

University of Wales Trinity St Davids

Research Proposal



Author: Dylan Huw Davies

Supervisor: Andrew Noble

Student Number: 2100937

Course Title: Motorsport Engineering

Date: 2/03/25

Contents

1.0 Introduction	4
2.0 Industrial Relevance	5
3.0 Literature Review	6
3.1.1 Introduction to Underbody Aerodynamics	6
3.2 Computational Fluid Dynamics (CFD) Validation	9
3.2.1 Domain Positioning /size	11
3.2.2 Part mesh/volume mesh size	11
3.2.3 Prism layers	11
3.2.4 Density box.....	11
3.2.5 Turbulence models and wall functions	11
3.2.6 Movind road and wheels	12
3.3 Flow visualisations methods	12
4.0 Legislative and Environmental Considerations	15
4.1 Regulations.....	15
4.2 Environmental Considerations	15
4.3 Challenges & Solutions.....	15
5.0 Proposed Methodologies.....	17
5.1 Computational Validation Simulation Approach.....	17
5.2 Gathering real data using pressure taps	18
5.3 Methodology for the Redesign of the Underbody for the Morgan Plus 4.....	19
5.3.1 Flat Floor Implementation	19
5.3.2 Rear Diffuser Optimization.....	20
5.3.2 Venturi Tunnels for Ground Effect Enhancement	20
5.3.3 Side Skirts for Airflow Containment.....	20
5.3.4 Rake Angle Adjustments	21
5.3.5 Expected Performance Improvements	21
6.0 Results Collection & Analysis	22

6.1 Type of Data to be Collected	22
6.2 Quantitative vs. Qualitative Data	22
6.3 Expected Data Format & Processing	23
6.4 Metrics for Data Assessment	23
6.5 Potential Issues in Data Collection & Analysis	23
6.6 Summary of Results Collection & Analysis	24
7.0 Project Planning/Timescales	25
8.0 Evaluation	27
8.1 Limitations and Challenges	27
8.2 Contributions and Outcomes	27
8.3 Conclusion	27
9..0 Reference	28

Figures

Figure 1 Diffuser effect on wake region (Buljac ,Kozmar, 2016).....	6
Figure 2 Effect of diffuser angle on Cd and Cl (Hu, Zhang, Yan, Zhao, 2011).	7
Figure 3 Diffuser with dirty underbody (Verus Engineering ,2019)	7
Figure 4 Side skirts	8
Figure 5 Morgan Plus Four CX-T (Vinten, 2021)	8
Figure 6 Distribution of pressure of a car's underbody diffuser and a venturi tube (Guerrero; Castilla; Eid, 2022)	9
Figure 7 Ahmed body (Peng, Alessio 2024)	10
Figure 8 Mira Car (Palaskar, Kumar, 2016)	10
Figure 9 DrivAer (Renda, 2021).....	10
Figure 10 Streamline airflow (Simscale ,2023).....	12
Figure 11 wake of an Ahmed body at different yaw angles (Sciacchitano, 2019).	13
Figure 12 path lines are often used to illustrate airflow routes around the vehicle.....	14
Figure 13 Delaney or octree mesh from ANSYS Fluent	18
Figure 14 DrivAer models, F-Fastback, E-Estate, N-Notchback (Ashton, 2015)	18

Tables

Table 1 Gantt chart Timework flow	26
---	----

1.0 Introduction

Aerodynamics is a key factor in vehicle performance, particularly in motorsport, where optimizing airflow can enhance speed, stability, and efficiency. This research focuses on the underbody aerodynamic development of the Morgan Plus 4, a heritage-inspired sports car, to implement performance-enhancing modifications while complying with Motorsport UK regulations.

The study aims to investigate and apply aerodynamic improvements, including a flat floor, optimized diffuser, venturi tunnels, and side skirts. Using Computational Fluid Dynamics (CFD) simulations, the research will analyse the impact of these modifications on drag coefficient (C_d) and lift coefficient (C_l) to enhance aerodynamic efficiency and stability.

Past research has demonstrated the benefits of optimizing underbody airflow. Knowles found that a flat floor generates ground effect, accelerating airflow to reduce pressure and increase downforce (Ehirim, Knowles, Saddington, 2018). Hu highlighted the role of diffuser geometry in improving pressure recovery and reducing wake turbulence (Hu, Zhang, Yan, Zhao, 2011). Guerrero showed that venturi tunnels enhance aerodynamic efficiency while maintaining compliance with regulations (Guerrero, 2022). This study will adapt these principles to the Morgan Plus 4's design constraints.

Regulatory and environmental considerations will be key aspects of this research. Motorsport UK imposes restrictions on ground clearance, diffuser dimensions, and aerodynamic elements, which must be adhered to throughout the redesign process. Additionally, reducing drag aligns with industry trends toward sustainability and fuel efficiency.

This research will validate CFD findings using benchmark models such as the DrivAer. By demonstrating that classic sports cars can benefit from modern aerodynamic advancements, this study will provide insights into improving traditionally styled vehicles within contemporary motorsport regulations.

2.0 Industrial Relevance

This research aligns with industry focus on performance optimization, fuel efficiency, and sustainability, particularly as manufacturers aim to reduce aerodynamic drag and enhance downforce for racing and road applications.

Studies, such as those by Yuan and Wang, have demonstrated that complex underbody geometries can increase drag by over 23% compared to optimized flat floors (Yuan, Wang, 2017). By investigating underbody modifications—including diffusers, flat floors, and Venturi tunnels—this research directly contributes to improving aerodynamic efficiency in motorsport applications.

Underbody aerodynamics has been extensively studied in motorsport, but limited research has been applied to heritage-inspired cars like the Morgan Plus 4. While modern manufacturers like Porsche and Ferrari integrate advanced underbody designs, For example, the Caterham C120 project emphasized a lightweight and agile design, focusing on maintaining the brand's signature aesthetics and driving experience, rather than incorporating complex underbody aerodynamics (*Caterham* ,2025). This research aims to bridge that gap, demonstrating that heritage designs can benefit from modern aerodynamic solutions without compromising brand identity.

Reducing drag through optimized underbody aerodynamics improves fuel efficiency, reducing emissions in internal combustion vehicles and extending the range in electric vehicles (Nyoni, 2024). As the industry shifts toward sustainable performance solutions, this research aligns with global trends in efficiency-focused vehicle development, ensuring that even legacy brands like Morgan can adapt to evolving environmental standards while maintaining performance integrity.

By conducting Computational Fluid Dynamics (CFD) simulations, this research minimizes the need for extensive wind tunnel testing, showing the advantages and disadvantages of CFD.

3.0 Literature Review

3.1.1 Introduction to Underbody Aerodynamics

There are many different aerodynamic underbody components that can be added or changed:

- Flat Floor
- Diffuser
- Venturi Tunnels
- Side Skirts
- Vortex Generators
- Underbody Splitter
- Ground Effect Channels
- Rear Venting Louvers
- Underbody Cooling Ducts

A study on diffuser design by Buljac, Kozmar, Džijan shows the importance of underbody design diffuser designs can significantly reduce the wake region (Figure 1), therefore reducing the overall drag of the vehicle and improving negative lift underneath the vehicle there for improving downforce (Buljac, Kozmar, 2016).

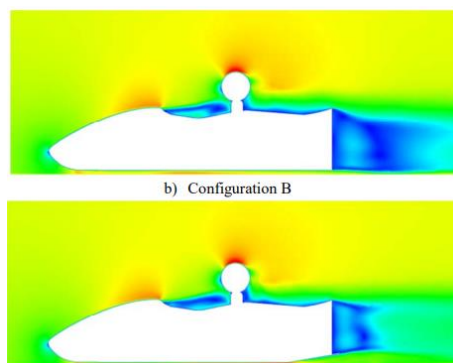


Figure 1 Diffuser effect on wake region (Buljac ,Kozmar, 2016).

Overall diffusers expand rear underbody airflow, accelerating air and creating low pressure, which increases downforce (Agarwal, 2020). Optimizing underbody flow is critical due to vortex formation. These vortices are caused by turbulent airflow at the vehicle's rear edges, creating spiralling air patterns that generate a force opposing the vehicle's motion. Diffusers can be used to minimize vortex formation (Dave, 2024).

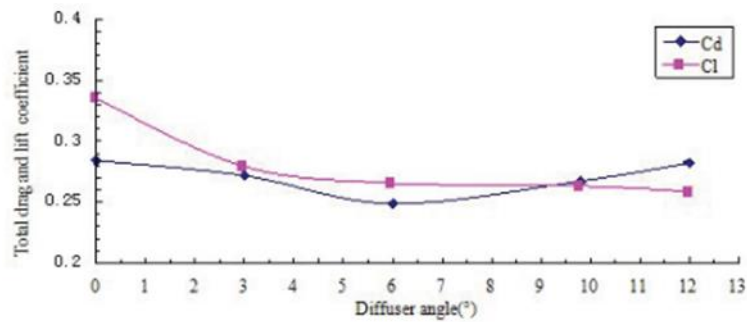


Figure 2 Effect of diffuser angle on Cd and Cl (Hu, Zhang, Yan, Zhao, 2011).

A study done by Hu, Zhang, Yan, and Zhao shows the effect of diffuser angle has on Cd and Cl values. Figure 2 Shows drag coefficient and lift coefficient plotted against increasing diffuser angles. Initially, both Cd and Cl decrease, indicating reduced drag and increased downforce, showing optimal range for aerodynamic efficiency. Beyond this point, Cd starts rising, likely due to flow separation, while Cl remains relatively stable (Hu, Zhang, Yan, Zhao, 2011).

Research by Knowles found that a flat floor creates downforce through ground effect accelerating airflow underneath and with a diffuser it further reduces underbody pressure, Pressure recovery, influenced by diffuser area ratios, geometry, and ride height, determines the static pressure increase. Overall, the diffuser decreases static pressure as airflow passes through, increasing downforce (Ehirim, Knowles, Saddington, 2018). More research by Versus Engineering concluded that a diffuser can reduce drag and improve performance even on a dirty underbody. However, a flat underbody does offer better efficiency (Verus Engineering , 2019).



Figure 3 Diffuser with dirty underbody (Verus Engineering ,2019)

Another component that improves underbody efficiency are Side skirts (Figure 4), which help regulate airflow beneath the car, improving aerodynamic efficiency. Their effectiveness depends on the



Figure 4 Side skirts (Stephen, 2021).

Vehicle's ground clearance, as higher placement reduces their impact. Additionally, side skirts create a separation between high-pressure and low-pressure airflow regions, keeping low pressure underneath the car from spilling (Stephen, 2021).

Recent CFD studies confirm the huge downforce potential of well-designed diffusers, especially when combined with side-skirt effects. Guerrero showed that adding sealing skirts to a 7° rear diffuser on a road car produced the highest downforce coefficient (-0.887 , a 1780% increase over a flat-floor baseline) while also yielding the best lift-to-drag efficiency (Guerrero, 2022). Similarly, Porcar found that mounting vertical endplate fins at the diffuser sides substantially improves its performance by channeling flow and preventing lateral spillover (Porcar, Toet, 2021). These works highlight that even small geometric additions can significantly enhance underbody suction and pressure recovery.

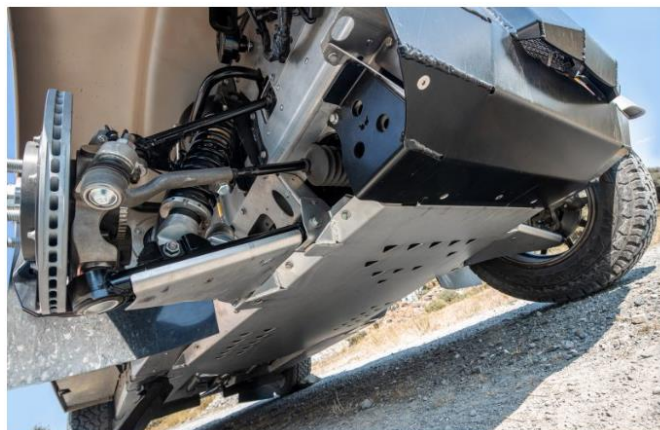


Figure 5 Morgan Plus Four CX-T (Vinten, 2021)

Figure 5 illustrates the underbody of the Morgan Plus Four CX-T, designed for the Dakar Rally. This image highlights the vehicle's structural constraints and defines the extent to which modifications to the underbody are feasible (Vinten, 2021). Another method to improve aerodynamic efficiency is by introducing venturi tunnels, shown in figure 6.

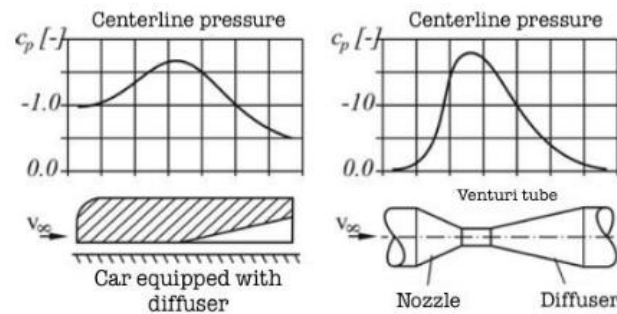


Figure 6 Distribution of pressure of a car's underbody diffuser and a venturi tube (Guerrero; Castilla; Eid, 2022)

In an atypical underfloor diffuser design, a specially contoured nozzle first profiles the air velocity. As the airflow moves into a constricted throat section, the Venturi effect accelerates (Figure 6) it to a peak velocity. Finally, an undertray decelerates the air, returning it to the free stream velocity. (Haris; Sapit; Hisyam; Mohamed, 2020).

A recent high-school research simulation indicated there is an optimal ride height where venturi tunnels minimize drag while maximizing downforce (Pothamsetti, n.d.). In experimental work, it was conducted an extensive wind-tunnel campaign on a diffuser-equipped bluff body to map the effect of rake. Their tests confirm that increasing rake angle consistently boosts underbody downforce across various ride heights, by promoting a stronger venturi effect at the rear (Kekus, 2021). This aligns with the practice of running a nose down rake to benefit the diffuser.

3.2 Computational Fluid Dynamics (CFD) Validation

Initial research is done on a suitable validation model, the following are researched:

- Ahmed body
- MIRA Car
- DrivAer

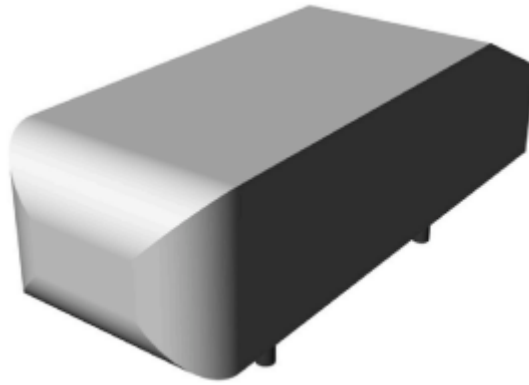


Figure 7 Ahmed body (Peng, Alessio 2024)

The Ahmed model is a simplified aerodynamic analysis tool. It features a blunt nose, box-like midsection, and slanted rear. It has a 40° slant angle, yielding a drag coefficient of 0.323 and lift coefficient of 0.0368 (Kamal, Ishak, Darlis, 2021).

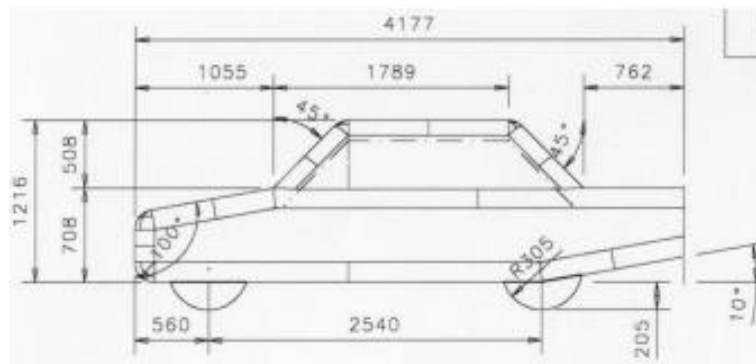


Figure 8 Mira Car (Palaskar, Kumar, 2016)

Palaskar and Kumar used the Mira car as a validation tool as compared to the Ahmed body, it was to simplified and the Mira car had more realistic features (Palaskar, Kumar, 2016), therefore making it more suitable for vehicle Computational fluid dynamics (CFD) validation.

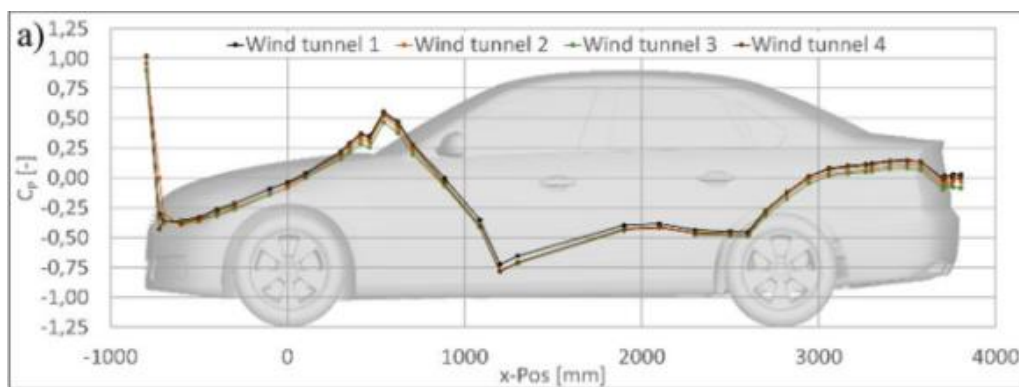


Figure 9 DrivAer (Renda, 2021).

Research by Renda found that the DrivAer model is a widely used benchmark in automotive aerodynamics research, serving as a reference for wind tunnel tests and CFD simulations. Initially introduced in 2012, it has been updated with features such as engine bay details, suspension components, and a refined underbody. Renda gathered Wind Tunnel data to show accuracy of the model for testing wind tunnel accuracy. Overall, the DrivAer model remains essential for CFD validation, making it a suitable CFD validation method (Renda, 2021).

3.2.1 Domain Positioning /size

A recommended starting point for domain positioning is at least 3 times the cars length at the front and 5 times behind it (Lanfrit, 2005).

3.2.2 Part mesh/volume mesh size

Independent mesh should be done on the part /volume mesh of the vehicle. This is a well know study, for example Ahmad and Abo-serie recommend reducing mesh size until a small difference in Cd between two successive mesh refinements is reached (Ahmad; Abo-Serie; Gaylard, 2009).

3.2.3 Prism layers

Research conducted by Lanfrit supports the use of 5 prism layers, this will be used as a starting point. To determine y^+ values a tool called Volupe is used for calculation of prism height to achieve a desired y^+ value (*Y+ Calculator CFD - Volupe.com*, 2024).

3.2.4 Density box

A study by Kang and Jun targeted large turbulence regions at the rear with density boxes to improve Cd and Cl precision (Kang, Jun, Park, Song, Kee, Kim, Lee, 2012). Another study enhanced flow separation accuracy around the car by employing a denser cell size (density box), a recommended starting point used is 2 vehicle lengths behind the body rear (Connor, Kharazi, Walter, Martindale, 2006).

3.2.5 Turbulence models and wall functions

K-omega SST model offers accuracy and reliability across a broader spectrum of flows, especially those featuring adverse pressure gradients, for example airfoils. The model incorporates turbulence shear stress transport in defining turbulent viscosity, these traits make it a more suited options compared to the BSL model (Menter, 2012). Other CFD literature emphasizes that the K-epsilon model provides easy of convergence and stability. It's known for its robustness and reliability, ensuring precise predictions of flow (Hu, Zhang, 2011). The model demonstrates accuracy in flows with recirculation, separation and adverse pressure gradients (Al Marzuki., 2017). Another study found that the K-epsilon RNG model (advanced version of K-epsilon) provides advanced accuracy flow separation and

turbulence flow in vehicle aerodynamics compared to K-epsilon (Ansari, 2017). Non-Equilibrium wall functions, an improvement from standard wall functions proposed by Kim and Choudhury account for pressure gradients and benefit for complex flows where separation and reattachment occur. Two-layer model with enhancements in the Enhanced wall functions can be used for both coarse and fine meshes (Wall, 2013). A key development is increased use of hybrid RANS/LES *models* (e.g. Detached Eddy Simulation, DES/DDES) to capture unsteady flow structures under the car. Rijns compared traditional steady RANS to DDES on a low-clearance race car model, both predicted similar pressure distributions overall, but DDES captured considerably more detail in highly unsteady regions (like separated diffuser flow and wake vortices) (Rijns, Teschner, 2023). The following five turbulence are researched based on their suitability for the drivair and can be validated based of real pressure data.

- K-omega SST
- Omega BSL
- K-Epsilon Realizable
- k-epsilon RNG
- Spalart-Allmaras model
- Detached Eddy Simulation, DES/DDES

3.2.6 Movind road and wheels

One study investigated the aerodynamic behaviour around a rotating and stationary 60% scale isolated wheel, with and without a moving ground plane. The research demonstrated that wheel rotation and ground movement substantially alter the flow structures, leading to more accurate predictions of aerodynamic forces. Notably, the study found that a rotating wheel with a moving ground plane exhibited different drag characteristics compared to stationary conditions (Rajaratnam, Walker, 2019). This study that moving wheels and road are crucial for an accurate study.

3.3 Flow visualisations methods

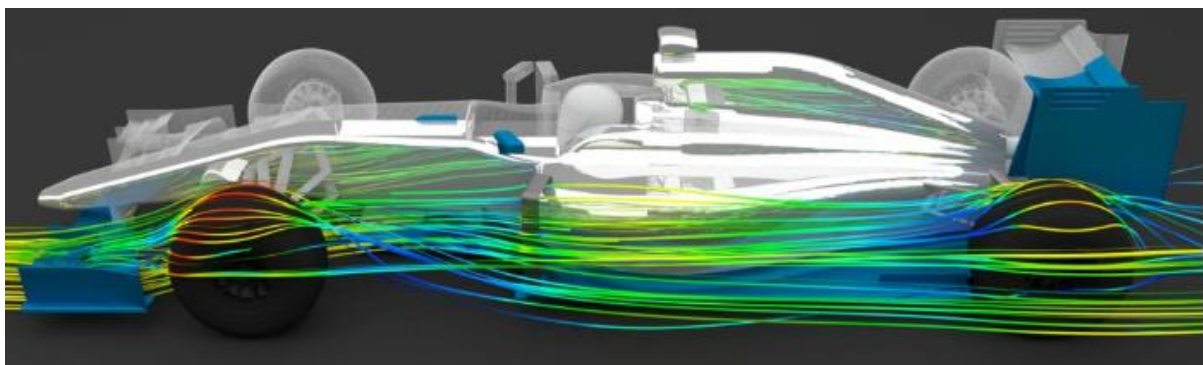


Figure 10 Streamline airflow (Simscale ,2023)

Streamlines are crucial for aerodynamic analysis as they visually represent the flow behaviour around a vehicle, highlighting areas of high and low pressure, flow separation, turbulence, and wake regions as seen in Figure 10. In motorsport and vehicle aerodynamics, streamlines help identify efficient airflow paths, improve downforce and drag reduction, and optimize underbody flow management (Simscale, 2023).

Plotting vorticity helps researchers see shear layers and vortex cross-sections around vehicles. For example, Sciacchitano used planar vorticity contours (Figure 11) to map the wake of an Ahmed body at different yaw angles. The contour plots of streamwise vorticity in vertical planes behind the model clearly show the twin C-pillar vortices and how they deform downstream (Sciacchitano, Giaquinta, 2019).

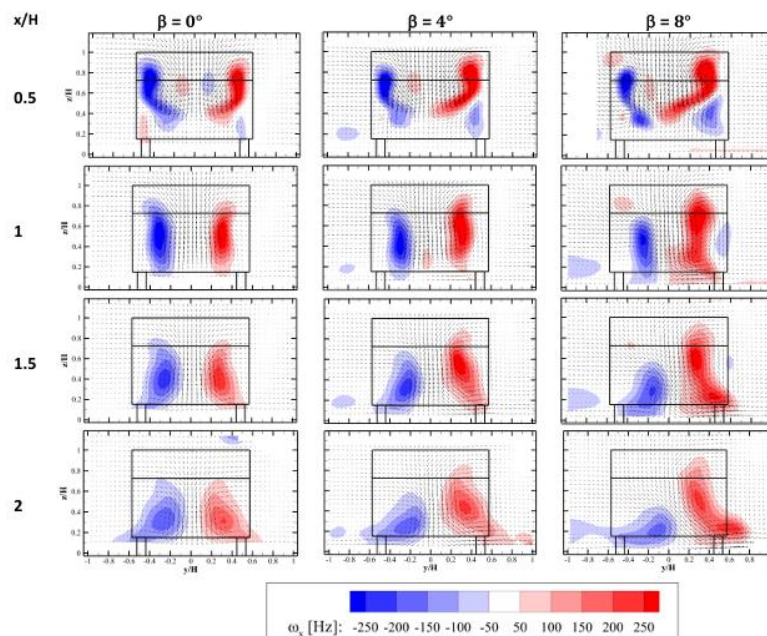


Figure 11 wake of an Ahmed body at different yaw angles (Sciacchitano, 2019).

In unsteady CFD simulations, path lines are often used to illustrate airflow routes around the vehicle. They help engineers see how air moves from front to rear.

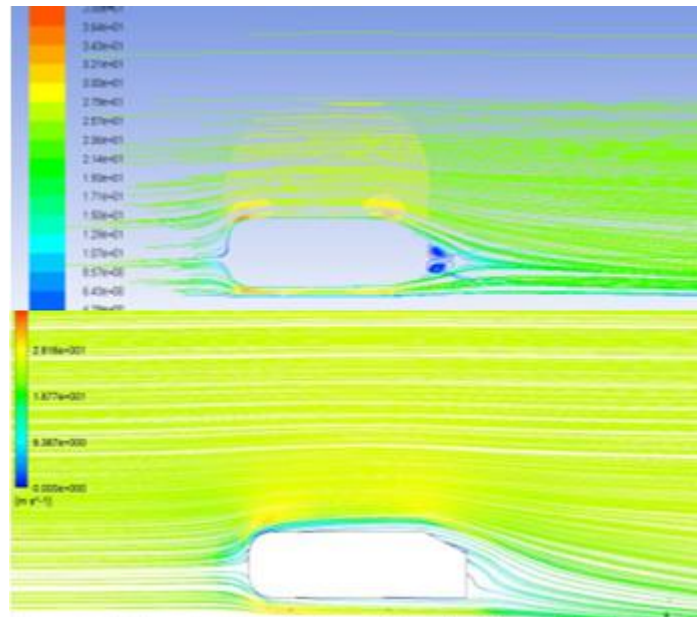


Figure 12 path lines are often used to illustrate airflow routes around the vehicle

For example, an aerodynamic study added an underbody diffuser to an Ahmed body and compared path line visuals with and without the diffuser as seen Figure 12 (Akash , Pranjal, 2018). Akash visualization method is a critical method to use.

4.0 Legislative and Environmental Considerations

The Morgan Plus Four, powered by a B48 turbocharged 2.0-litre engine (255 bhp), requires modifications that comply with Motorsport UK regulations while considering environmental impacts (*Morgan Plus 4*, 2025). This section outlines key underbody regulations and sustainability factors.

4.1 Regulations

Modifications must adhere to Motorsport UK guidelines (Motorsport UK Yearbook, 2025):

- Undertray: Must include drainage holes to prevent fluid accumulation.
- Ground Clearance & Skirts: A minimum 4 cm clearance is required, with skirts or bridging devices prohibited to ensure mechanical grip and suspension travel.
- Roll Cage: Must comply with Motorsport UK Section K for driver protection.
- Aerodynamics: Active devices are prohibited; modifications must remain fixed.
- Materials: Lightweight composites are permitted if they meet structural integrity and fire resistance standards.

Unlike FIA/WEC/GT regulations, club-level rules prohibit flexible or active aerodynamic components. Compliance ensures performance gains without breaching regulations.

4.2 Environmental Considerations

Aerodynamic improvements align with sustainability efforts by:

- Reducing drag and fuel consumption in ICE vehicles.
- Enhancing efficiency in electric/hybrid powertrains.
- Minimizing turbulence to reduce tire wear and energy losses.
- Promoting sustainable materials in line with FIA/Motorsport UK guidelines.

With motorsport shifting towards hybrid and electric platforms, optimizing aerodynamics can directly improve battery range and energy efficiency.

4.3 Challenges & Solutions

Regulatory restrictions pose design challenges, addressed through:

- Side skirts prohibition → Alternative airflow diverters or contoured panels.
- Active aero restrictions → Passive optimizations like vortex generators.

- Ground clearance limits → Balanced diffuser designs for performance and compliance.

Referencing the Motorsport UK Yearbook (2025) and consulting industry professionals ensures legal and effective modifications. By integrating compliance with sustainability, this research contributes to both competitive and environmental advancements in motorsport aerodynamics.

5.0 Proposed Methodologies

This research aims to compare generative design techniques for the redesign of a Morgan Plus 4's underbody to meet motorsport uk standards. The methodology will involve computational analysis (CFD), and experimental validation.

A structured multi-step approach will be applied, considering:

- Validating CFD methodology
- Computational simulations (CFD)
- Gathering pressure data using pressure taps
- Validating CFD results with real data
- Validation of results through industry standards

Additionally, the limitations and alternatives to each approach will be discussed to justify the selection of methods.

5.1 Computational Validation Simulation Approach

Computational tools provide accurate, repeatable, and cost-effective ways to assess aerodynamic efficiency. The following methods will be applied for a validation approach:

- Domain positioning study
- Domain size study
- Independent part and volume mesh study
- Prims layer study
- Density box study
- Delaney or octree
- Moving road and wheels
- Turbulence model and wall functions study

Independent mesh study is proposed as a methodology as it's been in previous study as found in the literature review, were abo-serie recommended the method (Ahmad; Abo-Serie; Gaylard, 2009). Density box are a method proposed by Kang and Jun to achieve more precise data, in turbulence flow areas (Kang, Jun, Park, Song, Kee, Kim, Lee, 2012).

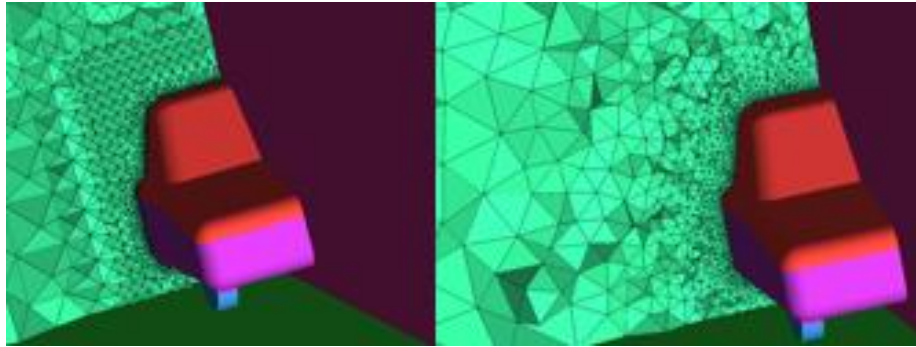


Figure 13 Delaunay or octree mesh from ANSYS Fluent

Figure 13 shows the difference between a Delaunay and Octree mesh. Delaunay meshing generates high-quality, well-shaped elements using triangulation, ideal for CFD due to its accuracy. Octree meshing subdivides space hierarchically, offering faster automation and adaptive refinement for complex domains. Delaunay excels in element quality, while Octree is preferred for speed and scalability. Both meshes must be studied to determine the most suited for the Morgan Plus 4 geometries.

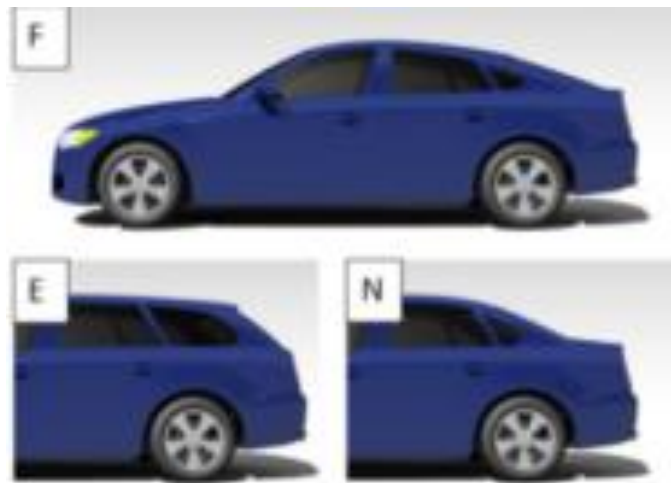


Figure 14 DrivAer models, F-Fastback, E-Estate, N-Notchback (Ashton, 2015)

Research gained in 3.0 Literature Review determined that the Notchback DrivAer model (Figure 14), is most suited for the validation study due to its realistic shape and underbody geometry. Research by Renda showed that there is multiple windtunnel data on the car that can be used for validation (Renda, 2021).

5.2 Gathering real data using pressure taps

Pressure data on the topside of the Morgan has already been collected using pressure taps however, there is no data on the underbody of the Morgan.

Pressure taps are commonly used in aerodynamic studies to measure surface pressure distributions, particularly in vehicle underbody research. A study by Moiola et al. investigated the accuracy of additively manufactured pressure taps, analysing leakage, measurement error, and optimization techniques. The study found that tap geometry, material properties, and manufacturing precision significantly impact measurement accuracy (Moioli, Reinbold, 2019), Showing pressure taps still have an error of percentage.

5.3 Methodology for the Redesign of the Underbody for the Morgan Plus 4

The underbody redesign for the Morgan Plus 4 aims to enhance aerodynamic efficiency based on key findings from previous research in vehicle aerodynamics. Studies show that underbody airflow management is a critical factor in reducing drag and increasing downforce, which directly influences vehicle performance and stability (Guerrero, 2022).

The redesign seeks to apply proven aerodynamic concepts to the Morgan Plus 4 while ensuring compliance with Motorsport UK regulations. The focus areas include:

- Reducing aerodynamic drag (C_d) to improve efficiency and speed.
- Increasing downforce (C_l) for enhanced cornering and stability.
- Managing airflow under the vehicle to optimize pressure distribution and minimize turbulence.
- Ensuring compliance with Motorsport UK rules, particularly regarding ground clearance and underbody modifications.

Based on findings from the literature review and CFD analysis, the following design modifications will be implemented:

5.3.1 Flat Floor Implementation

A flat underbody will be introduced to create a more controlled airflow beneath the car. Based on Knowles research showing that a flat floor accelerates airflow, reducing pressure underneath the vehicle and increasing downforce while minimizing drag, the Morgan Plus 4 redesign will incorporate a carefully structured flat floor (Ehirim, Knowles, Saddington, 2018). This will include strategic material choices and airflow management improvements to optimize performance.

The design considerations include:

- A smooth surface to reduce flow separation and turbulence.
- Drainage holes to comply with regulations while ensuring minimal disruption to airflow.

- Material adjustments to balance structural integrity and weight savings.

5.3.2 Rear Diffuser Optimization

Diffusers play a crucial role in managing underbody airflow and enhancing downforce. Hu found that increasing the diffuser expansion ratio improves pressure recovery, reducing drag and increasing negative lift (Hu, Zhang, Yan, Zhao, 2011). Based on these findings, a diffuser will be applied to the Morgan Plus 4, but with further refinements to optimize its performance. The redesign will focus on adjusting the expansion ratio, refining the diffuser geometry, and incorporating endplate extensions to enhance flow control and maximize aerodynamic efficiency.

The optimization process will involve:

- Testing different diffuser angles to find the ideal trade-off between drag and downforce.
- Evaluating endplate extensions to control vortex shedding and enhance stability.
- Integrating rake adjustments to optimize ground effect dynamics.

5.3.2 Venturi Tunnels for Ground Effect Enhancement

Studies, such as those by Guerrero, have shown that Venturi tunnels can accelerate airflow under the car, creating a pressure drop that enhances ground effect (Guerrero, 2022). Based on this, the Morgan Plus 4 redesign will incorporate some type of Venturi tunnels, refining their shape and positioning to maximize aerodynamic efficiency while adhering to ground clearance regulations. By carefully designing these tunnels, the redesign can achieve improved aerodynamic efficiency.

Key considerations include:

- Constriction points to maximize airflow acceleration without excessive drag.
- Compliance with minimum ground clearance regulations while optimizing tunnel geometry.
- Investigation of airflow interactions with the rear diffuser for maximum efficiency.

5.3.3 Side Skirts for Airflow Containment

While fully sealed side skirts are not permitted under Motorsport UK regulations, research by Stephen shows that carefully contoured airflow-guiding panels can minimize lateral airflow disruption, improving diffuser efficiency without violating regulations (Stephen, 2021). Applying this concept, the redesign will implement aerodynamically optimized side skirts that effectively channel airflow without forming a complete seal, ensuring both performance gains and regulatory compliance.

Proposed design modifications include:

- Aerodynamically contoured side panels to guide airflow without forming a complete seal.
- Managing pressure gradients to prevent high-pressure air from spilling into the low-pressure underbody region.
- Material selection for weight reduction while maintaining durability.

5.3.4 Rake Angle Adjustments

Research by Kekus highlights that increasing the rake angle (higher rear ride height) enhances the efficiency of diffusers by promoting better airflow expansion (Kekus, 2021). Reflecting on this, the Morgan Plus 4 will undergo rake angle tuning, testing different ride height configurations to achieve the optimal balance between aerodynamic performance and regulatory constraints.

To optimize this parameter:

- Different ride height configurations will be tested to find the best balance between drag reduction and downforce generation.
- Dynamic rake adjustments will be evaluated to maintain stability under load.
- Ground clearance constraints will be considered to ensure regulatory compliance.

5.3.5 Expected Performance Improvements

Based on research findings, the redesigned underbody is expected to:

- Reduce drag coefficient by streamlining underbody airflow and improving pressure recovery.
- Increase negative lift to enhance high-speed stability and cornering performance.
- Minimize wake turbulence, improving vehicle efficiency and predictability at speed.
- Ensure regulatory compliance, maintaining the integrity of the Morgan Plus 4's classic design while incorporating modern aerodynamic advancements.

6.0 Results Collection & Analysis

This section outlines the methodology for collecting, interpreting, and analysing data obtained through CFD simulations and, where applicable, experimental validation techniques. The results will be systematically assessed to ensure accuracy and reliability in evaluating the aerodynamic performance of the Morgan Plus 4's underbody modifications.

6.1 Type of Data to be Collected

The primary data collected will focus on the aerodynamic performance of the vehicle under various underbody configurations. This includes:

Aerodynamic Performance Metrics:

- Drag coefficient (C_d)
- Lift coefficient (C_l)
- Pressure distribution across the underbody and wake regions

Flow Visualization Data:

- Streamline patterns to observe flow separation and attachment
- Vorticity contours for wake structure analysis
- Q-criterion plots to visualize vortex formations

Pressure Measurements:

- Surface pressure distributions obtained from CFD simulations
- (If available) Pressure tap data for experimental validation

Grid Sensitivity Data:

- Impact of mesh refinement on simulation accuracy
- Convergence behaviour of C_d and C_l values

6.2 Quantitative vs. Qualitative Data

- Quantitative Data: Numerical values of C_d , C_l , pressure coefficients, velocity gradients, and turbulence intensity extracted from CFD simulations. These values will be used for comparison with existing literature and validated models.
- Qualitative Data: Visual representations of airflow characteristics, including streamlines, vorticity fields, and wake turbulence. These will support the numerical findings by illustrating flow behaviour and aerodynamic efficiency.

6.3 Expected Data Format & Processing

- CFD Data Outputs: Numerical data will be exported in CSV or MatLab format for further analysis.
- Graphical Representations: Pressure contours, velocity fields, and aerodynamic coefficient plots will be generated using post-processing tools such as ANSYS Fluent results.
- Residual Plots: Solver convergence will be assessed by tracking changes in aerodynamic coefficients over simulation time.
- Post-Processing: MATLAB will be used for statistical analysis and trend identification in aerodynamic metrics.

6.4 Metrics for Data Assessment

To evaluate the effectiveness of underbody modifications, data will be assessed against the following benchmarks:

Comparison with Validated Models:

- DrivAer model wind tunnel data to validate simulation accuracy
- Literature findings on diffuser performance, Venturi tunnels, and side skirt effects

Grid Sensitivity Analysis:

- Assessment of Cd and Cl convergence with increasing mesh resolution

Performance Improvements:

- Reduction in drag and increase in downforce relative to baseline Morgan Plus 4 geometries
- Analysis of flow separation and wake turbulence changes

6.5 Potential Issues in Data Collection & Analysis

While CFD provides a powerful tool for aerodynamic assessment, several challenges must be addressed:

CFD Accuracy Challenges:

- Selection of the appropriate turbulence model to balance computational cost and accuracy
- Potential errors in flow separation prediction due to meshing limitations

Experimental Validation Limitations:

- Lack of underbody pressure tap data for direct comparison

- Difficulty in replicating real-world conditions in simulations

Computational Constraints:

- High computational cost for fine-mesh simulations may impose optimization of simulation parameters
- Long processing times requiring resource management and efficiency strategies

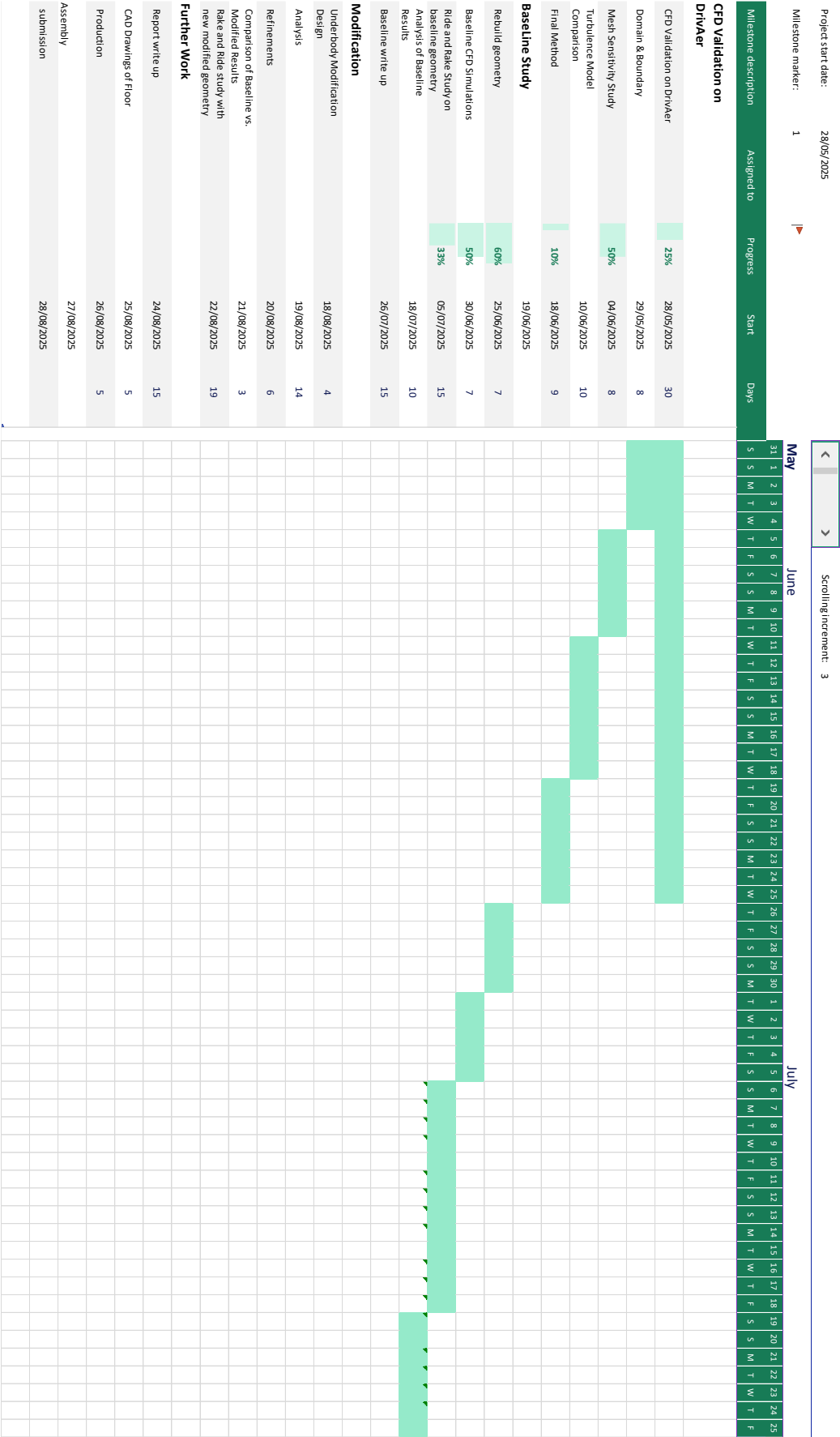
6.6 Summary of Results Collection & Analysis

This section outlines a structured approach to data collection and analysis, ensuring robust evaluation of aerodynamic modifications to the Morgan Plus 4. By leveraging CFD methodologies and benchmarking against validated models, the study aims to provide accurate and actionable insights into improving underbody aerodynamics for motorsport applications.

7.0 Project Planning/Timescales

Planning of the project can be seen in Table 1. This is a proposed timescale with anticipated tasks shown and their expected durations. Contingency time has been allowed by increases timescale for certain task where issue may go wrong. The grant chart has been made so there is no stoppage so one area does not bring the entire project to a halt.

Table 1 Gantt chart Timework flow



8.0 Evaluation

This research investigates aerodynamic optimization of the Morgan Plus 4, focusing on Computational Fluid Dynamics (CFD) to enhance underbody performance within Motorsport UK regulations. While offering improvements in drag reduction and downforce, certain limitations and challenges must be addressed.

8.1 Limitations and Challenges

CFD Validation: While CFD is efficient, the absence of wind tunnel testing may introduce uncertainties. Benchmark models like DrivAer will be used for validation (Abid, 2024).

Regulatory Constraints: Motorsport UK regulations restrict side skirts and enforce a minimum ground clearance of 4 cm, limiting optimization. Alternative solutions like airflow-guiding panels and vortex generators will be explored (Motorsport UK Yearbook, 2025).

Computational Challenges: High-resolution meshing improves accuracy but increases computational demand. Balancing efficiency and precision using density boxes and prism layers is essential (Pothamsetti, 2023).

8.2 Contributions and Outcomes

Despite constraints, this research contributes to motorsport aerodynamics by integrating modern aerodynamic techniques into heritage vehicles. Expected benefits include:

- Reduced drag, improving speed and efficiency (Zhang, 2023).
- Increased downforce, enhancing stability and cornering performance.
- Optimized pressure distribution, minimizing wake turbulence.
- Sustainability, aiding efficiency in electric and hybrid vehicles (SimScale, 2018).

8.3 Conclusion

This study offers a structured approach to improving underbody aerodynamics within regulatory constraints. Strategic validation, optimized meshing, and innovative solutions will enhance feasibility. Future work should include wind tunnel or track testing for further validation.

9.0 Reference

1. Yuan, Z., & Wang, Y. (2017). Effect of underbody structure on aerodynamic drag and optimization. *Journal of Measurements in Engineering*, 5(3), 194–204. <https://doi.org/10.21595/jme.2017.19210>
2. Buljac, A., Kozmar, H., & Džijan, I. (2016). Aerodynamic performance of the underbody and wings of an open-wheel race car. *Transactions of Famena*, 40(2), 19–34. [https](https://doi.org/10.21595/jme.2017.19210)
3. Agarwal, S. (2020). AERODYNAMICS OF VEHICLES. *International Journal of Advanced Research in Engineering and Technology (IJARET)*, 11(11), 1996–2001. <https://doi.org/10.34218/IJARET.11.11.2020.191>
4. Dave, D. (2024). ADVANCES IN VEHICLULAR AERODYNAMICS.
5. Hu, X., Zhang, R., Ye, J., Yan, X., & Zhao, Z. (2011). Influence of different diffuser angle on Sedan's aerodynamic characteristics. *Physics Procedia*, 22, 239–245. <https://doi.org/10.1016/j.phpro.2011.11.038/10.21278/>
6. Stephen, R. (2021). Research and Development of a Body Kit for Passenger Cars to Enhance Aerodynamic Efficiency Master's Final Degree Project.
7. Verus Engineering (2019) Is a Flat Underbody Necessary for a Rear Diffuser to Function?
8. Ehirim, O. H., Knowles, K., & Saddington, A. J. (2018). A review of ground-effect diffuser aerodynamics. *Journal of Fluids Engineering, Transactions of the ASME*, 141(2).
9. Lanfrit, M. (2005). Best practice guidelines for handling Automotive External Aerodynamics with FLUENT.
10. Ahmad, N. E., Abo-Serie, E., & Gaylard, A. (2010). Mesh Optimization for Ground Vehicle Aerodynamics. In *www.cfdl.issres.net* (Vol. 2, Issue 1).
11. Y+ Calculator CFD - Volupe.com (2024) Volupe.com. Available at: <https://volute.com/support/all-y-calculator-when-meshing-a-geometry-for-cfd-analysis/>.
12. Menter, F. R. (2012). Turbulence Modeling for Engineering Flows. www.ansys.com
13. Al, M., Marzuki, B., Azlan, S., Polytechnic, S., Firdaus, M., Azmi, M., Arzo, M., Bakar, A., Sultan, P., & Shah, A. (2017). Vehicle aerodynamics analysis of a multi-purpose vehicle using CFD. 12(7). <https://www.researchgate.net/publication/316935326>
14. Ansari, A. R. (2017). Article ID: IJMET_08_03_038 Cite this Article: Abdul Razzaque Ansari, CFD Analysis of Aerodynamic Design of Tata Indica Car. *International Journal of Mechanical Engineering and Technology*, 8(3), 344–355. <http://www.iaeme.com/IJMET/index.asp344http://www.iaeme.com/IJMET/issues.asp?JTyp>

[e=IJMET&VType=8&IType=3http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=8&IType=3](http://www.iaeme.com/IJMET/issues.asp?JType=IJMET&VType=8&IType=3)

15. Wall, A. (2013). Simulating the Volvo Cars Aerodynamic Wind Tunnel with CFD.
16. Ashton, N. and Revell, A., 2015. Comparison of RANS and DES methods for the DrivAer automotive body (No. 2015-01-1538). SAE Technical Paper.
17. Peng Qin, Alessio Ricci, Bert Blocken, (2024) CFD simulation of aerodynamic forces on the DrivAer car model: Impact of computational parameters, Journal of Wind Engineering and Industrial Aerodynamics, Volume 248, 2024,105711, ISSN 0167-6105, <https://doi.org/10.1016/j.jweia.2024.105711>.
18. Kamal, M. N. F., Ishak, I. A., Darlis, N., Maji, D. S. B., Sukiman, S. L., Rashid, R. A., & Azizul, M. A. (2021). A Review of Aerodynamics Influence on Various Car Model Geometry through CFD Techniques. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences, 88(1), 109–125. <https://doi.org/10.37934/arfmts.88.1.109125>
19. Palaskar, P.M., Kumar, V. and Vaidya, R., 2016. Methodology Development to Accurately Predict Aerodynamic Drag and Lift for Passenger Vehicles Using CFD (No. 2016-01-1600). SAE Technical Paper.
20. Moioli, M., Reinbold, C., Sørensen, K., & Breitsamter, C. (2019). Investigation of additively manufactured wind tunnel models with integrated pressure taps for vortex flow analysis. Aerospace, 6(10). <https://doi.org/10.3390/AEROSPACE6100113>
21. Vinten, D. (2021) Morgan Plus Four CX-T – Mad Maximilian - Car & Classic Magazine, Car & Classic Magazine. Available at: <https://www.carandclassic.com/magazine/morgan-plus-four-cx-t-mad-maximilian/> (Accessed: 26 February 2025).
22. Simscale (2023) What is CFD | Computational Fluid Dynamics?, SimScale. Available at: <https://www.simscale.com/docs/simwiki/cfd-computational-fluid-dynamics/what-is-cfd-computational-fluid-dynamics/>.
23. Guerrero, A., Castilla, R., & Eid, G. (2022). A Numerical Aerodynamic Analysis on the Effect of Rear Underbody Diffusers on Road Cars. Applied Sciences (Switzerland), 12(8). <https://doi.org/10.3390/app12083763>
24. Porcar, L., Toet, W., & Gamez-Montero, P. J. (2021). Study of the effect of vertical airfoil endplates on diffusers in vehicle aerodynamics. Designs, 5(3). <https://doi.org/10.3390/designs5030045>
25. Reducing Drag by Optimizing the Underbody with Ride Height in Formula 1 Varun Pothamsetti. (n.d.).

26. Kekus, P. (2021). Aerodynamics and Experimental Optimisation of Automotive Underbody Diffusers in the Presence of Rake.
27. Rijns, S., Teschner, T.-R., Blackburn, K., & Brighton, J. (2023). COMPARATIVE ANALYSIS OF RANS AND DDES METHODS FOR AERODYNAMIC PERFORMANCE PREDICTIONS FOR HIGH PERFORMANCE VEHICLES AT LOW GROUND CLEARANCES.
28. Sciacchitano, A., & Giaquinta, D. (2019). Investigation of the Ahmed body cross-wind flow topology by robotic volumetric PIV.
29. Akash MB , Pranjal Patil , Tej Pratap Singh (2018). Parametric Investigation of Effect of Diffuser Angle on the Flow Characteristics of an Ahmed Body Akash MB Pranjal Patil Tej Pratap Singh. www.ijert.org
30. Rajaratnam, E. and Walker, D., (2019) "Experimental and Computational Study of the Flow around a Stationary and Rotating Isolated Wheel and the Influence of a Moving Ground Plane," SAE Technical Paper 2019-01-0647.
31. Morgan Plus 4 (2025) evo. Available at: <https://www.evo.co.uk/morgan/plus-four> (Accessed: 27 February 2025).
32. Renda, G. (2021). Comparison of different turbulence models for the numerical simulation of the flowfield around the DrivAer Tesi di Laurea Magistrale in Aeronautical Engineering- Ingegneria Aeronautica.
33. Abid, A. (2024). Aerodynamic Phenomena of Controlling Passive Flow of a Time Attack Supercar. Engineering Reports.
34. Pothamsetti, V. (2023). Reducing Drag by Optimizing the Underbody with Ride Height in Formula 1. Research Archive.
35. Zhang, Z. (2023). Study on aerodynamic development in Formula One racing. Theoretical and Natural Science.
36. SimScale. (2018). Aerodynamics of Road Vehicles: CFD Analysis.
37. Motorsport UK Yearbook. (2025). Regulations for Circuit Racing.
38. Blocked (2025) Drivedesign.co.uk. Available at: <https://www.drivedesign.co.uk/designing-for-an-iconic-marque/> (Accessed: 2 March 2025).
39. Nyoni, E., & Chigumba, P. (2024). Aerodynamics and its role in enhancing fuel efficiency in automotive engineering. ~ 32 ~ International Journal of Automobile Engineering, 5(2). www.mechanicaljournals.com/ijae
40. Y+ Calculator CFD - Volupe.com (2024) Volupe.com. Available at: <https://volute.com/support/all-y-calculator-when-meshing-a-geometry-for-cfd-analysis/>.

