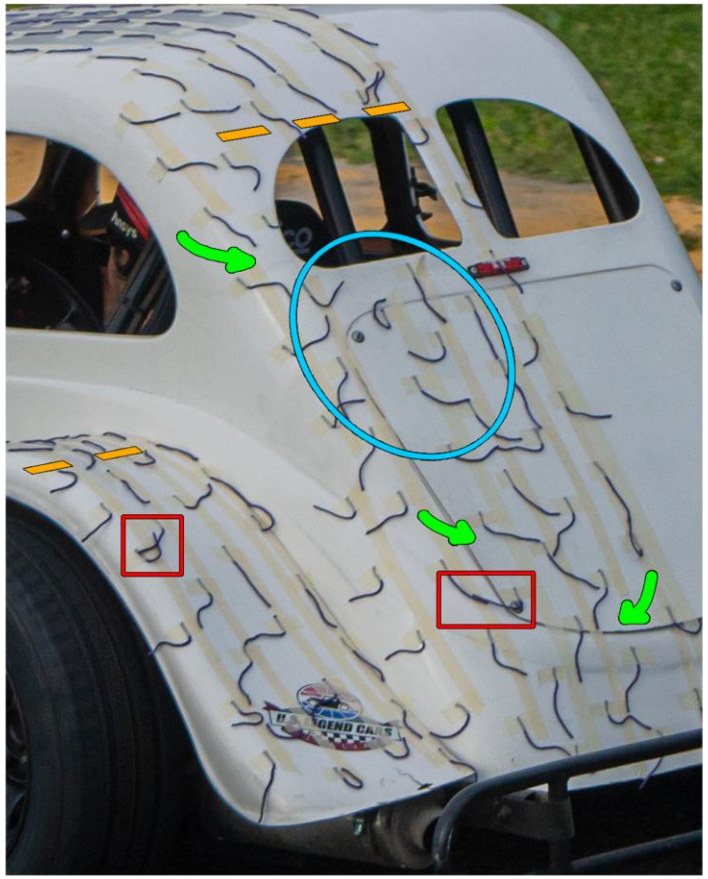
Roof and windshield



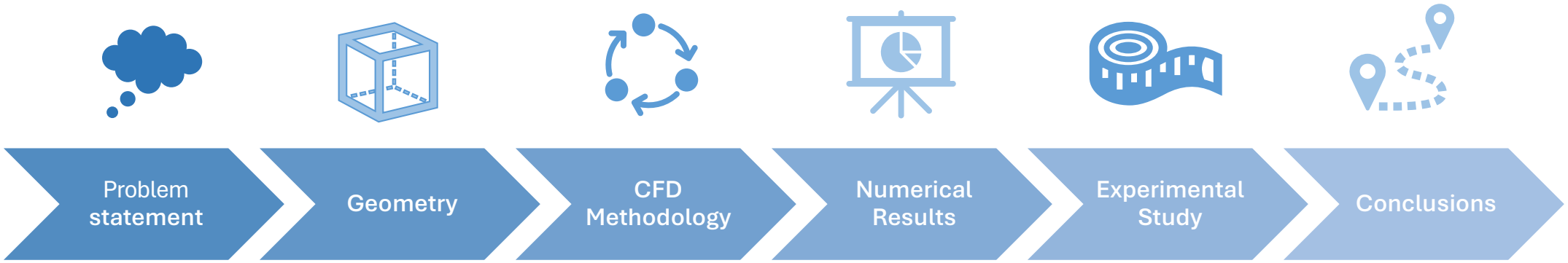
Experimental Study

Rear sections





Presentation outline



Conclusions

Conclusion

- Different chassis exhibit non-negligible differences in aerodynamic loads (especially in the C_D), with the Sedan chassis performing more favorably;
- Flow physics is complex due to the non-aerodynamic vehicle design but aligns with theoretical expectations;
- Spoiler introduction significantly reduces drag, especially on the Coupé chassis.

Limitations

- Differences in geometry between numerical and experimental studies (e.g., open windows during tuft test);
- Computational cost.

Future Developments

- Use of URANS/LES instead of RANS to better capture the unsteady flow phenomena;
- Possibility of studying the behavior at higher speeds;
- Possibility of directly solving boundary layer;
- Possibility to lengthen the simulation time;
- Possible inclusion of geometric details (e.g., rearview mirrors).

Acknowledgements

A special thanks to Adriano Giuseppe Monti and the staff of Castelletto Circuito for kindly providing access to the two Legends Cars and the track where the tests were conducted

Thank you for your attention!