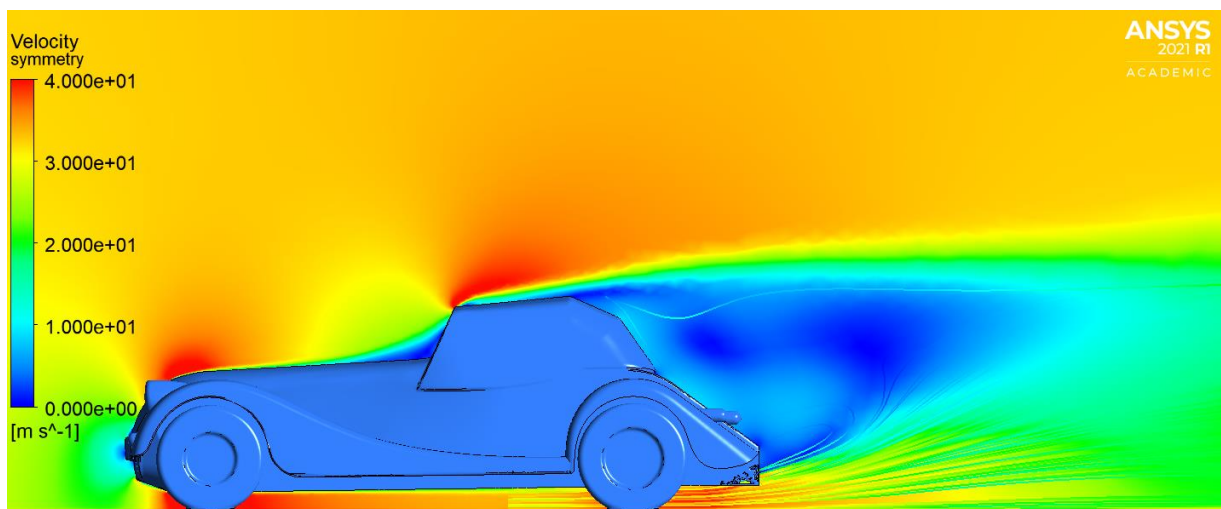


University of Wales Trinity St Davids

Aerodynamic Development and Validation of the Morgan Plus 4 through CFD Simulation



Author: Dylan Huw Davies

Supervisor: Andrew Noble and Tim Tudor

Student Number: 2100937

Course title: MSc Motorsport Engineering

Module code: ACEA7003

Date: 07/09/25

Declaration of Authenticity

The Thesis is the result of the author's original research. It has been composed by the author and has not been previously submitted for examination which has led to the award of a degree.

A handwritten signature in black ink, reading "D. Davies", enclosed within a black rectangular border. The signature is written in a cursive, flowing style.

Abstract

This dissertation optimises the underbody aerodynamics of a Morgan Plus 4 and validates the external-flow CFD workflow before transfer to the vehicle. The methodology is established on the DrivAer model with a moving ground and rotating wheels, domain sizing to best-practice guidance, a mesh-independence study, and Body-of-Influence (BoI) refinement of wake and separation regions. Multiple turbulence closures are assessed; the production simulations for the Morgan use the realizable k-epsilon model with wall functions (target y^+ in the log-law range). BoI application improves numerical behaviour (orthogonal quality increase and stable residual decay), and the setup is then applied to the Morgan geometry, which incorporates realistic packaging, rotating wheels, and a diffuser-first redesign with strakes and a front splitter to condition the underfloor inflow. The final package changes the baseline coefficients from $C_d = 0.4675$, $C_l = +0.0689$ to $C_d = 0.4959$, $C_l = -0.2436$, converting lift to net downforce with a modest drag rise. Pressure and velocity fields show a deeper underfloor suction region and sharper exit recovery relative to the baseline, consistent with the measured change in coefficients and indicating sensitivity to ride-height management to avoid diffuser stall.

The primary objective was to validate a CFD workflow and apply it to heritage-inspired vehicle geometry, developing diffuser and splitter concepts capable of reducing wake size and generating downforce within strict packaging limits. This work demonstrates that heritage brands such as Morgan can integrate modern aerodynamic practices without compromising brand identity,

providing both technical and industrial relevance for sustainable performance development in low-volume sports cars.

Contents

Abstract.....	2
Table of figures and tables	7
1.0 Introduction	10
1.0.1 Reasons and Relevance / Project Rationale	10
1.0.3 Objectives.....	11
2.0 Literature Review.....	12
2.1 Aerodynamic Improvement Paper/Journal Review	12
2.1 Splitter Ride-Height Effects	14
2.2 Aerodynamic Diffuser Design	14
2.2.1 Introduction	14
2.2.2 Importance of Diffuser and Splitter Design	14
2.2.3 Ramp Angle and Length	15
2.2.4 Role of Strakes in Multi-Channel Diffusers	15
2.2.5 Integration with Vehicle Aerodynamics	16
2.2.6 Critical Evaluation	16
2.2.7 Conclusion.....	16
2.3 Validation Research	17
2.3.1 Model Reseach.....	17
2.3.2 CFD Methodology Techniques.....	17
2.3.3 Turbulence Models	18
2.3.4 Moving Ground and Rotating Wheels	18
2.4 Mesh Quality.....	19
3.0 Validation study.....	20
3.1 Geometry and Experimental Reference Data	20
3.1.1 CAD Model Preparation and Rebuilding	21
3.1.2 Fluid Domain Geometry.....	21
3.2 Mesh Strategy and Setup Methodology	22
3.2.1 Mesh Quality Assessment.....	23
3.3 Setup	25
3.3.1 Turbulence Model Selection.....	25
4.0 Morgan Plus Four Study	26
4.1 Geometry and Mesh Methodology	26

4.1.1 Body of Influence (Bol)	26
4.1.2 Mesh Study	28
4.1.3 Prism Layers.....	30
4.1.4 Mesh Quality Improvements	32
4.2 Solution Setup for Morgan Geometry	33
4.2.1 Moving Ground and Rotating Wheels	33
4.2.2 Turbulence Model Setup and Analysis	34
4.2.3 Summary of Turbulence Model Findings	39
4.3 Tetrahedral vs Polyhedral.....	40
5 Baseline Analysis	44
5.1 Rear Analysis.....	44
5.1.1 Underbody Wake Development	45
5.1.2 Global Flow and Roof Separation	46
5.2 Quantitative Baseline Performance.....	47
5.3 Baseline Conclusions	48
6.0 Morgan Redesign	49
6.1 Convergence Problems and Model Rebuilding.....	50
6.2 Redesign Objectives	51
6.3 Aerodynamic and Bodywork Design Regulations.....	51
6.4 Diffuser Design	52
6.4.1 Initial Diffuser Design Analysis	54
6.5.1 Design Limitations	57
6.6 Diffuser Design three: Expanding Design Area	57
6.7 Redesign of Kick-Point.....	61
6.8 Diffuser Strakes – Effect on Underfloor Flow.....	62
6.9 Reducing Wheel Turbulence Intervention	65
6.10 Centre Diffuser Profile Change	67
7.0 Underbody Flow Management.....	69
7.1 Complete Flat Floor	69
7.2 Louvre Design on Wheel Arches.....	69
7.3 Conclusion	71
8.0 Splitter Design.....	72
8.1 Splitter Effect on Diffuser	72
8.2 Splitter Analysis	74
8.3 Splitter Design Two Analysis	74
8.3.1 Conclusion.....	76

8.4 Airfoil Splitter Design.....	77
8.4.1 Airfoil Results Analysis.....	78
8.4.2 Effect of Ride Height	79
8.4.3 Splitter Conclusion.....	80
8.5 Diffuser Analysis and Redesign After Splitter	81
8.5.1 Conclusion.....	85
9.0 Air Dam Design with Flat Splitter	86
9.1 Solution Problems Due to New Geometry	86
9.2 Flat Splitter and Air Dam Analysis	87
9.2.1 Leading-Edge Redesign.....	90
10.0.0 Conclusion.....	91
10.1 Limitations and Issues.....	92
10.2 Further Work.....	92
11.0.0 References	94

Table of figures and tables

Figure 1 small angled canards on MIRA Car (<i>Salhan, 2020</i>).	12
Figure 2 50mm overhang and no splitter, Velocity and pressure plot	12
Figure 3 Race Louvers on Front Wheel Arch- Mustang	13
Figure 4 vortex generator example	13
Figure 5 DrivAer car (left), MIRA car (right)	17
Figure 6 Mesh Quality (ANSYS Inc., 2012).	19
Figure 7 DrivAer Fastback with flat underbody CAD model	20
Figure 8 Domain Size For DrivAer	21
Figure 9 Orthogonal Quality 0.306	24
Figure 10 Cl convergence issue due to low mesh quality	24
Figure 11 Scaled Residuals not converging	27
Figure 12 Two density boxes to capture flow separation and the wake region	27
Figure 13 pressure plot. BOI vs no BOI	28
Figure 14 vortex core. BOI vs no BOI	28
Figure 15 Mesh Independent Study in Near BOI	29
Figure 16 Mesh independence study for the part mesh, showing convergence of lift (Cl) and drag (Cd) at a mesh size of 0.3 m.	30
Figure 17 Comparison of pressure coefficient distribution on the Morgan Plus 4 top surface using 3 and 5 prism layers ($k-\epsilon$) against experimental data.	31
Figure 18 Effect of part mesh refinement on pressure coefficient predictions with 5 prism layers compared to experimental data.	32
Figure 19 Final Mesh	32
Figure 20 Pressure Distribution Comparison with improved mesh strategy and y plus	33
Figure 21 Rotating Wheels	34
Figure 22 K-Omega Bsl, using 5 prism layers	35
Figure 23 20m/s vs 32 m/s	36
Figure 24 5 Prism vs Literature based research methodology	36
Figure 25 convergence issues with STT K-Omega	37
Figure 26 32m/s STT $k-\omega$ turbulence model with experimental pressure tap data for the Morgan Plus 4	37
Figure 27 realizable $k-\epsilon$ predictions with the pressure tap data	38
Figure 28 Pressure Distribution – CFD Mesh Improvements with Refined y^+ Resolution	38
Figure 29 Velocity contour comparison at the ground plane for polyhedral (Y) and tetrahedral (X) meshes	40
Figure 30 Velocity distribution through the wheel arch region for polyhedral (Y) and tetrahedral (X) meshes	41
Figure 31 Flow from wheel and roof separation contours comparing polyhedral (Y) and tetrahedral (X) meshes	41
Figure 32 Wake streamline comparison showing vortex structures for polyhedral (Y) and tetrahedral (X) meshes	42
Figure 33 Pressure coefficient comparison at the centerline and wheel arch: experimental data vs tetrahedral and polyhedral meshes	42
Figure 34 Baseline Analysis: Streamline From underneath rear car	44
Figure 35 Baseline Analysis of rear diffuser location and wake region	45
Figure 36 Baseline Morgan Plus 4 velocity contours showing wake development at different far-field widths (0.1 m, 0.3 m, 0.5 m, 0.7 m).	45

Figure 37 Plane positions.....	46
Figure 38 Pressure contour comparison of the Morgan Plus 4 at varying domain lengths (-1.0 m, -1.5 m, -1.8 m, -2.0 m).....	47
Figure 39 Morgan Plus Four fitted with roll cage	49
Figure 40 Open top wheel meshing issues	50
Figure 41 Convergence issues with open top CAD Model	50
Figure 42 Available Rear Packaging Space for Diffuser Design on the Morgan Plus 4, measured in workshop.....	53
Figure 43 Initial Diffuser Design.....	53
Figure 44 Holes in diffuser from meshing.....	54
Figure 45 Flow separation occurring early at diffuser profile	55
Figure 46 velocity plot, wake region analysis for first design	55
Figure 47 Diffuser Design two	56
Figure 48 Average Cd and Cl due to repeatability	56
Figure 49 Diffuser Design 3	58
Figure 50 Design 3, wake region	58
Figure 51 Streamlines in the wake region from design 3.....	59
Figure 52 Streamlines underneath the car from design 3.....	59
Figure 53 Streams along diffuser profile	60
Figure 54 turbulence flow bleeding from wheels, design 3.....	60
Figure 55 diffuser profile, velocity plot near rear wheel.....	61
Figure 56 Pressure distribution along diffuser	62
Figure 57 Pressure distribution along the diffuser centreline vs x from origin.; Case 6 (<i>magenta, with strakes</i>)	63
Figure 58 Velocity contour at x = -0.70 m (from origin), with strakes.....	64
Figure 59 Velocity contour at x = -0.60 m (from origin), with strakes.....	64
Figure 60 Velocity contour at x = -0.40 m (from origin), Strakes.....	65
Figure 61 compares the velocity distribution at x = -0.60 m for two diffuser layouts: the new design with additional strakes (left) and the original design with fewer strakes (right).	66
Figure 62 compares the velocity contours at x = -0.40 m for the new diffuser with additional strakes (left) and the original diffuser with fewer strakes (right).....	66
Figure 63 compares the velocity contours at 0.4 of the car width for the new diffuser with additional strakes (left) and the original diffuser with fewer strakes (right).....	67
Figure 64 compares the velocity field for the baseline diffuser (top left) and the revised design with a reshaped centre profile (top right), with the lower image showing the velocity difference between the two cases, at centre of car.	68
Figure 65 Underbody floor flow	69
Figure 66 Rear Wheel low velocity build up	70
Figure 67 Basic Louvre design.....	70
Figure 68 flow before entering diffuser . (left no splitter, right has a splitter)	72
Figure 69 flow at diffuser at 0.05m, showing lower velocity due to splitter. (left no splitter, right has a splitter).....	73
Figure 70 change in angle between the flat floor and the diffuser ramp	73
Figure 71 CFD velocity comparison of the Morgan front end without a splitter (left) and with the smoother leading-edge splitter (right).	74
Figure 72 Splitter Design two, with smooth profile	75
Figure 73 Velocity contour comparison beneath the Morgan Plus 4 with and without open splitter design	75

Figure 74 Velocity contours showing reduced splitter efficiency due to central hole design, leading to diminished front downforce	76
Figure 75 NACA 4412 profile (Airfoil Tools, 2024).....	77
Figure 76 70 NACA 4412 profile recreated in SolidWorks	78
Figure 77 Airfoil profile. Streamlines.....	78
Figure 78 Effect of splitter Ends.....	79
Figure 79 Airfoil profile. Cl and Cd vs ride height	80
Figure 80 KickPoint redesign (Left) old diffuser design (Right) New design	81
Figure 81 (Left) old diffuser design (Right) New design . Velocity plot at rear diffuser	82
Figure 82 Pressure distribution along car at centerline	82
Figure 83 velocity plot. flow separation still occurs. (Left) old diffuser design (Right) New design	83
Figure 84 Pressure distribution along car at 0.2m.....	84
Figure 85 Velocity Plot at 0.2m. (Left) old diffuser design (Right) New design	84
Figure 86 Effect of using warped face gradient correction. left(with) right(without)	86
Figure 87 Splitter and Air dam Analysis.....	87
Figure 88 Wake region of flat splitter and air dam design	87
Figure 89 Centerline pressure distribution. Air dam splitter vs diffuser design 6	88
Figure 90 Pressure Distribution at centerline and 0.2m	89
Figure 91 Velocity plot at 0.2m(width) near strakes	89
Figure 92 Leading edge redesign	90

Table 1 Moving Wheel Conditions AT 20 m/s	34
Table 2 Moving Wheel Conditions at 32m/s	34
Table 3 Comparison of Turbulence Models for Morgan Plus Four (Cd at 32 m/s)	39
Table 4 Difference in Experimental and CFD	48
Table 5 Roof Down Wheel Positions	49

1.0 Introduction

1.0.1 Reasons and Relevance / Project Rationale

This research aligns with the wider industry focus on performance optimisation, fuel efficiency, and sustainability, where manufacturers continually seek to reduce aerodynamic drag and enhance downforce for both road and competition vehicles. Studies such as Yuan and Wang (2017) have demonstrated that complex and unrefined underbody geometries can increase drag by more than 23% compared to optimised flat floors, underlining the significance of diffuser, flat floor, and Venturi tunnel design in overall aerodynamic efficiency. By investigating such underbody modifications, this project directly contributes to improved aerodynamic performance in applications ranging from track use to everyday driving.

While underbody aerodynamics has been extensively studied in motorsport, limited research has been directed toward heritage-inspired cars such as the Morgan Plus 4. Unlike modern manufacturers such as Porsche and Ferrari, which integrate advanced underbody features as standard, heritage-focused companies have typically prioritised style and driving character over aerodynamic optimisation. The Caterham C120 project, for example, emphasised agility and lightweight construction but deliberately avoided the adoption of complex underbody systems to preserve brand identity (Caterham, 2025). This creates a clear research gap: heritage vehicles stand to benefit from discreet aerodynamic enhancements that can improve performance without compromising their traditional design values.

Reducing drag through optimised underbody flow management not only enhances stability but also improves efficiency by lowering fuel consumption in combustion-powered vehicles and extending range in electric vehicles (Nyoni, 2024). As global regulations tighten and sustainability pressures grow, it is increasingly important for low-volume manufacturers such as Morgan to adopt efficiency-focused design practices. Ensuring aerodynamic effectiveness allows heritage brands to remain competitive while meeting modern environmental expectations.

Finally, the use of Computational Fluid Dynamics (CFD) provides a cost- and time-effective method for evaluating such design changes. By validating CFD methodology against established reference models before applying it to the Morgan Plus 4, this project minimises the need for extensive wind tunnel testing while also highlighting the advantages and limitations of numerical simulation in capturing complex underbody flow behaviour.

This rationale directly supports the project's objectives: to validate a CFD workflow, apply it to the Morgan Plus 4 within its packaging and regulatory constraints, and critically assess diffuser and splitter configurations capable of reducing wake size, lowering drag, and generating downforce in a manner consistent with both industrial feasibility and the heritage identity of the marque.

1.0.3 Objectives

Validation of CFD methodology – establish a robust external aerodynamics CFD workflow using the DrivAer model and Morgan, ensuring mesh independence, turbulence model suitability, and accuracy against published experimental data.

Application to Morgan Plus 4 geometry – apply the validated workflow to the Morgan platform, incorporating realistic packaging constraints, moving ground, and rotating wheels to replicate on-road aerodynamic conditions.

Underbody aerodynamic optimisation – design and evaluate diffuser and splitter configurations capable of reducing rear wake size, lowering lift, and converting the baseline car from a lift-producing to a downforce-producing state.

Comparative analysis of design iterations – critically assess how diffuser geometry, strakes, splitter profiles, and ride-height changes affect drag and lift coefficients, benchmarked against literature and experimental data.

Industrial and motorsport relevance – demonstrate how modern aerodynamic practices can be applied to a heritage-inspired sports car without compromising styling or packaging, providing recommendations aligned with Motorsport UK regulations and sustainable performance objectives.

2.0 Literature Review

2.1 Aerodynamic Improvement Paper/Journal Review

Research on front-end aerodynamic devices consistently highlights the trade-off between lift reduction and drag increase. Salhan (2020) demonstrated that splitters reduce front lift by creating a pressure differential across the plate, with an average lift reduction of 0.078 but a drag increase of 0.039, while small angled canards (Figure 1) achieved similar front balance with negligible drag penalty .

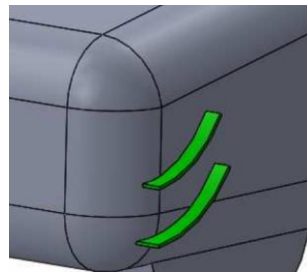


Figure 1 small angled canards on MIRA Car (Salhan, 2020).

Bhattacharjee et al. (2019) showed that splitter extensions up to 30 mm provide most of the downforce gain as seen in Figure 2, with diminishing returns and additional drag beyond this length, indicating a practical efficiency threshold.

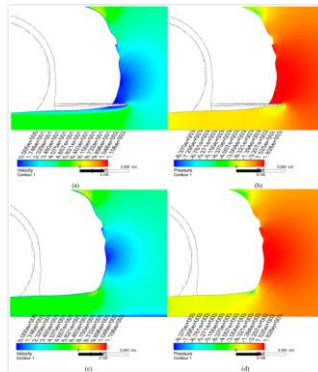


Figure 2 50mm overhang and no splitter, Velocity and pressure plot

More advanced concepts, such as the inflatable active splitter studied by Szudarek et al. (2022), dynamically reduced lift coefficient by up to 0.4 with little drag penalty, though effectiveness depended strongly on maintaining low ground clearance. Complementary work by Wright (2023) confirmed that splitter height is critical: moderate clearance improved efficiency by reducing lift, but excessive lowering disrupted flow and stability, while none of the configurations generated true downforce at 150 km/h. Collectively, these studies emphasise that while splitters can

meaningfully enhance front-end grip, efficiency depends on careful optimisation of length and ride height, with canards or active designs offering alternatives when drag penalties or regulatory constraints limit splitter use.

Race Louvers (2023) conducted a comparative CFD and wind tunnel study of three fender louvre configurations as seen in Figure 4 below.

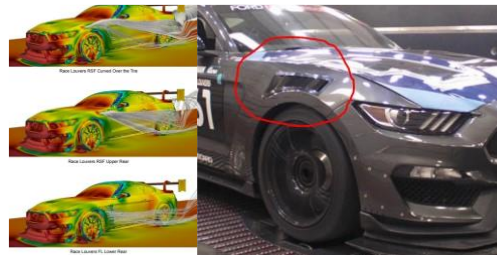


Figure 3 Race Louvers on Front Wheel Arch- Mustang

Showing that all vent placements improved front downforce with negligible drag penalty by releasing high-pressure air from the wheel wells. The optimal performance depended on liner treatment, a notched liner maximised downforce, whereas a fully removed liner increased cooling but slightly reduced aerodynamic benefit. Results also indicated diminishing returns beyond two vents, with medium-sized vents offering the best balance between packaging and function. Similarly, Kumar et al. (2013) investigated bump-shaped vortex generators (Figure 5) placed near roof ends and demonstrated that devices sized to local boundary-layer thickness (15–25 mm) delayed separation, reduced drag coefficient by up to 0.009, and lowered lift coefficient by 0.046.



Figure 4 vortex generator example

These findings confirm that passive devices, when positioned effectively, can narrow wake structures and enhance downforce without major drag penalties, making them relevant for managing flow separation around arches and roof edges.

2.1 Splitter Ride-Height Effects

Soares examined enclosed-wheel racing cars and found that downforce is more sensitive to ride-height than splitter length; a very low splitter may slightly increase downforce but significantly disrupt underbody airflow, leading to diminishing returns (Soares, 2018). Szudarek introduced the concept of an inflatable splitter and demonstrated that very low ground clearance significantly boosts front-axle downforce without increasing drag, but only when dynamic ride-height variations are managed carefully (Szudarek, 2022). Another numerical study using an Ahmed-body ground-effect diffuser model found that optimal downforce efficiency varies with ride-height, with 10 mm clearance combined with a 10° ramp angle achieving a performance peak. shallower or lower setups led to earlier flow separation and reduced suction (Um et al., 2021)

2.2 Aerodynamic Diffuser Design

2.2.1 Introduction

The rear diffuser is widely recognised as a central element of ground-effect aerodynamics, responsible for accelerating underbody flow and recovering pressure to generate downforce (Marklund, 2013; Gopalarathnam et al., 2019). In motorsport, its performance is often enhanced by vertical fences, or strakes, which divide the diffuser into multiple channels. These structures help stabilise flow, delay separation at higher ramp angles, and improve consistency under yaw and ride-height variation. Although the governing principles are well established, there is still no consensus on the most effective geometric arrangement of diffusers and strakes, with studies reporting different limits and performance trends depending on configuration and operating conditions.

2.2.2 Importance of Diffuser and Splitter Design

CFD studies on simplified geometries, such as the Ahmed body, consistently demonstrate that the diffuser is the primary driver of underbody pressure recovery, with optimised ramp angles of 20–30° producing the greatest downforce before separation occurs (Buscariolo et al., 2020). In contrast, experimental work on full-scale vehicles has reported more restrictive limits, with Katz (1995) noting that ramp angles above 10–12° often induce premature flow separation and drag rise. The influence of upstream devices further complicates diffuser performance. Cooper et al. (1998) showed that poorly integrated splitters disrupt momentum recovery by thickening the boundary layer, while Willemsen (2012) demonstrated that a diffuser, once correctly optimised

for slope, ride height and length, can deliver approximately 97 N/m of vertical load with little drag penalty. This supports the principle that splitters should be tuned only after the diffuser is established, forming the aerodynamic baseline for balance. More recent work reinforces the need for integration: Zhang et al. (2022) stressed that coordinated design of front and rear underbody elements is essential to avoid counterproductive interactions. Beyond geometry, diffuser efficiency is governed by pressure recovery metrics such as mean effective pressure coefficient, which are highly sensitive to ride height, expansion ratio and inlet flow quality. Studies suggest that these parameters often outweigh ramp angle in determining whether flow remains attached or separates, with factors such as openness and upstream conditioning proving decisive (Marklund, 2013; Gopalarathnam et al., 2019; Abood et al., 2024).

2.2.3 Ramp Angle and Length

Although HVAC and aerospace practice advises diffuser expansion angles below 7–10° to avoid separation (Kays and London, 1998), automotive ground-effect conditions permit considerably higher limits because of the moving ground plane and greater underfloor momentum. CFD and wind tunnel studies indicate that flows energised by strakes can remain attached up to 18–25° (Guerrero et al., 2022; Unni et al., 2017), although these values are highly sensitive to ride height and upstream flow quality. When critical thresholds are exceeded, stall can occur, with separation bubbles extending forward into the flat floor region (Marklund, 2013). Ramp angle alone, however, is not the most reliable performance metric. The expansion ratio—the outlet-to-inlet area ratio—has been shown to better predict pressure recovery, with moderate values between 1.2 and 1.5 generally offering the most robust behaviour across both open- and closed-sided configurations (Marklund, 2013; Gopalarathnam et al., 2019). Ride height adds further complexity: Rijns et al. (2023) demonstrated that multi-channel diffusers deliver peak downforce at intermediate clearances, whereas excessive lowering chokes the underfloor and reduces performance. Together, these findings indicate that diffuser effectiveness depends on the interplay between ramp angle, expansion ratio and ride height, requiring coordinated design with suspension kinematics to achieve stability across operating conditions.

2.2.4 Role of Strakes in Multi-Channel Diffusers

Strakes are commonly used to partition diffuser flow into discrete channels, reducing lateral contamination from wheel wakes and encouraging longitudinal vortices that re-energise the boundary layer, thereby delaying separation and permitting higher ramp angles (Unni et al., 2017; Yang et al., 2020; Nebot et al., 2025). Their effectiveness, however, is highly dependent on placement and geometry. Aligning strakes with high-momentum inflow regions and extending

them to the diffuser exit maximises pressure recovery (Unni et al., 2017), while heights close to or slightly above the local boundary-layer thickness balance separation control against added skin-friction drag (Santner et al., 2012). Small toe-in angles, typically 0–5°, can also direct vortex trajectories to promote reattachment (Nebot et al., 2025). Despite these benefits, strakes introduce trade-offs: they increase wetted area and friction, and excessive use narrows flow passages, raising local expansion ratios and risking diffuser stall (Gopalarathnam et al., 2019). Consequently, their design must balance stability gains under yaw and ride-height variation with efficiency losses due to drag and flow choking.

2.2.5 Integration with Vehicle Aerodynamics

Diffuser performance is highly dependent on its interaction with other aerodynamic components. Upstream, devices such as splitters and flat floors condition the boundary layer and redirect high-energy flow toward the diffuser inlet, while poor integration can thicken the boundary layer and reduce pressure recovery efficiency (Willemsen, 2012; Cooper et al., 1998). Downstream, parameters such as outlet height and the inclusion of kicker profiles have been shown to influence base pressure and wake development, further shaping overall performance (Marklund, 2013; Kang et al., 2012). Recent studies emphasise that optimal underbody behaviour cannot be achieved in isolation; Zhang et al. (2022) highlight the need for coordinated design between front and rear elements to prevent internal flow interference and achieve a balanced aerodynamic package.

2.2.6 Critical Evaluation

The literature consistently supports a diffuser-first design strategy, where the diffuser establishes the baseline flow field before splitter geometry is introduced. Nevertheless, existing research is constrained by steady-state assumptions and simplified geometries. Studies such as Cooper et al. (1998) and Willemsen (2012) demonstrate valuable principles but do not account for transient effects, yawed inflow, or dynamic ride-height variation. These limitations highlight the need for full-vehicle CFD and wind-tunnel testing under representative operating conditions to capture the complex interactions between splitters and diffusers.

2.2.7 Conclusion

The diffuser remains the principal determinant of underbody aerodynamic performance, with splitters serving primarily as flow-conditioning devices. Both experimental and computational studies reinforce the effectiveness of a diffuser-first development strategy, ensuring coherent underbody flow before front-end devices are tuned. However, a clear gap persists in the literature

regarding transient operation, yaw effects, and dynamic ride-height variation. Addressing these factors through full-vehicle CFD and wind-tunnel testing would provide a more realistic basis for optimising splitter–diffuser interaction under race conditions.

2.3 Validation Research

2.3.1 Model Research

The MIRA reference car was developed as a generic notchback geometry to provide a standardised platform for aerodynamic validation in wind tunnels and simulations (Howell, 2004). Its simple form enables short computational times and straightforward meshing, which explains its continued use for benchmarking. However, the model does not reproduce the complex flow behaviour of production vehicles, limiting its applicability to studies requiring detailed wake and underbody fidelity.

In contrast, the DrivAer model, introduced by the Technical University of Munich, combines realistic vehicle geometries with modular configurations for rear-end, wheel, and underbody layouts (Heft, Indinger and Adams, 2012). This flexibility allows it to replicate separation, wheel wake interactions, and other flow phenomena absent in the MIRA design. For this reason, the DrivAer is employed in the present study to validate the CFD methodology before application to the Morgan Plus 4 (Figure 6).

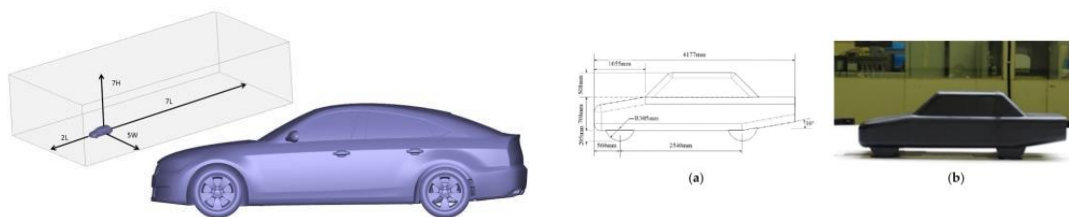


Figure 5 DrivAer car (left), MIRA car (right)

2.3.2 CFD Methodology Techniques

2.3.2.1 Boundary layers

For turbulence models that employ wall functions, such as the standard and realizable $k-\epsilon$, the wall-adjacent cell centre should fall within the log-law region of the boundary layer, corresponding to a y^+ between 30 and 300 (ANSYS Fluent Theory Guide, 2024). The SST $k-\omega$ model resolves the viscous sublayer directly, targeting a y^+ of approximately 1. This eliminates

dependence on wall functions and improves accuracy in flows with adverse pressure gradients and separation (Menter, 1994).

2.3.2.2 Meshing Methodologies

Mesh topology strongly influences accuracy and computational cost. Comparative studies of the DrivAer model indicate that polyhedral or hybrid prism–polyhedral meshes outperform tetrahedral and hexa-dominant approaches, delivering higher orthogonal quality, reduced skewness, and reliable results with fewer cells (Fotiadis et al., 2024; Sosnowski et al., 2018). Adaptive mesh refinement (AMR) provides further efficiency by concentrating cells in high-gradient regions such as separation lines, diffusers, and wheel wakes. Irigaray (2024) showed that AMR improved drag and pressure predictions while reducing total mesh size by avoiding unnecessary refinement in low-gradient areas. Similar results were reported by Cadence (2023), who demonstrated that targeting drag-sensitive regions such as the rear wake enhanced correlation with experimental data while maintaining manageable cell counts.

2.3.3 Turbulence Models

RANS models remain the most widely applied in industrial CFD because they offer a balance between accuracy and computational demand. By solving time-averaged equations, they are less intensive than LES or DNS, though their predictive capability varies with flow complexity.

The standard $k-\varepsilon$ model (Launder and Spalding, 1974) is widely validated for external aerodynamics and computationally efficient, but its assumption of isotropic turbulence leads to inaccuracies in separated flows and wakes. The realizable $k-\varepsilon$ improves upon this by introducing a variable C_μ and alternative dissipation-rate equation, giving better predictions in swirling and separated flows, although it remains limited in transition and detailed wake resolution (Shih and Liou, 1995). The SST $k-\omega$ model blends $k-\omega$ near walls with $k-\varepsilon$ in the free stream, resolving the viscous sublayer directly at $y^+ \approx 1$. It is effective in predicting separation and reattachment, making it popular for Ahmed body and DrivAer studies, but requires fine near-wall meshing and is sensitive to free-stream turbulence (Menter, 1994).

2.3.4 Moving Ground and Rotating Wheels

Accurate simulation of ground motion and wheel rotation is essential for external vehicle aerodynamics. Rejniak and Gatto (2020) demonstrated that including a moving ground substantially alters wake development, influencing drag and lift predictions. Wheel rotation introduces further complexity; Yamashita et al. (2018) reported that it can increase or decrease drag depending on how tyre rotation interacts with wheelhouses and the surrounding bodywork.

These effects are particularly relevant for geometries with open wheelhouses, where changes in wheel wake behaviour directly affect underbody pressure distribution and overall aerodynamic balance.

2.4 Mesh Quality

Mesh quality has a direct influence on numerical stability, convergence behaviour, and solution accuracy. According to ANSYS (2012), skewness should remain below 0.95 and orthogonal quality above 0.1 to ensure stable solutions (Figure 6). Aspect ratios are ideally below 35 for volume elements, although higher values can be tolerated within structured prism layers where curvature effects are minimal (Saeed et al., 2019).

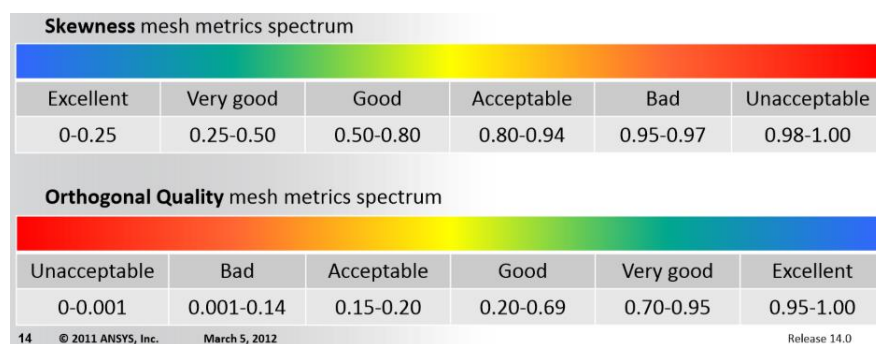


Figure 6 Mesh Quality (ANSYS Inc., 2012).

Element growth and transition also play a key role. Wang (2019) recommends a mesh growth rate of 1.0–1.2 to maintain smooth size variation, while Ferziger and Perić (2002) highlight that transition ratios of 0.2–0.3 between adjacent cells minimise abrupt size changes and reduce instability. During preliminary testing, a growth rate of 1.0 lowered orthogonal quality by approximately 6%, so 1.2 was adopted in this study.

In terms of topology, polyhedral meshes have consistently demonstrated advantages over tetrahedral grids. They retain higher orthogonal quality, are less sensitive to stretching, and require fewer cells to achieve comparable accuracy (Łomakin, Kudo and Oleinikova, 2018; Sosnowski et al., 2018). Further evidence shows that polyhedral meshes can match the accuracy of structured hexahedral meshes while reducing generation time (Lintermann et al., 2021).

These guidelines are applied here to ensure that both the DrivAer and Morgan Plus 4 meshes conform to recognised industry and academic standards for external aerodynamics CFD.

3.0 Validation study

The DrivAer model (Figure 7), created by the Technical University of Munich, provides a more realistic representation of passenger vehicle aerodynamics than the simplified MIRA Notchback. Its modular design incorporates detailed body contours, interchangeable rear-end configurations, and underbody layouts that replicate separation and wake structures typical of production vehicles (Heft, Indinger and Adams, 2012). The model is widely used in validation studies because of its extensive wind tunnel datasets and adaptability, making it a well-documented benchmark for CFD comparison. As such, it serves as an effective intermediate step between the idealised MIRA geometry and the complex, heritage-inspired form of the Morgan Plus 4.

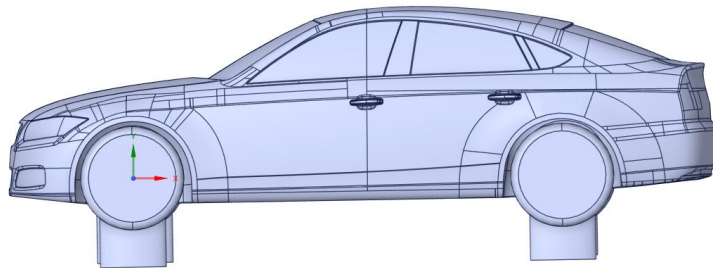


Figure 7 DrivAer Fastback with flat underbody CAD model

3.1 Geometry and Experimental Reference Data

The DrivAer fastback variant was selected for validation due to its strong representation in the literature and availability of both CFD and experimental datasets. Renda (2022) reports baseline drag and lift coefficients for the fastback with a flat underbody of $C_d = 0.261$ and $C_l = 0.083$, measured in a wind tunnel with an 11 m^2 open-jet test section equipped with boundary layer suction and tangential blowing systems to ensure uniform flow conditions. This configuration minimises interference effects and produces highly repeatable aerodynamic data, providing a robust benchmark for CFD comparison.

Qin and Ricci (2024) found close agreement between CFD and experimental values for the same configuration, reporting $C_d = 0.258\text{--}0.261$ and $C_l = 0.139$ for a 1:4 scale model. Strangfeld et al. (2013) obtained $C_d = 0.249$ and $C_l = 0.057$, while Renda (2022) achieved $C_d = 0.265$ and $C_l = 0.059$. Averaging these studies gives $C_d = 0.2583$ and $C_l = 0.0663$, which form the target values for this validation work, with a tolerance of $\pm 5\%$ for C_d and ± 0.01 for C_l .

3.1.1 CAD Model Preparation and Rebuilding

The imported DrivAer CAD model contained several geometric defects that would have undermined mesh quality and convergence. These included non-manifold edges within the wheel arches, gaps between the A-pillar and windscreen, intersecting surfaces at the wheel-floor interface, and untrimmed internal faces remaining from the modular assembly. Initial test meshes based on this geometry exhibited skewed elements, poor orthogonal quality, and prism-layer collapse.

To address these issues, the model was repaired in ANSYS SpaceClaim. Tools such as *Show Gaps* and *Stitch* were used to close discontinuities, *Imprint Faces* ensured clean boundaries for mesh control, and *Delete/Fill* removed redundant internal surfaces. *Merge Faces* was applied to simplify fragmented regions, improving uniformity and avoiding unnecessary refinement. In cases where features were too complex to mesh robustly, simplification was carried out while preserving the correct aerodynamic form.

This rebuilding process eliminated meshing errors, reduced total element count, and improved solver stability. The same workflow was subsequently applied to the Morgan Plus 4 model to ensure comparable mesh quality and consistent simulation reliability across both geometries.

3.1.2 Fluid Domain Geometry

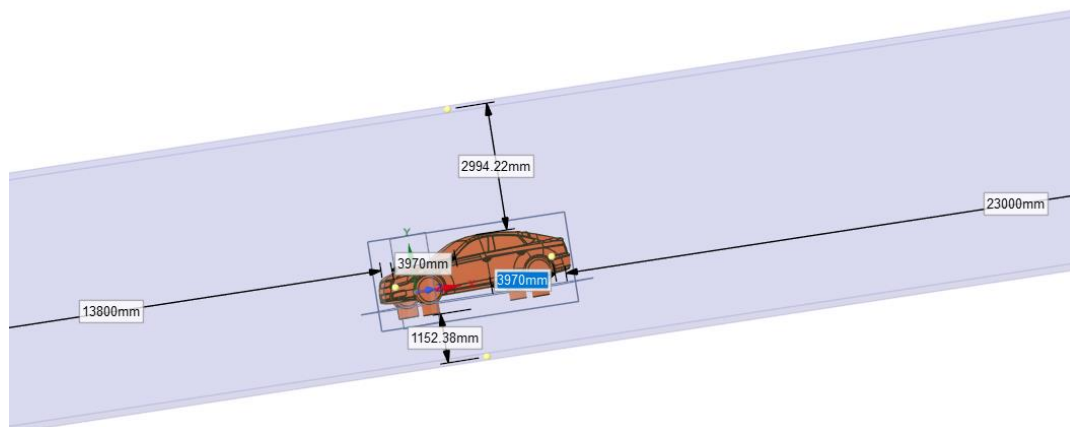


Figure 8 Domain Size For DrivAer

The fluid domain was constructed in accordance with best-practice guidelines from Lanfrit (2005), extending three vehicle lengths upstream and five downstream to capture inflow development and the complete wake structure. For the DrivAer fastback, with an overall length

of 4.613 m, this resulted in an inlet clearance of 13.8 m and an outlet length of 23 m (Figure 9). Vertical and lateral boundaries were based on the MIRA wind tunnel reference dimensions of 4.42 m in height and 7.94 m in width (MIRA, 2007), ensuring minimal blockage effects and close alignment with experimental conditions.

3.2 Mesh Strategy and Setup Methodology

The meshing approach was guided directly by the findings of the literature review, balancing resolution requirements with computational efficiency. A body surface size of 0.02 m and a finer wheel surface size of 0.01 m were applied, while local wheel blocks reduced poor-quality elements at the wheel–floor interface by approximately 60%.

Element growth and transition were controlled to ensure smooth variation. A growth rate of 1.2 was selected, as testing showed that a rate of 1.0 reduced mesh quality by around 6%, consistent with Wang’s (2019) recommendations. Ferziger and Perić (2002) advise a transition ratio of 0.2–0.3 between adjacent elements, which was also adopted to avoid abrupt size jumps and maintain numerical stability.

Mesh topology was chosen based on published validation work with the DrivAer model. Polyhedral or hybrid prism–polyhedral meshes have been shown to improve orthogonal quality and reduce skewness while requiring fewer cells to achieve equivalent accuracy compared to tetrahedral grids (Fotiadis et al., 2024; Sosnowski et al., 2018). Similar benefits are reported by Łomakin, Kudo and Oleinikova (2018), who found polyhedral meshes less sensitive to stretching, producing higher orthogonal quality and lower aspect ratios than tetrahedral meshes. These findings were confirmed in this study, where polyhedral meshes achieved superior alignment and lower distortion, reducing the risk of convergence issues.

Adaptive Mesh Refinement (AMR) was used to concentrate cells in regions with strong gradients, such as the wake, diffuser, and wheelhouses. Irigaray (2024) demonstrated that AMR improves drag and pressure prediction without inflating global cell counts, while Cadence (2023) showed that targeting drag-sensitive areas improves correlation with experiments while maintaining efficiency. These results supported the use of local refinement strategies in the present work.

Boundary layer resolution was achieved with five prism layers, giving $y^+ \approx 100$ for k – ε runs using a first layer height of 0.007 m. This configuration aligns with Lanfrit’s (2005) recommendations for external aerodynamics and proved sufficient in independence testing.

3.2 Mesh Strategy and Setup Methodology

The meshing approach applied here directly follows the literature review findings on mesh growth rates, topology, and quality metrics (2.0 Literature Review). A body surface size of 0.02 m and wheel surface size of 0.01 m balanced computational efficiency with near-wheel resolution. Wheel blocks were implemented to reduce low-quality elements at the wheel–floor interface by 60%.

Yue Wang (2019) recommends a growth rate between 1.0–1.2; a value of 1.2 was used after tests showed a 1.0 growth rate reduced mesh quality by 6%. Ferziger and Perić (2002) advise a transition ratio of 0.2–0.3 between adjacent elements to avoid abrupt size jumps, also consistent with the guidance in Section 2.3.3.

Comparative studies on the DrivAer model show that mesh topology significantly affects accuracy and cost. Fotiadis et al. (2024) found that polyhedral or hybrid prism–polyhedral meshes improve orthogonal quality, reduce skewness, and achieve accuracy at lower cell counts than tetrahedral grids—matching the polyhedral mesh benefits discussed in the literature review. Sosnowski et al. (2018) confirm that polyhedral meshes require fewer elements to match tetrahedral accuracy, reducing computational cost.

Adaptive mesh refinement (AMR) has also been applied successfully to DrivAer simulations. Irigaray (2024) demonstrated that focusing refinement in the wake, diffuser, and wheel wakes improves drag and pressure prediction while reducing total mesh size. Similarly, Cadence (2023) showed that targeted adaptation in drag-sensitive regions, particularly the rear wake and diffuser, improved drag correlation with experimental data while keeping total cell counts manageable.

Prism layers were included to capture boundary layer development, with $y^+ \approx 100$ for k – ε runs achieved using a first layer height of 0.007 m. Five prism layers were used, consistent with both independence tests and Lanfrit’s (2005) recommendations for external aerodynamics and the boundary layer setup described in the literature review.

3.2.1 Mesh Quality Assessment

Mesh quality was evaluated against recognised CFD stability criteria. The final mesh achieved an orthogonal quality of 0.306, comfortably above the 0.1 minimum stability threshold (ANSYS, 2012). Maximum skewness remained within the “good” range at 0.69, below the 0.95 limit. The recorded maximum aspect ratio of 52.97 exceeded the general guideline of 35 (Saeed et al., 2019), though this was confined to structured boundary-layer regions where flow curvature is low and higher values are acceptable.

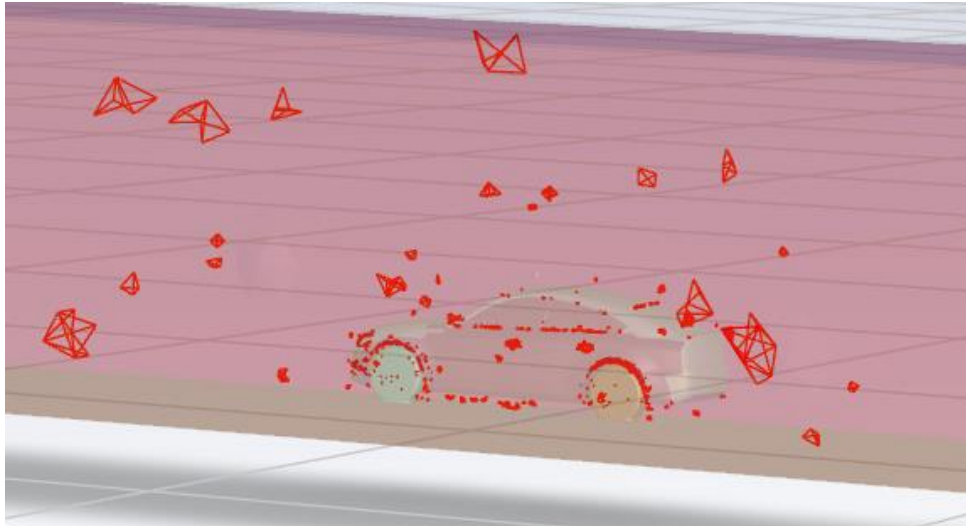


Figure 9 Orthogonal Quality 0.306

The use of polyhedral meshes was further justified by comparative studies. Łomakin, Kudo and Oleinikova (2018) highlighted that polyhedral cells are less sensitive to distortion and stretching, achieving orthogonal quality of 0.436 and an aspect ratio of 45.22, compared to higher distortion in tetrahedral meshes. Sosnowski and Krzywański (2018) also confirmed that tetrahedral grids require significantly more elements to match polyhedral accuracy, often causing convergence issues.

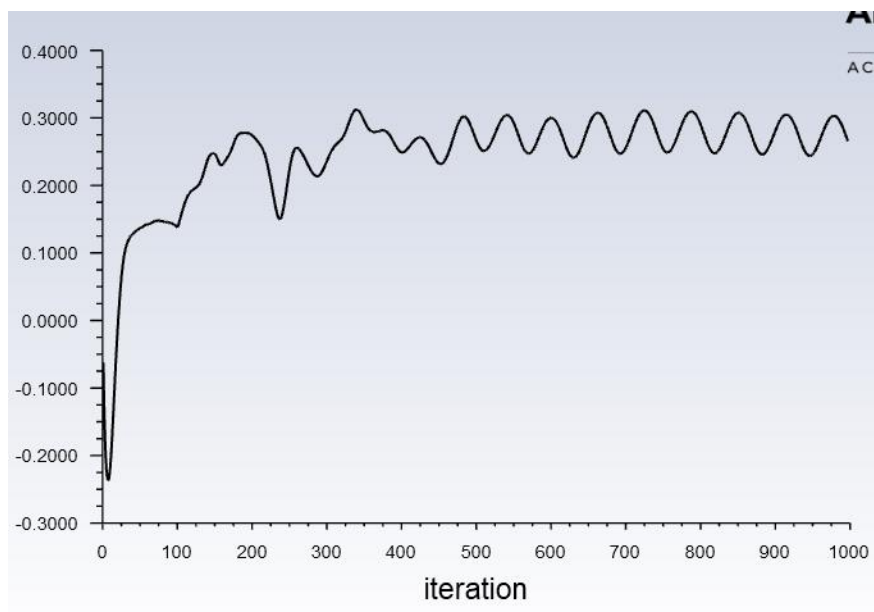


Figure 10 Cd convergence issue due to low mesh quality

Overall, the adopted strategy produced a mesh of sufficient robustness to ensure stability, accuracy, and efficient convergence, consistent with best practice for external aerodynamics CFD.

3.3 Setup

Consistent with best practices in external vehicle aerodynamics CFD, the solution began with a first-order scheme to ensure convergence stability, then transitioned to a second-order method for increased accuracy, particularly in resolving separation and wake flow features (Khalighi 2012; Strangfeld, 2016).

3.3.1 Turbulence Model Selection

The $k-\omega$ SST model was initially applied for its proven accuracy in capturing separation and diffuser performance (Menter, 1994). However, for primary validation runs, the realizable $k-\varepsilon$ model with non-equilibrium wall functions was selected, following findings that it provided stable convergence and accurate overall aerodynamic coefficients.

Setup can be seen in appendix.

4.0 Morgan Plus Four Study

The Morgan Plus Four exhibits compact proportions characteristic of a classic British roadster, with overall dimensions of approximately 3,830 mm in length, 1,650 mm in width, and 1,250 mm in height, as specified by the manufacturer. The wheelbase of 2,440 mm provides a balanced platform that influences both the vehicle's aesthetic proportions and its aerodynamic behaviour (Morgan Motor Company, 2024). These geometric parameters are essential inputs for defining the computational fluid dynamics (CFD) domain, particularly the width and height, which determine boundary-layer spacing and enclosure limits.

Experimental pressure data were obtained from physical testing using pressure taps placed on the upper body and wheel arches during on-road operation. These measurements provide a valuable benchmark but are not expected to align perfectly with CFD predictions. The main source of discrepancy arises from the soft-top roof, which undergoes local deformation under dynamic loading. This flexibility alters flow behaviour and can shift the location of separation points compared to the rigid, idealised CAD geometry used for simulation.

4.1 Geometry and Mesh Methodology

The fluid domain for the Morgan Plus Four was defined using the same principles established in the validation methodology. The enclosure dimensions were set at 11,336.73 mm upstream, 18,894 mm downstream, 3,169 mm above the roof, and 3,970 mm from each side of the vehicle. Wheel coordinates were carefully defined to ensure correct rotation centres, providing a realistic simulation of wheel motion.

Initial preparation of the CAD model focused on making the geometry CFD-ready to improve mesh quality. Repair tools in ANSYS SpaceClaim were used to close gaps, remove redundant features, and correct defects that would otherwise cause skewness and prism-layer collapse during meshing.

4.1.1 Body of Influence (BoI)

An initial test was conducted without Body of Influence (BoI) refinement to demonstrate its effect compared with simulations that include refinement zones. BoI enables local mesh refinement without altering the global mesh size, making it particularly effective for capturing boundary-layer development and wake structures (ANSYS Innovation Space, 2025). The IET (2020) also reported that refinement boxes improve resolution in high-gradient regions, ensuring that critical flow features are accurately resolved.

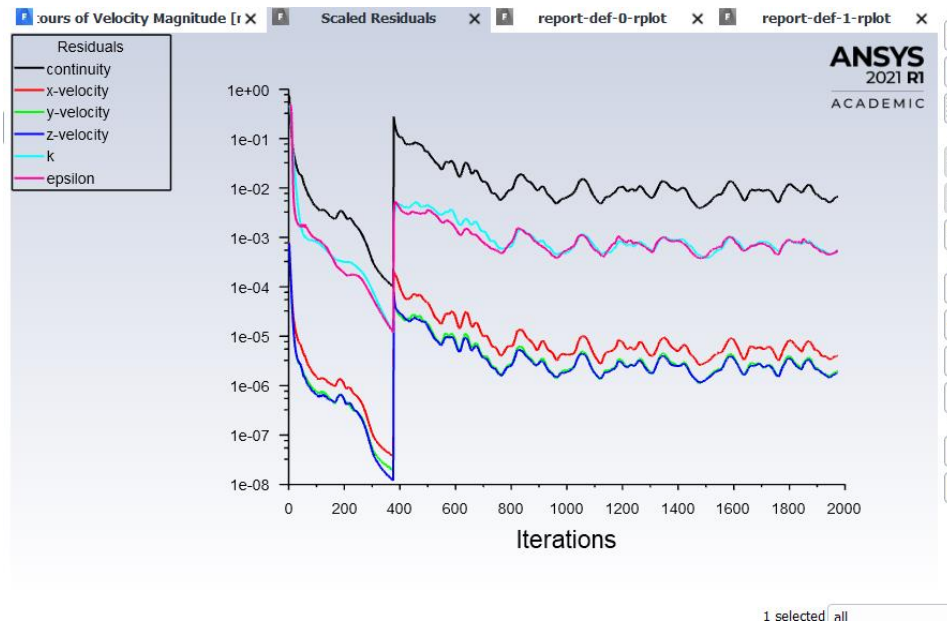


Figure 11 Scaled Residuals not converging

Figure 13 illustrates that omitting Bol refinement caused convergence problems, as the coarse mesh failed to resolve boundary layers, separation regions, and wake structures adequately. The solver was unable to consistently satisfy the governing equations, with turbulence (k , ϵ) and continuity residuals stagnating beyond 400 iterations even after switching to a second-order scheme. This highlights the importance of refinement in sensitive zones for stability, convergence, and predictive accuracy.

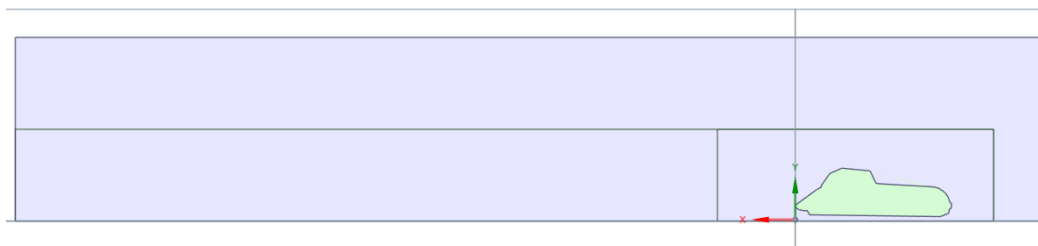


Figure 12 Two density boxes to capture flow separation and the wake region

Bol refinement was subsequently applied to the wake and separation regions using two density boxes with cell sizes of 0.1 m (near) and 0.3 m (far). The introduction of Bol refinement improved orthogonal quality by 77%, demonstrating a clear benefit for convergence behaviour.

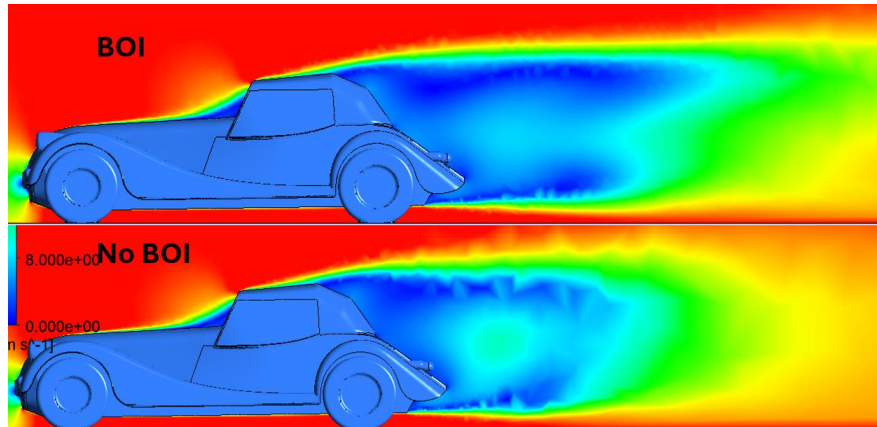


Figure 13 pressure plot. BOI vs no BOI

Pressure contours (Figure 15) show that the BoI-refined case generated a smooth and symmetric low-pressure wake, consistent with accurate drag prediction. By contrast, the non-refined case exhibited asymmetric pressure gradients and fragmented wake structures, which could lead to unstable solutions and unreliable coefficients.

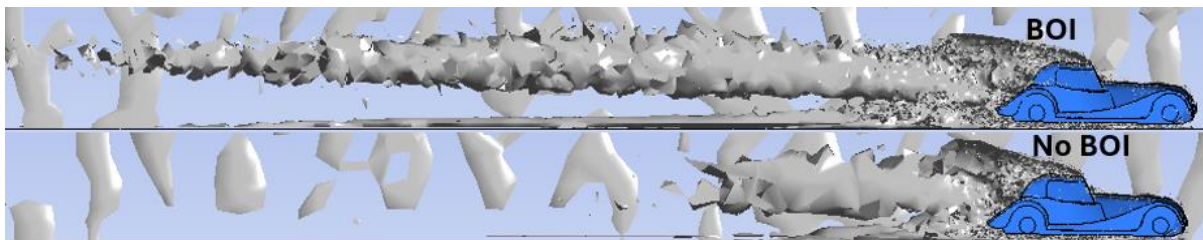


Figure 14 vortex core. BOI vs no BOI

The vortex core plots in Figure 16 further reinforce this finding. With BoI, long and coherent vortex structures were resolved, reflecting accurate turbulence modelling, whereas the non-refined mesh produced disjointed, poorly defined vortices. This confirms that BoI refinement is essential for robust wake capture and overall simulation accuracy.

4.1.2 Mesh Study

A mesh-independence study was carried out to confirm that aerodynamic coefficients stabilised within tolerance while avoiding unnecessary increases in computational cost. This approach followed best-practice recommendations from Ahmad and Abo-Serie (2019) and Gaylard (2018), with refinements tested in both the Body of Influence (BoI) regions and the part-mesh domain.

4.1.2.1 Bol Mesh Study

In the near-Bol region, drag coefficient (C_d) values stabilised for mesh sizes between 0.08 m and 0.15 m, remaining within the $\pm 0.05\%$ tolerance band. At coarser resolutions of 0.2 m, C_d diverged, indicating insufficient resolution of boundary layers and wake structures.

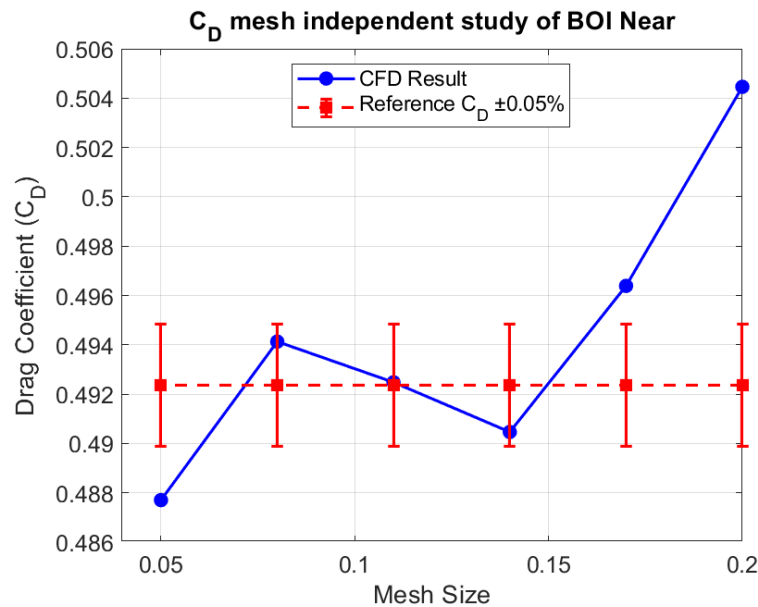


Figure 15 Mesh Independent Study in Near BOI

For the far-Bol region, a mesh size of 0.3 m provided the most effective balance between accuracy and computational cost. At this resolution, C_d stabilised within tolerance, with no measurable improvement gained from finer refinements. Larger cell sizes beyond 0.3 m caused C_d values to diverge, demonstrating inadequate wake capture. On this basis, 0.3 m was selected as the far-Bol mesh size for all full-vehicle simulations.

4.1.2.2 Part Mesh Study

The part-mesh independence study reinforced the conclusion from the Bol analysis. Both drag and lift coefficients stabilised at a mesh size of 0.3 m, with $C_d \approx 0.494$ and $C_l \approx 0.063$, both lying within the target tolerance range.

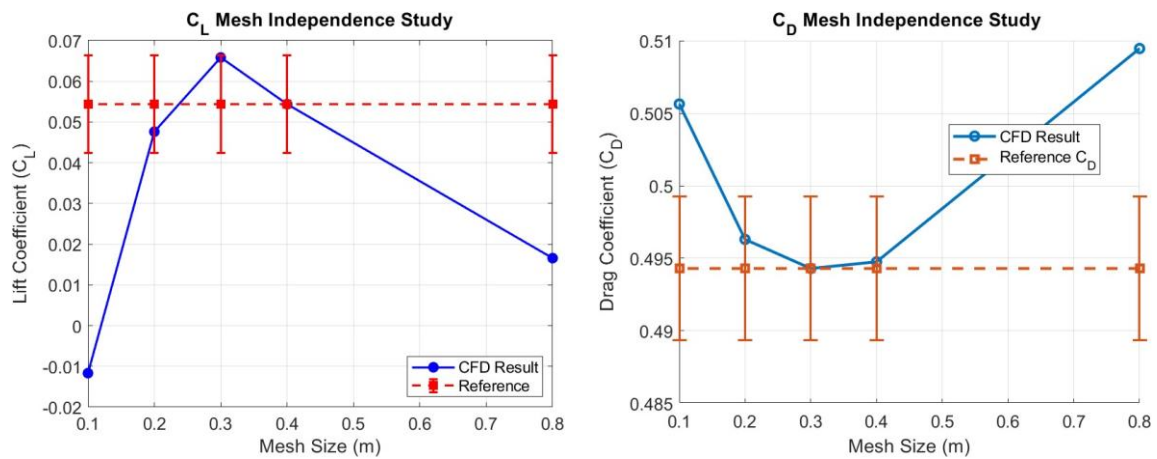


Figure 16 Mesh independence study for the part mesh, showing convergence of lift (C_L) and drag (C_D) at a mesh size of 0.3 m.

At coarser mesh resolutions above 0.5 m, the lift coefficient dropped noticeably, indicating reduced accuracy in resolving underbody flow and wake development. Drag divergence was also observed for mesh sizes between 0.7–0.8 m, confirming insufficient resolution. Finer meshes provided negligible improvements in coefficient prediction, further supporting the adoption of 0.3 m as the optimal resolution.

4.1.3 Prism Layers

Prism layers were applied to resolve near-wall gradients and capture boundary-layer development more accurately. A comparison was made between three and five prism layers using the realizable $k-\epsilon$ model, with results validated against experimental pressure-tap data.

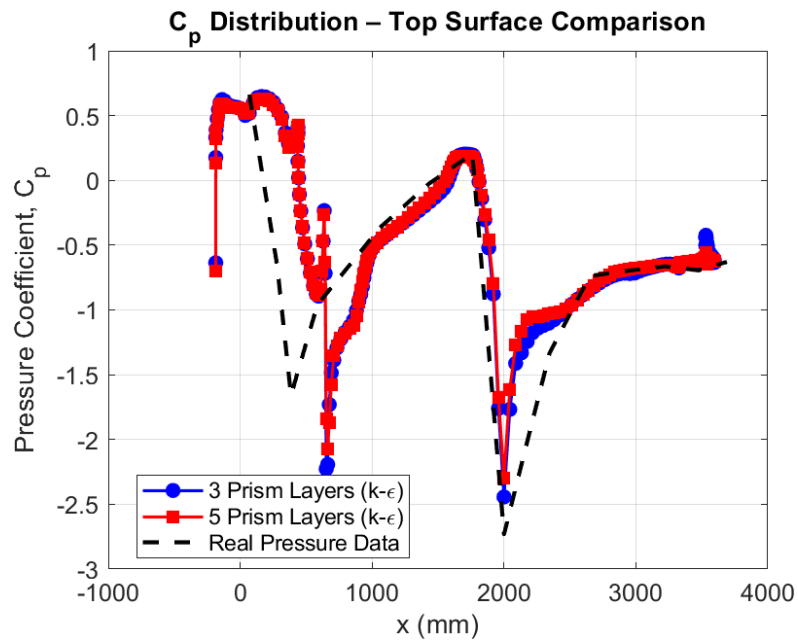


Figure 17 Comparison of pressure coefficient distribution on the Morgan Plus 4 top surface using 3 and 5 prism layers (k- ϵ) against experimental data.

Both configurations followed the experimental trend, but the five-layer mesh showed closer agreement in regions of sharp suction peaks and pressure recovery. This demonstrated that increasing prism resolution improves the capture of boundary-layer behaviour and enhances predictive accuracy.

The effect of local mesh refinement was also examined for the five-layer setup. Reducing the part-mesh size from 0.04 m to 0.02 m improved resolution of the wheel-arch suction peak compared with the coarser 0.03 m mesh.

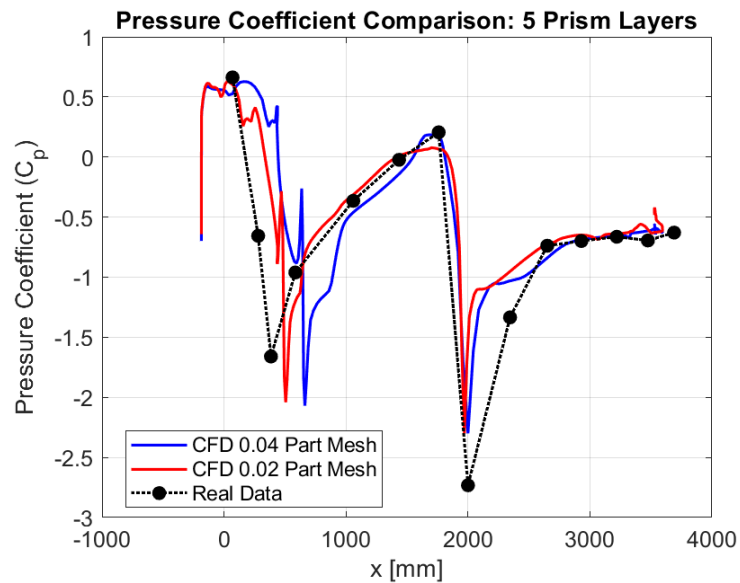


Figure 18 Effect of part mesh refinement on pressure coefficient predictions with 5 prism layers compared to experimental data.

This confirmed that additional refinement in critical regions can significantly improve the fidelity of C_p predictions, particularly around complex flow features such as the wheel arches.

4.1.4 Mesh Quality Improvements

Final mesh quality was assessed against recognised CFD stability criteria. The mesh achieved a maximum skewness of 0.69 and an orthogonal quality of 0.201, both of which fall within the “good” range according to ANSYS (2012). These results were obtained by applying targeted mesh-improvement tools after generation of the surface and volume meshes.

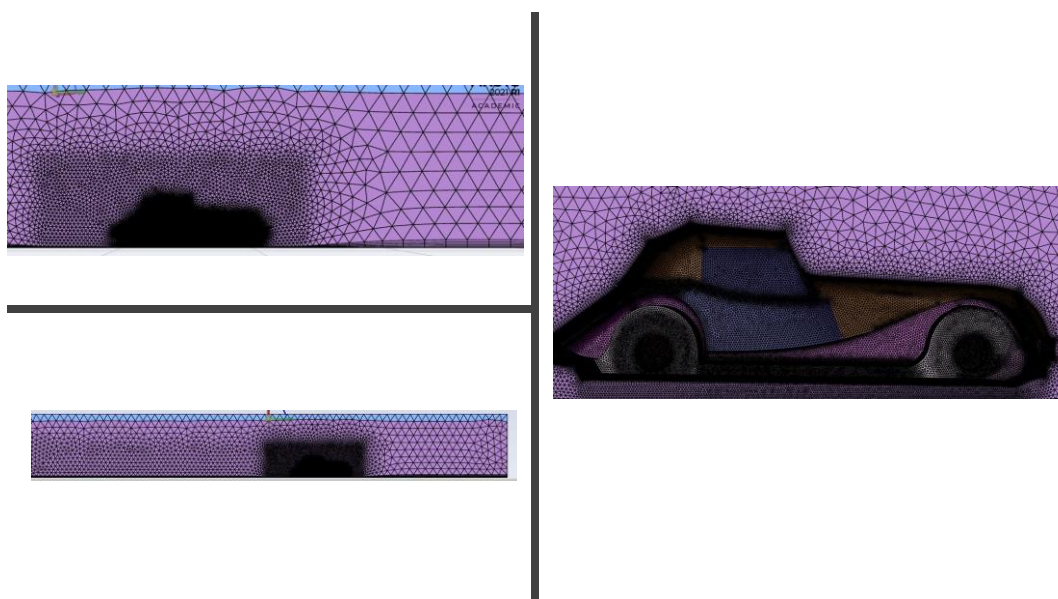


Figure 19 Final Mesh

The influence of these improvements was confirmed by comparing pressure distributions against experimental pressure-tap data. Using the refined mesh setup with five prism layers and an appropriate y^+ value, the realizable $k-\epsilon$ model achieved smoother predictions and closer alignment with experimental measurements.

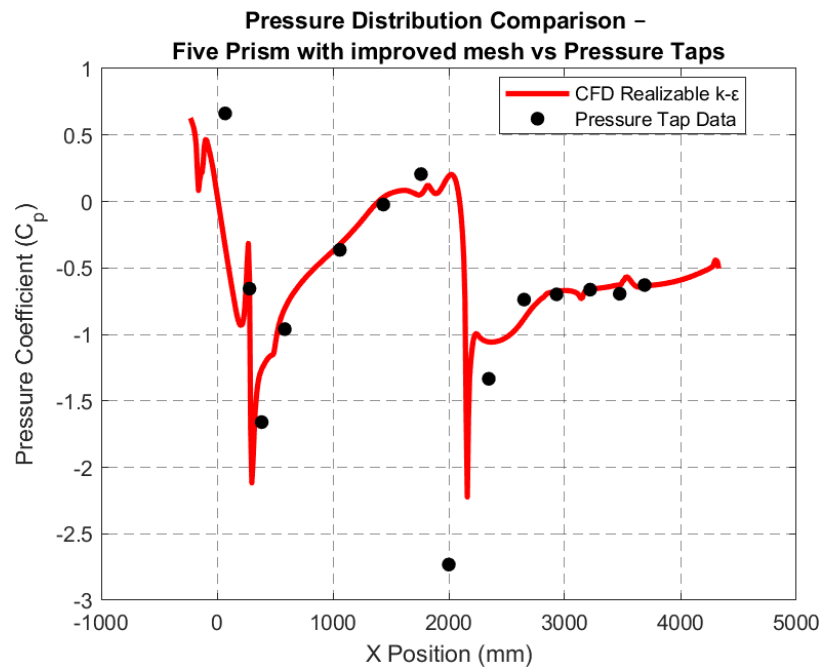


Figure 20 Pressure Distribution Comparison with improved mesh strategy and y plus

The enhanced near-wall resolution allowed the model to capture suction peaks and recovery regions more effectively, reducing discrepancies observed in earlier coarser meshes. This demonstrated that careful mesh refinement and boundary-layer treatment are essential for ensuring both numerical stability and physical accuracy in external aerodynamics simulations.

4.2 Solution Setup for Morgan Geometry

4.2.1 Moving Ground and Rotating Wheels

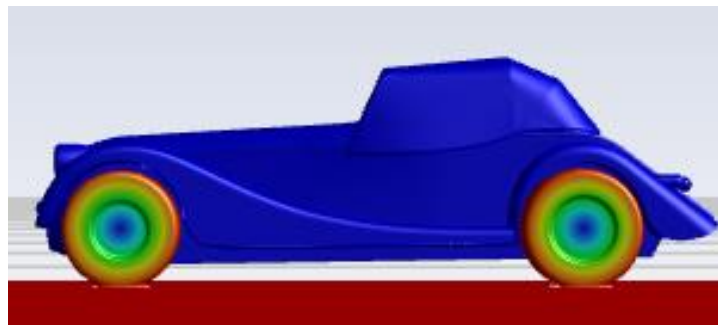
To replicate realistic on-road aerodynamics, the CFD setup included both a moving ground plane and rotating wheels. The ground speed was set equal to the free-stream velocity, with cases run at 20 m/s and 32 m/s. Wheel rotational speeds were calculated to match each velocity and are summarised in Table 1 and Table 2.

Table 1 Moving Wheel Conditions AT 20 m/s

Wheels	Speed (rads/s)	X	Y	Z	Rotation of axis
FW	62	-3.27346	0.28295	0.696	z
RW	62	-0.75326	0.28295	0.696	z

Table 2 Moving Wheel Conditions at 32m/s

Wheels	Speed (rads/s)	X	Y	Z	Rotation of axis
FW	95	-3.27346	0.28295	0.696	z
RW	95	-0.75326	0.28295	0.696	z

**Figure 21 Rotating Wheels**

The importance of these features is supported by prior research: Rejniak and Gatto (2020) demonstrated that a moving ground substantially alters wake development, while Yamashita et al. (2018) showed that wheel rotation can increase or decrease drag depending on its interaction with wheel arches and bodywork. Incorporating these effects ensured that the Morgan simulations reproduced the aerodynamic behaviour observed in the physical pressure-tap tests.

4.2.2 Turbulence Model Setup and Analysis

Turbulence models for the Morgan Plus Four were selected based on the literature review (see Section 2.3.3) and validation against the DrivAer model (see Section 3.0). Five models were tested:

- $k-\varepsilon$ RNG

- k - ε Realizable
- k - ω SST
- k - ω BSL
- Spalart-Allmaras

This range included industrially efficient k - ε formulations and low-Reynolds-number k - ω variants capable of resolving the viscous sublayer without wall functions.

4.2.2.1 k - ω SST and BSL

Menter (1994) introduced the SST k - ω model to improve predictions in flows with separation and strong pressure gradients. Unlike k - ε formulations, SST and BSL k - ω models resolve the viscous sublayer directly, requiring $y^+ \approx 1$ and fine prism layering. Studies of the DrivAer model recommend 15–20 prism layers at a growth rate of 1.2 for robust performance (Heft, Indinger and Adams, 2012; Strangfeld et al., 2013; Ahmed, 2019).

Figure 24 shows the pressure distribution predicted by the BSL k - ω model with five prism layers compared to experimental pressure-tap data. The model reproduced key suction peaks and pressure recovery trends, though some deviations occurred due to limited prism resolution and roof flexibility in physical testing.

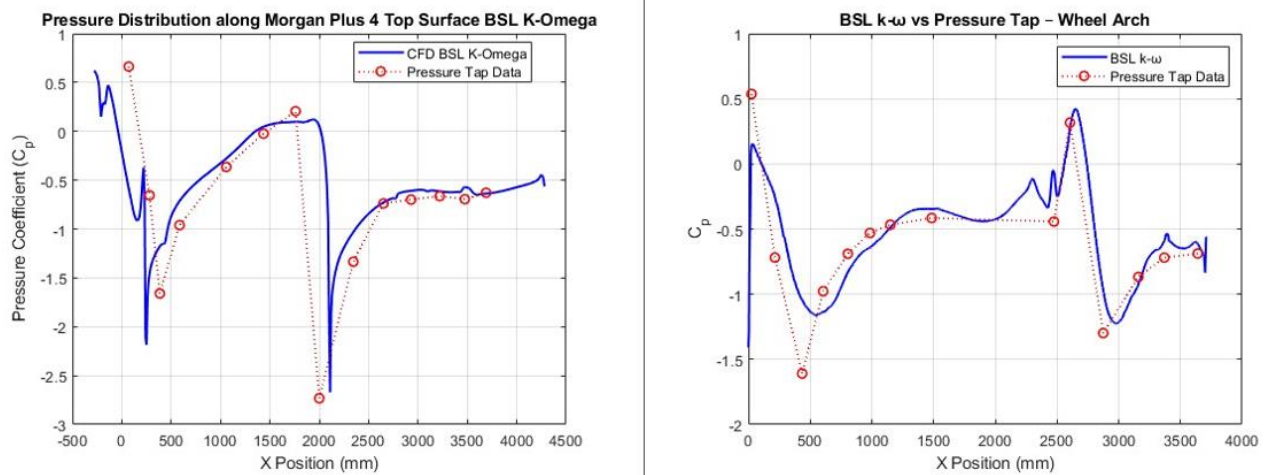


Figure 22 K-Omega Bsl, using 5 prism layers

Further comparisons are presented in Figure 25, where results at 20 m/s and 32 m/s are shown, and in Figure 26, which contrasts the five-prism-layer case with literature-based setups using up to 20 layers.

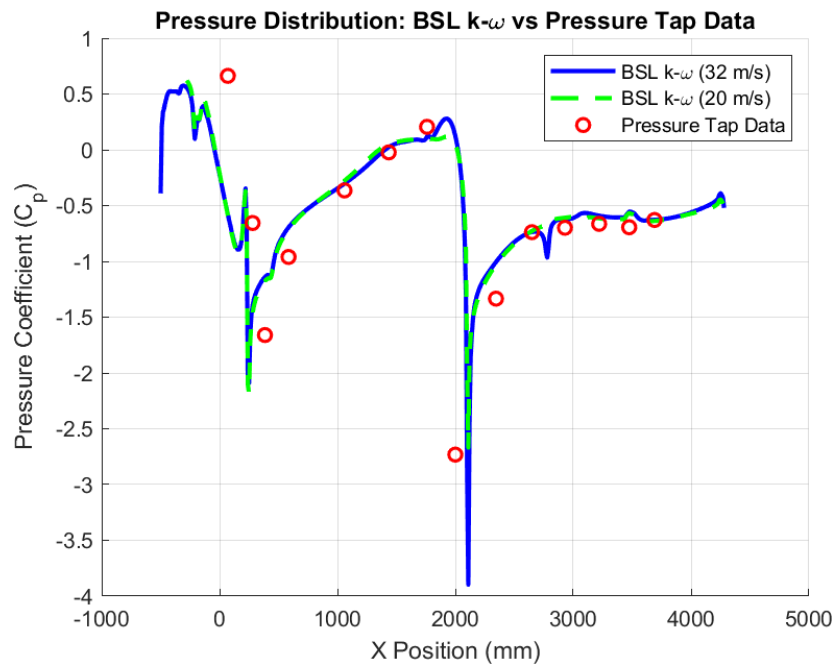


Figure 23 20m/s vs 32 m/s

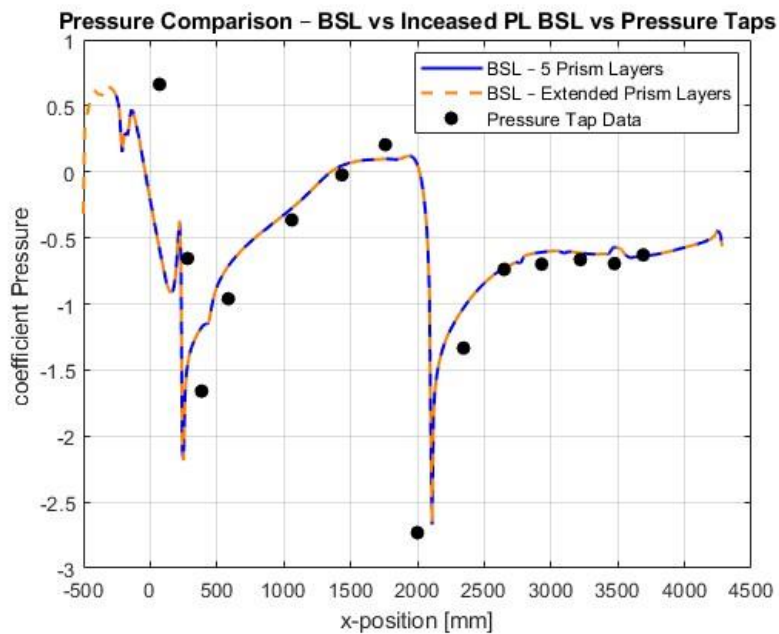


Figure 24 5 Prism vs Literature based research methodology

The SST $k-\omega$ model also demonstrated strong aerodynamic fidelity but struggled with convergence. Figure 27 shows residual stagnation above $1e+00$, while Figure 28 compares the SST $k-\omega$ predictions at 32 m/s with experimental pressure data, capturing the main trends but overpredicting suction in localised regions.

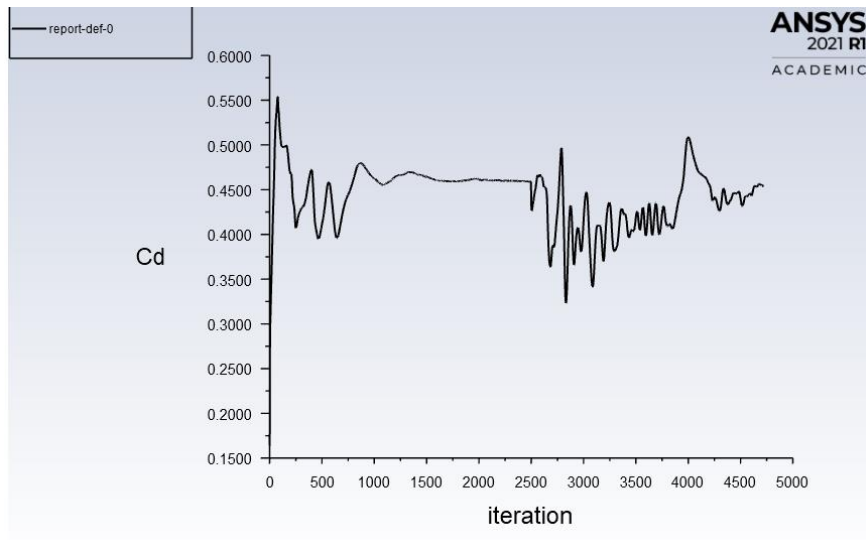


Figure 25 convergence issues with STT K-Omega

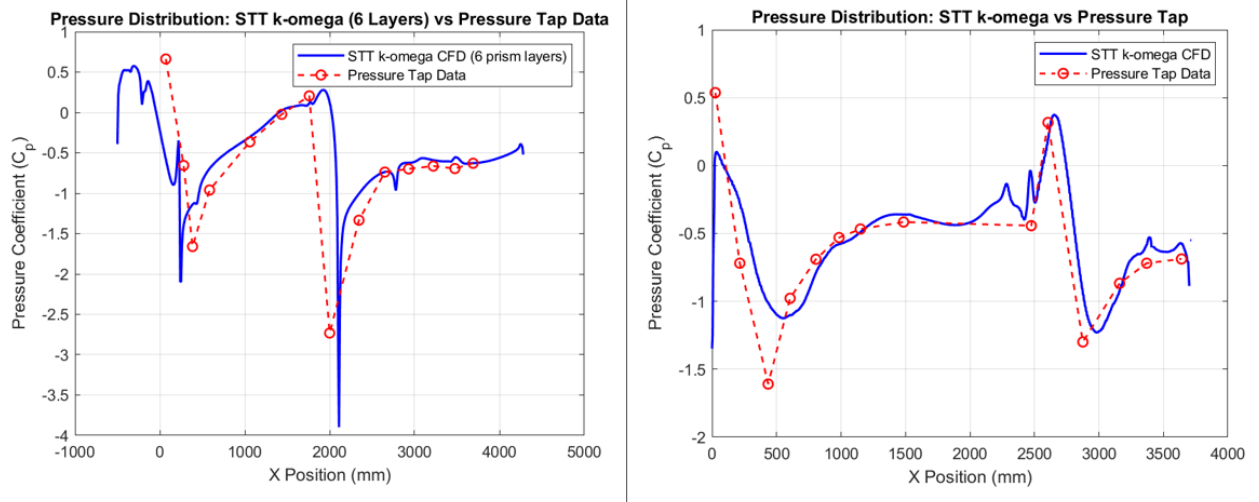


Figure 26 32m/s STT $k-\omega$ turbulence model with experimental pressure tap data for the Morgan Plus

4

4.2.2.2 Realizable $k-\epsilon$

The realizable $k-\epsilon$ model was also tested because of its strong convergence behaviour. Figure 29 shows the model's pressure predictions compared with experimental data. On the upper surface, the model reproduced nose suction and roof recovery trends, though it overpredicted suction magnitudes in certain regions. On the wheel arch, the main suction and recovery patterns were also captured, albeit with some phase shifts.

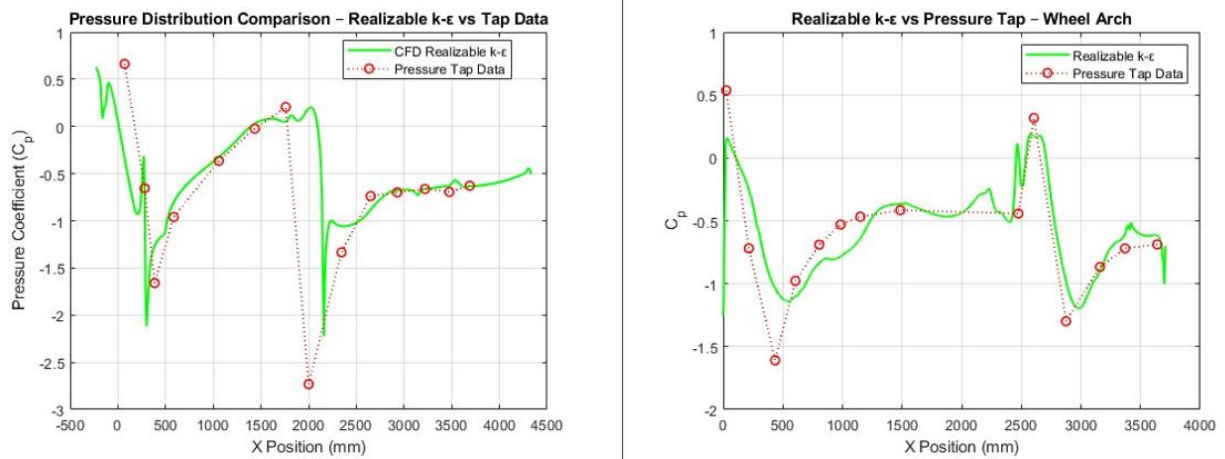


Figure 27 realizable $k\text{-}\epsilon$ predictions with the pressure tap data

The main limitation was sensitivity to near-wall resolution. The model requires wall functions with y^+ between 30–300, but this criterion was not consistently satisfied. Mesh refinement corrected this issue, as shown in Figure 30, where refined y^+ resolution improved suction-peak accuracy at the windscreen and smoothed pressure recovery.

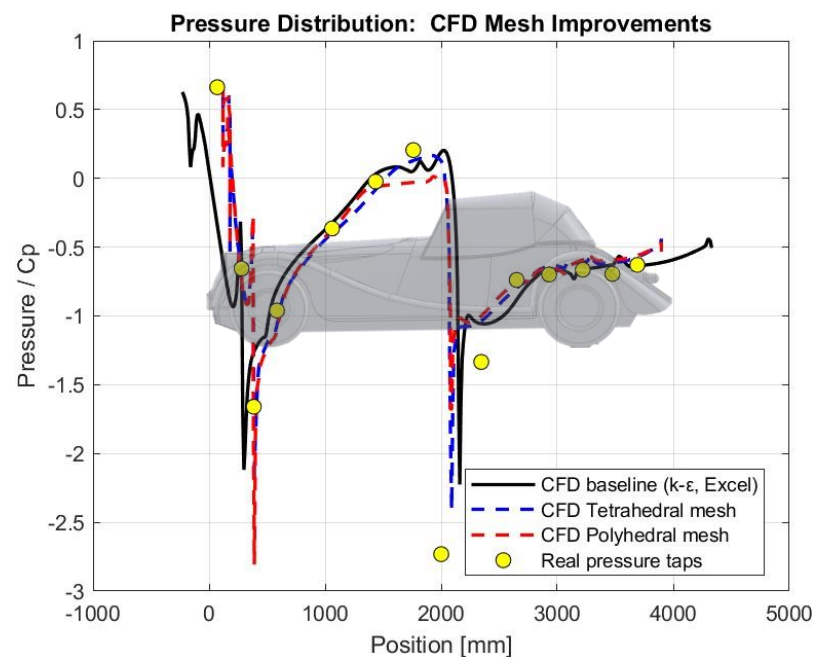


Figure 28 Pressure Distribution – CFD Mesh Improvements with Refined y^+ Resolution

4.2.2.3 Other Models

The Spalart–Allmaras and RNG $k\text{-}\epsilon$ models were also evaluated but quickly excluded. The Spalart–Allmaras model produced poor accuracy, consistent with Rumsey and Ying (2002), who

noted its limitations in separated automotive flows. The RNG $k\text{--}\epsilon$ failed to converge, matching reports of instability in external aerodynamics unless extremely fine near-wall resolution is applied (Wang et al., 2010). Both were therefore dismissed in favour of realizable $k\text{--}\epsilon$ and SST $k\text{--}\omega$, which performed more reliably (Heft et al., 2012; Strangfeld et al., 2013).

4.2.3 Summary of Turbulence Model Findings

Table 3 Comparison of Turbulence Models for Morgan Plus Four (Cd at 32 m/s)

Table X: Comparison of Turbulence Models for Morgan Plus Four (Cd at 32 m/s)				
Turbulence Model	Cd Predicted	Cd Experimental = 0.453	% Difference	Notes
k-ε RNG	0.5	0.453	10.40%	Stable; Cd slightly high
k-ε Realizable	0.472	0.453	4.20%	Good agreement; stable convergence
k-ε Standard	0.45	0.453	-0.7%	Closest match; Cd slightly underpredicted
k-ω SST	0.454	0.453	0.20%	Excellent Cd match; struggled with convergence
k-ω BSL	0.449	0.453	-0.9%	Underpredicted Cd; convergence moderate
Spalart-Allmaras	0.45	0.453	-0.7%	Acceptable Cd but poor flow accuracy

Overall, most turbulence models reproduced Cd within ±1% of the experimental benchmark, with the exception of the RNG $k\text{--}\epsilon$ model, which overpredicted by 10.4%. The SST $k\text{--}\omega$ model produced the closest Cd prediction (+0.2%) but suffered from convergence difficulties, limiting its efficiency for large-scale studies. The realizable $k\text{--}\epsilon$ model provided stable convergence and acceptable accuracy (+4.2%), making it the most practical choice for extended Morgan Plus Four simulations. Standard $k\text{--}\epsilon$, BSL $k\text{--}\omega$, and Spalart–Allmaras produced Cd values very close to experiment but showed deficiencies in flow-field resolution, particularly in separation and wake capture.

On balance, the realizable $k\text{--}\epsilon$ turbulence model was adopted as the primary closure for the Morgan Plus Four simulations due to its robustness and computational efficiency, while the SST

$k-\omega$ model was retained for selective high-separation cases where additional physical fidelity was required.

4.3 Tetrahedral vs Polyhedral

Tetrahedral meshes are widely used in external automotive CFD because of their ability to fit complex geometries. However, they are prone to stretching and skewness, which reduces mesh quality and can compromise convergence in sensitive regions such as wakes and diffusers. Sosnowski et al. (2018) found that tetrahedral meshes often require a higher element count to achieve accuracy comparable with alternative topologies, increasing computational cost. This behaviour was evident in the present study: the tetrahedral mesh produced stronger oscillations in the C_p distribution and less accurate agreement with experimental pressure taps.

By contrast, polyhedral meshes demonstrated improved stability and accuracy. Łomakin et al. (2018) reported that polyhedral cells maintain higher orthogonal quality and are less sensitive to distortion, improving convergence robustness. This was reflected in the current results, where the polyhedral mesh provided smoother pressure recovery and closer agreement with experimental data. The improvement in orthogonal quality from 0.219 (tetrahedral) to 0.306 (polyhedral) represented a $\sim 40\%$ rise, corresponding to better element alignment and reduced skewness, which enhanced solver stability and reduced numerical diffusion.

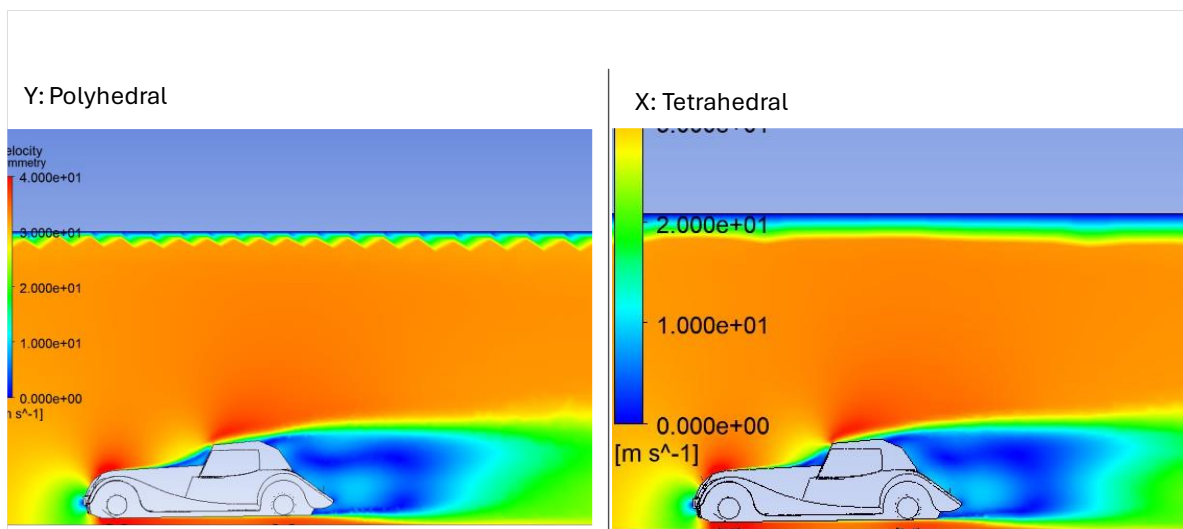
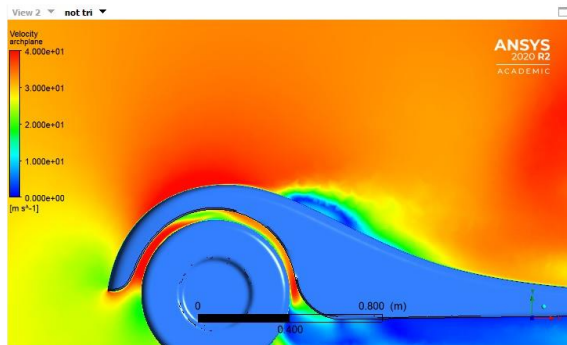


Figure 29 Velocity contour comparison at the ground plane for polyhedral (Y) and tetrahedral (X) meshes

Figure 31 compares velocity contours at the ground plane. Both meshes captured boundary-layer development consistently, indicating that global flow near the ground was resolved in a broadly similar way.

In the wheel arch region, shown in Figure 32, both meshes reproduced acceleration of flow around the arch. However, the polyhedral mesh predicted earlier flow separation and stronger turbulence beneath the vehicle. This indicates that while both approaches captured the dominant behaviour, the polyhedral mesh highlighted more aggressive underbody flow structures.

Y: Polyhedral



X: Tetrahedral

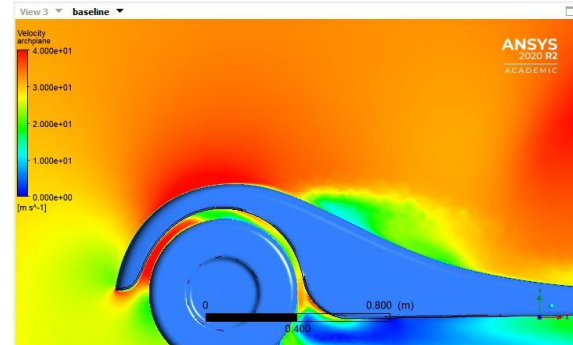
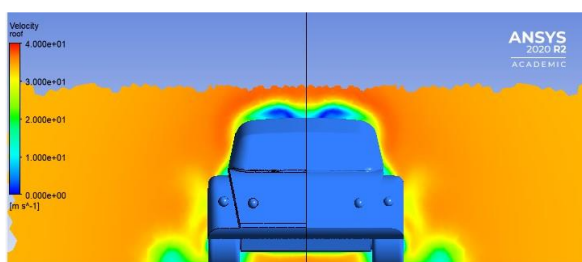


Figure 30 Velocity distribution through the wheel arch region for polyhedral (Y) and tetrahedral (X) meshes

Roof separation is illustrated in Figure 33. The tetrahedral mesh predicted stronger separation and a larger detached region over the roof, while the polyhedral solution showed smoother, less distinct detachment. This reflects known tendencies: tetrahedral meshes resolve sharper gradients, whereas polyhedral meshes dampen instabilities and yield more stable but diffused results (Łomakin et al., 2018; Sosnowski et al., 2018).

Y: Polyhedral



X: Tetrahedral

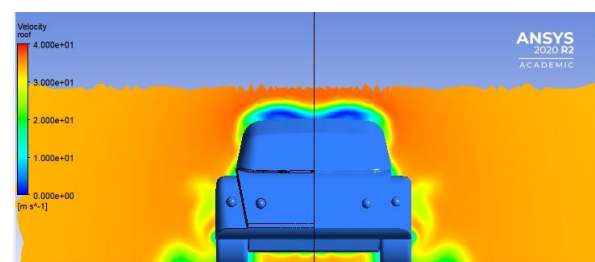


Figure 31 Flow from wheel and roof separation contours comparing polyhedral (Y) and tetrahedral (X) meshes

Wake behaviour is compared in Figure 34. Tetrahedral meshing produced sharper, more structured vortex cores, whereas polyhedral elements generated a more diffuse wake. Although the tetrahedral case captured finer detail, the polyhedral solution was more numerically stable.

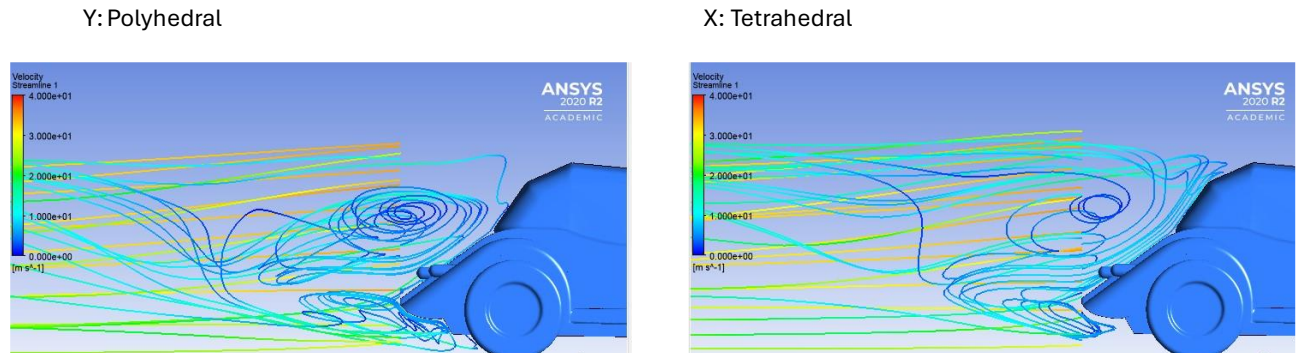


Figure 32 Wake streamline comparison showing vortex structures for polyhedral (Y) and tetrahedral (X) meshes

Finally, Figure 35 compares pressure coefficients at the centreline and wheel arch against experimental measurements. Both mesh types followed the experimental trend, with tetrahedral results showing sharper variations and polyhedral predictions appearing smoother. Notably, the polyhedral mesh gave a closer match at the arch, particularly between 2700–3000 mm, where it reproduced the pressure peak more accurately.

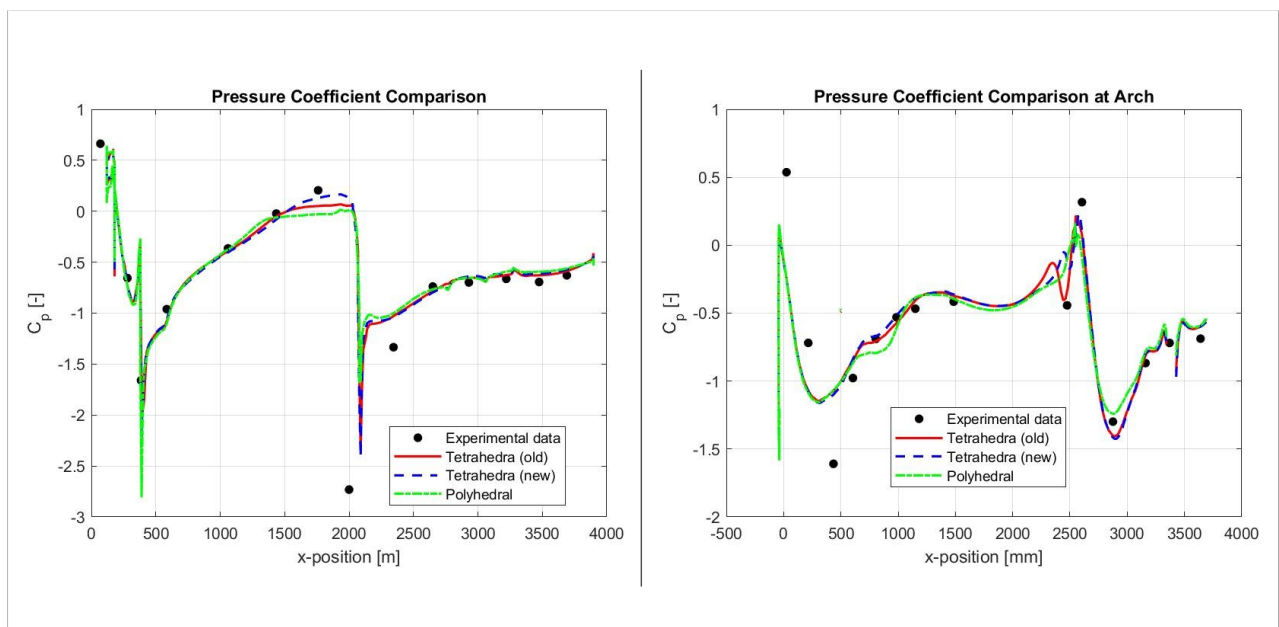


Figure 33 Pressure coefficient comparison at the centerline and wheel arch: experimental data vs tetrahedral and polyhedral meshes

The overall drag predictions confirmed these observations. The experimental C_d was 0.453, compared to 0.467 for the polyhedral mesh (+3.1%) and 0.479 for the tetrahedral mesh (+5.7%). While both approaches produced similar overall aerodynamic predictions, the polyhedral mesh yielded a closer match to experiment, smoother convergence, and reduced computational time by more than 50%. For these reasons, the polyhedral mesh was selected for all subsequent Morgan Plus Four simulations.

5 Baseline Analysis

The baseline aerodynamic performance of the Morgan Plus Four highlights fundamental weaknesses in both front and rear flow management. At the rear, the short diffuser ramp and obstructed underbody limit pressure recovery, preventing meaningful negative lift generation. At the front, the absence of a splitter or undertray extension restricts the ability to manage incoming flow and condition the boundary layer for downstream effectiveness. Together, these limitations explain the poor aerodynamic efficiency of the baseline vehicle and establish the foundation for subsequent redesign.

5.1 Rear Analysis

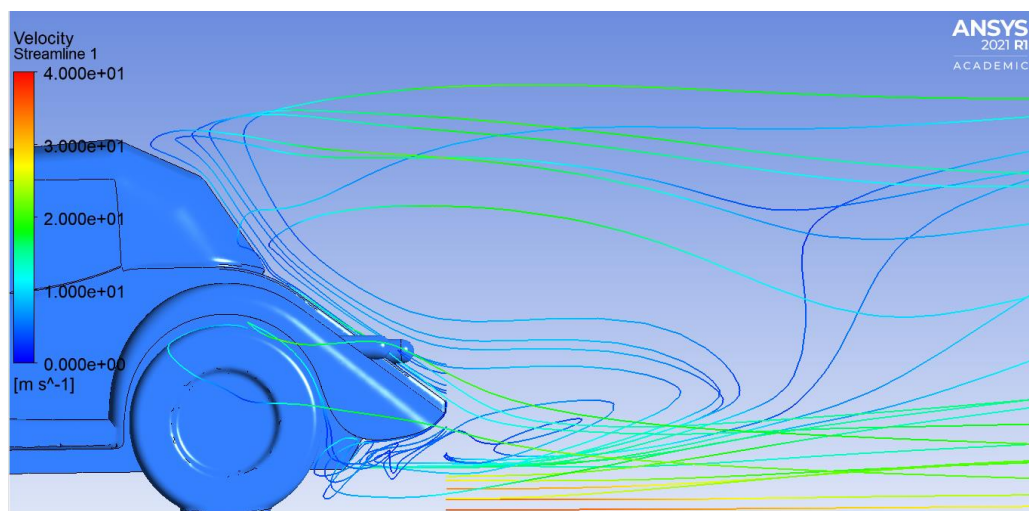


Figure 34 Baseline Analysis: Streamline From underneath rear car

The streamline plot in Figure 36 shows that the baseline geometry generates a large separated wake behind the underbody. Flow detaches early at the rear axle and rolls into recirculating vortices, expanding the wake and lowering base pressure. This behaviour reflects the literature on diffuser ramp length, where short and steep inclines cause premature stall (Katz, 1995; Marklund, 2013). As Buscariolo et al. (2020) noted, the diffuser dictates underbody pressure recovery, and the inability of the baseline Morgan geometry to sustain attached flow results in unsteady vortical structures dominating the wake, increasing drag while generating little negative lift.

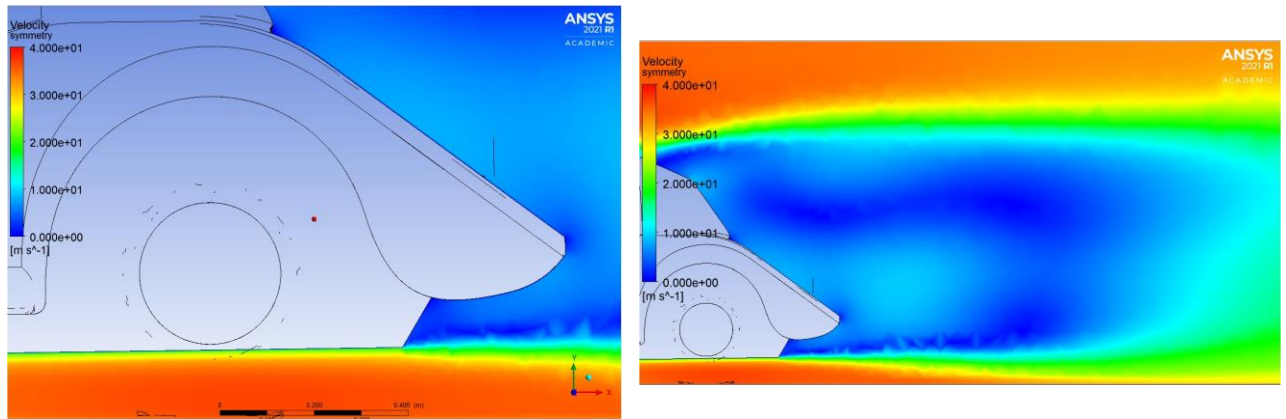


Figure 35 Baseline Analysis of rear diffuser location and wake region

Velocity contours in Figure 37 reinforce this finding. The diffuser exit flow decelerates sharply, leaving large low-velocity regions in the wake. Instead of accelerating smoothly through a gradual ramp, the underfloor stalls and expands, consistent with the separation seen in Figure 36. This behaviour matches the conclusions of Gopalarathnam et al. (2019), who identified expansion ratio and diffuser length as critical parameters. The outcome is a wide wake with low base pressure, confirming that the diffuser design in its current form cannot support effective ground-effect aerodynamics.

5.1.1 Underbody Wake Development

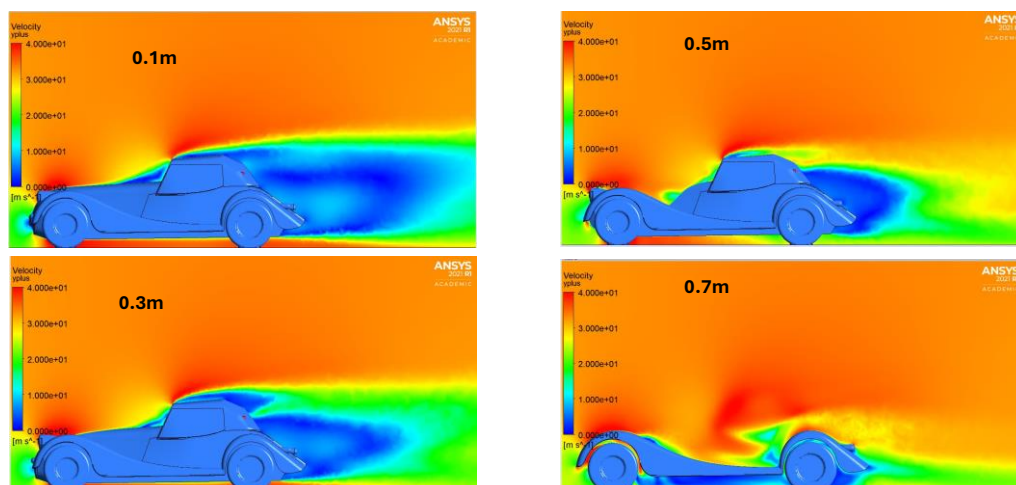


Figure 36 Baseline Morgan Plus 4 velocity contours showing wake development at different far-field widths (0.1 m, 0.3 m, 0.5 m, 0.7 m).

Figure 38 illustrates wake development at different lateral distances. At 0.1 m, the wake is largest, with a broad low-velocity region. By 0.3 m, partial recirculation begins, recovering some pressure. At 0.5 m, roof flow remains attached and wake size is reduced, though a recirculation zone persists. At 0.7 m, wheel-arch turbulence dominates, and a dead zone of low velocity forms beneath the chassis, contributing to positive lift. These results indicate that the underbody lacks the capacity to manage airflow effectively, confirming that diffuser redesign is necessary to improve wake control.

5.1.2 Global Flow and Roof Separation

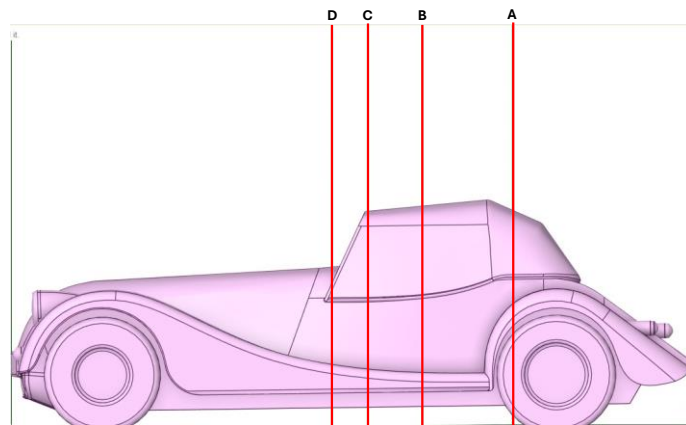


Figure 37 Plane positions

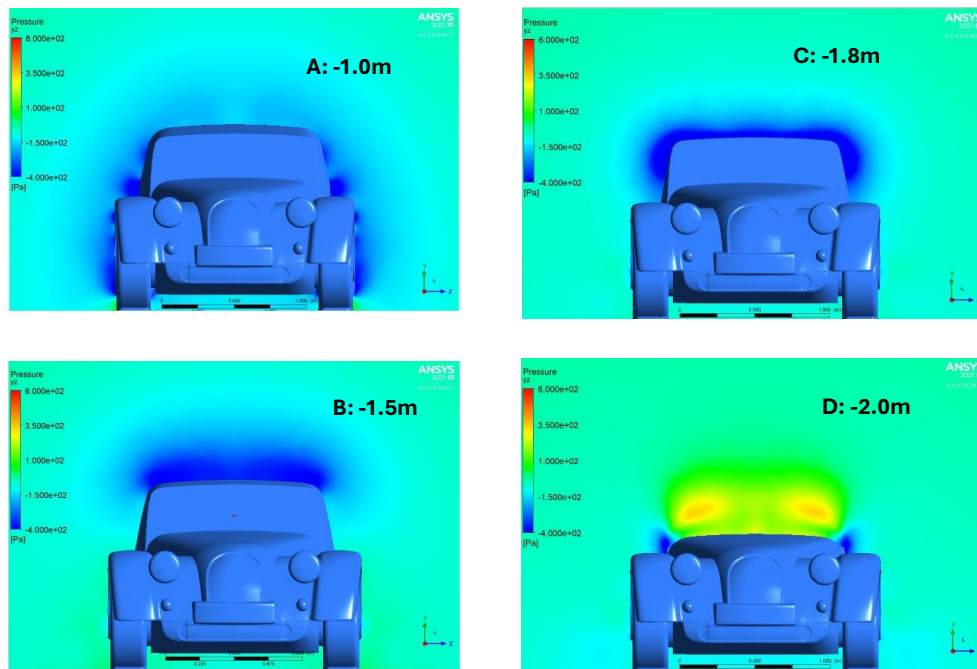


Figure 38 Pressure contour comparison of the Morgan Plus 4 at varying domain lengths (–1.0 m, –1.5 m, –1.8 m, –2.0 m).

Pressure contours in Figure 40 show pressure variation along the upper body. At Plane D (–2.0 m, windscreen base), a high-pressure stagnation zone loads the screen and A-pillars. At Plane C (–1.8 m, mid-roof), pressure drops rapidly as flow accelerates over the roof, producing a broad suction region. At Plane B (–1.5 m, rear roof), separation intensifies, expanding the low-pressure field. At Plane A (–1.0 m, behind the cabin), the wake is fully developed, with extensive low-pressure zones and recirculation. These findings demonstrate that the baseline Morgan produces lift over the upper body and a wide, turbulent wake, both of which reduce stability.

5.2 Quantitative Baseline Performance

To benchmark CFD against experiment, the baseline drag coefficient was compared with physical measurements from pressure-tap testing. The experimental C_d was 0.453, while the baseline CFD predicted 0.467, representing a +3.1% difference. This level of agreement demonstrates that the CFD methodology is sufficiently accurate to capture the Morgan’s aerodynamic behaviour.

Table 4 Difference in Experimental and CFD

Case	Cd	% Difference vs Experiment
Experimental	0.453	–
CFD Baseline	0.467	3.10%

No experimental lift coefficient (Cl) was available for comparison. However, the CFD baseline predicted small positive lift, consistent with observed roof separation and the absence of front-end flow control.

5.3 Baseline Conclusions

The baseline Morgan Plus Four exhibits poor aerodynamic performance, dominated by:

1. Rear underbody failure – short diffuser ramp and obstructed geometry causing premature separation, wide wake, and minimal negative lift.
2. Front-end weakness – absence of splitter or undertray flow conditioning, leading to stagnation at the windscreen base, strong suction over the roof, and positive lift.

Together, these features produced $C_d = 0.467$, within 3.1% of experimental measurement, validating the CFD setup but confirming that the baseline geometry cannot support effective ground-effect aerodynamics. Addressing these deficiencies through diffuser and splitter redesign is therefore essential. This analysis provides the foundation for Section 6, where redesign strategies are developed to improve aerodynamic efficiency while respecting the Morgan’s heritage styling constraints.

6.0 Morgan Redesign

The redesign stage began with remodelling of the Morgan Plus Four geometry to incorporate necessary safety and structural elements. A roll cage was added to the CAD model, which was a mandatory requirement, and the roof-down version was modified to include correct door and cage positioning (Figure 41).



Figure 39 Morgan Plus Four fitted with roll cage

A simplified driver model was also created, as the Morgan was to be simulated without a windscreen. Wheel rotation axes were corrected relative to the CAD reference frame, with updated positions listed in Table 5.

Table 5 Roof Down Wheel Positions

Wheels	x	y	z	Speed(Rad/s)
fw	-3.27791	0.30873	0.713	99.22
rw	0.75771	0.30873	0.713	95

6.1 Convergence Problems and Model Rebuilding

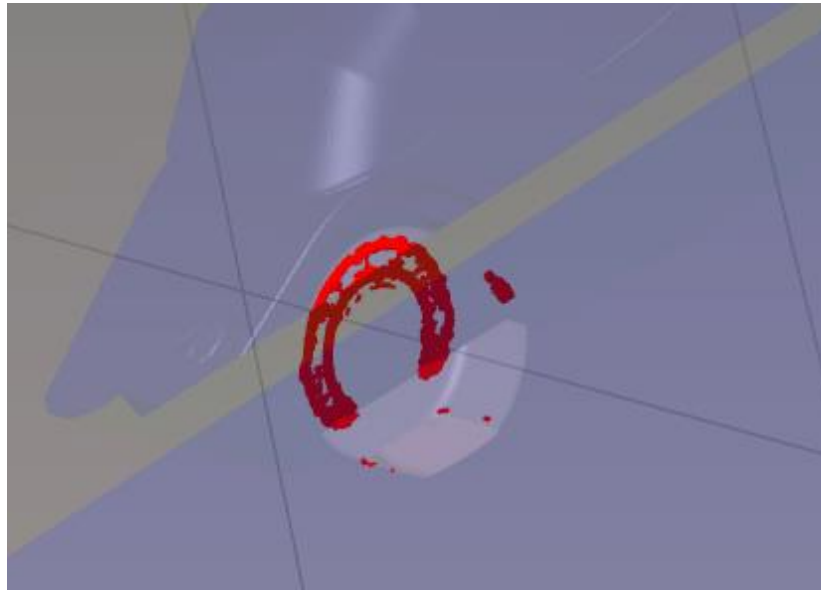


Figure 40 Open top wheel meshing issues

The first attempt to simulate the open-top roll cage configuration encountered significant meshing problems, particularly around the wheels. The initial mesh produced a maximum skewness of 0.713 and an orthogonal quality of only 0.084, which improved to 0.215 after remodelling the wheels. Despite these improvements, convergence remained unstable.

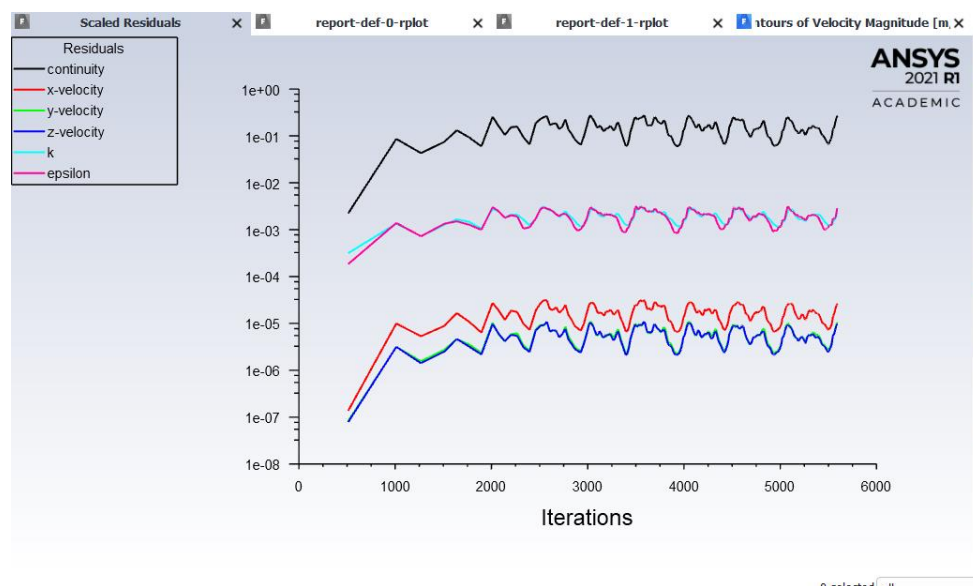


Figure 41 Convergence issues with open top CAD Model

As shown in Figure 43, residuals oscillated, and solutions repeated without stabilising. Further remodelling work was attempted, but results remained unpredictable and failed to converge.

within acceptable limits. The primary cause was traced to the roll cage geometry, which contained multiple errors and could not be merged into a single clean component. Additional issues occurred when transferring the model from SolidWorks to SpaceClaim, saving as a STEP file introduced surface distortions that persisted across different CAD platforms.

Given these persistent challenges, further development of the open-top configuration was postponed. The hard-top model was instead adopted for continued aerodynamic optimisation. This choice allowed diffuser and splitter redesign, identified in Section 5 as the main aerodynamic priorities, to progress within the project timeframe. The hard-top configuration was also expected to provide representative results for underbody and front-end flow behaviour, with only minor differences anticipated in the wake structure compared to the open-top model.

All subsequent geometry modifications were performed directly in ANSYS SpaceClaim to avoid the transfer errors encountered when moving models between different CAD platforms. Working within a single environment ensured consistent geometry handling and reduced the risk of introducing mesh-quality issues during preprocessing.

6.2 Redesign Objectives

The baseline analysis in Section 5 identified two critical aerodynamic shortcomings in the Morgan Plus Four: a rear diffuser prone to early separation and a lack of front-end flow conditioning. The redesign therefore focused on three objectives:

1. To optimise diffuser geometry by increasing ramp length where possible, improving expansion ratio, and integrating strakes to delay flow separation.
2. To introduce a front splitter capable of conditioning incoming flow, reducing front lift, and improving underbody suction without affecting diffuser performance.
3. To ensure that all modifications comply with Motorsport UK *Blue Book* regulations while respecting packaging constraints and the heritage-inspired design.

6.3 Aerodynamic and Bodywork Design Regulations

All redesign work was carried out in compliance with the Motorsport UK *Blue Book* (Section Q). These regulations place dimensional and safety constraints on aerodynamic devices used in circuit racing.

For front devices such as splitters, projection beyond the original bodywork is limited to 100 mm when viewed from above (Q.13.1.3). The maximum vehicle width, including aerodynamic additions, must not exceed 2000 mm (Q.13.1.2). Underbody elements must avoid sharp projections, and no aerodynamic component may extend below the chassis baseline (Q.13.1.3–4). Rear aerodynamic devices such as wings cannot extend more than 100 mm behind the rearmost bodywork and must not exceed vehicle width (Q.13.1.3b). All features must be rigidly mounted, with movable devices such as DRS explicitly prohibited (Q.13.1.5).

Visibility and lighting requirements also apply. Driver sightlines, lighting systems, and number plate visibility must remain unobstructed unless category rules state otherwise (Q.13.1.1, Q.13.3; Q.19). While ride height is not explicitly regulated, ground clearance is indirectly constrained by safety rules, requiring diffusers and splitters to avoid ground contact under dynamic conditions.

Modifications to bumpers, wheel arches, and vents are permitted, provided they present no sharp edges, are securely attached, and do not compromise structural integrity (Q.13.1.4). This allows louvres and arch cut-outs to be introduced where appropriate, provided safety and compliance are maintained.

6.4 Diffuser Design

Literature consistently supports prioritising diffuser optimisation before splitter development in order to establish a stable aerodynamic baseline. Buscariolo et al. (2020) demonstrated that diffuser geometry governs underbody pressure recovery and wake stability. Cooper et al. (1998) showed experimentally that splitter-first approaches risk disrupting downstream momentum recovery, reducing diffuser effectiveness. Willemsen (2012) reinforced this principle, concluding that the diffuser must be optimised first, with the splitter subsequently tuned for balance. Zhang et al. (2022) further emphasised the importance of integrated design to prevent counterproductive front–rear interactions.

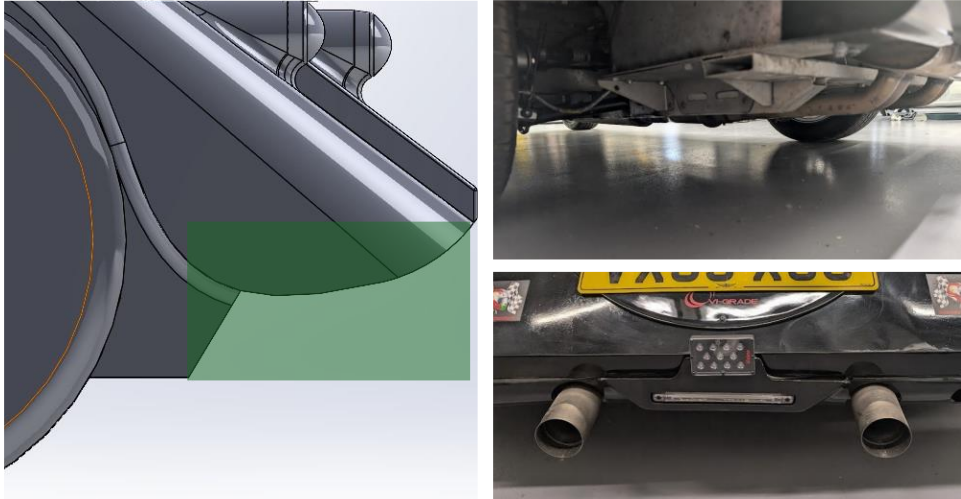


Figure 42 Available Rear Packaging Space for Diffuser Design on the Morgan Plus 4, measured in workshop

Physical inspection of the Morgan Plus Four (Figure 44) highlighted packaging constraints absent from the CAD model. Suspension members and chassis components intrude into the diffuser region, reducing effective throat height. In addition, the twin central exhaust outlets exit directly into the diffuser space. Both features would require cut-outs or geometric modifications, which could disrupt flow and compromise performance. This illustrates the importance of aligning CFD design concepts with real packaging feasibility.

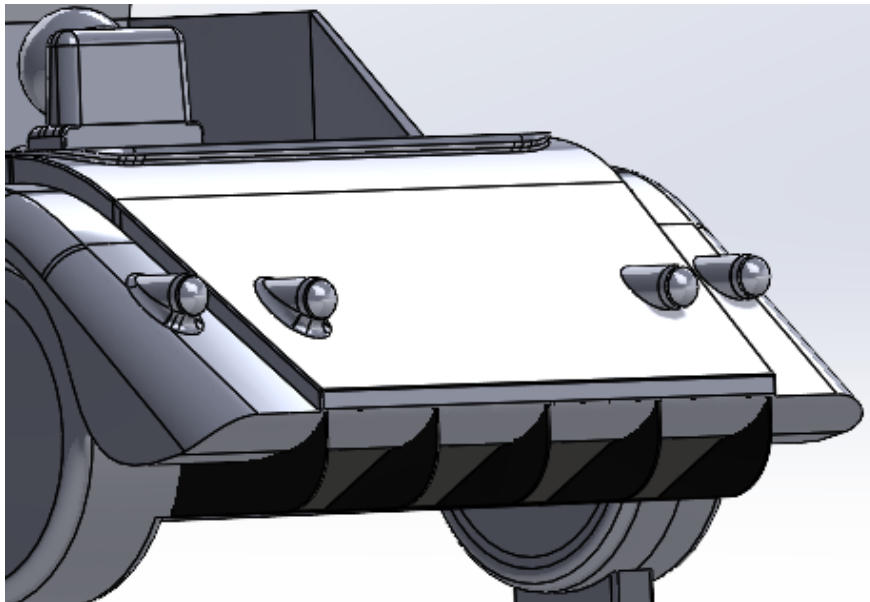


Figure 43 Initial Diffuser Design

The initial diffuser design was restricted by limited underbody length, resulting in a relatively steep ramp angle. Katz (1995) reported that diffuser ramp angles above 10–12° increase the likelihood of flow separation, a finding echoed by Marklund (2013). Given the Morgan's tight rear packaging, this steep angle poses a risk of stall and limits diffuser effectiveness in its baseline configuration.

6.4.1 Initial Diffuser Design Analysis

The first diffuser iteration produced an increase in negative lift of 0.0757 compared to the baseline, while the overall change in C_p was limited to approximately 1%. Although these results indicate a modest aerodynamic benefit, further inspection revealed issues with both meshing and geometry.

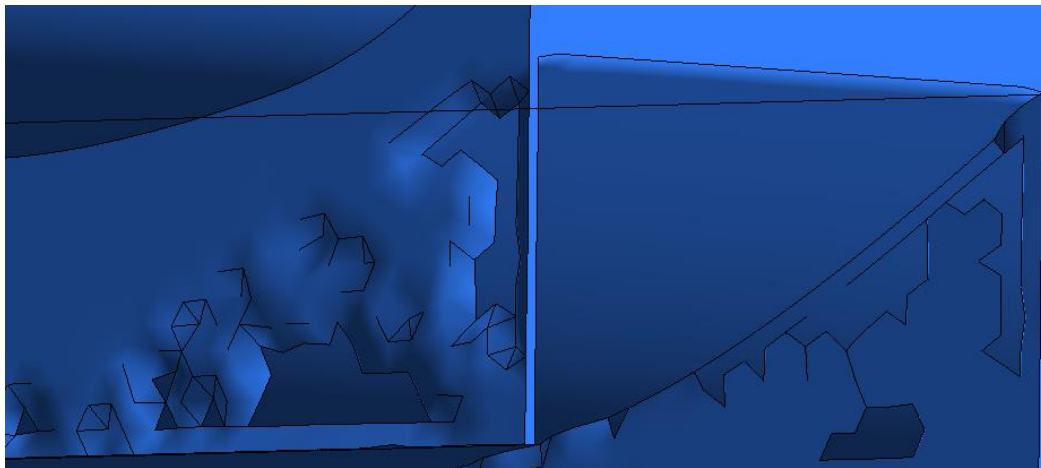


Figure 44 Holes in diffuser from meshing

As shown in Figure 46, meshing errors created gaps within the diffuser strakes. This occurred because the strake geometry was too fine for the global mesh settings. To avoid such errors in future iterations, either a local refinement strategy or thicker strake profiles must be applied.

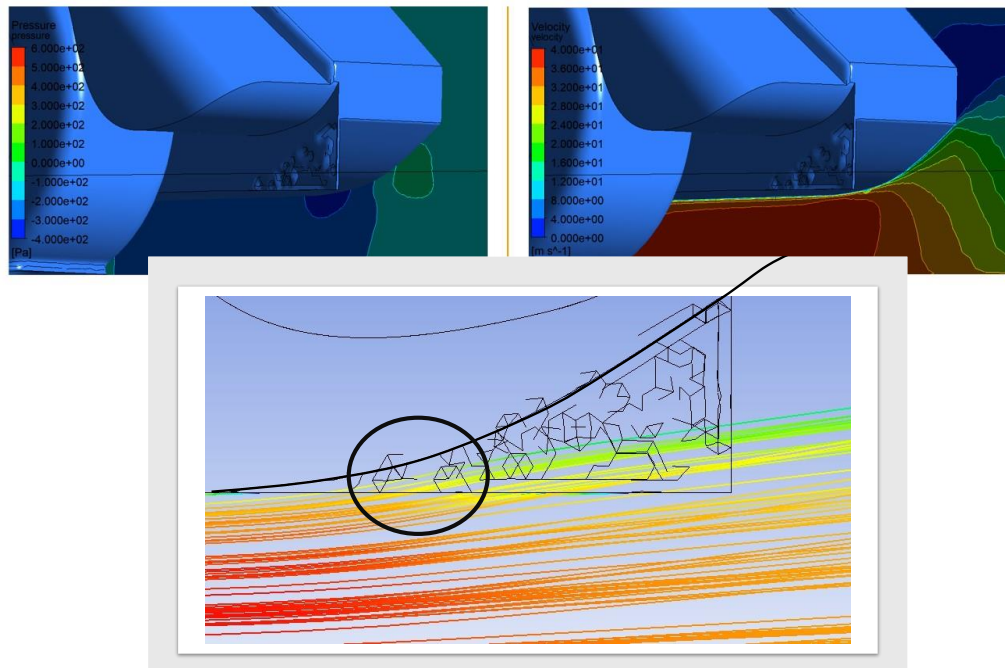


Figure 45 Flow separation occurring early at diffuser profile

Figure 47 highlights that separation occurred early along the diffuser ramp, suggesting that the design employed too aggressive a profile. Katz (1995) and Marklund (2013) both noted that steep diffuser ramp angles promote premature stall, supporting this observation. Future designs will therefore require smoother, more gradual expansion angles to sustain attached flow and enhance pressure recovery.

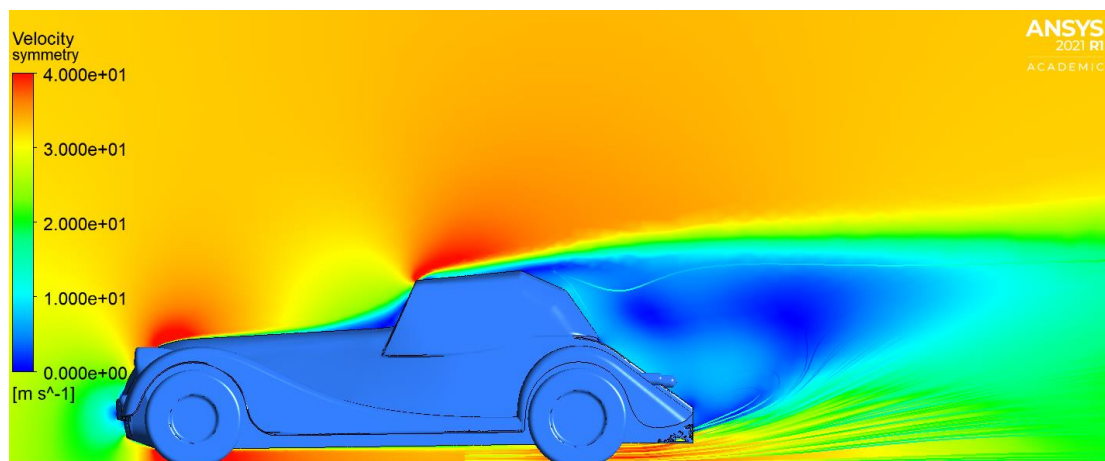


Figure 46 velocity plot, wake region analysis for first design

Despite these shortcomings, the velocity contours in Figure 48 show that the initial design reduced the wake region relative to the baseline, producing a more coherent flow structure. This confirms that the design direction is promising, but further refinements are necessary to maintain attachment throughout the diffuser length and achieve greater reductions in drag.

6.5 Diffuser Design Two

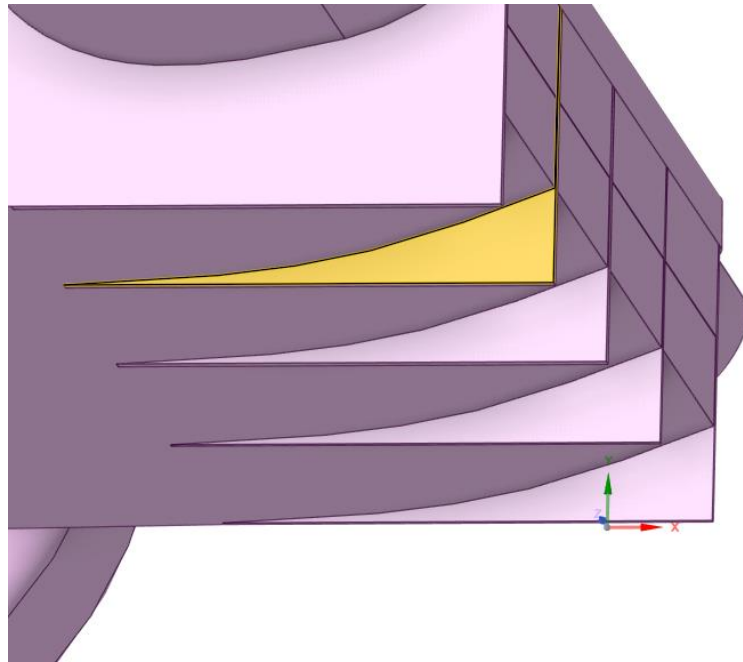


Figure 47 Diffuser Desing two

The second diffuser iteration introduced a revised ramp angle and increased strake thickness to 5 mm to improve meshing quality. These changes resolved some of the meshing issues observed in the first design, although CFD simulations produced repeating C_d and C_l values, requiring averages to be taken to establish reliable results (Figure 50).

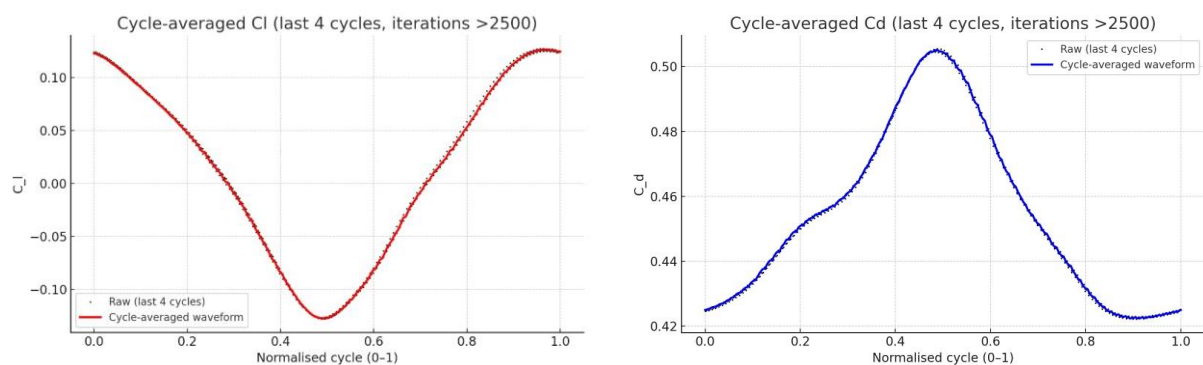


Figure 48 Average C_d and C_l due to repeatability

The averaged results indicated a reduction in drag of 0.163 and a decrease in lift of 0.0568 compared to the baseline. These values suggest that the new diffuser geometry provided measurable aerodynamic improvements. However, performance was still limited, as the kick point profile lacked sufficient sharpness to sustain attached low-pressure flow.

6.5.1 Design Limitations

Although the second diffuser design incorporated strakes to partition flow and promote boundary-layer energisation, its geometric proportions remained constrained. The diffuser length was too short to enable a gradual expansion, forcing either a steep ramp angle or reduced capacity for negative lift.

As discussed in Section 2.2.2.1, diffuser performance is strongly dependent on ramp length and angle. Buscariolo et al. (2020) reported that simplified geometries can sustain expansion angles of 20–30° before separation, while Katz (1995) observed that full-scale race cars typically stall beyond 10–12°. The Morgan’s packaging limitations placed this design between these thresholds but without adequate ramp length, attached flow could not be sustained.

Section 2.2.3 further emphasises that expansion ratio and ride height are equally important. The truncated length of the Morgan diffuser limited the expansion ratio, preventing the suction levels identified as optimal by Gopalarathnam et al. (2019) in the range of 1.2–1.5.

The addition of strakes aligned with best practice outlined in Section 2.2.4, yet their effectiveness was compromised by the steep, short ramp. Unni et al. (2017) showed that strakes improve boundary-layer stability only when integrated with a suitably long diffuser profile. In this design, the ramp geometry restricted vortex development, reducing their influence.

In summary, while the second diffuser design achieved reductions in drag and lift, its effectiveness was constrained by packaging limitations that forced a short, steep ramp. Without increasing ramp length and adopting a smoother expansion, the diffuser cannot provide sustainable underbody suction or maximise negative lift potential.

6.6 Diffuser Design three: Expanding Design Area

For a diffuser to operate effectively, the underbody geometry must provide sufficient space and clearance for flow development. As illustrated previously in Figure 44, the Morgan’s underbody packaging, including the rear suspension members, exhaust system and chassis layout, restricts the achievable diffuser length and ramp angle. Without modifications to these structures, any diffuser design will be limited in ramp length and expansion ratio. This prevents the establishment of stable low-pressure flow and meaningful downforce. Diffuser Design 3 was therefore developed to expand the available underbody region, incorporating a larger ramp length and expansion zone (Figure 51).

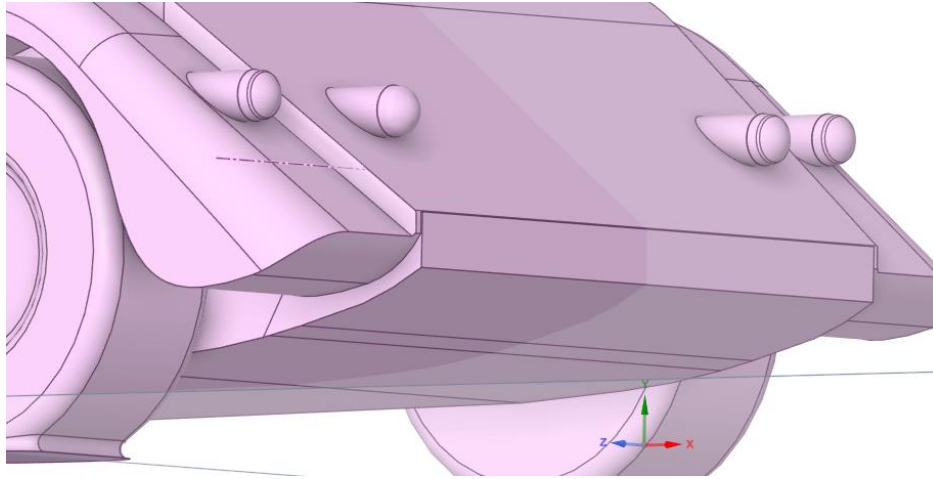


Figure 49 Diffuser Design 3

Compared with the baseline geometry, Design 3 reduced the size of the low-velocity wake directly behind the car. This indicates that it was beginning to energise the underbody flow (Figure 52). However, separation still occurred early along the diffuser ramp, which limited the suction effect and prevented the diffuser from operating at its full expansion ratio. The resulting pressure recovery remained weak.

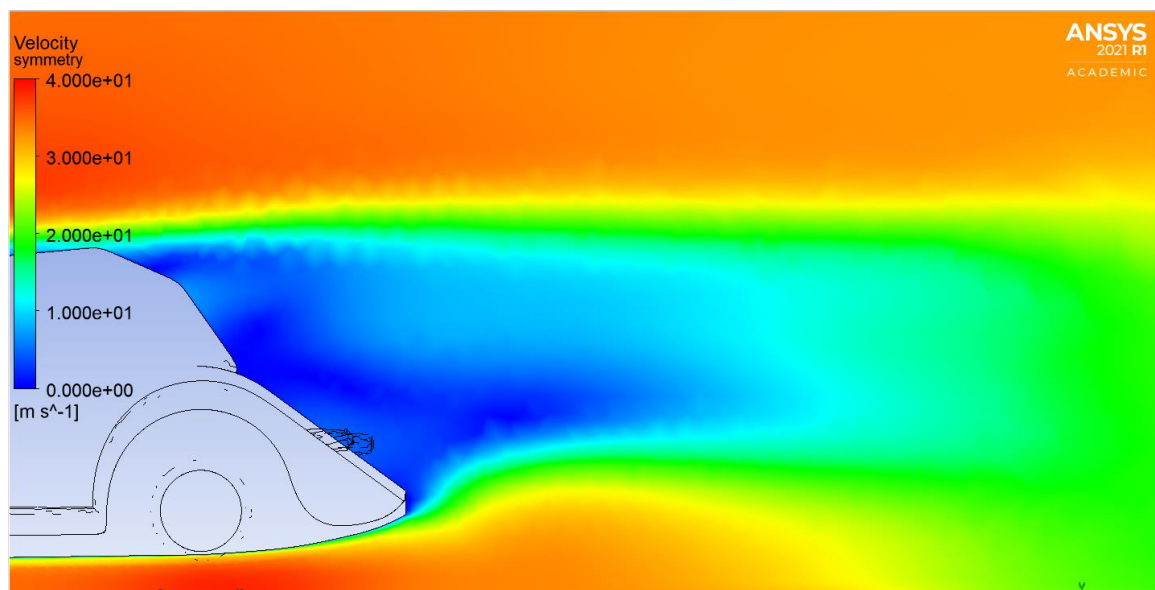


Figure 50 Design 3, wake region

The early separation was attributed to the steep ramp angle and the sharp transition at the diffuser entry. Both of these created an adverse pressure gradient that the boundary layer could not overcome. Although wake reduction was observed, the diffuser remained inefficient and did not produce the expected downforce levels. This highlighted the requirement for a smoother transition from flat floor to diffuser and a more gradual ramp angle to sustain attached flow.

Further analysis in Figure 53 revealed that turbulent flow from the rear wheels bled into the diffuser. This disrupted the suction effect and reduced negative lift.

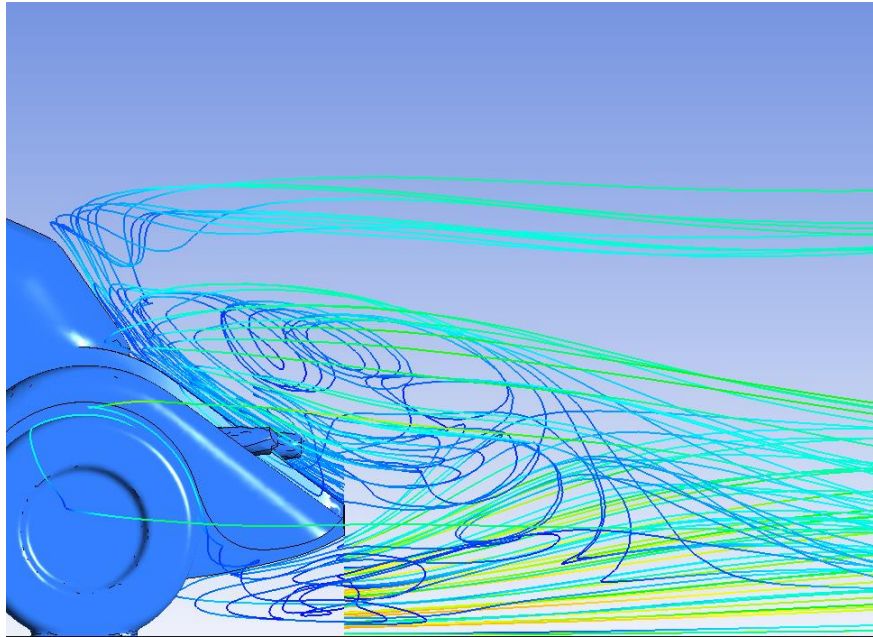


Figure 51 Streamlines in the wake region from design 3

Quantitatively, Diffuser 3 reduced drag by approximately 2%. However, flow restriction caused by the rear wheels meant that clean, high-energy airflow reached only the centre channels of the diffuser, as shown in Figure 54.

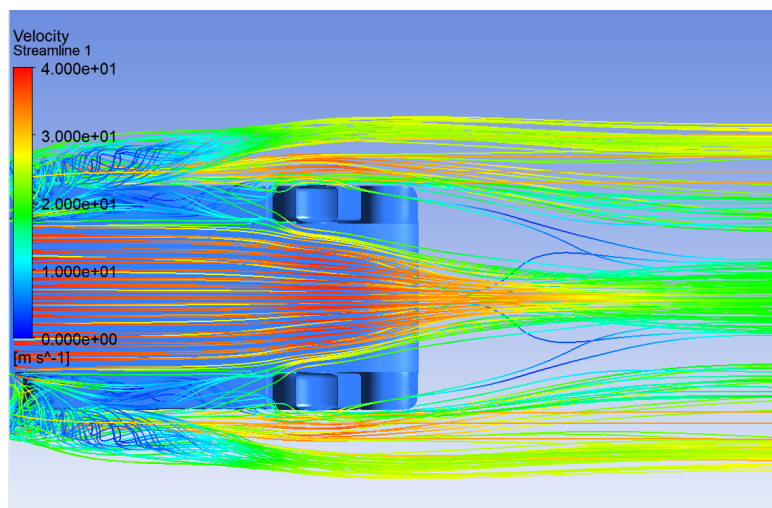


Figure 52 Streamlines underneath the car from design 3

Streamline plots through the centre of the diffuser (Figure 55) showed that flow remained relatively attached in this region. In contrast, separation near the rear wheels was driven by turbulence bleeding across the diffuser, as illustrated more clearly in Figure 56.

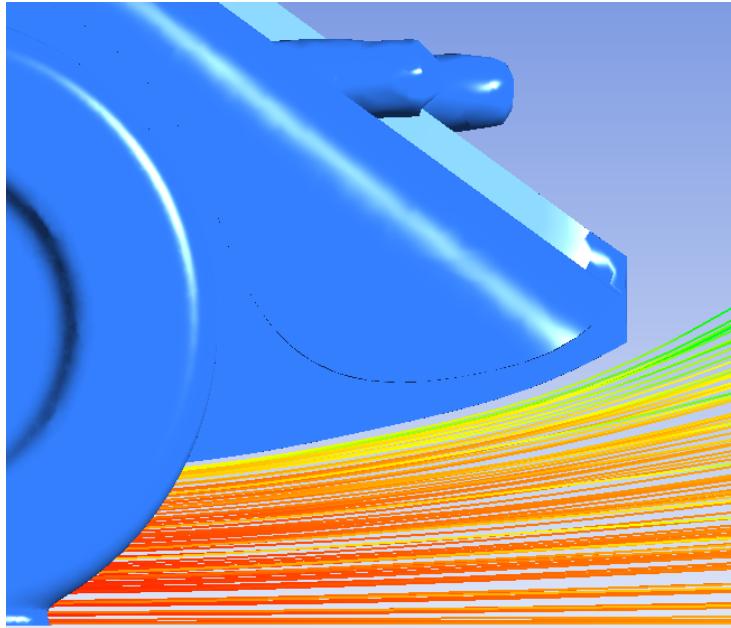


Figure 53 Streams along diffuser profile

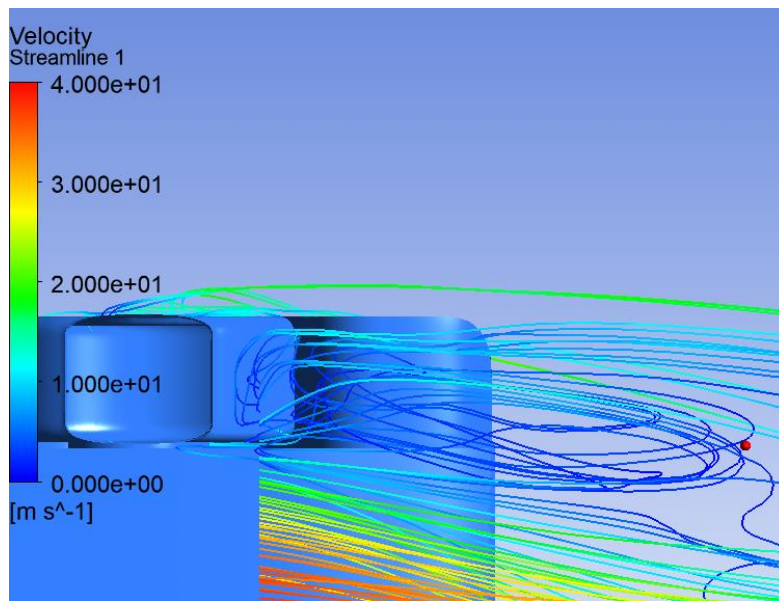


Figure 54 turbulence flow bleeding from wheels, design 3

These results confirm that while extending the diffuser ramp length reduced drag and improved central flow attachment, turbulence interaction from the wheels continued to undermine efficiency. A design strategy to limit cross-flow bleeding from the rear wheels must therefore be developed to improve diffuser performance and unlock greater downforce potential.

6.7 Redesign of Kick-Point

Due to the limited negative lift generated at the diffuser kick point in previous designs, the profile was reshaped as shown in Figure 57.

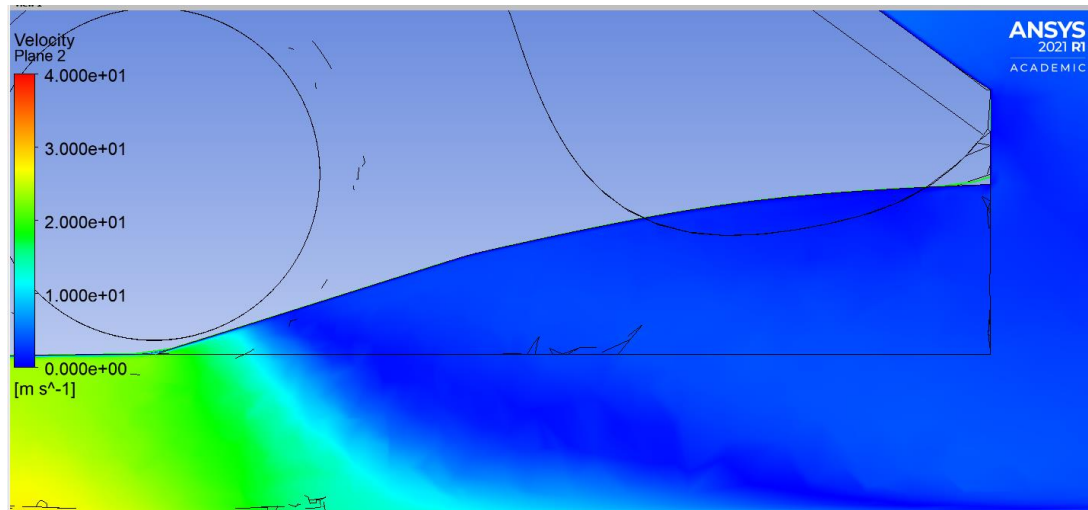


Figure 55 diffuser profile, velocity plot near rear wheel

The velocity plot near the rear wheel highlights turbulence generated at the tyre contact patch, with wheel wake spilling into the diffuser entry. This disturbed flow reduces the diffuser's ability to accelerate the underfloor air, creating localised low-velocity regions and promoting early separation along the ramp. The result is weaker suction at the kick point and reduced negative lift generation at the rear axle.

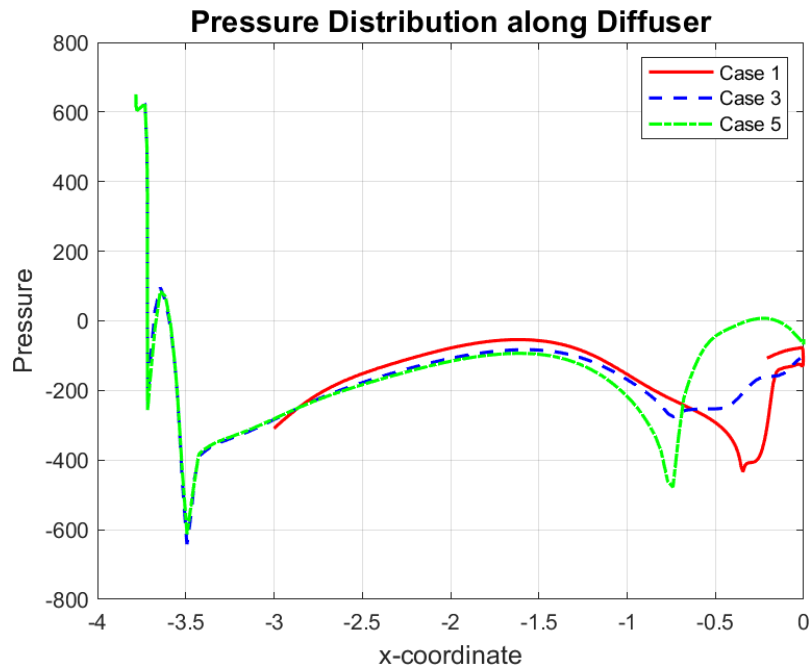


Figure 56 Pressure distribution along diffuser

Figure 58 shows the pressure distribution along the diffuser centreline for three design cases. The revised kick-point profile (Case 5, green line) exhibits a stronger pressure drop at the start of the diffuser compared with earlier designs. This indicates that the modification generates greater suction at the throat, which accelerates the underbody flow more effectively. Pressure recovery towards the exit is also smoother and more consistent than in the original Case 1 profile (red line), which suffered from early separation and a sudden rise in pressure near the exit.

Earlier designs produced limited negative lift at the kick point, as indicated by the flatter pressure curves in Case 1 and Case 3. By contrast, the revised kick-point geometry sustained a lower pressure region through the diffuser ramp and achieved a more stable recovery at the exit, improving aerodynamic efficiency and downforce generation.

6.8 Diffuser Strakes – Effect on Underfloor Flow

The inclusion of strakes altered how the diffuser was fed and how it recovered. Strakes divided the underfloor stream, kept the wheel wake out of the central channel, and suppressed lateral cross-flow that had been causing separation.

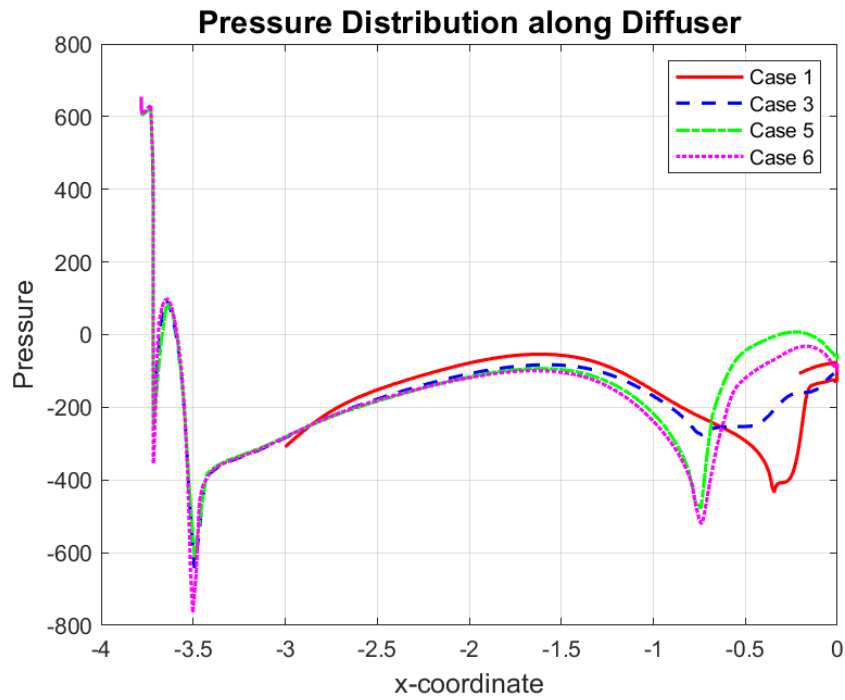


Figure 57 Pressure distribution along the diffuser centreline vs x from origin.; Case 6 (magenta, with strakes)

Figure 59 compares the pressure distribution with and without strakes. In the baseline case without strakes, a sharp pressure dip was followed by early recovery. With strakes fitted (magenta line), suction was sustained further downstream and recovery occurred more gradually. This confirmed that attachment was maintained deeper into the diffuser ramp and that separation was reduced.

CFD analysis showed that adding strakes decreased drag by 1.92%, demonstrating that improved flow management at the ramp reduced wake losses and supported cleaner pressure recovery.

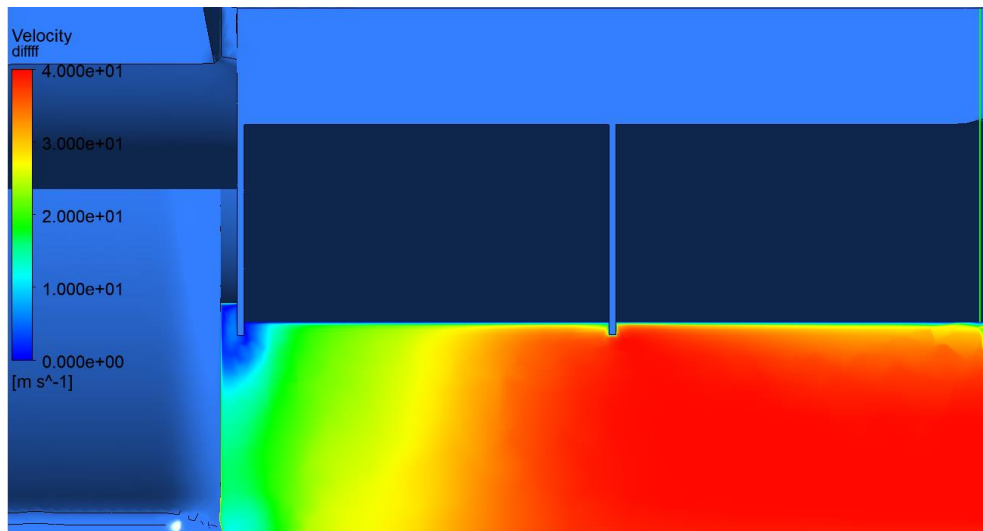


Figure 58 Velocity contour at $x = -0.70$ m (from origin), with strakes

Velocity contours at $x = -0.70$ m (Figure 60) illustrate the improved inlet condition. In the baseline case, wheel wake spilled into the centre channel, creating a low-velocity pocket. With strakes fitted, this disturbance was suppressed, resulting in a cleaner inflow.

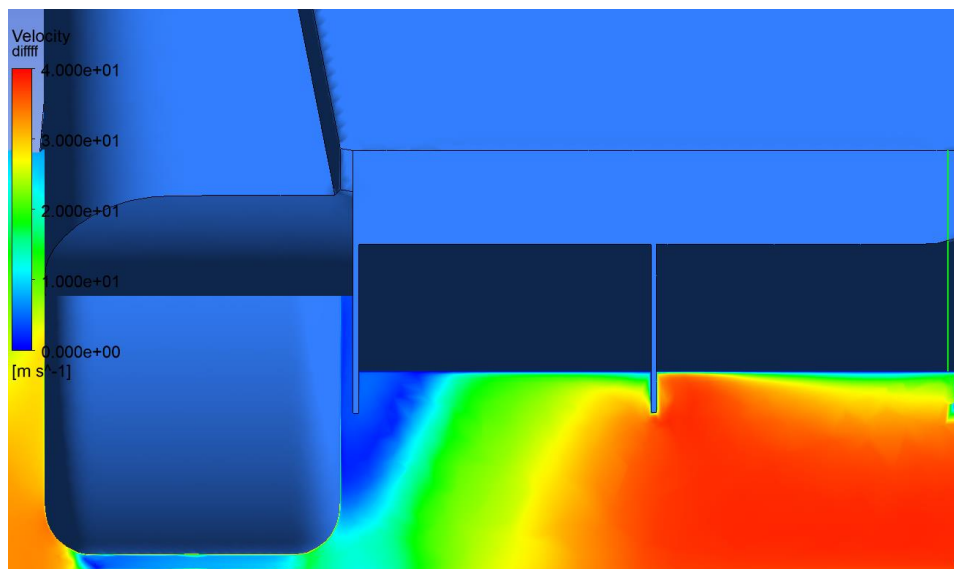


Figure 59 Velocity contour at $x = -0.60$ m (from origin), with strakes.

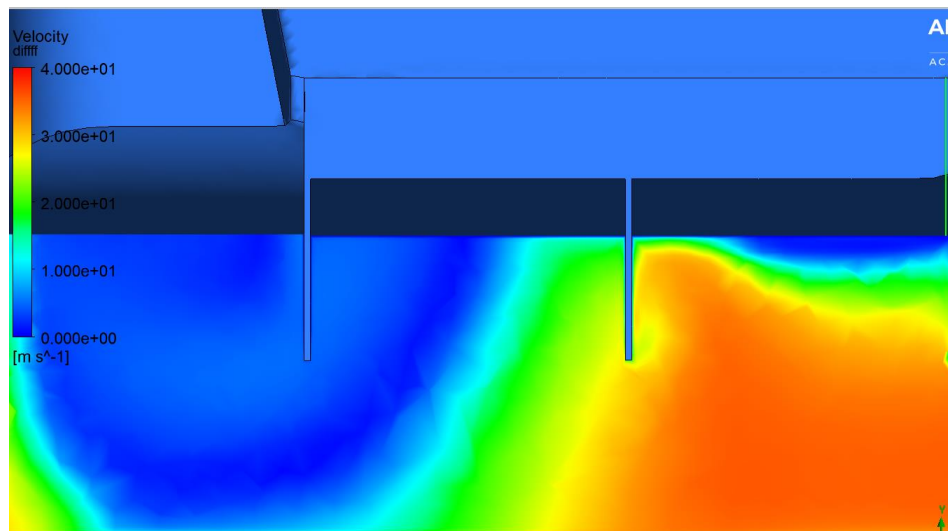


Figure 60 Velocity contour at $x = -0.40$ m (from origin), Strakes

Further downstream, Figures 61 and 62 show that the strakes delayed but could not eliminate separation. By $x = -0.40$ m, the flow quality was significantly reduced, with wheel-wake turbulence and boundary-layer growth weakening the underbody stream. This led to early pressure recovery, confirming that while strakes improved upstream stability, their influence diminished by the diffuser exit.

6.9 Reducing Wheel Turbulence Intervention

Additional strakes were introduced to further isolate the central diffuser channels from wheel-wake contamination. Figure 63 compares velocity contours at $x = -0.60$ m between the revised design (left) and the earlier configuration (right). In the new design, the extra dividers produced a more energised and uniform flow, with reduced low-velocity zones. This demonstrates that the additional strakes conditioned the inflow more effectively and minimised lateral bleeding.

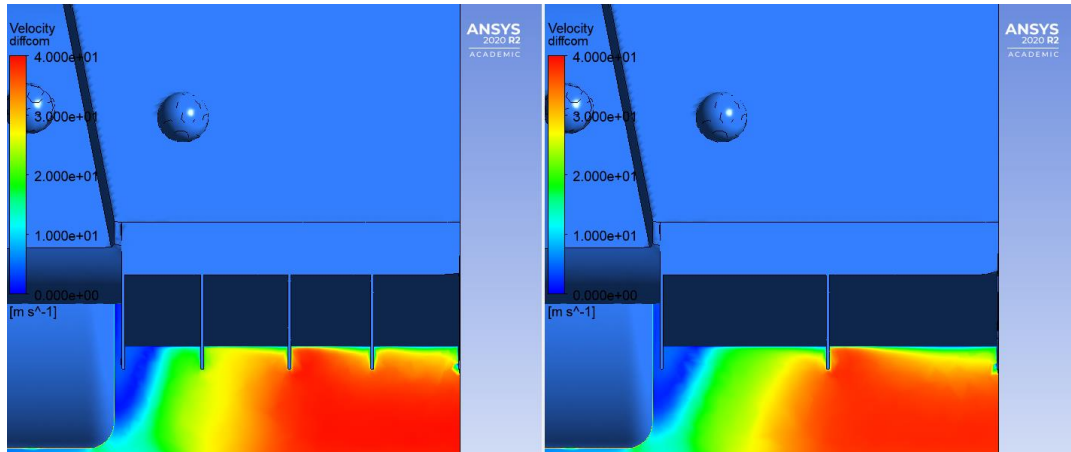


Figure 61 compares the velocity distribution at $x = -0.60$ m for two diffuser layouts: the new design with additional strakes (left) and the original design with fewer strakes (right).

Figure 64 shows the velocity field at $x = -0.40$ m, where the revised configuration produced smaller low-velocity pockets compared to the earlier diffuser. Figure 65 compares velocity contours at 0.4 of the vehicle width, with the revised design producing a more energised exit flow.

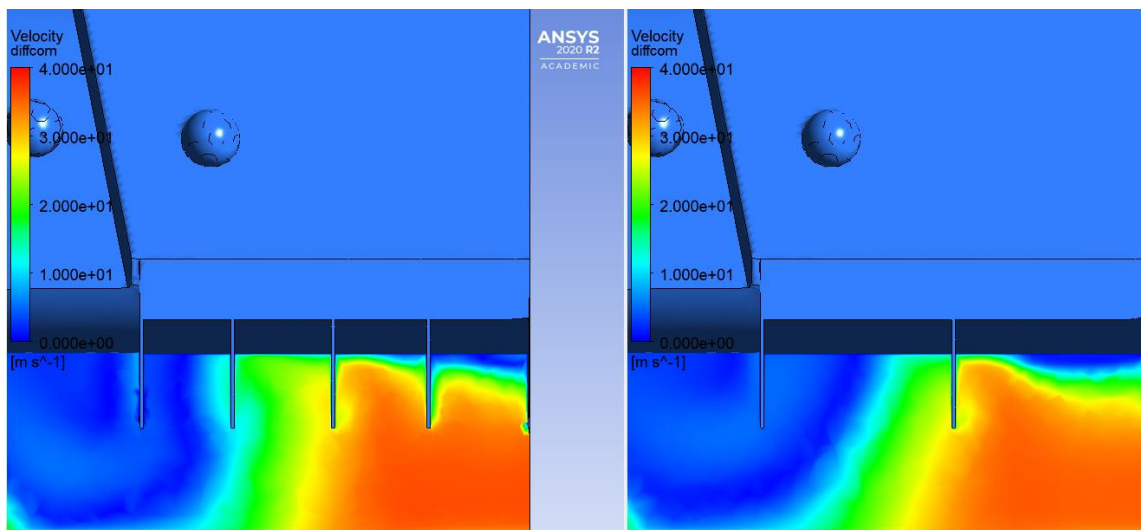


Figure 62 compares the velocity contours at $x = -0.40$ m for the new diffuser with additional strakes (left) and the original diffuser with fewer strakes (right).

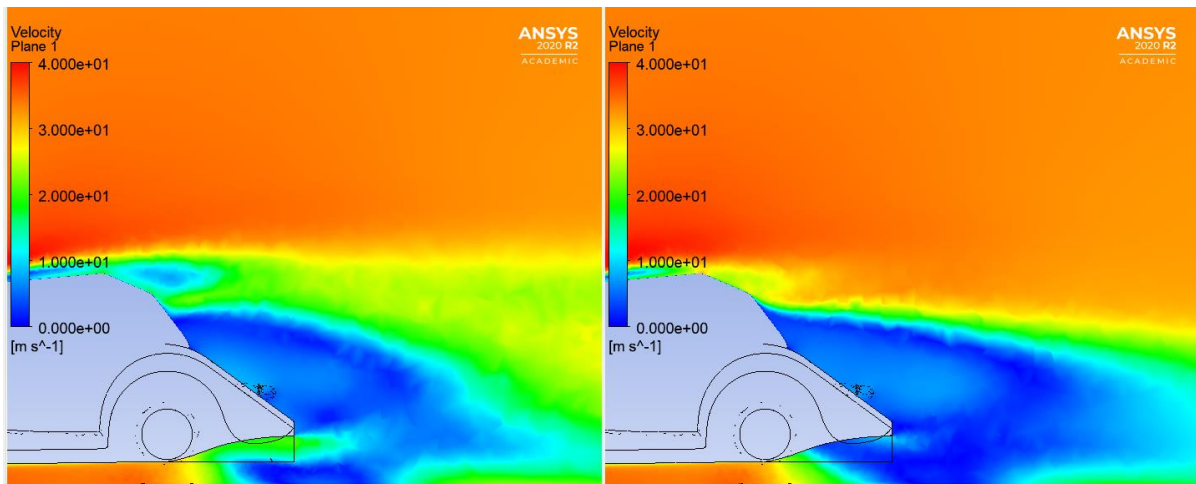


Figure 63 compares the velocity contours at 0.4 of the car width for the new diffuser with additional strakes (left) and the original diffuser with fewer strakes (right).

The revised diffuser configuration with additional strakes generated a lift coefficient of $C_l = -0.0222$, representing a significant improvement relative to the baseline $C_l = 0.0577$. This transition from a positive lift state to net downforce demonstrated the effectiveness of improved underbody partitioning. However, further refinement is required to balance this gain against drag penalties.

6.10 Centre Diffuser Profile Change

The final modification in this sequence was a reshaping of the diffuser centre profile. Figure 66 compares velocity contours for the baseline design (top left) and the revised centre profile (top right), with the lower image showing the velocity difference between the two.

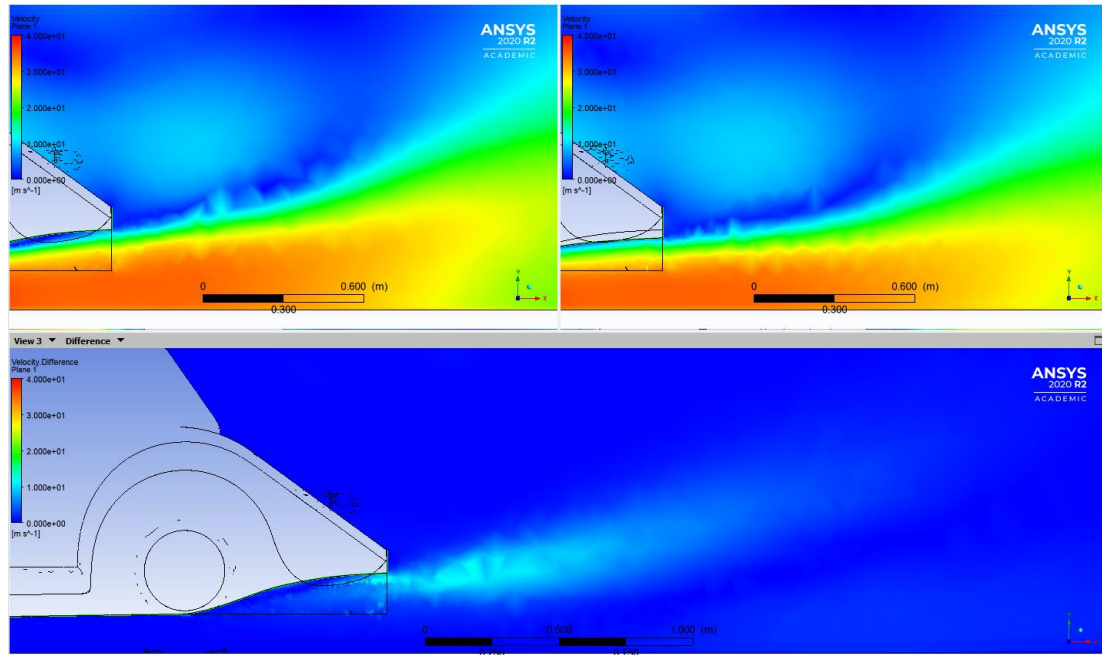


Figure 64 compares the velocity field for the baseline diffuser (top left) and the revised design with a reshaped centre profile (top right), with the lower image showing the velocity difference between the two cases, at centre of car.

The revised design incorporated a modified central ramp intended to maintain flow attachment for longer. The velocity contours demonstrated a more continuous high-speed band, with reduced low-velocity regions compared to the baseline. The difference plot confirmed these improvements, highlighting sustained higher velocity along the diffuser centreline and reduced wake development.

This change indicated improved flow stability in the core of the diffuser channel, where attachment was maintained more effectively. The reshaped centre profile therefore contributed to greater underbody suction and reduced separation losses.

7.0 Underbody Flow Management

7.1 Complete Flat Floor

CFD analysis of the baseline Morgan Plus Four highlighted that the underfloor region suffered from low velocity, particularly at the leading section of the floor (Figure 65). This behaviour was primarily caused by the alignment of the front wheel arch with the chassis baseline, which obstructed airflow entry beneath the car. As a result, most of the freestream was diverted around the sides and over the upper bodywork, leaving the underbody starved of mass flow. The outcome was the formation of a stagnation zone beneath the forward floor, which reduced diffuser effectiveness and prevented the generation of significant ground-effect downforce.

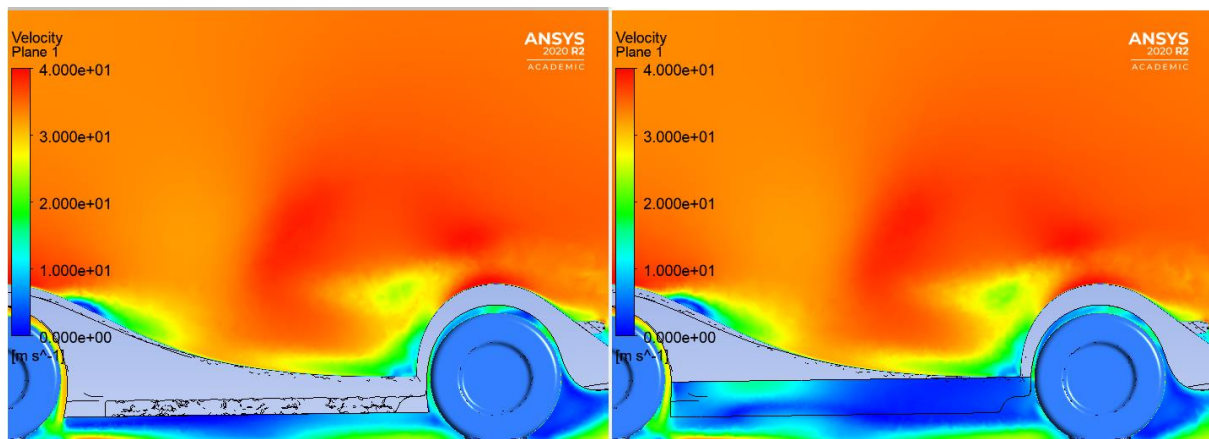


Figure 65 Underbody floor flow

Figure 65 compares velocity contours beneath the vehicle for the baseline configuration (left) and the redesigned flat floor (right). The redesigned floor shows a noticeable reduction in positive lift and is working closer to the floor. This improvement reduces the extent of stagnation at the front floor and increases the potential for ground-effect suction.

7.2 Louvre Design on Wheel Arches

Further CFD analysis of the Morgan Plus Four wheel arches revealed the accumulation of low-velocity, high-pressure flow inside the wheel wells. As shown in Figure 66, air within the rear arch became almost stagnant relative to the freestream, particularly at the upper rear section. This trapped flow created a pressure differential that contributed to rear axle lift and increased drag as it spilled around the tyre edges and into the diffuser, reducing its effectiveness.

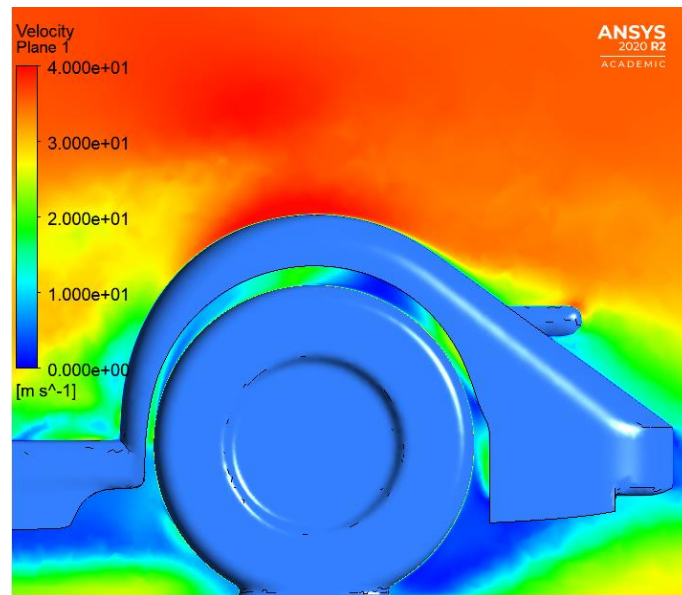


Figure 66 Rear Wheel low velocity build up

Installing louvres provides a controlled outlet for these high-pressure regions, venting them into the faster, lower-pressure external flow over the arches. This reduces axle lift and stabilises underbody flow. Race Louvers (2023) demonstrated through CFD and wind-tunnel testing that venting at the rear of wheel arches increases downforce with negligible drag penalty, with the most effective results achieved using medium-sized vents combined with a notched liner feed.

For the Morgan Plus Four, CFD suggested that the most effective louvre positions were at the upper rear of both front and rear arches, where pressure build-up was most pronounced (Figure 67).

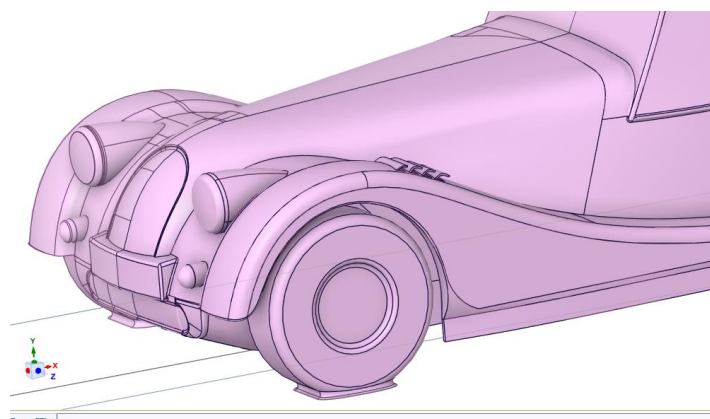


Figure 67 Basic Louvre design

Modelling the louvres in SpaceClaim presented challenges, as the thin blade profiles and cut-outs introduced intersecting surfaces within the bodywork. These caused non-manifold edges and overlapping geometry, which in turn led to meshing errors. Repeated repair and simplification

of the CAD were required to achieve watertight geometry. Even with these corrections, local skewness around the louvre blades remained problematic during meshing, demonstrating how small aerodynamic details can complicate CFD pre-processing.

The results showed that louvres did not provide measurable aerodynamic benefits in the front wheel arches, since flow in these regions did not generate significant pressure build-up that required venting. By contrast, the rear arches showed greater potential, as low-velocity air was consistently trapped within the cavity. Although full rear-louvre development could not be completed within the project timeframe, their inclusion represents a promising strategy for reducing pressure accumulation and improving diffuser efficiency in future work.

7.3 Conclusion

The underbody flow investigations demonstrated that the baseline Morgan Plus Four suffers from limited airflow beneath the chassis due to blockage at the front floor and pressure build-up within the wheel arches. Flattening the underfloor increased mass flow into the diffuser and reduced the extent of stagnation at the front of the chassis, providing a more energised stream for underbody suction.

Louvre designs were also assessed as a strategy to relieve high-pressure regions inside the arches. While the front arches showed little benefit, the rear arches exhibited significant low-velocity, high-pressure flow that disrupted diffuser performance. CFD confirmed that louvres at the upper rear arch positions offered the greatest potential for reducing pressure accumulation and improving wake stability, although detailed development was not completed within the project timeframe.

Overall, the flat floor configuration and rear louvre concepts represent effective approaches for conditioning the underbody flow. These results highlight the importance of clean diffuser feeding and reduced wheel-wake contamination, both of which directly influence rear-axle load and vehicle stability. The next stage of development therefore focused on front-end flow management through splitter optimisation, ensuring that the underbody could be fully utilised to generate downforce while maintaining regulatory compliance.

8.0 Splitter Design

The splitter plays a critical role in conditioning airflow entering the underbody and balancing aerodynamic load distribution between the front and rear of the car. Literature confirms that splitters generate downforce by creating a high-pressure region on the upper surface and a corresponding low-pressure region beneath, which effectively suctions the nose of the vehicle downwards (Salhan, 2020). The trade-off is that this pressure differential also increases drag, meaning splitter design must achieve a compromise between stability and efficiency.

Studies on splitter overhang length have shown that forward extension improves downforce but with diminishing returns beyond approximately 30 mm (Bhattacharjee et al., 2019). Wright and Al-Rawi (2023) also emphasised that splitter ride height has a strong influence on performance, with excessively low clearances causing wake disruption and reduced efficiency, whereas moderate ground clearance achieved the most favourable balance of downforce and drag. These findings highlight the need for the Morgan splitter to be sized for effective front load generation without excessive drag penalty, and positioned at a ride height consistent with the rake and suspension geometry defined in Section 4.0.

More advanced solutions, such as the inflatable splitter tested by Szudarek et al. (2022), demonstrated the benefits of dynamically reducing ground clearance to maximise suction at speed. Although impractical for this application, this reinforces the importance of maintaining close ground proximity under dynamic conditions to maximise splitter effectiveness.

8.1 Splitter Effect on Diffuser

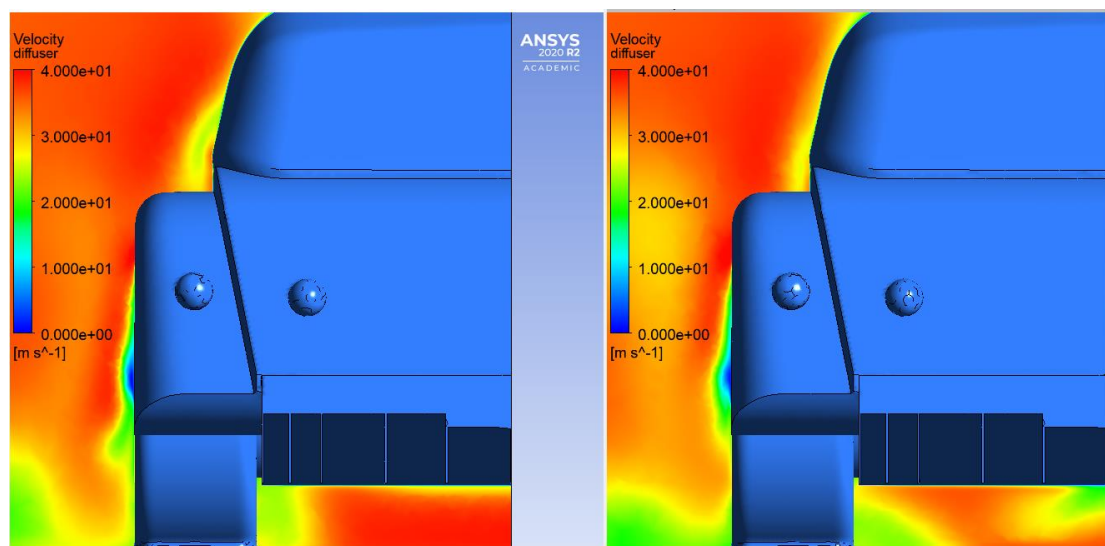


Figure 68 flow before entering diffuser . (left no splitter, right has a splitter)

CFD results in Figures 68 and 69 showed that flow velocity at the diffuser entry and throat was reduced when a splitter was added. Instead of accelerating smoothly into the diffuser, the air decelerated, reducing suction and lowering rear axle downforce.

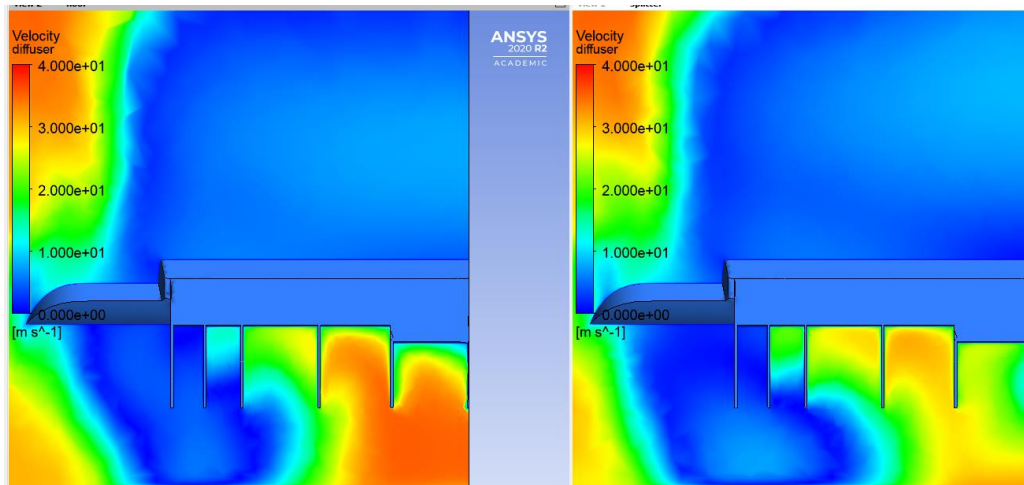


Figure 69 flow at diffuser at 0.05m, showing lower velocity due to splitter. (left no splitter, right has a splitter)

This behaviour was linked to the sharp change in angle between the flat floor and diffuser ramp, which made boundary-layer attachment difficult (Figure 70).

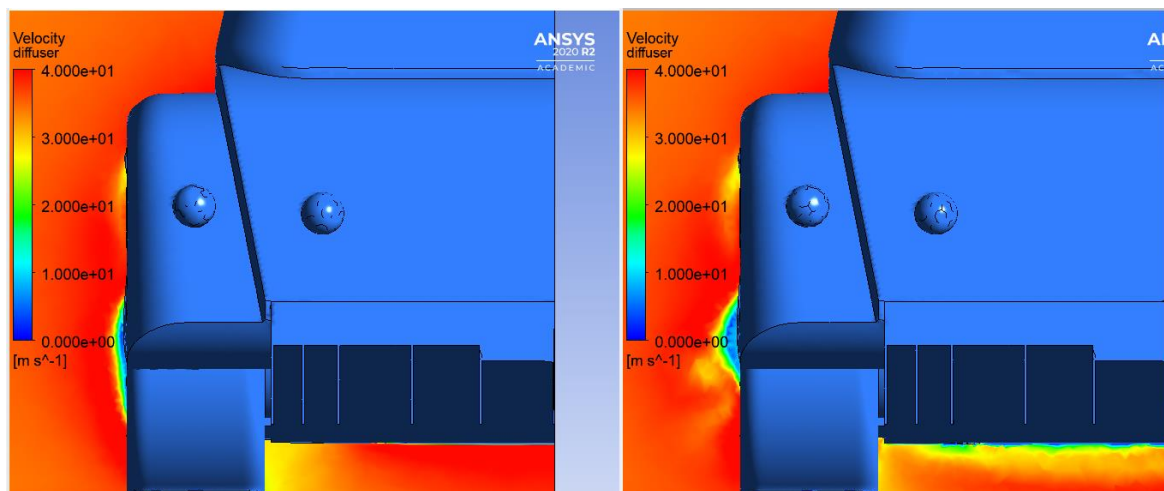


Figure 70 change in angle between the flat floor and the diffuser ramp

To improve diffuser performance, the underside requires a smoother transition into the ramp so that airflow can remain attached and maintain higher velocities, allowing stronger pressure recovery downstream.

8.2 Splitter Analysis

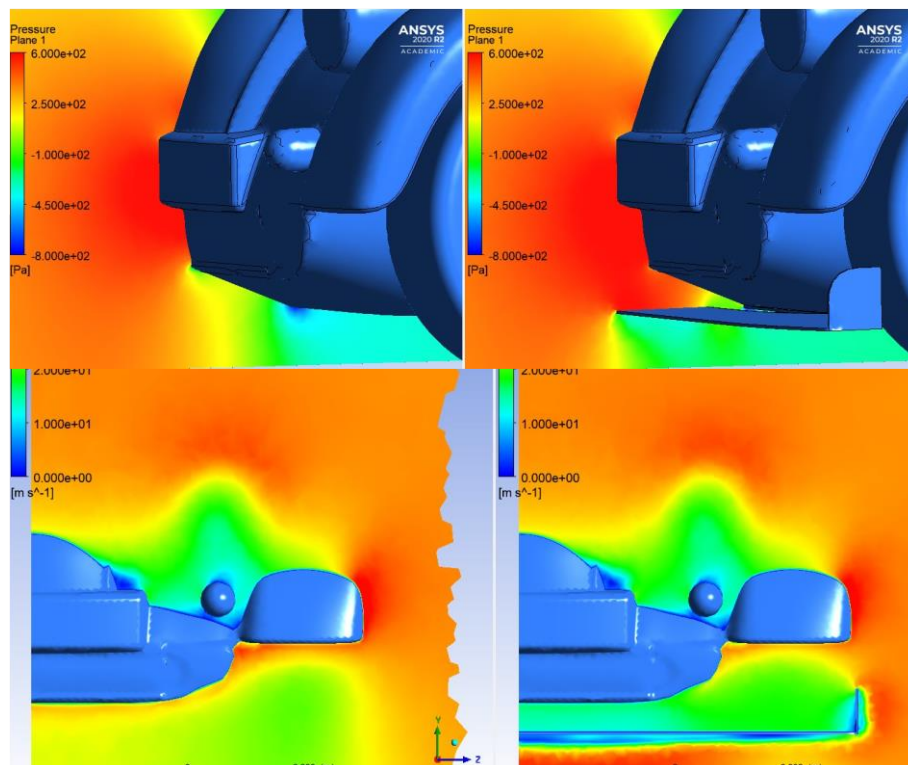


Figure 71 CFD velocity comparison of the Morgan front end without a splitter (left) and with the smoother leading-edge splitter (right).

Figure 71 highlights the stagnation bubble and local deceleration formed at the splitter nose. The sharp-edged design (right) created a steep adverse pressure gradient and small separation pockets, which spilled beneath the car and reduced the momentum feeding the floor. A smoother leading edge reduced this effect, confirming the importance of splitter nose shaping in managing inflow to the underbody.

8.3 Splitter Design Two Analysis

The second splitter concept incorporated a central opening intended to channel flow directly into the diffuser, with the aim of improving underfloor mass flow and pressure recovery (Figure 72).

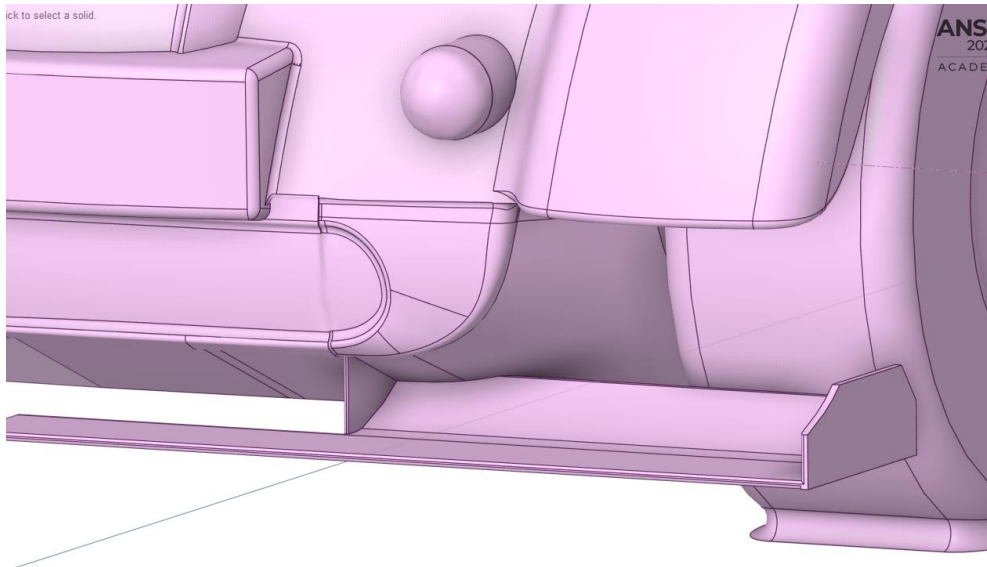


Figure 72 Splitter Design two, with smooth profile

The diffuser-only case produced a lift coefficient of $C_l = -0.0451$, demonstrating that the diffuser alone was able to generate a meaningful level of underbody downforce. When combined with the splitter featuring a central opening, the lift coefficient decreased further to $C_l = -0.0540$. However, this configuration reduced diffuser effectiveness, which required further analysis.

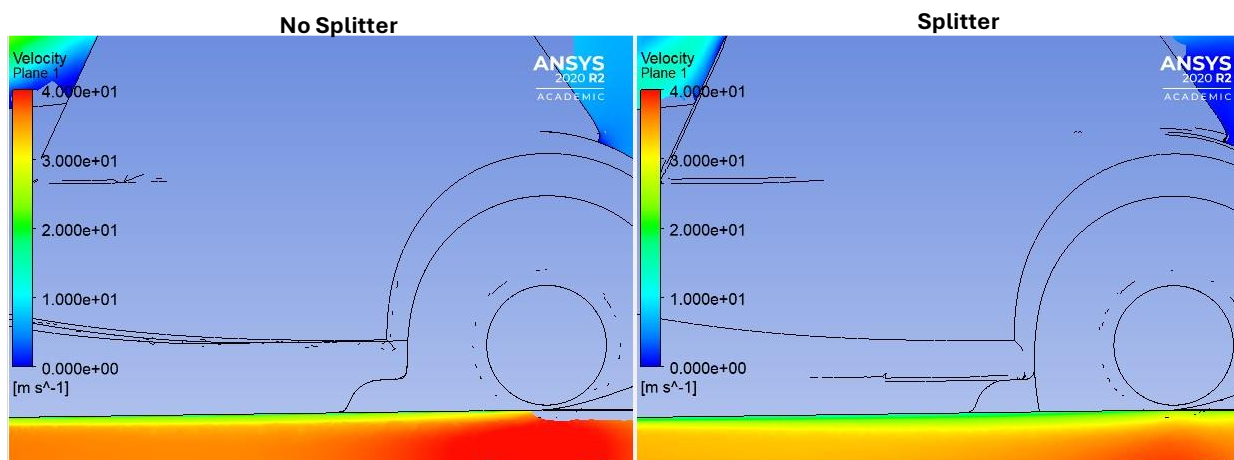


Figure 73 Velocity contour comparison beneath the Morgan Plus 4 with and without open splitter design

As shown in Figure 73, the no-splitter case allowed underfloor flow to accelerate uniformly, providing a relatively consistent velocity profile into the diffuser. In contrast, the splitter with the central opening disrupted this acceleration, resulting in lower velocity magnitudes immediately downstream.

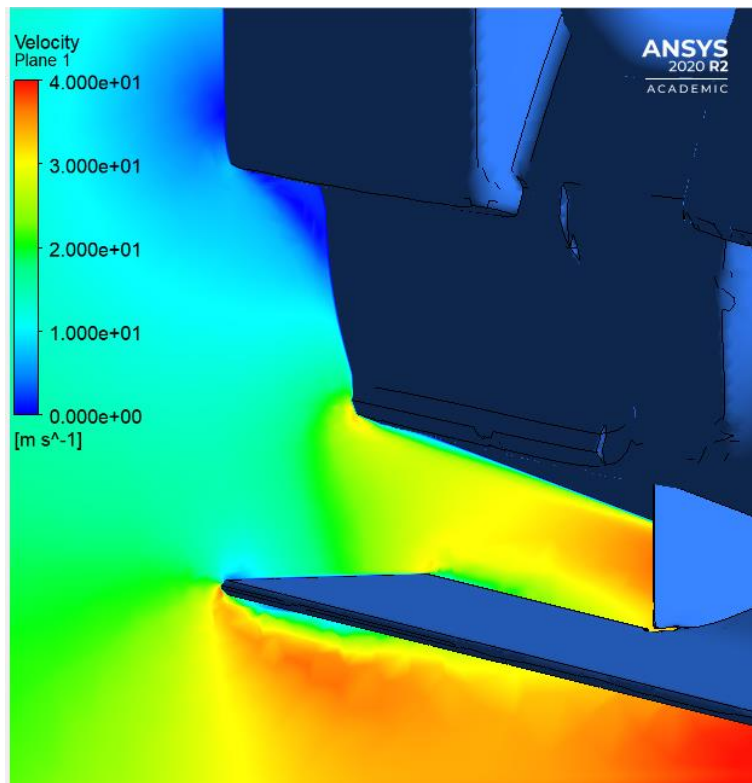


Figure 74 Velocity contours showing reduced splitter efficiency due to central hole design, leading to diminished front downforce

The opening reduced splitter efficiency by allowing high-energy flow to bleed through the cut-out rather than being forced underneath. This weakened the pressure differential between the upper and lower surfaces, reducing suction and diminishing the splitter's contribution to front-end downforce.

Although refinements were attempted, the central opening design consistently failed to deliver significant improvements in downforce generation and offered no clear advantage over the diffuser-only case. Consequently, the concept was deemed unviable for the Morgan Plus Four, and development was redirected towards alternative splitter geometries capable of delivering more stable aerodynamic performance.

8.3.1 Conclusion

The splitter studies confirmed that this device plays an important role in conditioning the airflow entering the underbody and balancing aerodynamic load between the front and rear axles. A sharp-edged splitter design created local separation and stagnation bubbles at the nose, which reduced momentum feeding into the underfloor and weakened overall performance. A smoother leading-edge profile reduced these effects, highlighting the importance of splitter shaping in maintaining flow quality.

The second design, incorporating a central opening, aimed to improve diffuser feeding but instead reduced underfloor acceleration and weakened the pressure differential across the splitter. As a result, the concept generated less suction on its underside and proved ineffective in sustaining downforce. Although refinements were attempted, the central opening design did not outperform the diffuser-only case and was therefore abandoned.

Overall, the results demonstrated that the splitter can contribute to front-end stability when carefully integrated, but aggressive or poorly positioned designs risk disrupting underbody flow and diminishing diffuser performance. Future splitter iterations should therefore prioritise smooth leading-edge geometry, appropriate overhang length, and ride height positioning to maximise benefits while avoiding unnecessary drag or rear-end performance loss.

8.4 Airfoil Splitter Design

Research into splitter development highlights the advantage of using an inverted aerofoil-shaped profile instead of a simple flat plate. The most commonly referenced geometry is the NACA 4412 section (Figure 75), which has been successfully applied in CFD studies on stock cars. When inverted and run at 0° angle of attack, the NACA 4412 significantly accelerated airflow beneath the splitter, increasing suction under the nose and generating higher front-axle downforce compared to a flat splitter of equal projection (Wrobel et al., 2017).

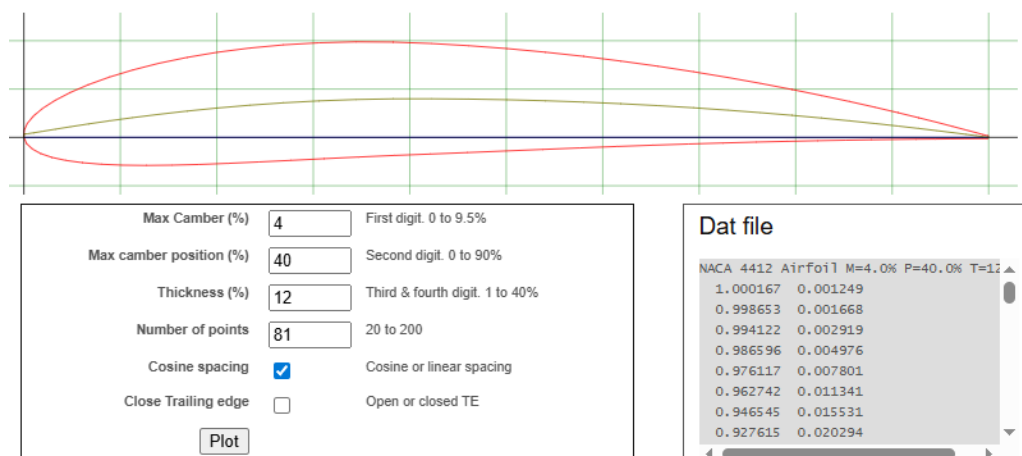


Figure 75 NACA 4412 profile (Airfoil Tools, 2024)

The advantages of an aerofoil-type splitter are clear. Firstly, the curved leading edge promotes smoother flow attachment, reducing stagnation and separation that typically occur with sharp-edged flat plates. Secondly, the cambered profile accelerates underbody flow, lowering pressure and enhancing downforce. Finally, cambered sections are aerodynamically more efficient than

flat plates at comparable Reynolds numbers, producing greater lift (or downforce when inverted) for the same frontal area (Pelletier and Mueller, 2000).

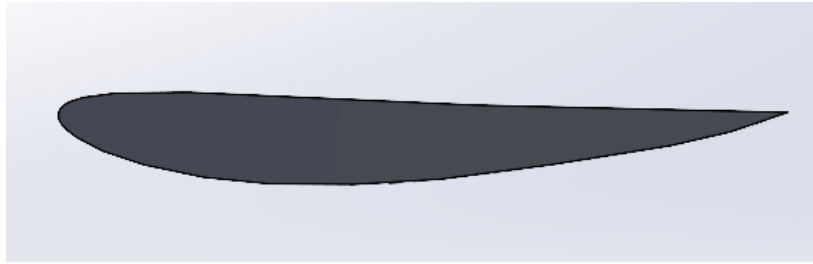


Figure 76 70 NACA 4412 profile recreated in SolidWorks

In summary, the inverted NACA 4412 was identified as the most suitable starting point for the Morgan splitter. It combined validated performance from prior research with ease of manufacture and compliance with Motorsport UK regulations.

8.4.1 Airfoil Results Analysis

The inverted NACA 4412 study demonstrated that the aerofoil-based splitter produced significant downforce while maintaining attached flow beneath the nose (Figure 77).

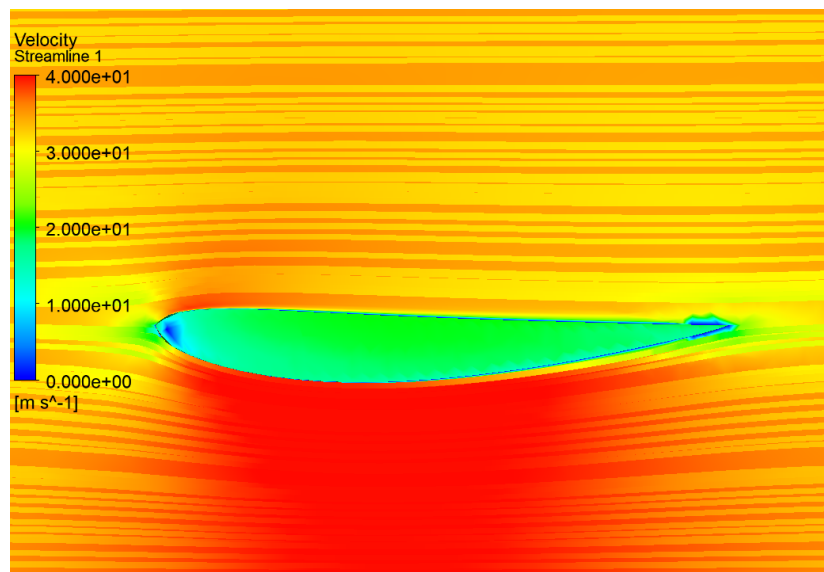


Figure 77 Airfoil profile. Streamlines

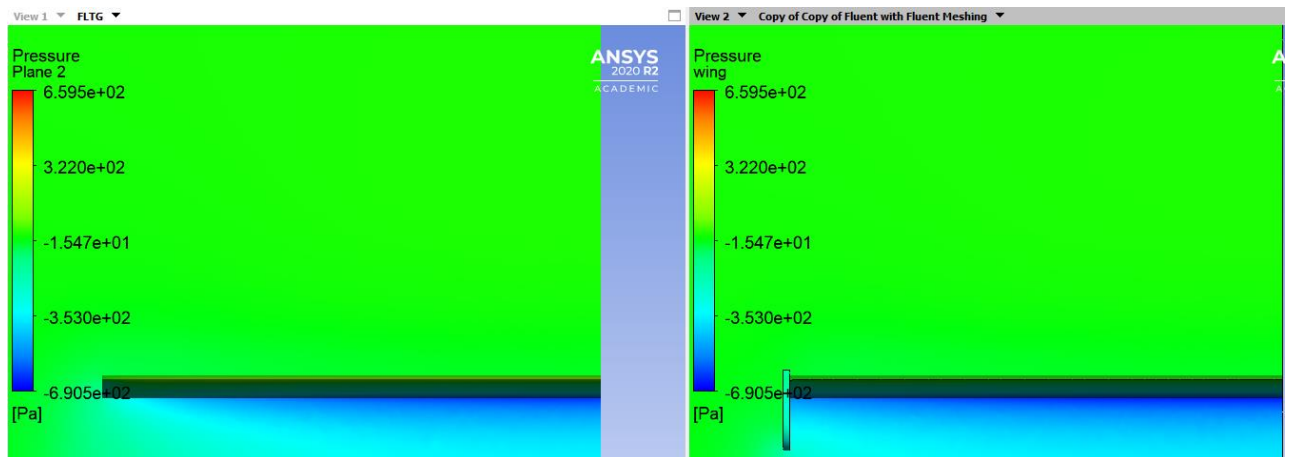


Figure 78 Effect of splitter Ends

Splitter end conditions were found to influence performance strongly. As shown in Figure 78, sealed ends reduced flow spillage, sustaining low pressure along the underside of the profile. This increased downforce generation but came at the cost of drag, which rose by 12.2%. The sealed-end configuration produced an increase in negative lift of 0.439. This highlighted the aerodynamic trade-off between greater downforce and increased drag.

8.4.2 Effect of Ride Height

Literature confirms that ride height has a decisive influence on splitter performance, with low but controlled clearances producing the most favourable aerodynamic balance (Soares, 2018; Szudarek, 2022; Um, 2021). Applying this to the Morgan, the aerofoil splitter was expected to be highly sensitive to ride height, since its curved profile accelerates flow most effectively when run closer to the ground. Lowering the splitter increased suction at the nose and boosted front axle load. However, excessively low clearance risked restricting diffuser inflow, reducing rear downforce and limiting overall efficiency.

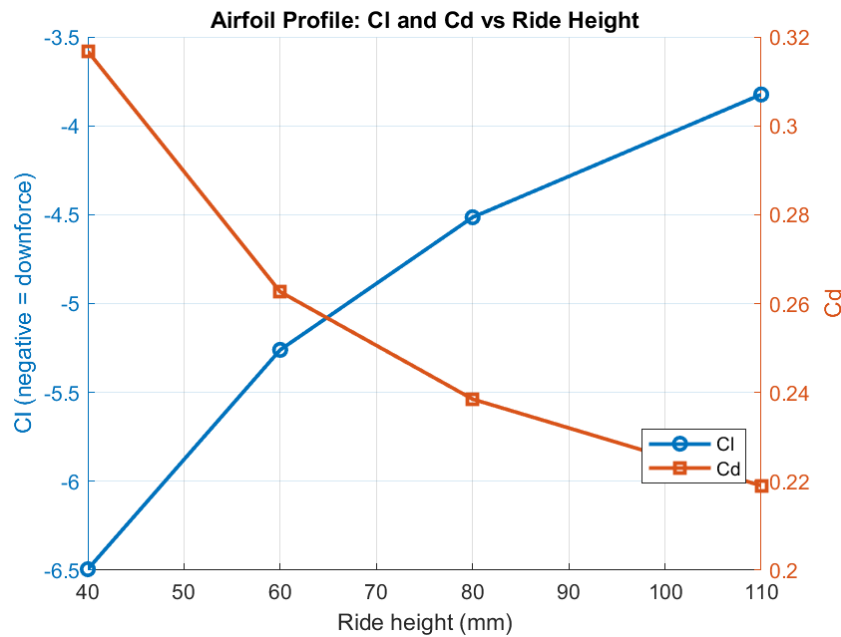


Figure 79 Airfoil profile. Cl and Cd vs ride height

As shown in Figure 79, the inverted NACA 4412 exhibited strong ride-height sensitivity. Reducing clearance from 110 mm to 40 mm shifted the lift coefficient from $Cl = -3.82$ to $Cl = -6.49$, confirming that lower ride height enhanced ground-effect suction and generated significantly greater downforce. This behaviour was consistent with findings by Džijan et al. (2019), who reported that proximity to the ground accelerates underfloor flow and strengthens suction beneath the body. However, this gain in downforce was accompanied by a drag increase, with Cd rising from 0.22 to 0.32 over the same ride height range. This reflected the aerodynamic trade-off consistently identified in splitter and underbody studies (Bhattacharjee et al., 2019; Salhan, 2020).

8.4.3 Splitter Conclusion

Several redesign iterations of the aerofoil splitter were tested, including open and closed configurations under rake conditions. The closed airfoil design achieved $Cd = 0.446$ and $Cl = -0.147$, while the open design produced $Cd = 0.450$ and $Cl = 0.095$. The closed case therefore delivered a 0.9% reduction in drag and a direct improvement in lift behaviour, shifting from a positive lift state to meaningful downforce. The absolute change in lift coefficient was 0.242, representing a clear transition from generating lift to producing downforce.

The results also indicated that the aerofoil splitter was more effective at generating downforce than the diffuser, even when the diffuser was provided with clean upstream flow. While diffuser performance depended heavily on pressure recovery and inflow quality, the splitter consistently

provided larger and more immediate aerodynamic gains. This justified finalising the closed aerofoil splitter configuration, which offered genuine front-end downforce with only a modest drag penalty.

8.5 Diffuser Analysis and Redesign After Splitter

With the splitter now acting as the dominant underbody device, the diffuser required re-evaluation to ensure that it could manage the altered upstream conditions. Redesigning the diffuser geometry was necessary to achieve effective pressure recovery and to maximise the efficiency of the overall underbody system. The revised concept adopted a sharper kick point, reducing overall diffuser length in line with available packaging constraints. Strakes were initially omitted in order to analyse the natural flow behaviour before introducing additional flow-management features.

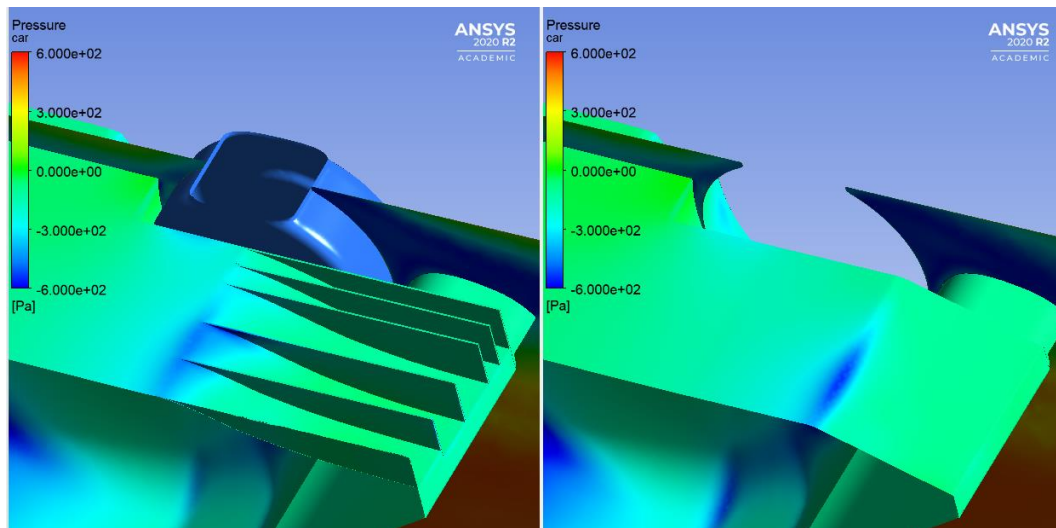


Figure 80 KickPoint redesign (Left) old diffuser design (Right) New design

Figure 80 shows that the revised diffuser with a sharper kick point generated lower pressure beneath the underbody compared with the earlier design, suggesting that the new configuration offered potential for enhanced suction in the throat region.

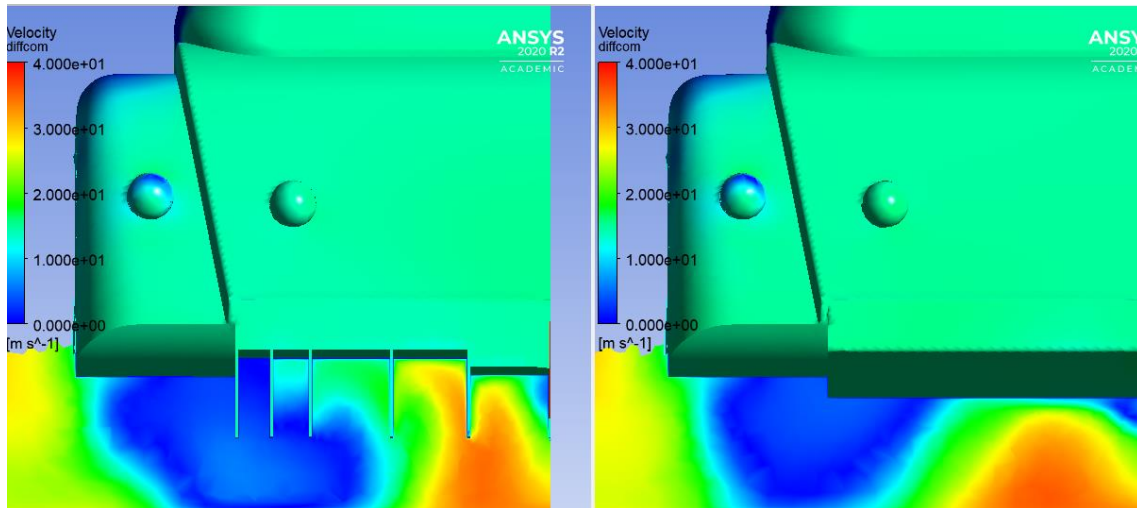


Figure 81 (Left) old diffuser design (Right) New design . Velocity plot at rear diffuser

As shown in Figure 81, flow management challenges persisted near the rear wheels, with significant low-velocity zones continuing to bleed into the diffuser. Central underbody flow also required improved attachment. These observations indicate that strakes will need to be introduced more strategically to stabilise the core diffuser channel and reduce cross-flow contamination.

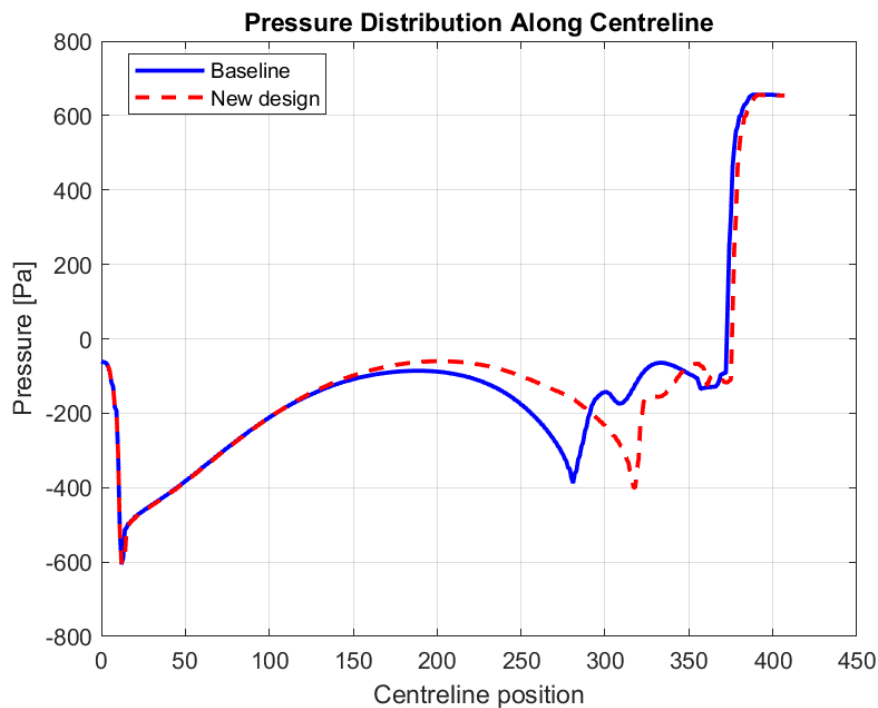


Figure 82 Pressure distribution along car at centerline

The baseline diffuser exhibited a sharp suction peak at approximately 300 mm along the centreline, followed by irregular and unstable pressure recovery (Figure 82). This behaviour

reflected flow separation and partial reattachment, reducing aerodynamic efficiency. The redesigned diffuser also experienced separation, but the suction peak was less extreme and recovery more progressive, producing a smoother overall profile.

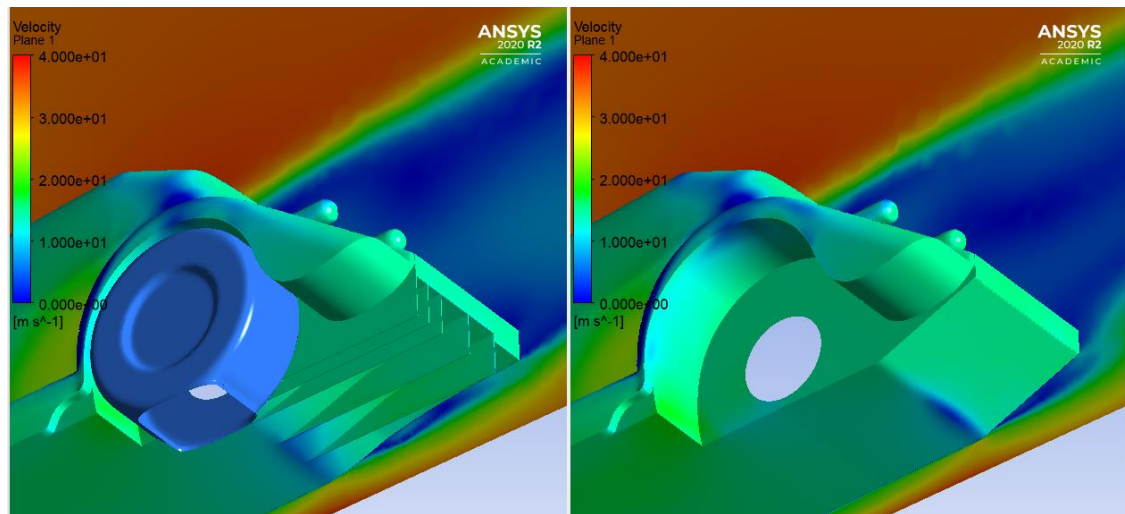


Figure 83 velocity plot. flow separation still occurs. (Left) old diffuser design (Right) New design

Velocity contours in Figure 83 reinforced the pressure results. In the baseline configuration, flow accelerated rapidly through the throat before breaking down into extensive recirculation along the ramp. This generated a fragmented wake and clear evidence of cross-flow contamination. The redesigned diffuser still displayed separation, but the recirculation zone was shorter and weaker, with more uniform acceleration and a cleaner exit flow. These results suggest improved pressure recovery and reduced wake turbulence, even though full attachment was not achieved.

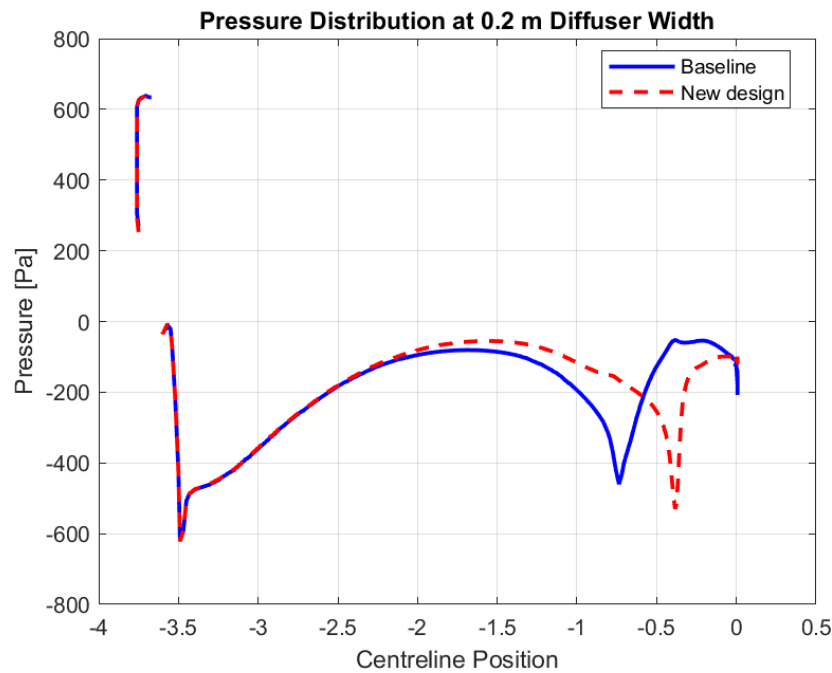


Figure 84 Pressure distribution along car at 0.2m.

At 0.2 m from the centreline (Figure 84), the redesigned diffuser produced a deeper suction peak than the baseline, confirming stronger downforce potential. However, the sharper recovery towards the exit indicated an increased risk of separation under adverse operating conditions.

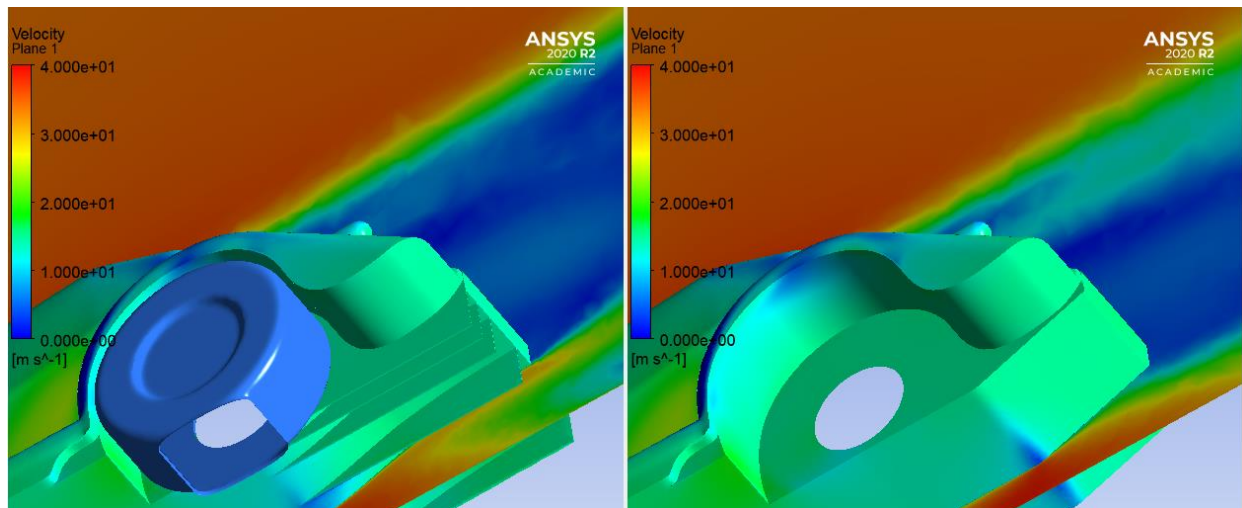


Figure 85 Velocity Plot at 0.2m. (Left) old diffuser design (Right) New design

The velocity contours at 0.2 m (Figure 85) supported this interpretation. The baseline diffuser maintained a smoother but weaker velocity field, while the redesigned version accelerated flow more aggressively along the ramp, generating stronger underbody suction but with evidence of localised distortion near the outlet. This trade-off suggests that the new diffuser improved

aerodynamic performance, but at the cost of greater sensitivity to ride height and transient flow variations.

8.5.1 Conclusion

The redesigned diffuser demonstrated clear improvements over the baseline geometry, producing a smoother pressure recovery profile and a more coherent exit flow. Although separation was still present, the recirculation zones were shorter and less severe, indicating greater aerodynamic stability. The deeper suction peaks observed confirmed that the new diffuser had increased downforce potential. However, the sharper kick point introduced greater sensitivity to ride height and operating conditions, which may compromise consistency in transient flow states.

Overall, the results suggest that while the redesigned diffuser improves underbody suction, further refinement is required to maintain attachment in the outer flow channels and to manage wheel-wake contamination. The integration of strakes in future iterations is expected to address these limitations by stabilising boundary-layer development and reducing cross-flow interference.

9.0 Air Dam Design with Flat Splitter

9.1 Solution Problems Due to New Geometry

Warped faces are a common issue when generating extruded prism layers around curved geometries such as the splitter, underfloor and diffuser. If left uncorrected, the Green–Gauss node-based gradient assumption treats these faces as planar, which introduces errors in wall-adjacent gradients. This results in oscillations in the force coefficients, slower convergence and higher uncertainty in the averaged drag and lift predictions.

To address this, Warped-Face Gradient Correction (WF-GC) was applied. By re-projecting gradients onto the true warped geometry, WF-GC improved the accuracy of face normals and fluxes. This produced smoother residuals and reduced the high-frequency jitter in force coefficients, as illustrated in Figure 86, where the corrected solution shows stable convergence compared to the uncorrected case. The method therefore ensured that the aerodynamic predictions were reliable, while only introducing a minimal increase in computational cost.

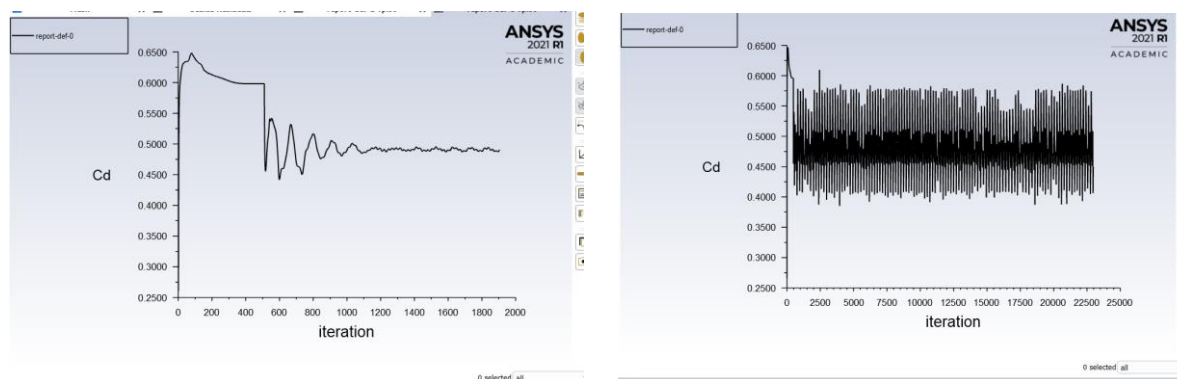


Figure 86 Effect of using warped face gradient correction. left(with) right(without)

9.2 Flat Splitter and Air Dam Analysis

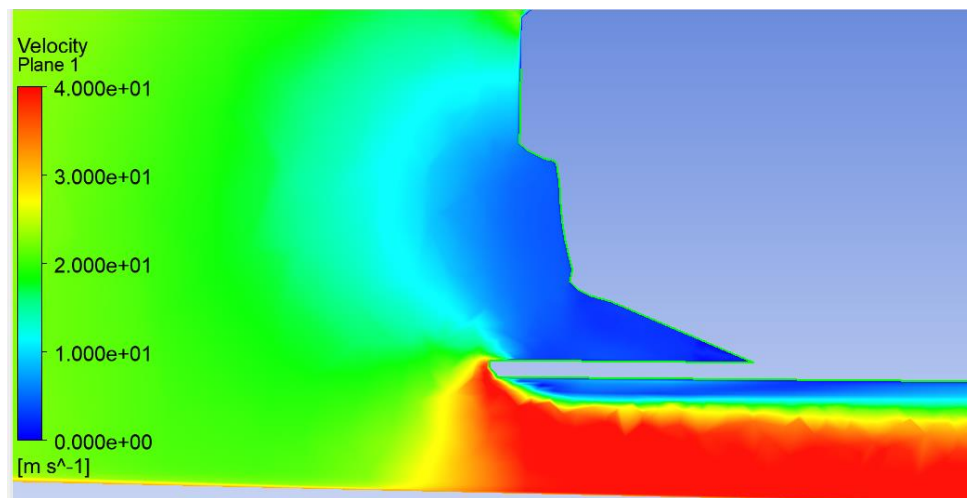


Figure 87 Splitter and Air dam Analysis

The pressure contours in Figure 88 show a high-pressure stagnation region forming on the underside of the splitter. This created the pressure differential needed for downforce, but also contributed to form drag. A steep gradient was visible along the vertical bodywork, where rapid pressure drop increased the risk of boundary-layer separation at the splitter edge. As highlighted in earlier studies, such separation can disrupt diffuser momentum recovery and reduce underbody efficiency (Cooper et al., 1998; Marklund, 2013).

To mitigate this, refinements such as a rounded leading edge or the addition of vertical strakes could smooth pressure recovery and stabilise vortical structures. Literature supports these strategies: Unni et al. (2017) and Nebot et al. (2025) demonstrated that strakes energise the boundary layer and delay separation, improving downstream diffuser performance.

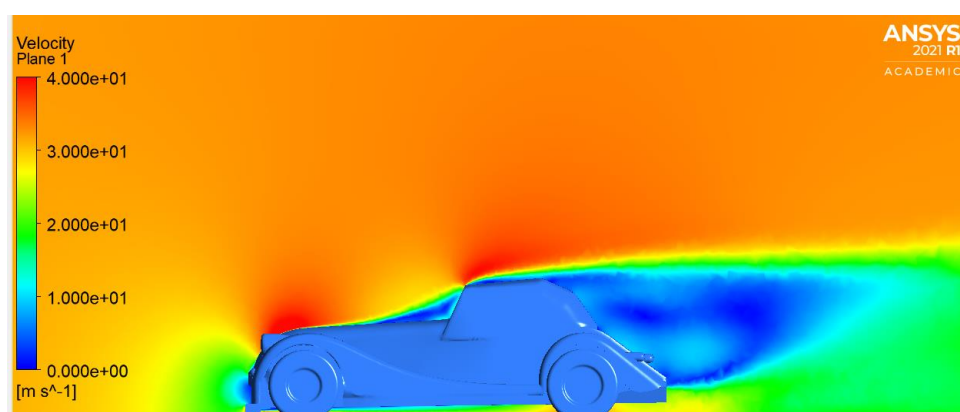


Figure 88 Wake region of flat splitter and air dam design

When the air dam and splitter were combined with the original diffuser profile and the updated strake geometry from Section 8.5, drag increased from $C_d = 0.4675$ to $C_d = 0.4904$, a 4.9% rise. However, the lift coefficient shifted dramatically from $C_l = +0.0689$ to $C_l = -0.2729$, representing a 496% swing and converting positive lift into substantial downforce. This trade-off highlights a key aerodynamic principle: a modest drag penalty can deliver a disproportionately large gain in stability through downforce. For the Morgan, such an improvement in high-speed balance is expected to outweigh the small loss in efficiency.

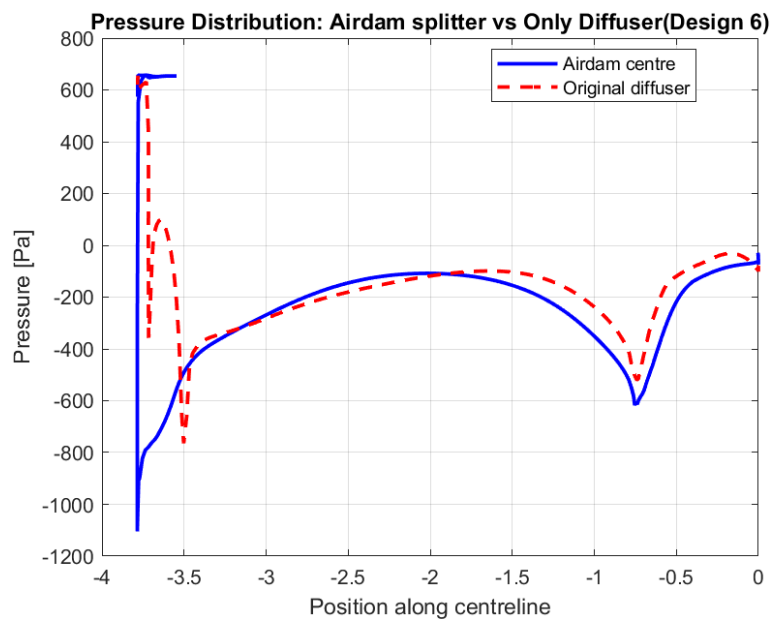


Figure 89 Centerline pressure distribution. Air dam splitter vs diffuser design 6

Figure 89 shows the pressure distribution beneath the Morgan for the new configuration compared with Diffuser 6 (used as the baseline due to its similar profile). A sharp pressure drop is observed at the splitter leading edge, followed by gradual recovery along the flat floor and a secondary low-pressure region at the diffuser throat. Importantly, the diffuser produced greater negative lift when combined with the splitter and air dam than when operating alone, confirming that the splitter promoted a positive aerodynamic interaction with the downstream flow.

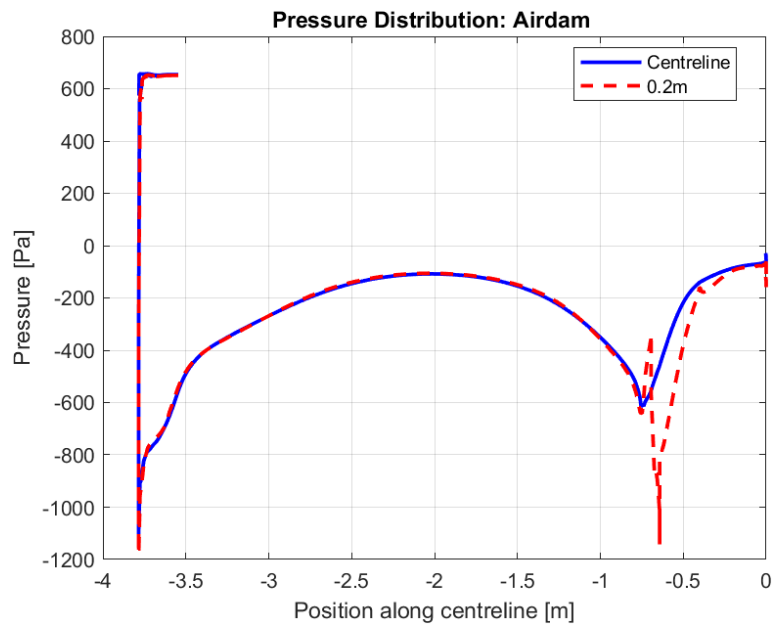


Figure 90 Pressure Distribution at centerline and 0.2m

Figure 90 compares the pressure distribution along the vehicle centreline and at 0.2 m from the centreline. Both traces show the same overall behaviour, confirming that the air dam effect was consistent across the span. At 0.2 m, the pressure minimum in the diffuser throat was slightly deeper, suggesting that the revised strakes helped to delay separation by re-energising the boundary layer.

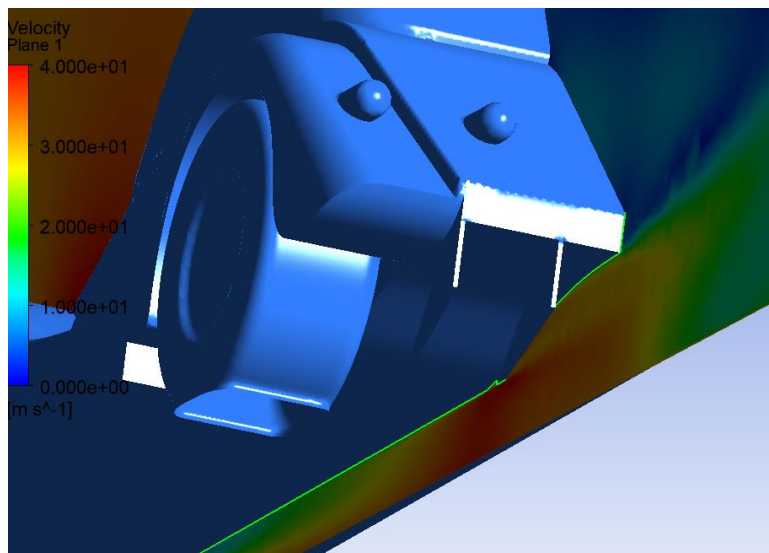


Figure 91 Velocity plot at 0.2m(width) near strakes

Velocity contours in Figure 91 support this finding, showing that flow accelerated most strongly adjacent to the strakes. The corresponding lower static pressure generated additional downforce,

confirming that the strakes were not only guiding the flow but also actively contributing to underfloor suction.

9.2.1 Leading-Edge Redesign

A modified splitter with a new leading edge was also tested (Figure 92). This configuration exhibited repeated convergence behaviour, with average values of $C_d = 0.4959$ and $C_l = -0.2436$ after 2500 iterations. However, the oscillatory behaviour suggested numerical artefacts rather than reliable aerodynamic performance. As a result, this design was excluded from further development, since the unstable convergence prevented accurate assessment of its aerodynamic characteristics.

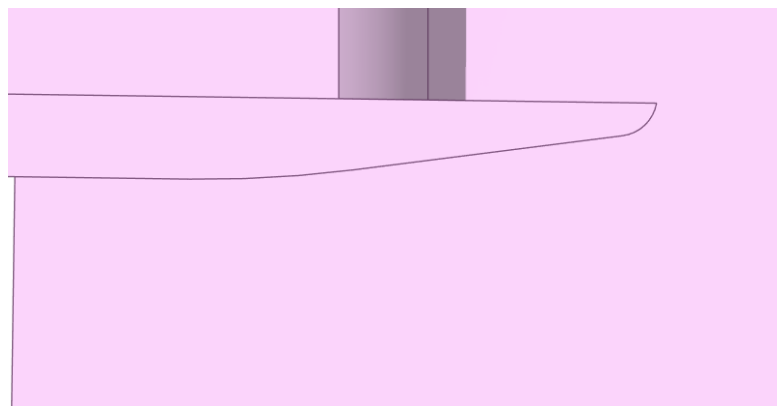


Figure 92 Leading edge redesign

10.0.0 Conclusion

The project aimed to improve the underbody aerodynamics of the Morgan Plus Four while confirming that a validated CFD process could be transferred from reference models to a production vehicle. Benchmarking against the DrivAer model and cross-checking with experimental pressure taps showed drag within $\pm 3\%$ and reproduced the main surface pressure distribution, giving confidence that the method was reliable for application to the Morgan.

In its baseline state the Morgan recorded $C_d = 0.4785$ and $C_l = +0.0577$, about 3% higher in drag than experiment and with net lift present. This reflected an underbody prone to diffuser stall and limited upstream flow conditioning. Iterative development improved the situation: diffuser re-profiling combined with strakes delayed stall and produced net downforce, with Diffuser 7 reaching $C_l = -0.0222$, a 138% swing compared to the baseline lift figure. These results are in line with Katz (1995), who highlighted ramp geometry as critical to stall onset, though suction levels were lower than those reported by Buscariolo et al. (2020) on simplified geometries due to Morgan's packaging limits.

Strakes proved effective in stabilising the central diffuser channel, supporting Unni et al. (2017), who noted the role of vortex structures in sustaining attachment. Splitter iterations gave smaller improvements, with Splitter 2 reducing lift to $C_l = -0.0540$, but their effect depended on diffuser stability. A flat floor reduced drag to $C_d = 0.4537$, almost identical to experimental values, in agreement with Marklund's (2013) findings on the importance of expansion ratio and ride height. The largest changes were achieved by the rake and nose redesigns. The rake case cut drag by 6.8% relative to baseline ($C_d = 0.446$) while producing moderate downforce ($C_l = -0.147$). The nose redesign generated $C_l = -0.2436$, a 320% swing from lift to downforce. These findings align with Zhang et al. (2022), who stressed that front-rear coordination is needed to sustain suction and limit wake growth.

It should be noted that the pressure coefficient (C_p) values were processed incorrectly. Although they highlighted relative performance trends, the absolute numbers cannot be considered reliable indicators of pressure recovery. Correct processing will be required in future to strengthen the connection between surface pressures and force coefficients.

Beyond the numerical improvements, these aerodynamic gains carry clear engineering implications. The conversion from positive lift to net downforce would enhance high-speed stability, reduce sensitivity to crosswinds, and improve driver confidence under braking and cornering. A 6–7% reduction in drag would translate to greater efficiency, improving fuel economy

in road use and lowering lap times in a motorsport setting. In this way, the findings extend beyond coefficient changes to practical performance benefits relevant to both heritage road cars and competitive applications.

Overall, the objectives set at the beginning of the project were achieved: the CFD workflow was validated, applied successfully to the Morgan geometry, and used to develop feasible underbody modifications that delivered meaningful aerodynamic improvements without compromising the vehicle's heritage identity.

10.1 Limitations and Issues

The main limitations of the study relate to packaging, experimental resolution, and data processing. The rear-exit exhaust and suspension geometry restricted diffuser length and expansion ratio, preventing the adoption of shallower ramps closer to the 1.2–1.5 ratios reported by Gopalarathnam et al. (2019) as optimal. Validation against pressure taps was valuable but offered limited resolution compared to full wind tunnel testing, making it difficult to fully capture local separation behaviour and wake growth.

The incorrect processing of C_p data reduced the accuracy of pressure recovery analysis. While useful for trend comparison, the absolute values could not be relied upon, weakening the link between surface pressure distribution and coefficient outcomes.

10.2 Further Work

Future work should address these issues through corrected pressure analysis, extended testing, and exploration of new aerodynamic concepts. A structured programme of rake–diffuser optimisation is recommended, with corrected C_p mapping and diffuser angle sweeps to map out stable operating envelopes. This should be supported by wind-tunnel campaigns capable of testing ride-height sensitivity, yaw, and transient wake development.

Packaging constraints should also be reconsidered. Relocating or reconfiguring the exhaust and suspension would release underbody volume, allowing longer and lower-angle diffusers to be tested. Beyond packaging, the exhaust itself offers aerodynamic potential. Exhaust-assisted diffuser blowing should be investigated, since directed jets can energise the boundary layer, delay stall, and lift base pressure at critical points (Gopalarathnam et al., 2019). Recent studies confirm that jet placement and characteristics alter drag, lift, and wake structure, supporting a controlled programme on the Morgan with fixed mass flow, temperature, and aim angle toward the ramp (Zaareer et al., 2023; Plogmann et al., 2023). Earlier work on underbody and cooling jets

also showed that rear-exit jets can improve pressure recovery and reduce drag (Lai et al., 2010; Sovran et al., 1986), reinforcing the case for measured trials aligned with the rake–ride matrix.

Finally, Motorsport UK Blue Book rules limit splitter projection and underfloor extensions. Homologation-compliant geometries should therefore be tested to establish the balance between aerodynamic benefit and regulatory acceptance.

In summary, the next stage should refine C_p analysis, pursue systematic rake–diffuser testing, expand experimental validation, and evaluate exhaust-assisted diffuser blowing under real packaging and regulatory constraints. These steps are essential to move the proof of concept into practical application for industry and motorsport.

11.0.0 References

Airfoil Tools (2024) *NACA 4-Digit Airfoil Series*. Available at: <http://airfoiltools.com/airfoil/naca4digit> (Accessed: 28 August 2025).

Abood, M.S., et al. (2024) 'Ground effect of an inverted double element wing–diffuser system', *Heliyon*.

Abood, M.S., Hashim, F.M. and Hussein, A.M. (2024) 'Ground effect of an inverted double-element wing–diffuser system', *Heliyon*, 10(5), e32717. <https://doi.org/10.1016/j.heliyon.2024.e32717>

Ahmad, N.E., Abo-Serie, E. and Gaylard, A. (2010) 'Mesh Optimization for Ground Vehicle Aerodynamics', www.cfdlissres.net, 2(1).

ANSYS Inc. (2012) *Mesh Quality Recommendations*. ANSYS Fluent Release 14.0 Documentation. ANSYS Support Portal.

ANSYS Inc. (2024) *ANSYS Fluent Theory Guide*. Release 2024 R1. Canonsburg, PA: ANSYS Inc., Section 6.3.2.

ANSYS Innovation Space (2025) 'Local Sizing with Bol and Fol'. *ANSYS Innovation Courses*, Lesson 2 and Lesson 6. Accessed July 2025.

Bhattacharjee, S., Arora, B.B. and Kashyap, V. (2019) 'Optimization of race car front splitter placement using CFD', SAE Technical Paper 2019-01-5097.

Buscariolo, F.F., Assi, G.R.S. and Sherwin, S.J. (2020) 'Computational study on an Ahmed Body equipped with simplified underbody diffuser', *Journal of Wind Engineering & Industrial Aerodynamics*, 206, 104375.

Buscariolo, F., D'Aero, L., Greco, C. and Di Bernardino, A. (2020) 'CFD Investigation on Aerodynamic Interaction Between Diffuser and Rear Wing in Ground Effect', SAE Technical Paper 2020-01-0657. <https://doi.org/10.4271/2020-01-0657>

Cadence (2023) 'Fidelity Pointwise and IsimQ Mesh Adaptation for Accurate Aero Vehicle Drag Prediction'. Available at: <https://resources.system-analysis.cadence.com/blog/fidelity-pointwise-and-isimq-mesh-adaptation-for-accurate-aerovehicle-drag-prediction> (Accessed: 14 August 2025).

Cooper, K.R., Bertenyi, T., Dutil, G., Syms, J. and Sovran, G. (1998) 'The aerodynamic performance of automotive underbody diffusers', SAE Technical Paper 980030.

Džijan, I., Nizić, D. and Rujnić-Sokele, M. (2019) 'Aerodynamic Forces Acting on a Race Car for Various Ground Clearances and Rake Angles', *Tehnički vjesnik – Technical Gazette*, 26(1), pp. 14–19. <https://doi.org/10.17559/TV-20180406111242>

Ferziger, J.H. and Perić, M. (2002) *Computational Methods for Fluid Dynamics*. 3rd edn. Berlin: Springer.

Fotiadis, I., Skaperdas, E. and Iordanidis, G. (2024) *CFD Mesh Strategies for DrivAer Model Variants*. BETA CAE conference presentation. Available at: https://www.beta-cae.com/events/c6pdf/3B_2_Fotiadis.pdf (Accessed: 14 August 2025).

Gopalarathnam, A., Taamallah, S., Murugappan, A. and Oweis, G. (2019) 'A Review of Ground-Effect Diffuser Aerodynamics', *ASME Journal of Fluids Engineering*, 141(2), 020802. <https://doi.org/10.1115/1.4041768>

Guerrero, A., Vilar, J., Llopis, E. and García, A. (2022) 'A Numerical Aerodynamic Analysis on the Effect of Rear Diffusers on an SUV', *Applied Sciences*, 12(8), 3763. <https://doi.org/10.3390/app12083763>

Heft, A., Indinger, T. and Adams, N. (2012) *Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations*. SAE Technical Paper 2012-01-0168.

Heft, A., Indinger, T. and Adams, N.A. (2012) 'Introduction of a new realistic generic car model for aerodynamic investigations', SAE Technical Paper 2012-01-0168. <https://doi.org/10.4271/2012-01-0168>

Heft, A., Indinger, T. and Adams, N.A. (2012) 'Introduction of a new realistic generic car model for aerodynamic investigations', SAE Technical Paper 2012-01-0168. <https://doi.org/10.4271/2012-01-0168>

Howell, J. and Potts, I. (2004) 'Automotive Aerodynamics: The MIRA Reference Car', *Proceedings of the MIRA International Vehicle Aerodynamics Conference*. Nuneaton: MIRA Ltd.

IET Intelligent Transport Systems (2024) 'CFD study of a passenger car'. *IET Intelligent Transport Systems* (accepted 13 October 2020). Wiley.

Irigaray, A. (2024) 'Comparison of adaptive mesh refinement strategies for automotive CFD applications', *Computers & Fluids*, 275, 106034. <https://doi.org/10.1016/j.compfluid.2024.106034>

Kang, S.O., Park, J.H., Park, H., Kim, Y. and Kim, C. (2012) 'Active Translating Rear Diffuser for Drag Reduction', SAE Technical Paper 2012-01-0173. <https://doi.org/10.4271/2012-01-0173>

Katz, J. (1995) *Race Car Aerodynamics: Designing for Speed*. Cambridge, MA: Bentley Publishers.

Katz, J. (1995) *Race Car Aerodynamics: Designing for Speed*. Cambridge, MA: Bentley Publishers.

Kays, W.M. and London, A.L. (1998) *Compact Heat Exchangers*. 3rd edn. New York: McGraw-Hill.

Khalighi, M., Shost, M., Iaccarino, G. and Hamman, C. (2012) 'Unsteady aerodynamics and flow control for a realistic vehicle underbody', SAE Technical Paper 2012-01-0170. <https://doi.org/10.4271/2012-01-0170>

Kumar, M.J., Dubey, A., Chheniya, S. and Jadhav, A. (2013) 'Effect of vortex generators on aerodynamics of a car: CFD analysis', *International Journal of Innovation in Engineering and Technology (IJJET)*. Available at: https://www.researchgate.net/publication/316855385_Effect_of_Vortex_generators_on_Aerodynamics_of_a_Car_CFD_Analysis

Lai, C.G., Chen, H., Gao, Z. and Zhang, B. (2010) 'Investigation of aerodynamic performance due to the underbody and engine-cooling exit flow', SAE Technical Paper 2010-01-0290. <https://doi.org/10.4271/2010-01-0290>

Lanfrit, M. (2005) *Best Practice Guidelines for Handling Automotive External Aerodynamics with FLUENT*. Fluent Deutschland GmbH.

Launder, B.E. and Spalding, D.B. (1974) 'The numerical computation of turbulent flows', *Computer Methods in Applied Mechanics and Engineering*, 3(2), pp. 269–289. [https://doi.org/10.1016/0045-7825\(74\)90029-2](https://doi.org/10.1016/0045-7825(74)90029-2)

Łomakin, V.F., Kudo, K. and Oleinikova, M. (2018) 'Polyhedral meshing in numerical analysis of conjugate heat transfer', *EPJ Web of Conferences*, 180, 02096.

Marklund, J. (2013) *Under body and Diffuser Flows of Passenger Vehicles*. Chalmers University.

Marklund, J. (2013) *Under-body and Diffuser Flows of Passenger Vehicles*. Master's Thesis. Chalmers University of Technology. Available at: <http://publications.lib.chalmers.se/records/fulltext/177909/177909.pdf>

Mears, A.P. (2004) *The Aerodynamic Characteristics of an Exposed Racing Car Wheel*. PhD thesis. Loughborough University. <https://www.academia.edu/113732638>

Menter, F.R. (1994) 'Two-equation eddy-viscosity turbulence models for engineering applications', *AIAA Journal*, 32(8), pp. 1598–1605. <https://doi.org/10.2514/3.12149>

Menter, F.R. (1994) 'Two-equation eddy-viscosity turbulence models for engineering applications', *AIAA Journal*, 32(8), pp. 1598–1605. <https://doi.org/10.2514/3.12149>

MIRA (2007) *Aerodynamic Wind Tunnel Facilities Users' Handbook*. www.mira.co.uk.

Nebot, E.P., Duran, J., Bonilla, J. and Valero, E. (2025) 'Investigating the Impact of Structural Features on F1 Car Diffuser Performance using CFD', *Mathematics*, 13(9), 1455. <https://doi.org/10.3390/math13091455>

Pinto, C.P., Marimuthu, S., Rajendran, P. and Rajadurai, M. (2025) 'Computational analysis of blended winglet designs to reduce the wake turbulence on the Airbus A380 wingtip', *Energy*, 6(4), 68.

Pishey, K. (2020) *Wheel Envelope Definition and Analysis for Vehicle Architecture Packaging*. MSc thesis. Chalmers University of Technology. <https://odr.chalmers.se/handle/20.500.12380/301154>

Plogmann, J., Stauffer, C., Dimopoulos Eggenschwiler, P. and Jenny, P. (2023) 'URANS simulations of vehicle exhaust plumes with insight on remote emission sensing', *Atmosphere*, 14(3), 558. <https://doi.org/10.3390/atmos14030558>

Qin, P., Ricci, A. and Blocken, B. (2024) 'CFD simulation of aerodynamic forces on the DrivAer car model: Impact of computational parameters', *Journal of Wind Engineering and Industrial Aerodynamics*, 248, 105711. <https://doi.org/10.1016/j.jweia.2024.105711>

Race Louvers LLC (2023) *Fender Louver CFD Analysis & Wind Tunnel Report*. Available at: <https://racelouvers.com/content/Race-Louvers-Fender-Vent-CFD.pdf>

Rejniak, R. and Gatto, A. (2020) 'Influence of rotating wheels and moving ground use on unsteady wake', *Flow, Turbulence and Combustion*, 106(1), pp. 109–137. <https://doi.org/10.1007/s10494-020-00180-8>

Renda, G. (2022) *Comparison of different turbulence models for the numerical simulation of the flowfield around the DrivAer*. Master's thesis. Politecnico di Milano. POLItesi.

Rijns, S., Hurst, R., Lewis, R. and Dominy, R. (2023) 'Aerodynamic Analysis of the Ride Height Dependency of a Multichannel Diffuser', SAE Technical Paper 2023-01-5064. <https://doi.org/10.4271/2023-01-5064>

Rumsey, C.L. and Ying, S.X. (2002) 'Prediction of high lift flows using turbulence models', *Progress in Aerospace Sciences*, 38(2), pp. 145–180. [https://doi.org/10.1016/S0376-0421\(01\)00020-9](https://doi.org/10.1016/S0376-0421(01)00020-9)

Saeed, S., Saleem, M. and Niazi, A. (2019) 'Effect of Mesh Quality on CFD Results for an Airfoil', *International Journal of Science and Research (IJSR)*, 8(4), pp. 1405–1408.

Salhan, P. (2020) *Computational Study of the Front-end Downforce Enhancing Aerodynamic Elements in Sports Cars*.

Santner, C., Fasel, H. and Lang, M. (2012) 'Low-Profile Vortex Generators in an S-shaped Diffuser', *ASME Journal of Turbomachinery*, 134(1), 011022. <https://doi.org/10.1115/1.4003251>

Shih, T.H., Liou, W.W., Shabbir, A., Yang, Z. and Zhu, J. (1995) 'A new $k-\epsilon$ eddy-viscosity model for high Reynolds number turbulent flows—model development and validation', *Computers & Fluids*, 24(3), pp. 227–238. [https://doi.org/10.1016/0045-7930\(94\)00032-T](https://doi.org/10.1016/0045-7930(94)00032-T)

Sosnowski, M., Krzywanski, J., Grabowska, K. and Gnatowska, R. (2018) 'Polyhedral meshing in numerical analysis of conjugate heat transfer', *EPJ Web of Conferences*, 180, 02096.

Sosnowski, M., Krzywanski, J., Grabowska, K. and Gnatowska, R. (2018) 'Polyhedral meshing in numerical analysis of conjugate heat transfer', *EPJ Web of Conferences*, 180, 02096. <https://doi.org/10.1051/epjconf/201818002096>

Sovran, G., Morel, T. and Mason, W.T. (1986) 'Car underside, upperbody and engine-cooling system interactions—contributions to aerodynamic drag', SAE Technical Paper 860212. <https://doi.org/10.4271/860212>

Strangfeld, C., Wieser, D., Schmidt, H. J., Woszidlo, R., Nayeri, C. and Paschereit, C. (2013) 'Experimental Study of Baseline Flow Characteristics for the Realistic Car Model DrivAer', SAE Technical Paper 2013-01-1251. <https://doi.org/10.4271/2013-01-1251>

Strangfeld, C., Wieser, D., Schmidt, H.-J., Woszidlo, R., Nayeri, C. and Paschereit, C. (2013) 'Experimental study of baseline flow characteristics for the realistic car model DrivAer', SAE Technical Paper 2013-01-1251. <https://doi.org/10.4271/2013-01-1251>

Strangfeld, S., Wassen, E. and Krämer, E. (2016) 'Aerodynamic simulation of the DrivAer model', *Automotive and Engine Technology*, 1(1), pp. 43–58. <https://doi.org/10.1007/s40322-016-0074-3>

Szudarek, M., Kamieniecki, K., Tudruj, S. and Piechna, J. (2022) 'Towards Balanced Aerodynamic Axle Loading of a Car with Covered Wheels – Inflatable Splitter', *Energies*, 15(15), 5543.

Unni, T.P.A., Mathur, A. and Jindal, S. (2017) 'Numerical Investigation on Aerodynamic Effects of Vanes and Flaps on a Diffuser', SAE Technical Paper 2017-01-2163. <https://doi.org/10.4271/2017-01-2163>

Vilén, M. (2017) *Determining Tyre Clearance Using Modelling Clay*. Bachelor's thesis. Metropolia University of Applied Sciences. <https://www.theseus.fi/handle/10024/124169>

Wang, X., Khalighi, B. and Iaccarino, G. (2010) 'Large-eddy simulation of automotive near-wall aerodynamics', *ASME Journal of Fluids Engineering*, 132(12), 121103. <https://doi.org/10.1115/1.4002892>

Wright, C. and Al-Rawi, M. (2023) 'One-Way Fluid Structure Interaction of a Utility Vehicle Splitter using CFD Modelling', *CFD Letters*, 15(12), pp. 34–43. <https://doi.org/10.37934/cfdl.15.12.3443>

Yamashita, T., Sano, Y., Hori, A. and Asanuma, T. (2018) 'Effects of Moving Ground and Rotating Wheels on Aerodynamic Drag of a Two-Box Vehicle', SAE Technical Paper 2018-01-0730. <https://doi.org/10.4271/2018-01-0730>

Yang, J., Zhang, H., Li, X. and Yang, Z. (2020) 'Separation control in a conical diffuser with Kármán-VG', *International Journal of Thermal Sciences*, 149, 106194. <https://doi.org/10.1016/j.ijthermalsci.2019.106194>

Yue Wang (2019) 'How can I understand mesh quality and simulation results in CFD?' *CFD Online / ResearchGate*.

Zaareer, M.N.M., Mourad, A.-H.I., Darabseh, T., Khan, S. and Elgendi, M. (2023) 'Impact of a vehicle exhaust pipe position on the lift and drag coefficients: 2D and 3D simulations', *International Journal of Thermofluids*, 18, 100321. <https://doi.org/10.1016/j.ijft.2023.100321>

Zhang, T., Chen, J., Xu, M., Zhao, Y. and Yang, Y. (2022) 'Numerical investigation on the aerodynamic characteristics of a sports car with a rear diffuser', *Energy Reports*, 8, pp. 2026–2036.

Zhang, Z., Yu, H., Chen, G. and Wang, W. (2022) 'Joint Research on Aerodynamic Characteristics and Handling Stability of Racing Car Under Different Body Attitudes', SAE Technical Paper 2022-01-0883. <https://doi.org/10.4271/2022-01-0883>