

On-site full-scale race car aerodynamics analysis using the Ring-of-Fire

Pablo I. Álvarez García, Adrian Grille Guerra*, Andrea Sciacchitano

Delft University of Technology, Faculty of Aerospace Engineering, Delft, The Netherlands

* Correspondent author: a.grilleguerra@tudelft.nl

Keywords: Vehicle Aerodynamics, Field Experiments, Automotive Diffuser, Tyre-wake, 3D LPT, PIV, Ring of Fire

ABSTRACT

This study explores the use of the Ring of Fire methodology for investigating the aerodynamics of a Formula Student race car using Lagrangian Particle Tracking (LPT) via the Shake-the-Box (STB) algorithm. Applied to the DUT24 developed by Formula Student Team Delft, the approach enables high-resolution reconstruction of the three-dimensional velocity and pressure fields around a fully aerodynamically equipped vehicle. The investigation focuses on two critical regions: the underbody diffuser and the wakes shed by the rotating tyres. Experiments were conducted on a 60 m test track at approximately 12 m/s using Helium-Filled Soap Bubbles as neutrally buoyant tracers. For the diffuser study, three cameras and four LEDs were installed in a ditch beneath the car, defining a measurement volume of $300 \times 400 \times 350$ mm, while tyre-wake measurements employed a lateral camera and LED arrangement, yielding a $400 \times 800 \times 450$ mm volume. Results reveal strong acceleration beneath the front wing and diffuser, with peak velocities exceeding twice the free-stream ($u/U_\infty > 2.2$). Diffuser strakes generate coherent streamwise vortices promoting flow attachment and downforce, while tyre wakes are highly three-dimensional and unsteady, with larger wakes from the front tyres, strongly influenced by the front-wing. Pressure reconstruction using omnidirectional integration reproduced physically consistent distributions, capturing diffuser recovery and tyre-wake stagnation zones. Comparison with CFD demonstrates good agreement in overall flow topology and acceleration, while differences were observed in vortex strength and tyre rotation effects. Overall, the full-scale application of the Ring of Fire methodology successfully captures the complex aerodynamics, including high-velocity underbody flows, coherent diffuser vortices, and unsteady tyre wakes, establishing its effectiveness as a robust tool for on-site vehicle aerodynamic investigation.

1. Introduction

Race car aerodynamics plays a central role in vehicle performance, stability, and efficiency, with downforce increasing tyre-road friction, enhancing cornering speed, and reducing lap times. The underbody is particularly significant, contributing up to 50% of total downforce, while the front and rear wings provide the remainder (Toet, 2013). Simultaneously, wheel-induced flow contributes approximately 30% of the vehicle's drag (Fackrell & Harvey, 1975), highlighting the

importance of the diffuser and tyre wakes in aerodynamic optimization. These regions influence vortex formation, pressure recovery, and wake development, yet their highly three-dimensional and unsteady flows remain challenging to measure and predict accurately.

The exploitation of ground effect, where proximity to the track accelerates underbody flow and lowers pressure, is fundamental for downforce generation. Diffuser performance depends on the interplay between ground effect, diffuser geometry, and vortex dynamics. Flow separation and vortex breakdown at high diffuser angles can reduce efficiency, while stable vortices promote attached flow and in turn enhance pressure recovery (Cooper et al., 1998; Ruhrmann and Zhang, 2003). Geometric parameters such as diffuser length, ramp angle, and ride height critically influence aerodynamic forces, with small changes inducing non-linear variations and hysteresis in downforce (Senior, 2002; Katz and Garcia, 2002; Jowsey and Passmore, 2010). Multi-strake diffusers, vortex generators, and blown diffusers have been employed to stabilize vortices and optimize performance (Parra and Kontis, 2006; Zhang et al., 2022; Holland et al., 2025).

Wheel rotation further complicates aerodynamic behaviour. Rotating tyres modify local flow patterns, generally increasing wake height and altering vortex structures compared to stationary wheels (Fackrell, 1974; McManus and Zhang, 2006). Wake vortices, including upper, ground, and hub-related vortices, strongly influence drag and lift, with rotation reducing lift and modifying base pressure distributions (Merker et al., 1991; Katz and Morey, 2008; Josefsson et al., 2022; Gérardin et al., 2014). Accurately capturing these effects remains critical for both performance prediction and design optimization.

Predicting diffuser and tyre-wake behaviour remains challenging despite advances in computational and experimental techniques. Computational Fluid Dynamics (CFD) provides valuable insights, but is limited in highly three-dimensional, unsteady regions due to trade-offs between accuracy and computational cost (Cooper et al., 1998; Knowles, 2005). Conventional experimental methods, such as wind tunnel testing, are constrained by optical access, scaling effects, and simplified geometries. Recent innovations in experimental fluid mechanics offer new opportunities. Helium-Filled Soap Bubbles (HFSB, Scarano et al., 2015) with the Shake-The-Box Lagrangian Particle Tracking algorithm (Schanz et al., 2016) enable, three-dimensional quantitative flow visualizations over domains of the order of the cubic meter, which have been extended to the study of the aerodynamics of transiting objects via the Ring of Fire (RoF) approach (Spoelstra et al., 2019, Hüttig et al., 2025). Early applications to scaled automotive models have demonstrated the potential of these techniques (Battezzore et al., 2024; Holland et al., 2025), but underbody diffuser flows and rotating tyre wakes remain largely unexplored, especially for full-scale vehicles.

This study applies three-dimensional LPT combined with RoF to the DUT24 Formula Student race car, focusing on the underbody diffuser and rotating tyre wakes. By reconstructing velocity and pressure fields and comparing results with CFD predictions, the work seeks to deepen understanding of ground-effect aerodynamics, vortex dynamics, and wake interactions, contributing to the optimization of high-performance race car design.

2. Methodology

2.1. Experimental setup and testing routine

The experiments were performed in the Flow Physics and Technology Laboratories of Delft University of Technology in the Netherlands. The vehicle used in the experimental campaign was the DUT24, designed and manufactured by the Formula Student Team Delft. The vehicle dimensions are around 3 metres long, 1.6 metres wide, and 1 metre tall, equipped with a full aerodynamic package that can be divided into five groups, as depicted in Figure 1: (I) the front wing, which is the first flow-interacting element, redirecting airflow downstream while efficiently generating downforce; (II) the chassis, which serves as the primary structural framework; (III) the floor and diffuser, designed to exploit ground-effect aerodynamics for maximum downforce with minimal drag; (IV) the rear wing, which generates substantial downforce, though at a significant aerodynamic drag cost; and (V) the wheels and bull-wings, positioned along the chassis sides, critically influencing downstream airflow.

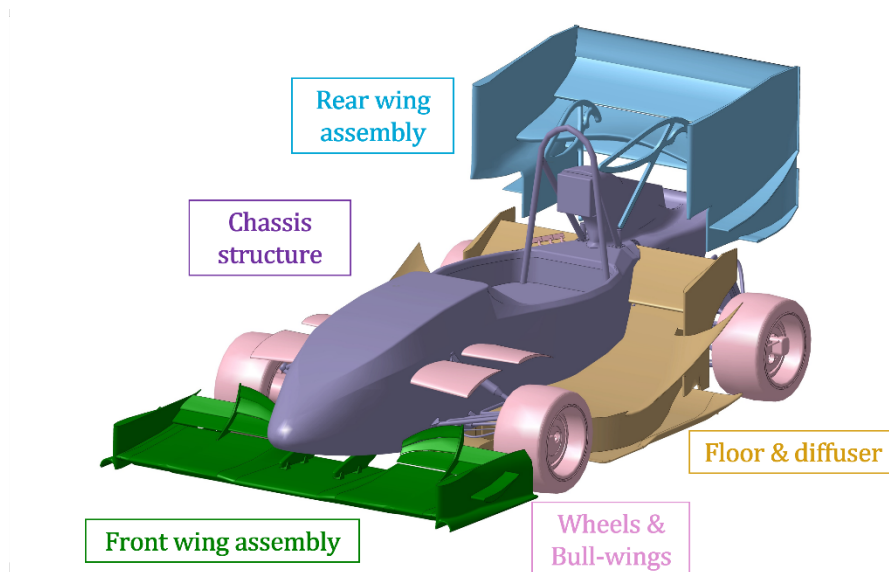


Figure 1. DUT24 geometry and aerodynamic package.

The experimental setup designed to provide visual access to the car's underbody, improving it from previous experiments (Holland et al., 2025), is shown in Figure 2 (a), while the measurement volume achieved in this specific experiment is depicted in Figure 2 (b).

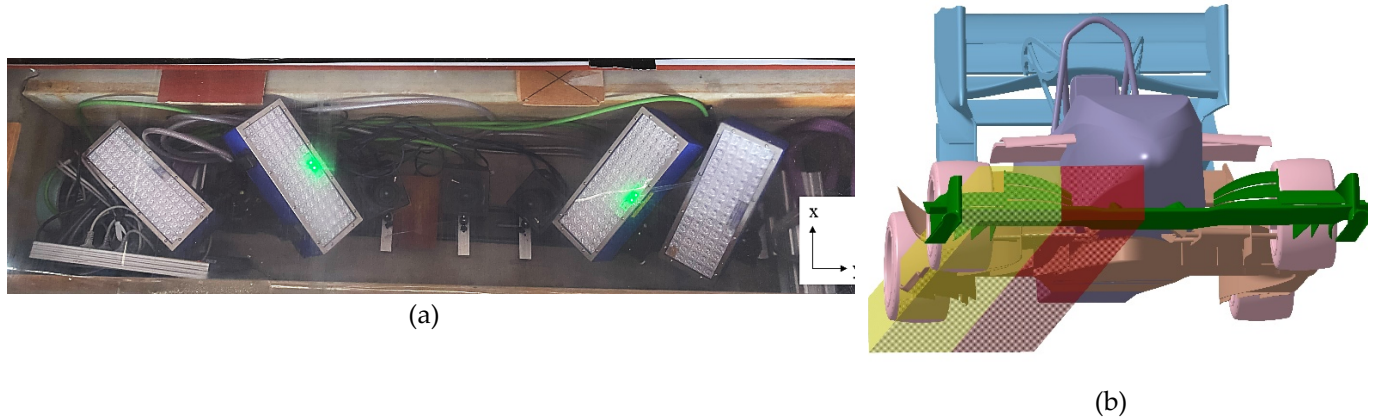
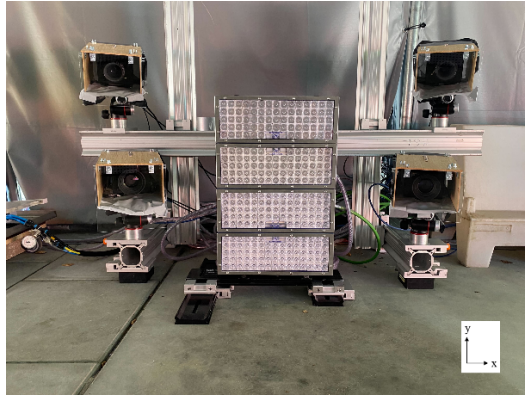


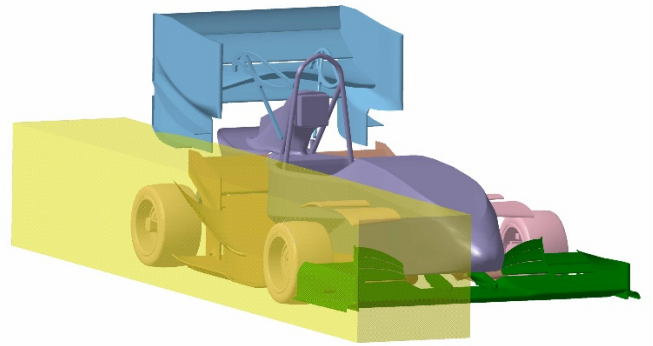
Figure 2. Diffuser study: (a) Camera and LED set-up and (b) Measurement volumes.

A concrete shell with an internal volume of 2 m (length) by 43 cm (width) by 40 cm (height) was placed in the ground, with its longest dimension perpendicular to the direction of motion of the car. Inside the shell, three Photron *Mini AX100* high-speed cameras (12 bit, 20 μm pixel pitch, 1,024 $\times$ 1,024 pixels) and four LaVision *LED Flashlight 300* light sources were mounted. The cameras, located in the centre of the ditch, faced directly to the test-track, being their overlapping field of view the determining factor that derived the dimensions of the measurement volume. With this setup, the field of view did not cover the full diffuser width; hence, multiple car passes were required, whereby the vehicle passed through the measurement domain at different lateral locations. A total of 20 runs were done for the inner section (red, in Figure 2 (b)), and another 20 for the outboard region (yellow, in Figure 2 (b)).

For the tyre-wake analysis, the setup and measurement volumes differ from the diffuser study. This is illustrated in Figure 3 (a) and Figure 3 (b), respectively. In this case, the setup was mounted on the side of the test track, pointing to the centre of it. Four cameras aligned in a two-by-two configuration, and four light sources vertically piled up were employed.



(a)



(b)

Figure 3. Tyre wake study: (a) Camera and LED set-up and (b) Measurement volume.

The specifications for both experiments are found in Table 1:

Table 1. Experiment set-up and parameters.

Category	Parameter	Diffuser Study	Tyre-Wake Study
Imaging - Cameras	Model	3× Photron <i>FastCAM Mini AX 100</i>	4× Photron <i>FastCAM Mini AX 100</i>
	Lens	3× AF Nikon f=20 mm, f# = 22	2× AF Nikon f=50 mm, 2× AF Nikon f=60 mm, f# = 16
	Acq. Freq. (kHz)	3	2
Illumination	LED	4×LaVision <i>LED Flashlight 300</i>	4× LaVision <i>LED Flashlight 300</i>
Tracer particles	Type	Neutrally-buoyant Helium-filled Soap Bubbles	Neutrally-buoyant Helium-filled Soap Bubbles
Seeding Container	Location	Outdoor Tent (9×5×4 m)	Outdoor Tent (9×5×4 m)
Measurement volume dimensions	X (mm)	300	400
	Y (mm)	400	800
	Z (mm)	350	450

The flow was seeded with helium-filled soap bubbles (HFSB) flow tracers released by an in-house seeding rake comprising 204 generators (Scarano et al., 2022). To achieve sufficient concentration of flow tracers, the HFSBs were confined in a measurement tent of dimensions 9×5×4 m. The entrance and exits could be opened and closed to accumulate the bubbles and then allow the passage of the car. Running the seeder for approximately 1 minute was required to achieve a

sufficient bubble density that could ensure high-quality measurement. A schematic image of the track is found in Figure 4, showing all the elements involved for both cases.

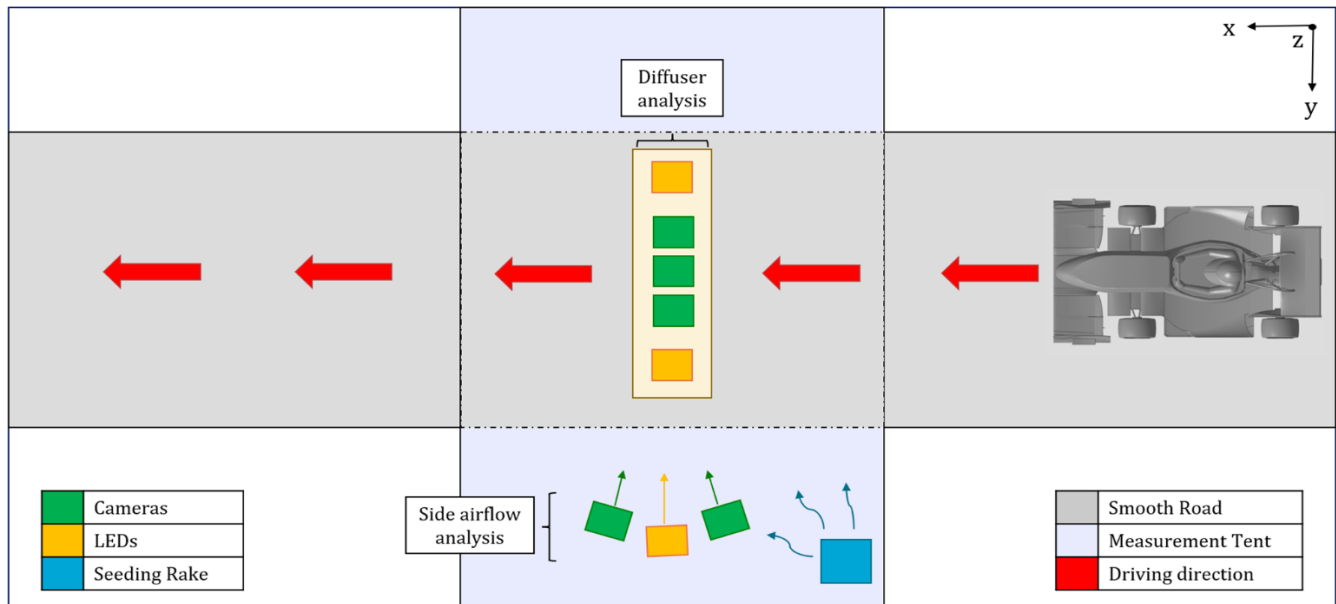


Figure 4. Track layout schematic.

For conducting the camera calibration, a 2D calibration plate with fiducial markers was located in the measurement volume. A few images were taken at different angles and positions, in order to reconstruct the 3D space accurately. After camera calibration and volume self-calibration (Wieneke, 2008), the data acquisition could start by operating the HFSB generator with the curtains of the tent down. For each passage, 1000 images were captured at 3 and 2 kHz, respectively. The procedure was repeated 40 times for the diffuser test (20 for each width), and 30 for the tyre wake evaluation.

2.2. Data analysis

To remove unwanted light reflections and enhance particle detectability, images were pre-processed following the approach introduced by Grille Guerra et al. (2025). Then, the images were processed via the Shake-The-Box LPT algorithm (Schanz et al., 2016) to obtain three-dimensional particle tracks. Image recording and processing was conducted with the *DaVis 11* software from LaVision GmbH.

A Galilean transformation was applied to convert all position and velocity data from the stationary laboratory frame to the moving vehicle frame, similar to Holland et al. (2025). The transformation repositioned the origin at the car's centreline and ground level, using the measured vehicle speed to express the flow field relative to the car's motion. Consequently, the reference system was

adapted to correspond to a conventional wind-tunnel static frame of reference. The system of reference was set at the leading edge of the front wing (x), the vehicle's centreline (y), with zero height at the ground (z). The x axis follows the car's movement, while the y axis is aligned with the wheel axis, resulting in a positive z value pointing upwards from the ground.

The velocity fields from multiple passages of the car are transformed into the car's reference frame and normalized by the vehicle's instantaneous speed to correct for run-to-run velocity variations. Subsequently, the data are time-averaged and interpolated onto a Cartesian grid following the spatial averaging method described by Agüera et al. (2016). A bin size of 40 mm was used for the diffuser case and 50 mm for the tyre wake, both with a 75% overlap between neighbouring bins, resulting in effective spatial resolutions of 10 mm and 12.5 mm, respectively.

Pressure fields are reconstructed following the hybrid methodology introduced by Jux et al. (2020). The domain is first separated into irrotational and rotational regions, with Bernoulli's equation applied in the former and the Navier–Stokes–based pressure gradients integrated across the latter. Ensemble-averaged PIV data provide both convective and Reynolds stress terms, while viscous contributions are neglected due to the high Reynolds number regime. Pressure fields are then reconstructed iteratively, with irrotational values serving as boundary conditions for integration into the rotational zones.

The on-site 3D-LPT experiments on the transiting formula student car are used to validate the numerical simulations by CFD conducted by the TU Delft formula student team. Such simulations employed a RANS framework, with the Shear Stress Transport (SST) $k-\omega$ turbulence model; the governing equations were solved using the Finite Volume Method with a structured mesh.

3. The flow field inside the underbody

3.1. Velocity field

The underbody flow of DUT24 is characterised by strong acceleration beneath the front wing and flat floor, followed by peak velocities at the diffuser throat and progressive deceleration during expansion. This acceleration–deceleration cycle generates a low-pressure region that sustains downforce. Streamwise vortices, initiated by the diffuser strakes, play a decisive role in maintaining attached flow despite the steep expansion, while also redistributing momentum vertically and contributing additional suction through their low-pressure cores. Figure 5 (a) illustrates the overall evolution of the streamlines of the flow in the underbody region, while Figure 5 (b) showcases the variation of the velocity field along the front-wing, floor and diffuser. The LPT measurements confirm that the flow remains attached along the diffuser ramp at the tested velocity ($U_\infty = 12$ m/s), with no evidence of large-scale separation or stagnation. CFD

simulations reproduce the main mechanisms—acceleration beneath the front wing, peak velocity at minimum ride height, and pressure recovery in the diffuser—but underestimate flow momentum in key regions. In particular, CFD predicts weaker upwash and a pronounced velocity deficit in the upper diffuser, whereas the PIV data shows coherent vortices sustaining higher momentum across the diffuser height. The velocity peak values are more than twice the free-stream velocity value, and are achieved underneath both front-wing and diffuser, directly translating into a considerable downforce generation. This is visible in Figure 6. The noise present in the LPT experimental data arises from the presence of ghost tracks caused by the carbon fibre surface of the DUT24, which is highly reflective and tends to be incorrectly triangulated as real particle displacements.

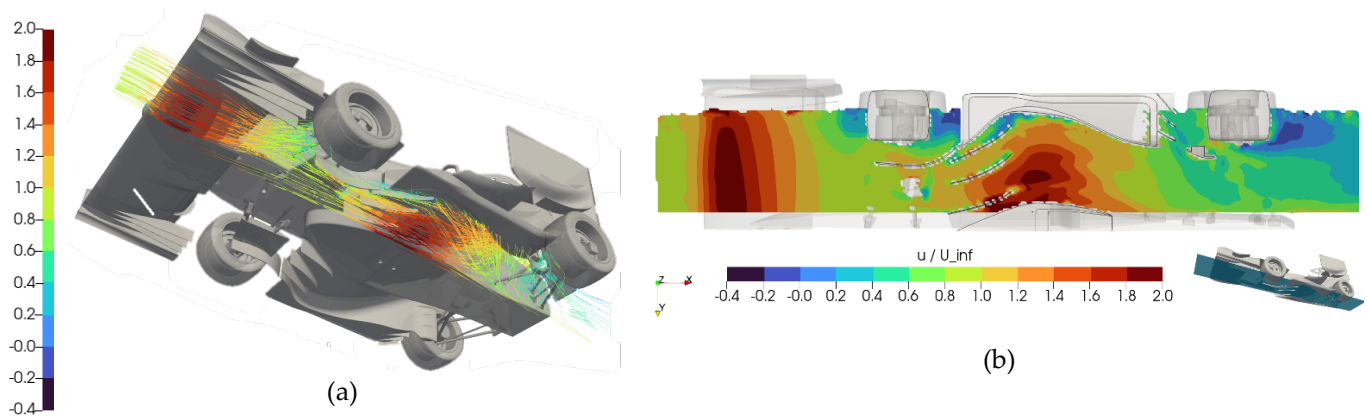
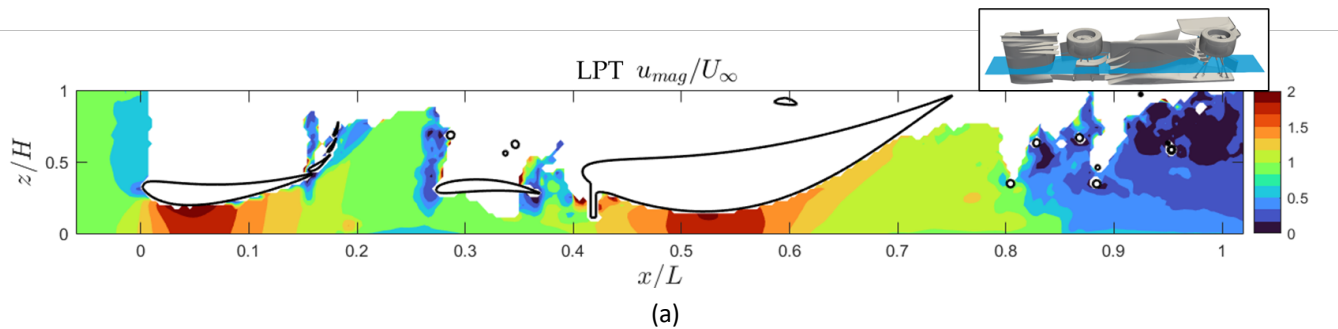


Figure 5. Velocity field for in the underbody region: (a) streamlines color-coded by streamwise velocity component and (b) streamwise velocity component contours in a section cut at $z = 50$ mm from the ground.



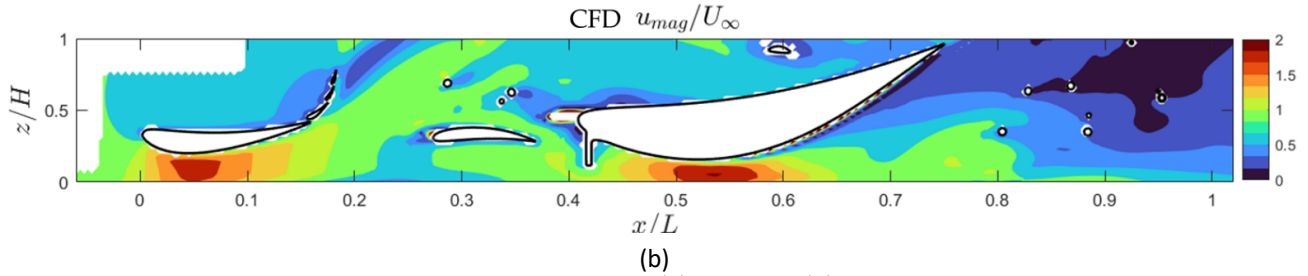


Figure 6. Velocity magnitude comparison for (a) LPT and (b) CFD data, $y = -350$ mm slice.

Furthermore, strake-generated vortices emerge as coherent structures at the diffuser inlet, gradually stretching downstream while losing intensity. LPT results reveal three dominant vortices that later merge into larger structures, delaying breakdown and ensuring attached flow along the diffuser's steep surfaces (although visually limited by the measurement domain dimensions). An additional vortex forms near the diffuser side edge due to lateral inflow driven by pressure differences between the underbody and the outer flow. CFD again captures the presence but underestimates the strength of these vortices, often missing the outer vortex entirely. Q-criterion visualisations reinforce this finding: experimental vortices remain coherent throughout the diffuser, whereas simulated ones dissipate prematurely. Figure 7 illustrates the comparison between experimental and simulated data for the diffuser-generated vortices.

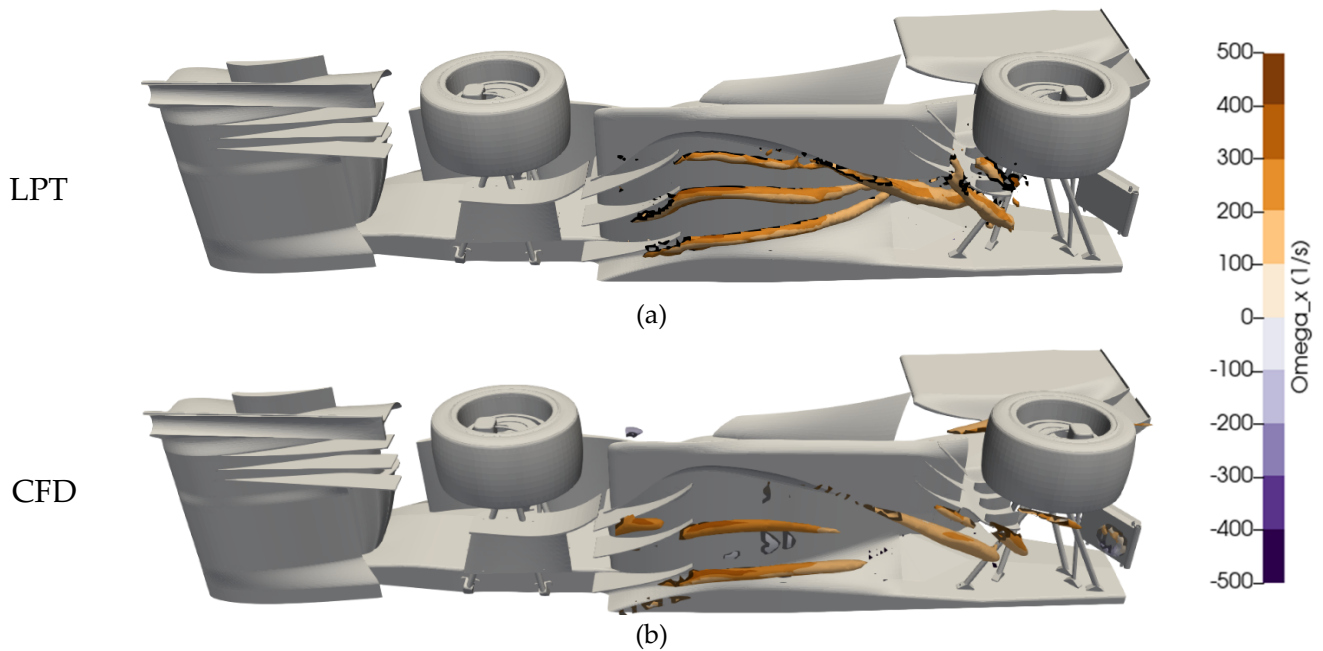


Figure 7. Q-criterion iso-surfaces for $Q = 5 \cdot 10^3$ ($1/s^2$), coloured by streamwise vorticity ω_x ($1/s$) for (a) LPT and (b) CFD.

Overall, the results confirm that DUT24's diffuser achieves stable flow attachment under ground-effect operation, with strake-induced vortices providing critical flow control. While CFD captures

global trends, it underpredicts vortex strength and diffuser momentum transfer, highlighting the limitations of RANS modelling for such strongly three-dimensional, vortical flows.

3.2. Pressure field

The reconstructed pressure field under the DUT24 highlights the principal aerodynamic mechanisms driving underbody performance. Using the non-dimensional pressure coefficient $C_p = (p - p_\infty)/q_\infty$, the flow field was integrated according to the hybrid Bernoulli–gradient method described previously, partitioning rotational and irrotational regions based on a total-pressure threshold of $C_{p,\text{tot}} < 0.9$. Within the irrotational domain, static pressure was obtained directly from Bernoulli’s equation, while in rotational regions it was recovered through integration of the convective and Reynolds-stress terms of the momentum equation. Contours of the static pressure coefficient are given in Figure 8 inside a wall-normal slice.

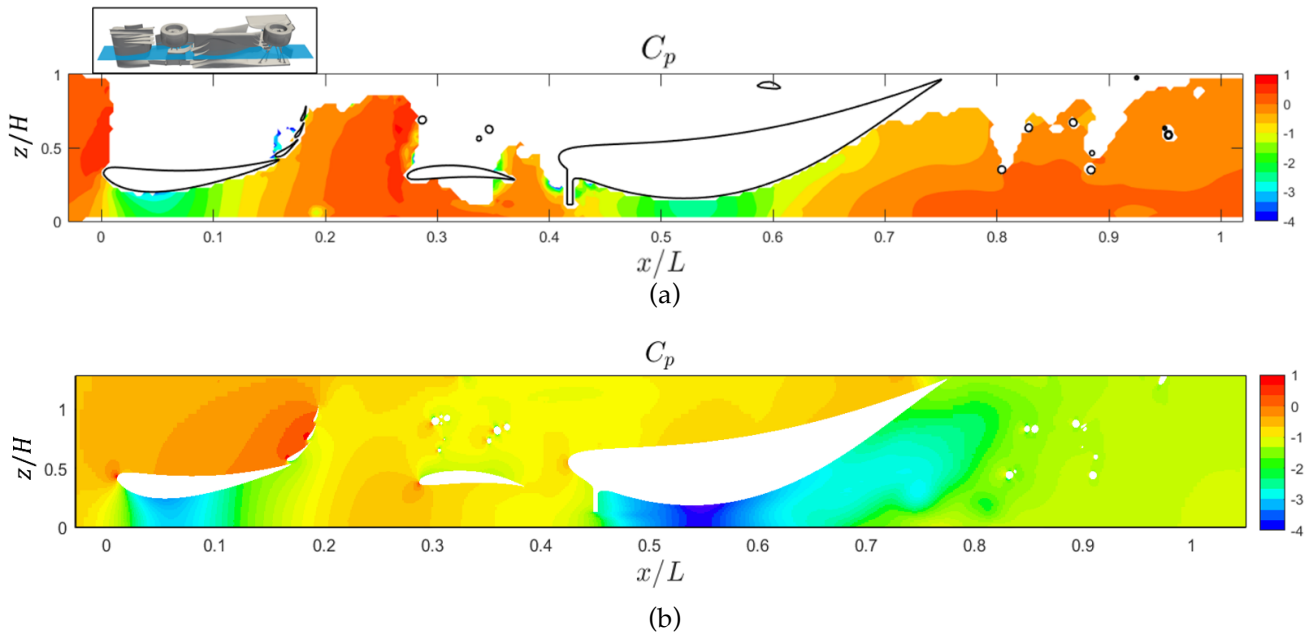


Figure 8. Comparison of static pressure field inside the underbody for (a) LPT and (b) CFD data at $y = -350$ mm.

Upstream, $C_p \approx 0$ indicates freestream conditions, while beneath the front wing the flow accelerates rapidly, producing a sharp suction peak consistent with the velocity field. This low-pressure zone contributes significantly to front-axle downforce and is intensified by ground effect due to the wing’s proximity to the floor. At the diffuser inlet, the pressure remains negative, with the most pronounced suction occurring near the minimum ground clearance, reflecting local flow acceleration necessary to conserve mass. From this throat region, the static pressure rises gradually without encountering an adverse gradient large enough to cause separation, confirming an

efficient pressure recovery process. The lowest C_p values, around -4 in the CFD data and slightly higher in the reconstructed field, occur beneath the inner strakes where vortices emerge and its low-pressure core effect is strongest. Both PIV and CFD reveal the same qualitative trend: a smooth recovery toward the outlet, with the experimental data showing slightly higher C_p due to noise and spatial resolution limits.

4. The flow field on the side of the vehicle

4.1. Velocity field

The wakes generated by the rotating tyres of DUT24 are highly three-dimensional, dominated by separation at the wheel shoulders and strong ground interaction. Immediately downstream of the front tyres, a broad low-momentum region forms, with negligible streamwise velocity and extensive recirculation. These wakes expand vertically and spanwise, interacting with both the diffuser entry and the lower surfaces of the bull-wings. Similar to the diffuser study, streamlines were extracted from the velocity field to visualize flow structures, found in Figure 9.

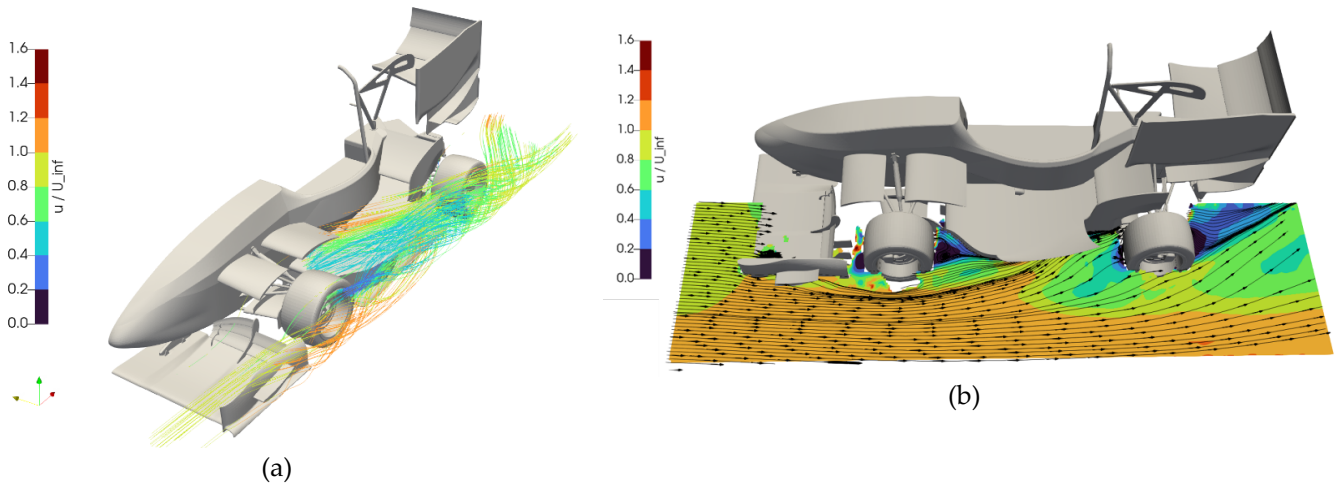


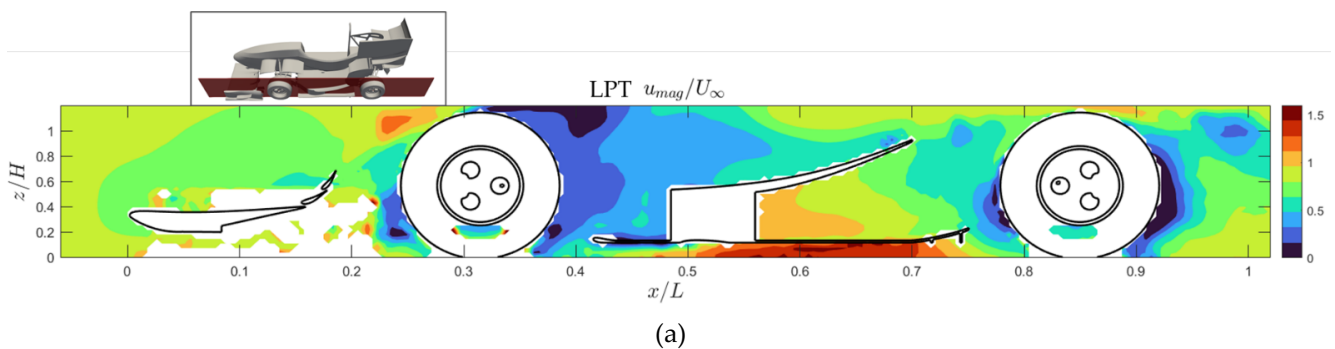
Figure 9. Velocity field for on the side of the car: (a) streamlines color-coded by streamwise velocity component and (b) streamwise velocity contours with streamlines in a section cut at $z = 150$ mm from the ground.

The velocity fields reveal sharp momentum deficits at the front wheels, with stagnation displaced below mid-height by tyre rotation and jetting effects. Flow recovery is aided by the vortices shed from the front wing that deflect low-energy fluid outward, conserving momentum near the chassis. Behind the rear wheels, wakes are smaller and recover faster, as the incoming flow has an increased turbulent nature, influenced by upstream elements. Flow jetting from the contact patch further lifts the wake, reducing its extent near the ground.

When comparing the velocity field results obtained for both LPT and CFD cases, more insights about the tyre-wake data can be obtained. Figure 10 illustrates the differences in shape and magnitude of the wakes. In the case of the experimental results, both wakes tend to be pushed upwards due to the lifting effects of the jet exiting the rear of the tyre. In the CFD case, even though the rotation of the wheels was set as a tangential velocity boundary condition, the solver is not correctly accounting for the viscous jetting effect, resulting in flow stagnation at lower heights, and in considerably wider wakes.

Despite these differences in detail, the overall topology of the flow field is captured in both approaches, with the main wake cores, recirculation regions consistently reproduced. The CFD simulation, in contrast, systematically over-predicts the wake size and underestimates the recovery velocity and the jet-induced upward deflection, pointing to the need for improved wheel modelling.

The wake topology is governed by a series of coherent vortical structures. At the front of the vehicle, vortices originating from the wing endplates, flaps, and rotating tyres interact with hammerhead and Y250-like features (Granados-Ortiz et al., 2023), collectively deflecting the wake outboard and upward. This mechanism reduces the wake's influence in the central flow region. As illustrated in Figure 11, the front wing sheds multiple clockwise vortices that act to lift the front-tyre wake, mitigating its interference with downstream aerodynamic components. Counterclockwise vortices, generated by the inner section of the front wing and the bull-wings, complement this process by further displacing the tyre wake outward. In addition, wheel rotation induces a jetting effect that produces a vortex pair, which quickly convects upward, reinforcing the overall upward deflection of the near-wake region.



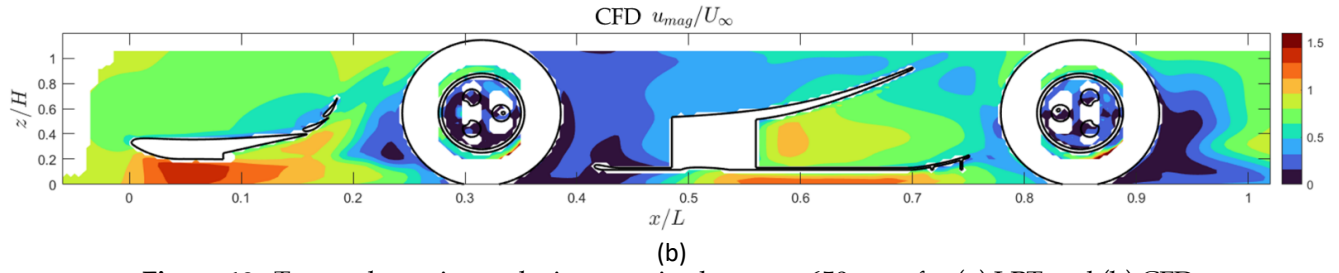


Figure 10. Tyre wake region: velocity magnitude at $y = -650$ mm, for (a) LPT and (b) CFD.

Comparisons with CFD confirm the main wake structures and rotational sense but reveal systematic differences. Simulations without wheel rotation predict larger wakes with weaker momentum recovery and diminished vortex strength, while experiments show coherent vortices persisting further downstream. Despite these discrepancies, both approaches capture the essential topology: strong separation at the tyres, rapid outboard deflection of the front wake, and inboard contraction of the rear wake under side-diffuser influence.

Overall, the results highlight tyre wakes as dominant sources of drag and unsteadiness in open-wheel aerodynamics. Effective aerodynamic optimization relies on upstream vortex systems and diffuser-induced structures to redirect and contract the wakes, thereby improving downstream flow quality and supporting diffuser and rear-wing performance.

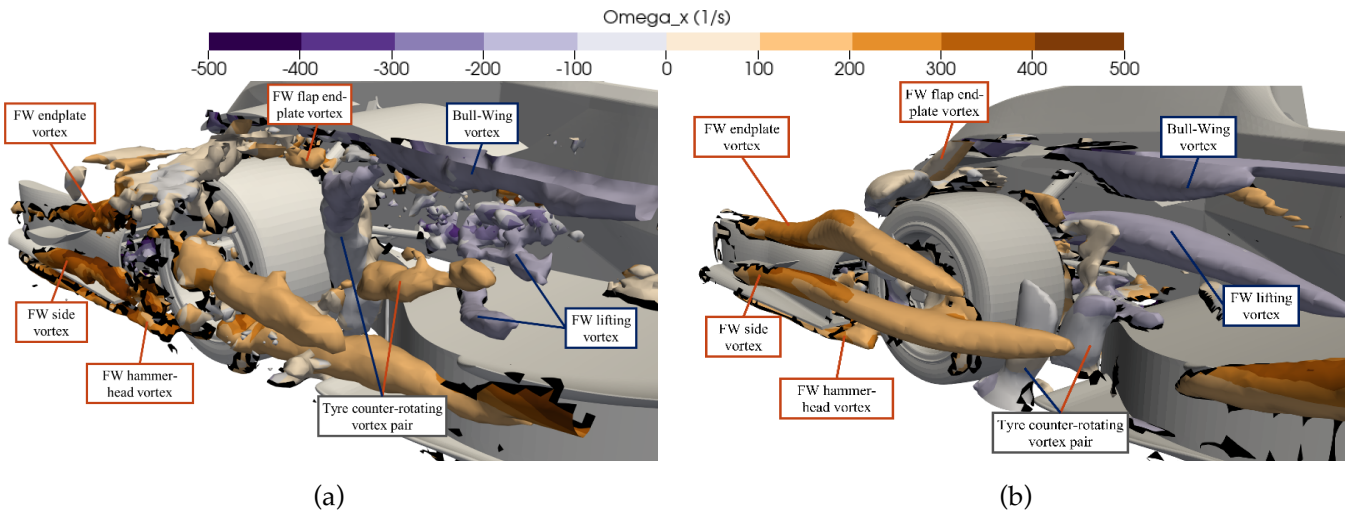


Figure 11. Vortex system behind front tyre, Q-criterion isosurfaces $Q = 10^3$ ($1/s^2$) coloured by streamwise vorticity ω_x ($1/s$) for (a) LPT and (b) CFD.

4.2. Pressure field

The pressure field around the tyres complements the diffuser results, exposing how local flow structures govern the wake topology and overall pressure balance. The reconstructed C_p maps shown in Figure 12 indicate a well-defined high-pressure stagnation region ahead of each tyre and

an extended low-pressure core immediately downstream, directly corresponding to the velocity deficit and coherent vortices identified earlier. The front-tyre stagnation point lies below mid-height, while the rear tyre exhibits a broader high-pressure region and a deeper wake deficit. In the PIV data, tyre rotation produces an upward jetting effect that drives the wake vertically and outboard, promoting pressure recovery and reducing wake size. The CFD simulations fail to reproduce correctly the effects of rotation, thus missing this flow feature and therefore predicting a broader, lower-pressure wake. Both datasets, however, capture the principal pressure gradients: the upstream stagnation region, the low-pressure separation bubble behind the tyres, and the subsequent recovery influenced by diffuser upwash.

Overall, the reconstructed pressure results show strong agreement between PIV and CFD in terms of wake topology and stagnation placement, despite quantitative discrepancies attributed to measurement noise and rotational modelling.

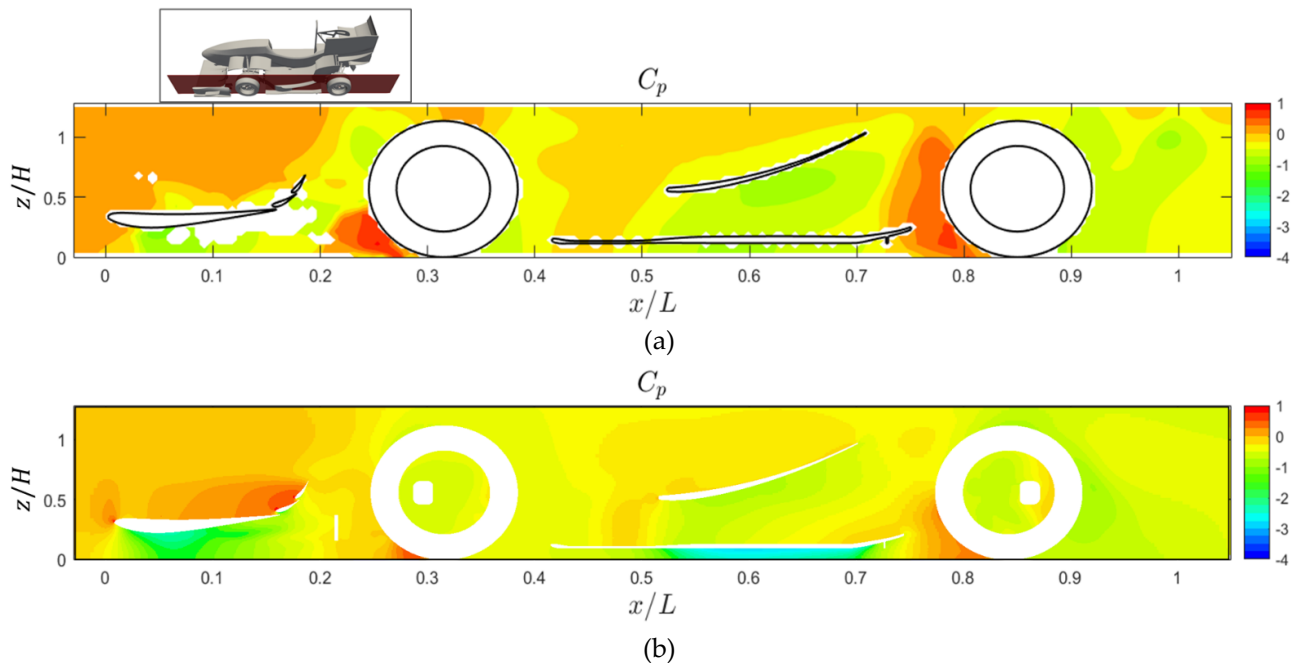


Figure 12. Comparison of static pressure field on the side of the car for (a) LPT and (b) CFD data at $y = -350$ mm.

5. Conclusions

The Ring of Fire system was successfully adapted for full-scale aerodynamic measurements on the DUT24 Formula Student car to investigate the flow both in the underbody region and on the vehicle's side. The underbody analysis revealed peak velocity up to $u/U_\infty > 2.2$ at the diffuser inlet, confirming efficient downforce generation and stable attached flow without vortex

breakdown. Diffuser strakes produced coherent streamwise vortices that maintained the flow attached and increased underbody performance. Tyre-wake measurements showed highly three-dimensional unsteady flow, with the front wake being wider and more energetic than the rear. Upstream elements, such as the front wing and side diffuser, were found to mitigate wake effects and enhance overall aerodynamic efficiency.

CFD comparisons matched experimental flow trends but underpredicted vortex strength and diffuser momentum conservation. Pressure recovery patterns were consistent, though quantitative discrepancies highlighted limitations in both CFD coupling and PIV-based pressure reconstruction.

Acknowledgements

The authors would like to thank the members of the Formula Student Team Delft for their support during the experiments.

References

- Agüera, N., Cafiero, G., Astarita, T., & Discetti, S. (2016). Ensemble 3D PTV for high resolution turbulent statistics. *Measurement Science and Technology*, 27(12), 124011.
- Battegazzore, A., Grille Guerra, A., & Sciacchitano, A. (2024). On-Site Characterization of Automotive Diffuser Aerodynamics by 3D LPT. In: *Proceedings of the 21st International Symposium on the Application of Laser and Imaging Techniques to Fluid Mechanics*.
- Cooper, K. R., Bertenyi, T., Dutil, G., Syms, J., & Sovran, G. (1998). The aerodynamic performance of automotive underbody diffusers. In *SAE Technical Papers*. SAE International.
- Fackrell, J. E. (1974). The Aerodynamics of an Isolated Wheel Rotating in Contact With the Ground. PhD thesis, University of London, U.K.
- Fackrell, J. E., & Harvey, J. K. (1975). The Aerodynamics of an Isolated Road Wheel. In B. Pershing (Ed.), *Proceedings of the Second AIAA Symposium of Aerodynamics of Sports and Competition Automobiles* (pp. 119–125). Los Angeles, CA, U.S.A.
- Gérardin, F., Gentric, C., & Midoux, N. (2014). Particle dispersion in the near-wake of an isolated rotating wheel: Experimental and CFD study. *Journal of Aerosol Science*, 76, 58–74.
- Granados-Ortiz, F.-J., Morales-Higueras, P., & Ortega-Casanova, J. (2023). 3D CFD simulation of the interaction between front wheels & brake ducts and optimised five-element F1 race car front wings under regulations. *Alexandria Engineering Journal*, 64.
- Grille Guerra, A., Porcar Galan, L., Sciacchitano, A., & Scarano, F. (2025). Treatment of light reflections in 3D PIV systems. *Measurement Science and Technology*, 36(11), 115301.

- Holland, C., Grille Guerra, A., & Sciacchitano, A. (2025). On-Site 3D LPT for Automotive Underbody Aerodynamics. In: *Proceedings of the International Symposium on the Application of Laser and Imaging Techniques to Fluid Mechanics*.
- Hüttig, S., Kühn, M., Gericke, T., Bloem, G., Sciacchitano, A. & Akkermans, R.A. (2025). On-road vehicle aerodynamics with a large-scale stereoscopic-PIV setup: “the Ring of Fire”. *Experiments in Fluids*, 66(6), p.125.
- Jowsey, L., & Passmore, M. (2010). Experimental study of multiple-channel automotive underbody diffusers. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, 224(7), 865-879.
- Josefsson, E., Hobeika, T., Sebben, S., & Urquhart, M. (2022). Investigation of Tyre Pattern Effect on the Aerodynamics of a Passenger Vehicle. *Journal of Fluids Engineering*. DOI: 10.1115/1.4054851.
- Jux, C., Sciacchitano, A., & Scarano, F. (2020). Flow pressure evaluation on generic surfaces by robotic volumetric PTV. *Measurement Science and Technology*, 31(10), 104001.
- Katz, J., & Garcia, D. (2002). Aerodynamic Effects of Indy Car Components. SAE Technical Paper 2002-01-3311.
- Katz, J., & Morey, F. (2008). Aerodynamics of Large-Scale Vortex Generator in Ground Effect. ASME. *J. Fluids Eng.* July 2008; 130(7): 071101.
- Knowles, R. D. (2005). Monoposto Racecar Wheel Aerodynamics: Investigation of Near-Wake Structure & Support-Sting Interference. PhD thesis, Cranfield University.
- McManus, J., & Zhang, X. (2006). A Computational Study of the Flow Around an Isolated Wheel in Contact With the Ground. *Journal of Fluids Engineering*, 128, 520–530. DOI: 10.1115/1.2175158.
- Merker, E., Breuer, N., Berneburg, H., & Emmelmann, H. (1991). On the Aerodynamic Interference due to the Rotating Wheels of Passenger Cars. SAE Technical Paper 910311.
- Parra, F. L., & Kontis, K. (2006). Aerodynamic effectiveness of the flow of exhaust gases in a generic Formula One car configuration. *The Aeronautical Journal*, 110(1110), 573–586.
- Ruhrmann, A., & Zhang, X. (2003). Influence of Diffuser Angle on a Bluff Body in Ground Effect. *Journal of Fluids Engineering*, 125(2), 332–338. DOI: 10.1115/1.1537252.
- Scarano, F., Ghaemi, S., Caridi, G. C. A., et al. (2015). On the Use of Helium-Filled Soap Bubbles for Large-Scale Tomographic PIV in Wind Tunnel Experiments. *Experiments in Fluids*, 56, 42.
- Scarano, F., Schneiders, J. F. G., Saiz, G. G., & Sciacchitano, A. (2022). Dense velocity reconstruction with VIC-based time-segment assimilation. *Experiments in Fluids*, 63(6).
- Schanz, D., Gesemann, S., & Schröder, A. (2016). Shake-The-Box: Lagrangian Particle Tracking at High Particle Image Densities. *Experiments in Fluids*, 57, 70.

- Senior, A. E. (2002). The Aerodynamics of a Diffuser Equipped Bluff Body in Ground Effect. PhD thesis, University of Southampton.
- Spoelstra, A., de Martino Norante, L., Terra, W., Sciacchitano, A., & Scarano, F. (2019). On-site Cycling Drag Analysis with the Ring of Fire. *Experiments in Fluids*, 60, 90.
- Toet, W. (2013). Aerodynamics and aerodynamic research in Formula 1. *Aeronautical Journal*, 117(1187).
- Wieneke, B. (2008). Volume self-calibration for 3D particle image velocimetry. *Experiments in Fluids*, 45, 549–556.
- Zhang, G., He, J., Zhou, F., Wang, Z., Chen, D., Yu, Y., Wang, D., Zhang, Y. (2022). Application of Strake Structure in Aerodynamic Design of Formula Racing. In: *Proceedings of China SAE Congress 2020: Selected Papers* (Lecture Notes in Electrical Engineering, vol. 769, pp. 87–97). Springer, Singapore.