

Cranfield University



Analysis of Active Aerodynamics for High-Performance Vehicles

Steven Rijns

Faculty of Engineering and Applied Sciences
Transport Systems
Advanced Vehicle Engineering Centre

Doctor of Philosophy

Academic Year: 2022–2025

Supervisors: Dr Tom-Robin Teschner and Dr Kim Blackburn

March 2025

Cranfield University



Faculty of Engineering and Applied Sciences
Transport Systems
Advanced Vehicle Engineering Centre

Doctor of Philosophy

Academic Year: 2022–2025

Analysis of Active Aerodynamics for High-Performance Vehicles

Steven Rijns

Supervisors: Dr Tom-Robin Teschner and Dr Kim Blackburn

March 2025

© Cranfield University 2025. All rights reserved. No part of this publication may be reproduced without the written permission of the copyright owner.

Academic Integrity Declaration

I declare that:

- The thesis submitted has been written by me alone.
- The thesis submitted has not been previously submitted to this university or any other.
- That all content, including primary and/or secondary data, is true to the best of my knowledge.
- That all quotations and references have been duly acknowledged according to the requirements of academic research.

I understand that to knowingly submit work in violation of the above statement will be considered by examiners as academic misconduct.

Abstract

The pursuit of greater efficiency and performance drives advancements in the automotive and motorsport industries, with active aerodynamics emerging as a promising approach due to their ability to dynamically adapt aerodynamic characteristics to specific operating conditions. However, their development presents challenges, including the need for practical yet accurate simulation methodologies, a deeper understanding of vehicle aerodynamics in dynamic conditions, and a comprehensive assessment of their performance potential. This research addresses these challenges through interdependent studies. A cost-effective Computational Fluid Dynamics (CFD) workflow is developed and validated against experimental and high-fidelity simulation data, complemented by a structured wind tunnel correlation process to ensure reliable aerodynamic predictions. Yaw and cornering effects on flow field characteristics and aerodynamic performance are analysed using wind tunnel experiments and CFD simulations. Finally, active aerodynamic configurations, including 2D systems capable of modulating aerodynamic balance longitudinally and laterally, are designed and examined using minimum lap time simulations to assess performance gains, optimal control strategies, and dependencies on vehicle setup. The CFD workflow demonstrates high predictive accuracy across various aerodynamic conditions, with the structured correlation process improving experimental data interpretation and validation. However, conditions critically dominated by highly unsteady flow phenomena require higher-fidelity simulations. Yaw and cornering conditions induce significant flow field alterations, including underbody interference, enhanced upper surface flow acceleration, and asymmetric wake structures, leading to substantial downforce and drag penalties. Active aerodynamic systems provide significant performance benefits across diverse scenarios, with 2D systems consistently outperforming conventional designs by prioritising aerodynamic loads on underloaded tyres to improve total grip. Overall, this research advances numerical methodologies, deepens understanding of vehicle aerodynamics in dynamic conditions, and demonstrates the performance potential of various active aerodynamic designs. The work establishes a foundation for optimising vehicle performance with active aerodynamic systems, supporting future research and industry innovations in automotive and high-performance vehicle engineering.

Acknowledgements

I would like to express my sincere gratitude to the individuals who have supported and contributed to the successful completion of this research. First and foremost, I would like to thank Professor James Brighton for providing me with the opportunity to undertake this PhD within the Advanced Vehicle Engineering Centre at Cranfield University.

I am especially grateful to my supervisor, Dr Tom-Robin Teschner, for his continuous support and engagement throughout this research. His expert guidance in computational fluid dynamics was instrumental in establishing a strong methodological foundation, and his thoughtful feedback played a key role in shaping the quality of this work. I particularly appreciated his genuine interest in the project and his consistent willingness to make time for discussions. I would also like to thank my second supervisor, Dr Kim Blackburn, whose input during our occasional meetings was equally appreciated.

I would like to thank Dr Anderson Ramos Proenca for his essential support in organising the wind tunnel experiments and processing the data, which formed a critical part of this research. I am also grateful to Lynton Banks-Davies and Karl Gerhard for the preparation of the experiments and technical support throughout the wind tunnel campaigns. Special thanks go to Dr Efstathios Siampis for providing the initial minimum lap time simulation framework and for his valuable input on vehicle modelling, which was central to the performance-focused aspects of this work. Finally, I would like to acknowledge the anonymous reviewers of my journal papers, whose constructive feedback helped strengthen the publications and refine this thesis.

Lastly, I am truly grateful to my parents, sister, and partner for the continuous love and support they have shown throughout my PhD journey. In particular, I would like to thank my parents for inspiring me, believing in me, and encouraging me to pursue unique opportunities to achieve my goals. The continued guidance and motivation you have provided throughout my life and during this PhD have truly meant a lot to me and will always stay with me.

Table of Contents

Academic Integrity Declaration	i
Abstract	ii
Acknowledgements	iii
Table of Contents	iv
List of Figures	x
List of Tables	xiv
List of Abbreviations	xv
List of Symbols	xvi
1 Introduction	1
1.1 Context	1
1.2 Aims and Objectives	4
1.3 Thesis Structure	5
1.4 Chapter Overview	6
1.5 Research Contributions	8
1.6 Publications	10
2 Development of a CFD Workflow and Wind Tunnel Correlation Process	12
2.1 Introduction	13
2.2 Methodology	14
2.2.1 Vehicle Geometry	14
2.2.2 Experimental Validation Data	15
2.2.3 Computational Fluid Dynamics Simulations	15
2.2.3.1 Domains and Boundary Conditions	15
2.2.3.1.1 Wind Tunnel Installation	16
2.2.3.1.2 Open-Road Installations	17
2.2.3.2 Mesh	18
2.2.3.2.1 Description	18
2.2.3.2.2 Grid Convergence	19
2.2.3.2.3 DDES - High Near-Wall Resolution Mesh	20

2.2.3.3	Solve	20
2.2.3.3.1	RANS	20
2.2.3.3.2	DDES	21
2.2.4	Wind Tunnel Correlation Process	21
2.2.4.1	Blockage Effect and Correction Methods	22
2.2.4.2	Overhead Measurement System Interference	22
2.3	Results and Discussion	23
2.3.1	Turbulence Model Assessment	23
2.3.1.1	Force Coefficients	24
2.3.1.2	Pressure Distribution	25
2.3.1.3	Slant and Spoiler	28
2.3.1.4	Diffuser	30
2.3.1.5	Wake	32
2.3.1.6	Domain Sensitivity	34
2.3.1.7	Assessment Summary	34
2.3.2	Wind Tunnel Correlation Process	36
2.3.2.1	Blockage Effect	36
2.3.2.2	Blockage Correction Methods	38
2.3.2.3	Overhead Measurement System Interference	39
2.4	Conclusion	40
	References	41
3	Aerodynamic Analysis and Turbulence Model Evaluation at Varying Ride Heights	47
3.1	Introduction	48
3.2	Methodology	49
3.2.1	Vehicle Geometry	49
3.2.2	Experimental Validation Data	50
3.2.3	Computational Fluid Dynamics Simulations	50
3.2.3.1	Domain and Boundary Conditions	50
3.2.3.2	Mesh	52
3.2.3.3	Solve	53
3.3	Results and Discussion	53
3.3.1	RANS vs DDES	54
3.3.2	Ride Height Analysis	57
3.3.2.1	Underbody Pressure Distribution	57
3.3.2.2	Multichannel Diffuser Flow	58
3.4	Conclusion	61
	References	62
4	Aerodynamic Analysis of Yaw Conditions	67
4.1	Introduction	68
4.2	Methodology	69
4.2.1	Vehicle Geometry	69

4.2.2	Wind Tunnel Experiments	71
4.2.2.1	Cranfield University 8x6 Wind Tunnel	71
4.2.2.2	Aerodynamic Forces	71
4.2.2.3	Wake Total Pressure	72
4.2.2.4	Surface Pressure	74
4.2.3	Computational Fluid Dynamics Simulations	76
4.2.3.1	Domain and Boundary Conditions	76
4.2.3.2	Mesh	77
4.2.3.3	Solve	78
4.3	Validation	78
4.3.1	Aerodynamic Forces	78
4.3.2	Surface Pressure	80
4.3.3	Wake Total Pressure	80
4.3.4	Summary	83
4.4	Results and Discussion	83
4.4.1	Aerodynamic Forces	83
4.4.1.1	Lift Coefficients	83
4.4.1.2	Drag Coefficients	85
4.4.1.3	Side Force Coefficients	86
4.4.2	Flow Field	86
4.4.3	Pressure Distribution	88
4.4.3.1	Vehicle Body	88
4.4.3.2	Slant	91
4.4.3.3	Rear Wing	93
4.4.4	Wake Characteristics	94
4.4.4.1	0° Yaw	94
4.4.4.2	5° Yaw	95
4.4.4.3	10° Yaw	96
4.4.4.4	15° Yaw	96
4.4.4.5	20° Yaw	97
4.5	Conclusion	100
	References	101
5	Aerodynamic Analysis of Cornering Conditions	107
5.1	Introduction	108
5.2	Methodology	109
5.2.1	Vehicle Geometry	109
5.2.2	Operating Conditions	110
5.2.3	Wind Tunnel Experiments	111
5.2.4	Computational Fluid Dynamics Simulations	113
5.2.4.1	Domain and Boundary Conditions	113
5.2.4.2	Mesh	115
5.2.4.3	Solve	117
5.2.4.4	Post-Processing	117
5.3	Validation	117

5.4	Results and Discussion	119
5.4.1	Aerodynamic Forces	119
5.4.2	Vehicle Aerodynamics	121
5.4.3	Rear Wing Aerodynamics	125
5.4.4	Rear Wing Interaction	128
5.5	Conclusion	130
	References	131
6	Aerodynamic and Steady-State Performance Analysis of Active Aerodynamics During Cornering	135
6.1	Introduction	136
6.2	Methodology	138
6.2.1	Vehicle Geometry	138
6.2.2	Wind Tunnel Experiments	140
6.2.3	Operating Conditions	141
6.2.4	Computational Fluid Dynamics Simulations	141
6.2.4.1	Domain and Boundary Conditions	141
6.2.4.2	Mesh	142
6.2.4.3	Solve	144
6.2.4.4	Post-Processing	145
6.2.5	Vehicle Dynamics Model	145
6.2.5.1	Vehicle Model	147
6.2.5.2	Centrifugal Force	148
6.2.5.3	Vertical Tyre Forces	148
6.2.5.4	Tyre Model	151
6.2.5.5	Tyre Characteristics	152
6.3	Results and Discussion	153
6.3.1	Baseline Vehicle Configuration	153
6.3.2	Active Aerodynamic Load Balancing Vehicle Concepts	156
6.3.2.1	Control Strategy	156
6.3.2.2	Split Rear Wing	158
6.3.2.3	Tilt Rear Wing	164
6.3.2.4	Split Rear Wing vs Tilt Rear Wing	170
6.4	Conclusion	171
	References	172
7	Lap Time Performance Analysis of Active Aerodynamics	177
7.1	Introduction	178
7.2	Methodology	180
7.2.1	Vehicle Geometry	180
7.2.2	Computational Fluid Dynamics Simulations	181
7.2.2.1	Domain and Boundary Conditions	181
7.2.2.2	Mesh	181
7.2.2.3	Solve	182
7.2.2.4	Aerodynamic Systems Interdependencies	183

7.2.3	Vehicle Model	184
7.2.3.1	Tyres	186
7.2.3.2	Load Transfer	188
7.2.3.3	Aerodynamics	189
7.2.3.4	Powertrain	191
7.2.4	Optimisation Framework	191
7.2.4.1	Optimal Control Problem	191
7.2.4.2	Non-Linear Programming	193
7.2.4.3	Setup	193
7.3	Results and Discussion	195
7.3.1	Aerodynamic Performance	195
7.3.2	Vehicle Performance	197
7.3.3	Optimal Control of Aerodynamic Systems	200
7.3.4	Vehicle Design Parameters	203
7.4	Conclusion	204
	References	205
8	Lap Time Performance Analysis of Active Aerodynamics Combined With Active Torque Distribution	210
8.1	Introduction	211
8.2	Methodology	212
8.2.1	Vehicle Geometry	212
8.2.2	Vehicle Model	213
8.2.2.1	Tyres	215
8.2.2.2	Aerodynamics	217
8.2.2.3	Powertrain	219
8.2.3	Optimisation Framework	220
8.3	Results and Discussion	222
8.3.1	Aerodynamic Performance	222
8.3.2	Lap Times	224
8.3.3	Vehicle Design Parameters	224
8.3.4	Vehicle Performance	225
8.3.5	Control Inputs	227
8.3.6	Tyre Characteristics	231
8.4	Conclusion	234
	References	235
9	General Discussion	240
9.1	Synthesis of Research Phases	240
9.1.1	Phase 1: Development of Methodological Foundation	240
9.1.1.1	Approach	240
9.1.1.2	Key Findings	241
9.1.1.3	Comparative Remarks	241
9.1.1.4	Summary	242

9.1.2	Phase 2: Aerodynamic Analysis and Active Aerodynamic Design Evaluation	242
9.1.2.1	Approach	243
9.1.2.2	Key Findings	243
9.1.2.3	Comparative Remarks	244
9.1.2.4	Summary	245
9.1.3	Phase 3: Performance Optimisation and Analysis of Active Aerodynamic Systems	246
9.1.3.1	Approach	246
9.1.3.2	Key Findings	246
9.1.3.3	Comparative Remarks	247
9.1.3.4	Summary	248
9.2	Research Contributions and Implications	249
9.3	Future Work	252
10	Conclusion	254

List of Figures

2.1	Illustration of the <i>DrivAer hp-F</i> model.	15
2.2	Illustration of the <i>WT</i> computational domain.	17
2.3	Illustration of the <i>wTS</i> computational domain.	17
2.4	Illustration of the <i>woTS</i> computational domain.	18
2.5	View of the medium-resolution mesh on the <i>wTS</i> installation.	19
2.6	Dynamic pressure comparison between the <i>WT</i> and <i>wTS</i> installations. . .	23
2.7	Lift and drag coefficient comparison on the <i>WT</i> installation.	24
2.8	Lift and drag coefficient error on the <i>WT</i> installation.	25
2.9	Upper body pressure distribution on the <i>WT</i> installation.	26
2.10	Upper body pressure contours on the <i>WT</i> installation.	26
2.11	Underbody pressure distribution on the <i>WT</i> installation.	27
2.12	Underbody pressure contours on the <i>WT</i> installation.	28
2.13	Streamwise velocity contours on the slant of the <i>WT</i> installation.	29
2.14	Streamwise velocity profiles on the slant of the <i>WT</i> installation.	30
2.15	Streamwise shear stress contours on the diffuser of the <i>WT</i> installation. .	31
2.16	Streamwise velocity contours on the diffuser of the <i>WT</i> installation. . . .	31
2.17	Wake total pressure contours on the <i>WT</i> installation.	32
2.18	Wake pressure losses and vertical position on the <i>WT</i> installation.	33
2.19	<i>wTS</i> installation forces normalised by <i>WT</i> installation results.	35
2.20	Surface pressure comparison between the <i>WT</i> and <i>wTS</i> installations. . .	36
2.21	Quantitative wake comparison between the <i>WT</i> and <i>wTS</i> installations. . .	37
2.22	Qualitative wake comparison between the <i>WT</i> and <i>wTS</i> installations. . .	38
2.23	Correction methods on lift and drag coefficients.	39
2.24	Flow field comparison between the <i>wTS</i> and <i>woTS</i> installations.	39
3.1	Illustration of the <i>DrivAer hp-F</i> model.	49
3.2	Description of the multichannel diffuser.	50
3.3	Illustration of the computational wind tunnel domain.	51
3.4	View of the mesh strategy	52
3.5	Comparison of lift and drag coefficients between RANS and DDES. . . .	54
3.6	Comparison of underbody pressure contours between RANS and DDES. .	55
3.7	Comparison of vorticity in the diffuser between RANS and DDES. . . .	56
3.8	Pressure contours on the underbody.	57
3.9	Pressure distribution along a near-symmetry line on the underbody. . . .	58
3.10	Vorticity contours on frontal planes in the diffuser.	59
3.11	Streamwise shear stress contours on the diffuser.	59
3.12	Velocity contours on frontal planes in the diffuser.	60

4.1	Illustration of the <i>DrivAer hp-F</i> model configurations.	70
4.2	Experimental setup in the 8x6 Wind Tunnel at Cranfield University. . . .	71
4.3	Repeatability of the aerodynamic force measurements.	72
4.4	Experimental setup for the wake total pressure measurements.	73
4.5	Repeatability of the wake total pressure measurements.	74
4.6	Locations of the surface pressure taps.	75
4.7	Repeatability of the surface pressure measurements.	75
4.8	Illustration of the computational wind tunnel domain.	76
4.9	View of the mesh strategy.	77
4.10	Experimental and numerical aerodynamic force coefficients.	79
4.11	Experimental and numerical surface pressure coefficient distributions. . .	81
4.12	Experimental and numerical wake total pressure coefficient distributions. .	82
4.13	Experimental total lift, front axle lift, and rear axle lift coefficients. . . .	84
4.14	Experimental drag and side force coefficients.	86
4.15	Numerical flow field visualisations.	87
4.16	Numerical surface pressure coefficient distributions on the vehicle body. .	90
4.17	Experimental surface pressure coefficient distributions on the slant. . . .	92
4.18	Numerical surface pressure coefficient distributions on the rear wing. . . .	93
4.19	Experimental total pressure coefficient distributions on wake planes. . . .	99
5.1	Illustration of the <i>DrivAer hp-F</i> model.	110
5.2	Illustration of a steady-state cornering condition.	111
5.3	Moving ground setup in the 8x6 Wind Tunnel at Cranfield University. . .	112
5.4	Open-road computational domain with cornering boundary conditions. . .	114
5.5	Illustration of the computational wind tunnel domain.	114
5.6	Illustration of the overset component mesh interfaces.	116
5.7	Contour plot of element sizes in the open-road domain.	116
5.8	Lift and drag coefficient comparison from experiments and simulations. .	118
5.9	Pressure distributions from open-road and wind tunnel simulations. . . .	119
5.10	Lift coefficients for the vehicle and rear wing.	120
5.11	Drag coefficients for the vehicle and rear wing.	121
5.12	Pressure coefficient distribution in straight-line conditions.	122
5.13	Pressure coefficient distribution in a 5.75L radius corner.	122
5.14	Pressure coefficient distribution in a 2.9L radius corner.	123
5.15	Wake structure visualisations in straight-line and cornering.	124
5.16	Total pressure coefficient distribution and streamlines on wake planes. .	125
5.17	Pressure coefficient distribution at the rear wing with vortex labels. . . .	126
5.18	Pressure coefficient distribution on the rear wing's upper surface.	126
5.19	Pressure coefficient distribution on the rear wing's lower surface.	127
5.20	Pressure coefficient distribution on the vehicle's upper surfaces.	129

6.1	Flowchart of the chapter's research process.	139
6.2	Illustration of the <i>DrivAer hp-F</i> configurations.	139
6.3	Moving ground setup in the 8x6 Wind Tunnel at Cranfield University. . .	140
6.4	Computational domain with cornering boundary conditions.	142
6.5	Illustration of the overset component mesh interfaces.	143
6.6	Contour plot of element sizes near the vehicle.	144
6.7	Vehicle dynamics model workflow.	145
6.8	Schematic illustration of the vehicle model.	147
6.9	Schematic illustration of lateral load transfer.	149
6.10	Lateral tyre force vs. slip angle and vertical load.	153
6.11	Experimental results of aerodynamic forces on the baseline <i>DrivAer hp-F</i> . .	154
6.12	Cornering performance of the baseline <i>DrivAer hp-F</i>	155
6.13	Lateral tyre forces for the baseline <i>DrivAer hp-F</i>	156
6.14	Downforce coefficient map with highlighted operating lines and settings. .	157
6.15	Downforce and drag coefficients for the split rear wing concept.	159
6.16	Pressure distribution on the split rear wing concept's (5,5) state.	160
6.17	Pressure distribution on the split rear wing concept's (1,5) state.	160
6.18	Downforce on in- and outside tyres of the split rear wing concept.	161
6.19	Aerodynamic roll moment coefficients for the split rear wing concept. . .	162
6.20	Maximum cornering velocities for the split rear wing concept.	163
6.21	Lateral tyre forces for the split rear wing concept.	164
6.22	Downforce and drag coefficients for the tilt rear wing concept.	165
6.23	Pressure distribution on the tilt rear wing concept's (5,5) state.	166
6.24	Pressure distribution on the tilt rear wing concept's (1,5) state.	166
6.25	Downforce on in- and outside tyres of the tilt rear wing concept.	167
6.26	Aerodynamic roll moment coefficients for the tilt rear wing concept. . . .	168
6.27	Maximum cornering velocities for the tilt rear wing concept.	169
6.28	Lateral tyre forces for the tilt rear wing concept.	170
7.1	Illustration of the high-performance vehicle concept.	180
7.2	Illustration of the computational domain.	181
7.3	Contour plot of element sizes near the vehicle.	182
7.4	Aerodynamic systems interdependencies on lift, drag and side force. . . .	183
7.5	Schematic illustration of the vehicle model.	184
7.6	Influence of step size on lap time and computation time.	194
7.7	Influence of individual elements on aerodynamic performance.	196
7.8	Influence of front and rear wing actuation on aerodynamic performance. .	197
7.9	Optimal racing line for configuration 1.	198
7.10	Velocity, delta velocity, steering input, and wheel torque profiles.	199
7.11	GG-diagrams of each configuration compared to configuration 1.	200
7.12	Optimal aerodynamic control inputs and load transfer profiles.	202

8.1	Illustration of the high-performance vehicle concept.	213
8.2	Schematic illustration of the vehicle model.	214
8.3	Influence of individual elements on aerodynamic performance.	223
8.4	Optimal racing lines for C1 and C1-ATD.	225
8.5	Understeer characteristics for all vehicle configurations.	226
8.6	Delta velocity profiles for all vehicle configurations.	226
8.7	GG-diagrams for all vehicle configurations.	227
8.8	Steering angle and total wheel torque for all vehicle configurations.	228
8.9	Individual wheel torque profiles for C4-ATD.	229
8.10	Aerodynamic control input profiles for all vehicle configurations.	230
8.11	Tyre forces for the original and optimised nominal loads.	231
8.12	GG-diagrams with original and optimised tyres.	233
8.13	Tyre forces during cornering exit acceleration and trail braking.	233

List of Tables

2.1	Main dimensions of the <i>DrivAer hp-F</i> model.	15
2.2	Overview of the wind tunnel air properties.	16
2.3	Overview of the mesh properties.	19
2.4	Grid convergence study results.	20
3.1	Main dimensions of the <i>DrivAer hp-F</i> model.	50
3.2	Overview of the wind tunnel air properties.	52
4.1	Main dimensions of the <i>DrivAer hp-F</i> model.	70
4.2	Overview of the reference operating conditions for CFD simulations. . . .	77
5.1	Main dimensions of the <i>DrivAer hp-F</i> model.	110
5.2	Overview of the CFD simulation settings for the open-road domain. . . .	115
6.1	Main dimensions of the <i>DrivAer hp-F</i> configurations.	140
6.2	Overview of the CFD simulation settings for each cornering condition. . .	143
6.3	Overview of the vehicle parameters.	146
6.4	Overview of the tyre model coefficients.	151
6.5	Operating settings for the split front wing.	157
6.6	Operating settings for the split rear wing.	158
6.7	Operating settings for the tilt rear wing.	158
7.1	Overview of the CFD simulation settings.	182
7.2	Overview of the vehicle parameters.	185
7.3	Overview of the tyre model coefficients.	188
7.4	Overview of the aerodynamic relationship coefficients.	190
7.5	Overview of the rate limits and regularisation weight factors.	194
7.6	Lap time results and aerodynamic settings for all vehicle configurations. .	197
7.7	Optimal vehicle design parameter results.	203
8.1	Overview of the vehicle configurations.	212
8.2	Overview of the vehicle parameters.	215
8.3	Overview of the tyre model coefficients.	217
8.4	Overview of the aerodynamic relationship coefficients.	219
8.5	Overview of the rate limits and regularisation weight factors.	222
8.6	Lap time and optimal design parameter results.	224
8.7	Lap times with original and optimised tyres.	232

List of Abbreviations

2D	Two-Dimensional
AALB	Active Aerodynamic Load Balancing
ATD	Active Torque Distribution
AWD	All-Wheel-Drive
BL1	First Boundary Layer Suction System
BL2	Second Boundary Layer Suction System
CFD	Computational Fluid Dynamics
COG	Centre of Gravity
CORD	Cranfield Online Research Data
COR	Centre of Rotation
CTU	Convective Time Unit
DDES	Delayed Detached Eddy Simulation
DES	Detached Eddy Simulation
DRS	Drag Reduction System
Exp	Experiment
GCI	Grid Convergence Index
IPOPT	Interior Point Optimizer
LES	Large Eddy Simulation
MAD	Mean Absolute Deviation
MLTP	Minimum Lap Time Problem
NLP	Non-Linear Program
OCP	Optimal Control Problem
OS	Overhead Strut
RANS	Reynolds-Averaged Navier-Stokes
RE	Richardson Extrapolation
RB	Rolling Belt
SRS	Scale-Resolving Simulation
URANS	Unsteady Reynolds-Averaged Navier-Stokes
WT	Wind Tunnel
WS	Wheel Supports
wTS	with Top Strut
woTS	without Top Strut

List of Symbols

Global Symbols

Symbol	Description	Unit
A_{ref}	Vehicle reference area	m^2
b_{RW}	Rear wing span	m
c_{RW}	Rear wing chord length	m
C_{D}	Drag coefficient	-
C_{L}	Lift coefficient	-
C_{S}	Side force coefficient	-
C_{p}	Static pressure coefficient	-
C_{pT}	Total pressure coefficient	-
CS	Car scale	%
F_{D}	Drag force	N
F_{L}	Lift force	N
F_{S}	Side force	N
g	Gravitational acceleration	m/s^2
H	Vehicle height	m
h_0	Reference ride height	m
I_{T}	Turbulent intensity	%
L	Vehicle length	m
l_{T}	Turbulent length scale	m
P_{atm}	Atmospheric pressure	Pa
P_{dyn}	Dynamic pressure	Pa
P_{op}	Operating pressure	Pa
P_{s}	Static pressure	Pa
P_{t}	Total pressure	Pa
T	Temperature	$^{\circ}\text{C}$
t	Time	s
U_{∞}	Freestream velocity	m/s
v	Velocity	m/s

Symbol	Description	Unit
W	Vehicle width	m
x, y, z	Longitudinal, lateral, and vertical coordinates	m
y^+	Non-dimensional wall distance	-
μ	Dynamic viscosity	kg/(m·s)
ρ	Density	kg/m ³
ω	Angular velocity	rad/s

Chapter-Specific Symbols

Chapter 2

Symbol	Description	Unit
C_{Dc}	Corrected drag coefficient	-
C_{Lc}	Corrected lift coefficient	-
f	Discrete solution	-
h	Ride height	m
h_c	Average cell size	mm
N	Number of cells	-
p	Order of accuracy	-
q_c	Corrected dynamic pressure	Pa
q	Measured dynamic pressure	Pa
r	Refinement factor	-
χ	Non-dimensional time-step	-

Chapter 3

Symbol	Description	Unit
h	Ride height	m
L_D	Diffuser length	m
U	Flow velocity	m/s
χ	Non-dimensional time-step	-

Chapter 4

Symbol	Description	Unit
Q	Q-criterion	s ⁻²

Chapter 5

Symbol	Description	Unit
R	Corner radius	m
x_c, y_c	Longitudinal and lateral coordinates of the corner centre	m
ψ	Angle of the relative airflow	deg

Chapter 6

Symbol	Description	Unit
a	Lateral acceleration	m/s ²
B_y, C_y, D_y, E_y	Stiffness factor, shape factor, peak value, curvature factor	-
d	Distance between roll axis and COG	m
df_z	Normalised change in vertical wheel load	-
F_c	Centrifugal force	N
$F_{DF,fl}, F_{DF,fr}$	Aerodynamic downforce at front left and right wheels	N
$F_{DF,rl}, F_{DF,rr}$	Aerodynamic downforce at rear left and right wheels	N
F_{Sf}, F_{Sr}	Aerodynamic side force at front/rear axle	N
$F_{Tf,1}, F_{Tr,1}$	Lateral load transfer due to front/rear unsprung mass	N
$F_{Tf,2}, F_{Tr,2}$	Lateral load transfer via suspension links at front/rear axle	N
$F_{Tf,3}, F_{Tr,3}$	Lateral load transfer due to roll moment at front/rear axle	N
F_y	Lateral tyre force	N
$F_{yf,req}, F_{yr,req}$	Required lateral force at front and rear axles	N
$F_{Z,fl}, F_{Z,fr}$	Vertical force at front left and right wheels	N
$F_{Z,rl}, F_{Z,rr}$	Vertical force at rear left and right wheels	N
F_{Z0}	Nominal vertical wheel load	N
g	Gravitational acceleration	m/s ²
h	Roll centre height at COG	m
h_f, h_r	Front and rear roll centre height	m
h_{COG}	COG height	m
h_{uf}, h_{ur}	Front and rear unsprung mass height	m
$k_\phi, k_{\phi f}, k_{\phi r}$	Total, front, and rear axle roll stiffness	Nm/rad
l	Wheelbase length	m
l_f, l_r	Distance from front/rear axle to COG	m
m	Total vehicle mass	kg
m_b	Sprung mass of vehicle body	kg
m_d	Driver mass	kg
m_s	Total sprung mass	kg

Symbol	Description	Unit
m_{uf}, m_{ur}	Front and rear unsprung mass	kg
M_f, M_r	Roll moment reacted at front/rear axle	Nm
$M_{A,rc}$	Aerodynamic roll moment about roll centre	Nm
$p_{CY1}...p_{VY4}$	Pacejka tyre model coefficients	-
t_f, t_r	Front and rear track width	m
V	Vehicle velocity	m/s
W_f, W_r	Front and rear static wheel loads	N
α_f, α_r	Front and rear wheel slip angle	rad
α_A, α_T	Angle of attack and tilt angle of tilt wing	deg
α_I, α_O	Angles of the inside and outside split wing elements	deg
β	Vehicle sideslip angle	rad
γ	Wheel camber angle	rad
δ	Steering angle	rad
$\lambda_{\mu y}... \lambda_{Ey}$	Pacejka tyre model scaling factors	-
ϕ	Sprung mass roll angle	rad

Chapters 7 - 8

Symbol	Description	Unit
k	Subscript for longitudinal (x), lateral (y), or vertical (z) direction in the vehicle model	-
i	Subscript for front (f) or rear (r) axle location in the vehicle model	-
j	Subscript for left (l) or right (r) wheel position in the vehicle model	-
w	Superscript for left front wing angle (FW-L), right front wing angle (FW-R), rear wing angle (RW), and rear wing tilt (TW) in the vehicle model	-
u	Subscript for lift (L), drag (D), or side (S) force in the vehicle model	-
v	Subscript for front, rear, left or right force distribution location in the vehicle model	-
$\mathbf{b}$	Vector defining vehicle start and finish conditions	-
$\mathbf{B}$	Weight matrix	-
B_k, C_k, E_k	Stiffness, shape, and curvature factors in direction k	-
C_f	Friction coefficient	-
$C_{L,ij}$	Lift coefficient for wheel ij	-

Symbol	Description	Unit
$C_{u,v}, C_{u,v}^0$	Combined and nominal coefficients for force component u, v	-
$C_{\Gamma y, i}$	Fraction of total lateral load transfer reacted at axle i	-
d	Distance between roll axis and COG	m
$D_{k, ij}$	Peak value in direction k for wheel ij	-
ds_k	Track segment length with k indexing track segments	m
$\mathbf{f}$	Vector function describing vehicle dynamics	-
F_D, F_D^0	Total and nominal aerodynamic drag force	N
$F_{k, ij}$	Tyre force in direction k at wheel ij	N
$F_{L, ij}$	Aerodynamic lift force at wheel ij	N
F_R	Rolling resistance force	N
$F_S, F_{S, i}$	Aerodynamic side force on the entire vehicle and axle i	N
F_{z0}	Nominal vertical wheel load	N
$\mathbf{h}$	Vector function defining optimisation constraints	-
h_{COG}	COG height	m
h_i	Roll centre height at axle i	m
h_{RW}	Rear wing height	m
$h_{u, i}$	Unsprung mass height at axle i	m
I_w	Wheel rotational inertia	kg m ²
I_z	Yaw moment of inertia	kg m ²
J	Objective function	-
$k_\phi, k_{\phi i}$	Total and axle i roll stiffness	Nm/rad
l	Wheelbase length	m
l_i	Distance from axle i to COG	m
ℓ	Indexing term for polynomial expressions	-
m	Total vehicle mass	kg
m_s	Total sprung mass	kg
$m_{u, i}$	Unsprung mass at axle i	kg
n	Distance perpendicular to track centreline	m
N	Number of track segments	-
$N_{u,v}^w$	Polynomial order for force component u, v of aerodynamic element w	-
$\mathbf{p}$	Vector of vehicle design parameters	-
P_{D2}	Pacejka coefficient for friction variation with load	-
$P_{e, \max}$	Maximum electric motor power	kW
$P_{u,v}^w$	Polynomial coefficient for force component u, v of aerodynamic element w	-

Symbol	Description	Unit
r	Vehicle yaw rate	rad/s
r_m	Regularisation weight factor for the m -th derivatives of optimisation variables	-
s	Travelled distance along track centreline	m
s_0, s_f	Start and finish track positions	m
S_f	Scaling factor for distance-based equations	-
$\mathbf{S}_{f,k}$	Vector function of objective values at collocation points with k indexing track segments	-
s_{ij}	Slip ratio of wheel ij	-
$s_{k,j}$	Collocation points with k indexing track segments and j the Legendre polynomial order	-
T_b	Braking torque averaged over all wheels	Nm
T_e	Electric motor torque	Nm
$T_{e,max}$	Maximum electric motor torque	Nm
T_{ij}	Applied wheel torque at wheel ij	Nm
t_w	Track width	m
$\mathbf{u}$	Vector of control inputs	-
U_k	Discrete control inputs with k indexing track segments	-
v_{ij}	Effective velocity of wheel ij	m/s
v_x, v_y	Vehicle longitudinal and lateral velocity	m/s
$\mathbf{x}$	Vector of vehicle states	-
X_k	Discrete vehicle states with k indexing track segments	-
$\mathbf{y}$	Vector of auxiliary variables	-
Y_k	Discrete auxiliary variables with k indexing track segments	-
α_{ij}	Slip angle at wheel ij	rad
α_w	Operating angle of aerodynamic element w	deg
γ_b	Brake torque distribution	-
γ_g	Final gear ratio	-
γ_{ij}	Active torque distribution for wheel ij	-
Γ_k	Total load transfer in direction k	N
$\Gamma_{k,ij}$	Load transfer in direction k at wheel ij	N
$\gamma_{k\phi}$	Roll stiffness distribution	-
γ_T	Drive torque distribution	-
γ_{z0}	Normal tyre load shift	-
δ	Steering angle	rad
$\Delta C_{u,v}^w$	Coefficient for the effect of aerodynamic element w on force component u, v	-

Symbol	Description	Unit
ε	Angle relative to track centreline tangent	rad
κ	Track centreline curvature	m^{-1}
μ_0	Tyre friction coefficient at nominal load	-
μ_{ij}	Tyre friction coefficient at wheel ij	-
$\Omega_{e,max}$	Maximum electric motor speed	RPM
Ω_{ij}	Angular velocity of wheel ij	rad/s

Chapter 1

Introduction

This thesis is presented in a paper-format style, where each chapter constitutes a self-contained study, including its specific abstract, introduction with literature review, methodology, discussion of results, and conclusion. Collectively, the chapters cover the overarching aims and objectives of this research. This chapter begins by introducing the research context in Section 1.1, followed by a statement of the aims and objectives in Section 1.2. It then outlines the thesis structure in Section 1.3 and provides an overview of the individual chapters in Section 1.4. Finally, it provides a summary of the research contributions in Section 1.5 and a list of related publications in Section 1.6.

1.1 Context

The motorsport industry has long been recognised as a platform for innovation, driving the development of cutting-edge methodologies and technologies that often extend beyond the race track and find application in the mainstream automotive sector. Among the many disciplines contributing to this innovation, aerodynamics has consistently played a pivotal role in shaping vehicle performance. The discipline of aerodynamics focuses on the vehicle's interaction with airflow, which significantly influences key aspects such as handling, efficiency, and overall performance.

The aerodynamic performance of high-performance vehicles is typically quantified by downforce and drag. Downforce is a downward force exerted by airflow onto the vehicle that enhances grip. Drag, on the other hand, is a resistive force exerted by airflow that opposes the vehicle's forward motion. While downforce is crucial for improving dynamic performance in demanding driving scenarios such as cornering, it comes at the cost of higher drag, which reduces the vehicle's efficiency and top speed.

The design of high-performance vehicles incorporates various features to manipulate airflow and generate the desired aerodynamic forces. These include components such as splitters, diffusers, spoilers, front wings, rear wings, and specially designed underbodies. Although the aerodynamic performance of these components has been improved over time, static design implementations inherently impose a trade-off between downforce and drag, requiring design compromises to balance performance and efficiency.

These limitations are addressed by the evolution from static to active aerodynamic systems, which can adjust the configuration of aerodynamic devices based on the vehicle's operating conditions to improve performance. Active aerodynamic systems have already gained significant interest from the automotive and motorsport industries, where common implementations of this technology are active splitters, spoilers, or wings, which transition between a high-downforce and low-drag state in response to a condition threshold.

A profound example is the Drag Reduction System (DRS) in Formula One, introduced in 2011, which enables the rear wing to switch to a low-drag configuration on designated high-speed track sections and within specified overtake conditions to improve straight-line performance. The next advancement in Formula One will be the introduction of active front- and rear wings in 2026, which can be operated in a high-downforce mode for maximum dynamic performance and a low-drag mode to improve straight-line speed.

The next step in the evolution of active aerodynamic systems involves the transition in functionality from discrete operating settings and threshold-based control to systems with continuous configurability, capable of adjusting in real-time and continuously across their excursion range. This capability allows aerodynamic performance to be precisely tailored to the vehicle's specific operating condition, with the potential to significantly enhance safety, efficiency, handling, and overall performance in both the automotive and motorsport industries.

Additionally, while conventional systems primarily influence longitudinal aerodynamic balance (front-to-rear), recent innovations by manufacturers such as Zenvo and Lamborghini have introduced systems capable of affecting lateral aerodynamic balance (left-to-right) as well. Referred to as "2D active aerodynamic systems" in this research, these systems aim to increase downforce on tyres that become offloaded by lateral load transfer during cornering. This capability could potentially offer unprecedented levels of stability and performance in dynamic driving scenarios.

Despite the promising potential of advanced active aerodynamics, publicly available research remains limited in scope, leaving substantial gaps in their design, evaluation, and fundamental understanding for vehicle applications. While each chapter of this thesis provides an extensive literature review to support its specific research objectives, the overarching research gaps emerge across three main themes that are summarised below.

Firstly, the development of active aerodynamic systems requires reliable and cost-effective Computational Fluid Dynamics (CFD) simulations, supported by robust wind tunnel correlation practices. Existing methodological studies have predominantly concentrated on turbulence model evaluation and wind tunnel correlation for simplified geometries or passenger vehicles, with a primary focus on drag. These insights are not directly transferable to the highly complex flow fields of high-performance vehicles, which typically feature various downforce-generating devices. As a result, the predictive accuracy of widely used turbulence models and the effects of wind tunnel conditions on high-performance vehicle aerodynamics remain insufficiently investigated, leaving a critical gap in understanding the capabilities of available methods for high-performance vehicle applications.

Secondly, the design of effective active aerodynamic systems requires a comprehensive understanding of their behaviour under dynamic operating conditions, such as yaw and cornering. Most aerodynamic studies focus on straight-line scenarios, with research on dynamic conditions typically limited to simplified geometries or passenger vehicles. The findings from such work cannot be directly applied to the more intricate flow fields of high-performance vehicles. Furthermore, while isolated aerodynamic devices have been shown to exhibit pronounced sensitivity to oncoming flow conditions, the combined interaction of such devices with the vehicle body under dynamic conditions remains largely unexplored. This limits the understanding of their influence on overall aerodynamic performance, leaving a significant gap in the knowledge of their vehicle-integrated aerodynamic behaviour in realistic dynamic operating scenarios.

Lastly, exploring the performance potential of active aerodynamic systems and demonstrating their performance-enhancing mechanisms requires comprehensive evaluations under representative driving scenarios. Existing research has mainly examined conventional aerodynamic configurations and often relied on simplified performance simulations. While these studies have provided indicative trends, they have not addressed the critical interactions between aerodynamic loads and vehicle dynamics with sufficient fidelity, offering only limited insight into the mechanisms driving the performance gains. Consequently, in-depth analyses of active aerodynamics, particularly advanced 2D systems, using comprehensive and representative simulation frameworks remain scarce, leaving a critical gap in understanding their true potential and the underlying mechanisms that translate aerodynamic functionality into vehicle performance improvements.

This PhD thesis addresses these challenges with a series of self-contained studies that combine CFD simulations, wind tunnel experiments, active aerodynamic system designs, and performance optimisation frameworks. Together, these studies advance the understanding of active aerodynamic systems and demonstrate their potential benefits for high-performance vehicle applications. The subsequent sections will outline the research aims and objectives, along with the adopted structure and a summary of the key contributions.

1.2 Aims and Objectives

This research aims to explore the performance potential of active aerodynamic systems for high-performance vehicles in realistic driving scenarios. To support this overarching aim, the research seeks to establish a cost-effective workflow for aerodynamic assessments and investigate the impact of dynamic operating conditions on the aerodynamic performance of high-performance vehicles. These supporting aims provide the foundation for analysing both conventional and innovative 2D active aerodynamic systems, capable of modulating aerodynamic balance both longitudinally and laterally. The following objectives are formulated to achieve these research aims:

1. Develop and validate a cost-effective Computational Fluid Dynamics (CFD) workflow for high-performance vehicle aerodynamics, addressing meshing strategies, turbulence model selection, and wind tunnel correlation methodologies.
2. Investigate the aerodynamic characteristics of various high-performance vehicle configurations in yaw conditions using wind tunnel experiments and CFD simulations.
3. Analyse the aerodynamic behaviour of a high-performance vehicle and its rear wing in both straight-line and cornering conditions using wind tunnel experiments and CFD simulations.
4. Design multiple active aerodynamic systems, including 2D concepts, and evaluate their influence on aerodynamic balance and cornering performance by coupling CFD-derived aerodynamic data with vehicle dynamics simulations.
5. Assess the performance potential of the most promising 2D and conventional active aerodynamic systems, identified in prior analyses, through minimum lap time simulations that combine detailed vehicle modelling with comprehensive CFD-derived aerodynamic data.
6. Examine the influence of integrating active aerodynamics with active torque distribution on vehicle performance through minimum lap time simulations incorporating detailed vehicle and powertrain models combined with comprehensive CFD-derived aerodynamic data.

1.3 Thesis Structure

This thesis is structured into three interdependent research phases, each essential to achieving the overarching aims. It progresses from establishing a robust methodological foundation to conducting aerodynamic analyses and evaluating active aerodynamic designs in dynamic operating conditions, ending with the optimisation and performance assessment of active aerodynamic systems. Each phase builds upon the previous, ensuring a cohesive and systematic investigation of high-performance vehicle aerodynamics and the impact of active aerodynamic systems. The research phases are detailed below, with an overview of the individual chapters provided in Section 1.4.

The first phase establishes a methodological framework for aerodynamic analyses of high-performance vehicles. Chapters 2 and 3 focus on evaluating and validating CFD simulation methods and wind tunnel correlation strategies, leading to the development of a cost-effective numerical workflow. This phase ensures that all subsequent investigations are built upon a rigorous and validated methodology, balancing numerical accuracy with computational efficiency to effectively capture high-performance vehicle aerodynamics.

Building on the methodological groundwork, the second phase investigates aerodynamic behaviour under dynamic operating conditions, specifically in yaw and cornering scenarios. Chapters 4 and 5 employ extensive wind tunnel experiments and CFD simulations to assess how these conditions influence aerodynamic performance and flow field characteristics. Additionally, Chapter 6 examines the influence of various active aerodynamic systems on vehicle aerodynamics and performance during cornering, establishing a foundation for the third phase.

The third phase advances toward a performance-focused investigation of active aerodynamic systems, building on the methodological foundation and insights from the preceding phases. Chapter 6 identifies promising active aerodynamic configurations, which are subsequently evaluated in Chapters 7 and 8 using a minimum lap time simulation framework. These studies analyse the influence of various active aerodynamic systems on overall vehicle performance across different powertrain configurations. This final phase bridges aerodynamics and vehicle performance, ensuring a comprehensive assessment of active aerodynamic systems for high-performance vehicle applications.

1.4 Chapter Overview

Chapter 2 addresses Objective 1 by developing a cost-effective CFD workflow for aerodynamic analyses of high-performance vehicles. The workflow is established through grid independence and Reynolds-Averaged Navier-Stokes (RANS) turbulence model evaluation studies, validated against experimental data and high-fidelity Delayed Detached Eddy Simulation (DDES) results. Additionally, the chapter introduces a numerical approach for assessing and correcting wind tunnel-induced effects. Serving as the methodological foundation, this chapter supports the CFD simulations and wind tunnel correlation processes employed in the subsequent chapters.

Chapter 3 contributes to Objective 1 by evaluating the predictive capabilities of the established cost-effective RANS-based CFD methodology against experimental data and high-fidelity DDES results at critically low ride heights, where unsteady flow phenomena dominate. The study identifies strengths and limitations of each approach, demonstrating the necessity of high-fidelity simulations in highly unsteady flow conditions. Additionally, the chapter utilises the DDES simulations to analyse the impact of ride height on aerodynamic performance, focusing on the vortex dynamics in multichannel diffusers and their evolution with changing ride height.

Chapter 4 shifts the focus to operating conditions, addressing Objective 2 by analysing the aerodynamic behaviour of three high-performance vehicle configurations in yaw conditions. The chapter presents wind tunnel measurements of aerodynamic forces, surface pressures, and wake pressures across a range of yaw angles. Complemented by CFD simulations employing a methodology built upon previous chapters, the study provides a detailed analysis of yaw-induced flow field alterations and their impact on aerodynamic performance.

Chapter 5 expands the analysis of operating conditions by investigating the aerodynamic behaviour of a high-performance vehicle with a rear wing in cornering scenarios, addressing Objective 3. The chapter presents straight-line moving ground wind tunnel experiments measuring aerodynamic forces across a range of rear wing angles, along with CFD simulations using an extended methodology from previous chapters to replicate cornering conditions. The study provides a detailed analysis of the influence of cornering on the vehicle's flow field, including aerodynamic interactions with the rear wing, and the resulting effects on aerodynamic performance.

Chapter 6 shifts the focus to active aerodynamics, addressing Objective 4. It introduces 2D active aerodynamic concepts, specifically split and tilt wing designs, which can modulate aerodynamic balance both longitudinally and laterally. The study employs CFD simulations using the methodology established in the previous chapter to evaluate their aerodynamic behaviour in various cornering conditions. Additionally, the CFD data is coupled with a steady-state vehicle dynamics model to assess the impact of each 2D active aerodynamic concept on cornering performance. By comparing the aerodynamic characteristics and performance potential of these designs, this chapter establishes the foundation for the performance-focused analyses in the following chapters.

Chapter 7 advances the investigation of active aerodynamic systems by assessing their performance in representative on-track conditions, thereby addressing Objective 5. The study extends an established minimum lap time simulation framework through the integration of CFD-derived aerodynamic data with a high-fidelity vehicle model, enabling the optimisation and evaluation of multiple active aerodynamic configurations. The analysis is performed on the Circuit de Barcelona-Catalunya to quantify performance improvements, characterise the underlying control strategies, and examine vehicle setup considerations for both conventional and 2D active aerodynamic systems.

Chapter 8 presents a complementary research direction, formulated in collaboration with the Vehicle Dynamics and Control group at Cranfield University, which addresses Objective 6 by investigating the performance potential of active aerodynamic systems when integrated with advanced powertrain architectures capable of active torque distribution (ATD). Building upon the minimum lap time simulation framework established in the preceding chapter, this study extends its capabilities to enable performance optimisation across various aerodynamic and torque distribution configurations. The analysis quantifies individual and synergetic lap time gains, investigates the underlying performance-enhancing mechanisms, and examines the influence of vehicle setup and tyre characteristics on their effectiveness.

Chapter 9 provides a general discussion on the research as a whole. It begins by synthesising the research phases, outlining their aims, methodologies, and key findings, while drawing comparisons between studies to offer a broader reflection on the research. It then discusses the research contributions and their practical implications before concluding with identified avenues for future work. Finally, Chapter 10 presents a high-level overview of the research conducted in this thesis.

1.5 Research Contributions

The research contributions and their implications are discussed in detail in Section 9.2. However, to provide an early overview, a summary is presented below.

1. Development of a cost-effective CFD workflow for high-performance vehicle aerodynamics, integrating a refined meshing strategy, turbulence model evaluation, and wind tunnel correlation methodology. This framework provides a structured reference for practical aerodynamic simulations, supporting both research and industry applications.
2. Identification of critical ride height dependencies in high-performance vehicle aerodynamics through numerical analysis, revealing vortex-driven diffuser performance mechanisms and the onset of vortex breakdown with its detrimental effects at critically low ride heights. These findings advance knowledge of underbody aerodynamics and inform their design for enhanced performance and stability.
3. Characterisation of yaw-induced aerodynamic effects through combined experimental and numerical investigations, identifying key flow field alterations and their impact on aerodynamic performance. These findings deepen the understanding of vehicle aerodynamics in yaw conditions and provide insights for improving aerodynamic design in dynamic operating conditions.
4. Analysis of the aerodynamics of a high-performance vehicle with a rear wing in straight-line and cornering conditions using wind tunnel experiments and CFD simulations, revealing corner-induced flow field changes and their impact on aerodynamic performance. This study refines cornering simulation methodologies and advances the understanding of vehicle aerodynamics during cornering, informing vehicle and rear wing design for improved performance in dynamic driving conditions.
5. Design and evaluation of two 2D active aerodynamic systems, developed as split and tilt wing concepts, using CFD simulations and steady-state cornering performance analysis, demonstrating their ability to modulate aerodynamic balance both longitudinally and laterally, along with their impact on cornering performance. These findings advance the understanding of 2D active aerodynamic systems and support research aimed at enhancing vehicle performance through active aerodynamics.
6. Optimisation and analysis of vehicle performance with active aerodynamic systems, including 2D configurations, using a minimum lap time simulation framework, identifying performance gains, optimal control strategies, and vehicle setup considerations. These findings advance knowledge on the performance potential of active aerodynamics and provide a foundation for further research and development on active aerodynamics for automotive and high-performance vehicles.
7. Optimisation and evaluation of active aerodynamics combined with active torque distribution using minimum lap time simulations, identifying individual and combined performance gains, key performance-enhancing mechanisms, and optimal vehicle setup and tyre characteristics. The findings advance the understanding of active aerodynamics integrated with different powertrain architectures and support future research into their application in automotive and high-performance vehicles.

8. Expansion of the limited experimental database on vehicle aerodynamics through publicly available datasets comprising wind tunnel measurements of forces, surface pressures, and wake pressures in yaw conditions, as well as force measurements at varying rear wing angles under moving ground conditions. These datasets serve as essential validation resources for aerodynamic research and development, with their active use in industry highlighting their practical relevance.
9. Extension of an existing minimum lap time simulation framework, enhancing its capabilities to optimise vehicle performance across active aerodynamic and torque distribution configurations, alongside parametric optimisation of vehicle setup and tyre characteristics. This publicly available extension serves as a valuable resource for future research on active aerodynamics and related vehicle systems, as well as for supporting decision-making in vehicle development and pre-event setup planning.

1.6 Publications

The chapters of this thesis correspond to peer-reviewed publications, with Chapters 2 to 7 already published, while Chapter 8 has been submitted and is under review at the time of writing. Minor augmentations have been made across chapters to enhance coherence and consistency. Citations are provided at the start of each chapter and an overview of the publications is presented below.

Journal Papers

1. Rijns S., Teschner TR, Blackburn K, Brighton J. Aerodynamic analysis of the ride height dependency of a high-performance vehicle equipped with a multichannel diffuser in ground effect. SAE Technical Paper 2023-01-5064. 2023. Available from: <https://doi.org/10.4271/2023-01-5064>.
2. Rijns S, Teschner TR, Blackburn K, Brighton J. Integrated numerical and experimental workflow for high performance vehicle aerodynamics. SAE Technical Paper 2024-01-5016. 2024. Available from: <https://doi.org/10.4271/2024-01-5016>.
3. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Experimental and numerical investigation of the aerodynamic characteristics of high-performance vehicle configurations under yaw conditions. *Physics of Fluids*. 2024;36(4):045112. Available from: <https://doi.org/10.1063/5.0196979>.
4. Rijns S, Teschner TR, Blackburn K, Brighton J. Effects of cornering conditions on the aerodynamic characteristics of a high-performance vehicle and its rear wing. *Physics of Fluids*. 2024;36(4):045119. Available from: <https://doi.org/10.1063/5.0204204>.
5. Rijns S, Teschner TR, Blackburn K, Brighton J. Performance analyses of active aerodynamic load balancing designs on high-performance vehicles in cornering conditions. *Physics of Fluids*. 2024;36(8):085199. Available from: <https://doi.org/10.1063/5.0222780>.
6. Rijns S, Teschner TR, Blackburn K, Siampis E, Brighton J. Optimising vehicle performance with advanced active aerodynamic systems. *Vehicle System Dynamics*. 2025; 1-29. Available from: <https://doi.org/10.1080/00423114.2025.2505619>.
7. Rijns S, Teschner TR, Siampis E, Blackburn K, Brighton J. Performance optimisation of high-performance vehicles with active aerodynamics and active torque distribution [in review]. Advanced Vehicle Engineering Centre. Cranfield University. 2025.

Conference Papers

1. Rijns S, Teschner TR, Blackburn K, Brighton J. Comparative analysis of RANS and DDES methods for aerodynamic performance predictions for high performance vehicles at low ground clearances. UKACM 2023 Conference. Coventry, UK: UKACM; 2023.

Datasets

1. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Dataset DrivAer hp-F: force measurements in yaw conditions. Cranfield: CORD; 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/22436>
2. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Dataset DrivAer hp-F: surface pressure measurements in yaw conditions. Cranfield: CORD; 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/23606>
3. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Dataset DrivAer hp-F: wake total pressure measurements in yaw conditions. Cranfield: CORD; 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/23609>
4. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. DrivAer hp-F: spoiler & rear wing configurations geometry pack. Cranfield: CORD; 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/22433>
5. Rijns S, Teschner TR, Blackburn K, Brighton J. Dataset DrivAer hp-F: force measurements at various rear wing angles of attack. Cranfield: CORD; 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/22438>
6. Rijns S, Siampis E, Teschner TR, Blackburn K, Brighton J. Minimum lap time simulation framework: active aerodynamics and torque distribution. Cranfield: CORD; 2025. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/23604>

Chapter 2

Development of a CFD Workflow and Wind Tunnel Correlation Process

This chapter was originally published as:

Rijns S, Teschner TR, Blackburn K, Brighton J. Integrated numerical and experimental workflow for high performance vehicle aerodynamics. SAE Technical Paper 2024-01-5016. 2024. Available from: <https://doi.org/10.4271/2024-01-5016>.

Abstract

The automotive and motorsport industries continually strive to enhance the aerodynamic efficiency and performance of vehicles. The aerodynamic research and development processes necessitate a well-balanced use of accurate yet cost-effective simulation tools in combination with wind tunnel experiments. This study evaluates the predictive capabilities of four Reynolds-Averaged Navier-Stokes (RANS) turbulence models against experimental data and high-fidelity Delayed Detached Eddy Simulation (DDES) results for the *DrivAer hp-F*, a high-performance variant of the widely adopted DrivAer model. Additionally, wind tunnel-induced effects on the vehicle's aerodynamic performance are investigated, and a novel Computational Fluid Dynamics (CFD) based blockage correction method is introduced to assess the applicability of conventional methods. The turbulence model evaluation reveals that among the RANS models, the $k-\omega$ SST model generally exhibits the highest relative accuracy to experimental force trends and the best agreement with more detailed flow predictions from DDES simulations. The numerical analysis of wind tunnel-induced effects shows that blockage increases downforce and drag by approximately 9% and 16%, respectively. Furthermore, interference effects from the overhead measurement system increase downforce by 4% and drag by 8%. The CFD-based correction method demonstrates that conventional methods estimate the dynamic pressure rise induced by blockage within about 3% but do not account for local effects on downforce and drag individually. Overall, this research advances prior work on automotive applications by contributing to the development of aerodynamic research methodologies tailored to the complex flow fields of high-performance vehicles.

Keywords: Aerodynamics; CFD; RANS; DDES; Wind Tunnel; Correlation; DrivAer

2.1 Introduction

The highly competitive nature of motorsport drives rapid advancements in methodologies and technologies, with innovations often influencing the broader automotive industry. A key focus in both sectors is the development of advanced aerodynamic designs to optimise vehicle efficiency and performance. Effective development of such designs requires the combined usage of Computational Fluid Dynamics (CFD) simulations and wind tunnel experiments. However, due to commercial sensitivity, publicly available knowledge on high-performance vehicle aerodynamics and the integration of numerical and experimental workflows remains limited.

Turbulence modelling is crucial in CFD, requiring a balance between computational efficiency and predictive accuracy (1). Reynolds-Averaged Navier-Stokes (RANS) models are widely used in the automotive and motorsport industries due to their relatively low computational cost. Research on automotive applications has shown that while RANS models provide reasonable accuracy in drag and surface pressure predictions, they generally struggle to capture lift forces and unsteady flow behaviour (2, 3, 4, 5, 6, 7). Furthermore, their predictive accuracy varied significantly, not only between different models but also for the same model when applied to different vehicle geometries, preventing consensus on the most suitable RANS model for automotive aerodynamic predictions.

Scale-Resolving Simulation (SRS) methods, particularly hybrid RANS and Large Eddy Simulation (LES) models such as Detached Eddy Simulation (DES) and its variants, have been shown to improve accuracy in capturing complex unsteady aerodynamic phenomena (2, 6, 7, 8, 9, 10, 11, 12, 13). Compared to RANS, these models demonstrated greater accuracy and consistency in predicting aerodynamic forces and capturing unsteady flow regions, such as wakes, with higher fidelity. However, their significantly higher computational cost limits their practicality for aerodynamic research and development in fast-paced environments.

Beyond numerical modelling, accurate correlation between CFD simulations and wind tunnel experiments is essential for aerodynamic development. While most CFD simulations assume open-road conditions, wind tunnel testing introduces facility-specific effects that can significantly alter aerodynamic performance. Among these, blockage and measurement system interference are key factors. Blockage effects arise due to test section confinement, which increases local flow velocity around the vehicle model (14, 15). Various blockage correction methods exist to correct for these effects, typically relying on generic geometrical relationships between the tested model and wind tunnel facility (16). Measurement system interference, on the other hand, can affect aerodynamic performance by obstructing airflow and thereby disturbing the vehicle's local flow field (17, 18, 19).

Through numerical recreation of open-road and wind tunnel conditions, studies on passenger vehicles and trucks have shown that blockage effects increase lift and drag forces, as well as intensify flow separation, with the severity of these effects typically growing with vehicle size (20, 21, 22, 23). However, the effectiveness of blockage correction methods has remained inconsistent across different vehicle geometries and wind tunnel facilities, suggesting that a universal correction approach may not be feasible (20, 21, 22, 24).

Similarly, measurement system interference effects on lift and drag forces have been shown to vary considerably depending on vehicle shape and the aerodynamic sensitivity of downstream components (17, 18, 19).

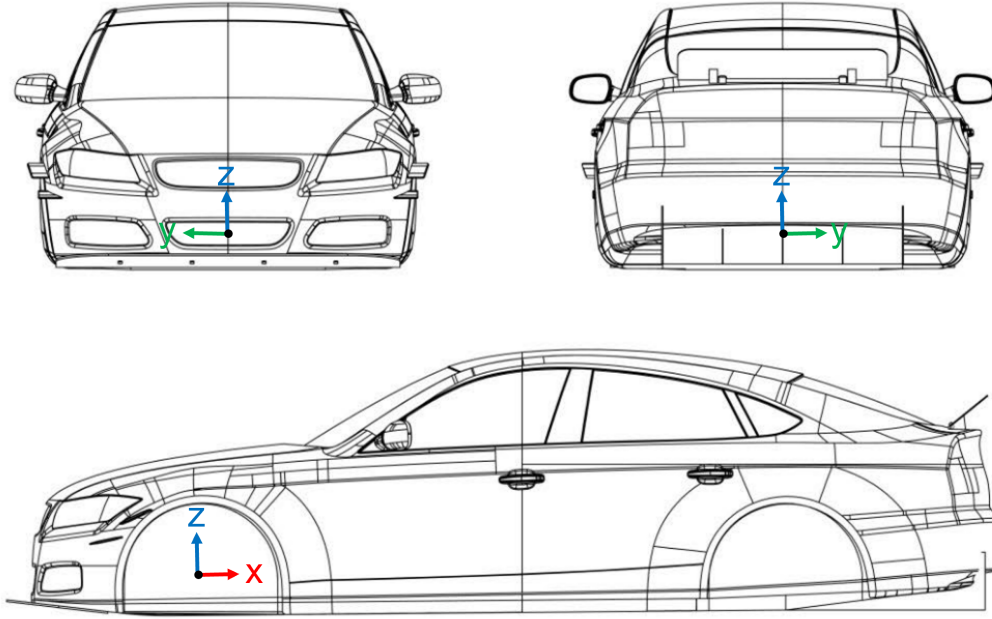
While extensive research has evaluated turbulence models for automotive aerodynamics, most studies were focused on passenger vehicles, with an emphasis on drag force predictions. Investigations into wind tunnel conditions have demonstrated the feasibility of numerically examining facility-induced effects, but such analyses were also largely limited to passenger vehicles, with blockage correction methods and measurement interference assessments primarily targeting drag. In contrast to passenger vehicles, high-performance vehicles employ various aerodynamic devices to generate downforce, resulting in considerably more complex flow fields. Consequently, the predictive accuracy of commonly used turbulence models and the impact of wind tunnel effects on the intricate flow fields and downforce characteristics of high-performance vehicles remain largely underexplored.

This study addresses these gaps by systematically evaluating the predictive capabilities of four commonly used RANS turbulence models for high-performance vehicle aerodynamics, benchmarking their predictions against experimental force measurements and high-fidelity Delayed Detached Eddy Simulation (DDES) results. Additionally, CFD simulations of both open-road and wind tunnel conditions are conducted to assess the influence of blockage and measurement system interference effects on aerodynamic performance. A novel CFD-based approach is presented to evaluate the accuracy and applicability of conventional blockage correction methods. Through detailed analyses of aerodynamic performance, turbulence models' predictive capabilities, and challenges in aligning numerical and experimental results, this research advances the methodological framework for high-performance vehicle aerodynamics. The findings provide critical insights to enhance aerodynamic development workflows, benefiting both academic research and industrial applications.

2.2 Methodology

2.2.1 Vehicle Geometry

Aerodynamic research in the automotive industry often relies on standardised reference vehicle models, such as the DrivAer model (25). This study employs the *DrivAer hp-F* reference model (26, 27), a high-performance variant of the DrivAer Fastback configuration. The *DrivAer hp-F*, depicted in Figure 2.1, is a 35% scale model featuring a smooth underbody, enclosed wheel cavities, and side mirrors. It incorporates additional aerodynamic devices, including a front splitter, forebody strakes, a rear spoiler, and an underbody diffuser. The main geometric parameters of the model are summarised in Table 2.1, with further details provided by Soares et al. (26).

Figure 2.1: Illustration of the *DrivAer hp-F* model.

2.2.2 Experimental Validation Data

The numerical results in this study are validated against wind tunnel experiments conducted by Soares et al. (26, 28) on the *DrivAer hp-F* in the 8x6 Wind Tunnel at Cranfield University. The test section was modified to include a moving ground and a two-stage boundary layer suction system. Aerodynamic forces were measured across eight different ride heights at a freestream velocity of 40 m/s, using an overhead balance system with a maximum full-scale uncertainty of ± 0.90 N for lift and ± 0.18 N for drag. The experimental measurements were conducted under a blockage ratio of approximately 10.2%.

2.2.3 Computational Fluid Dynamics Simulations

2.2.3.1 Domains and Boundary Conditions

CFD simulations are performed across three installations to systematically evaluate aerodynamic performance and the influence of wind tunnel conditions. Given the experimental straight-line setup, the time-averaged flow field is expected to approximate symmetric behaviour.

Table 2.1: Main dimensions of the *DrivAer hp-F* model.

Parameter	Symbol	Value	Unit
Car scale	CS	35	%
Length	L	1.6144	m
Width	W	0.6273	m
Ride height	h_0	0.0455	m
Area	A_{ref}	0.2556	m ²

This assumption enables the use of half-car models, which have been shown to reduce mesh size by over 40% while keeping aerodynamic force coefficient deviations within 1% of full-car simulations (1, 29), thereby improving computational efficiency with minimal loss of accuracy.

Studies on multichannel diffusers, like the one on the *DrivAer hp-F*, have shown that the channels suppress asymmetric flow structures observed in plane diffusers at low ride heights (30, 31), which further supports the half-car simplification. While DDES simulations inherently resolve unsteady turbulence, which does not obey symmetry instantaneously (32), this study focuses exclusively on time-averaged flow quantities, where the influence of turbulent fluctuations along the symmetry plane is negligible. The strong agreement between the DDES predictions and experimental data, as will be demonstrated in Section 2.3.1.1, further confirms that the symmetry simplification is an appropriate and computationally efficient modelling strategy for this study.

2.2.3.1.1 Wind Tunnel Installation

The computational wind tunnel domain, referred to as the *WT* installation, is designed to closely replicate the 8x6 Wind Tunnel at Cranfield University. Due to the limited availability of facility specifications, the domain is modelled as a straight tunnel, omitting the contraction and diffuser sections. The half-domain, shown in Figure 2.2, has a cross-sectional height of 1787 mm and a width of 1219 mm, incorporating 431.05 mm long corner fillets. Following established best practices in automotive aerodynamics (33), the velocity inlet is positioned three vehicle lengths (L) upstream, while the pressure outlet is placed $5L$ downstream, resulting in a total domain length of 14904 mm. These dimensions ensure adequate flow development ahead of the vehicle and prevent outlet conditions from interfering with the near-wake region.

To replicate experimental conditions, the top and side walls are modelled as no-slip stationary walls, allowing for boundary layer development. The moving ground system is implemented as a translational moving wall, and the overhead measurement system is explicitly included in the computational model to replicate the reported blockage ratio of approximately 10.2%. The inlet and moving ground velocities match the experimental freestream velocity of $U_\infty = 40$ m/s, corresponding to a Reynolds number of 4.21×10^6 . Air properties reported in Table 2.2 are maintained for consistency with experimental conditions.

Table 2.2: Overview of the wind tunnel air properties.

Parameter	Symbol	Value	Unit
Air density	ρ	1.1678	kg/m ³
Air dynamic viscosity	μ	18.377e-6	kg/(m·s)
Atmospheric pressure	P_{atm}	99982	Pa
Dynamic pressure	P_{dyn}	934.18	Pa
Tunnel temperature	T	25.1	°C
Turbulent intensity	I_T	0.052	%

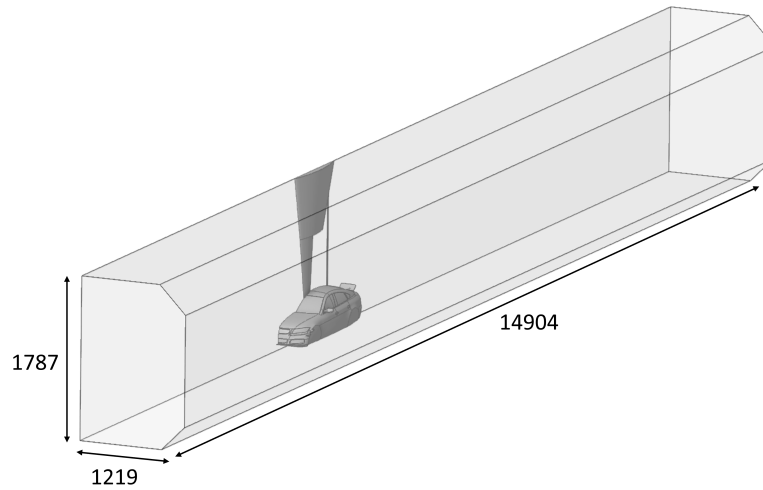


Figure 2.2: WT installation: vehicle model with top strut in the wind tunnel domain.

2.2.3.1.2 Open-Road Installations

To systematically evaluate the influence of wind tunnel conditions on the aerodynamics of the *DrivAer hp-F*, two open-road installations are modelled using the same computational domain but with different vehicle configurations. The first configuration, referred to as *wTS* installation (with Top Strut), includes the overhead measurement system, as depicted in Figure 2.3. This case allows for direct comparison with the *WT* installation to assess the influence of blockage effects induced by the wind tunnel walls. The second configuration, denoted as *woTS* installation (without Top Strut), excludes the overhead measurement system, as shown in Figure 2.4. This setup represents a more realistic open-road scenario, allowing for comparison with the *wTS* installation to examine the impact of measurement system interference effects.

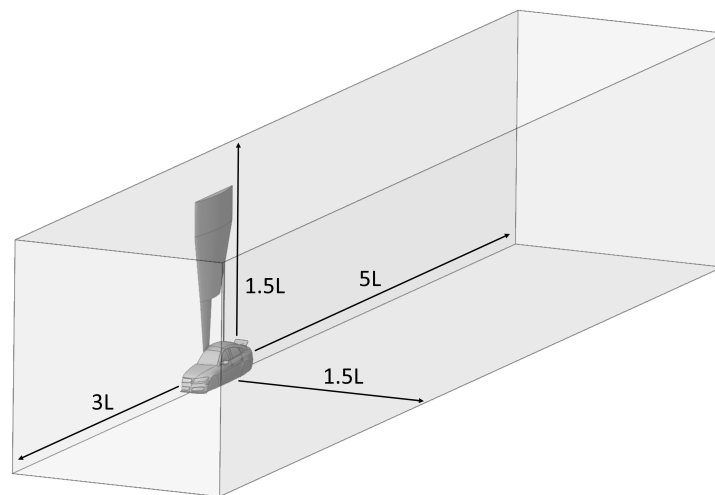


Figure 2.3: *wTS* installation: vehicle model with top strut in the open-road domain.

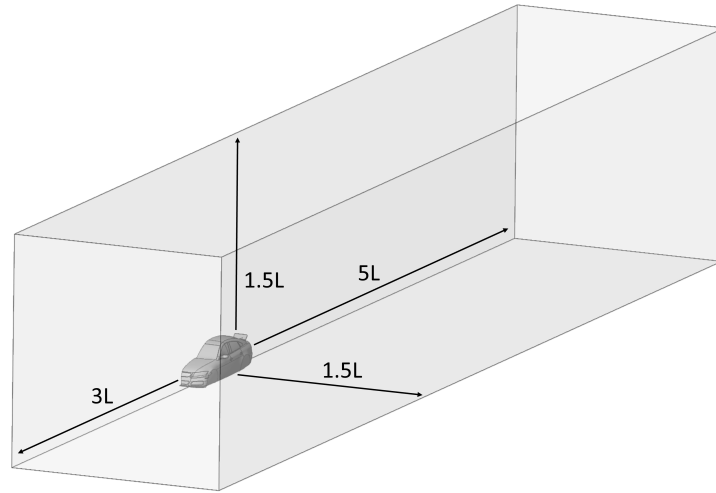


Figure 2.4: woTS installation: vehicle model without top strut in the open-road domain.

The open-road computational domain is designed to replicate real-world driving conditions while maintaining numerical efficiency. As in the wind tunnel domain, the velocity inlet is positioned $3L$ upstream, and the pressure outlet is placed $5L$ downstream of the vehicle. The top and side boundaries are located $1.5L$ from the vehicle, reducing the blockage ratio to around 1%, and are treated as symmetry planes to prevent artificial boundary layer formation. To maintain consistency across simulations, the freestream velocity of $U_\infty = 40$ m/s, modelling of the moving ground, and air properties are kept constant across all domains.

2.2.3.2 Mesh

2.2.3.2.1 Description

Unstructured poly-hexcore meshes are generated in ANSYS Fluent Meshing, an approach which has been shown to reduce total mesh size by approximately 40% compared to conventional hexcore methods, while decreasing computational solve time by around 14% for simulations on a standard DrivAer model (34). A medium-resolution mesh is constructed based on optimised element sizes for scaled vehicle models (29), with base and surface element sizes set at approximately 7.4% and 0.45% of the vehicle length, respectively. To accurately capture the complex geometries of aerodynamic add-on devices and small-scale features, local mesh refinements reduce surface element sizes by over 80%.

A medium near-wall treatment is employed, with y^+ values generally around 150 and incorporating 4–6 inflation layers on both the vehicle and wind tunnel walls. This approach ensures an adequate resolution of the turbulent boundary layer while maintaining a practical balance for RANS simulations, adhering to the recommended y^+ range of 30–300 for such models (35). Additionally, refinement zones are implemented following best practices from the 1st Automotive CFD Prediction Workshop (36), enhancing flow field resolution around the vehicle, as illustrated in Figure 2.5.

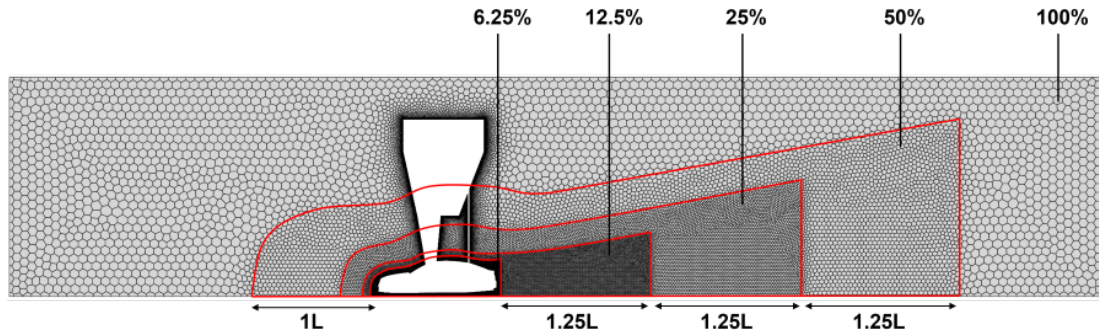


Figure 2.5: View of the medium-resolution mesh on the symmetry plane of the *wTS* installation, highlighting refinement zones in red with element sizes expressed as a percentage of the base mesh size.

2.2.3.2.2 Grid Convergence

A grid convergence study is performed using the Grid Convergence Index (GCI), as introduced by Roache (37). To evaluate grid dependency, two additional meshes with varying refinement levels are generated alongside the medium-resolution mesh. These grids follow the systematic refinement approach for unstructured meshes proposed by Celik et al. (38), which recommends refinement factors r (-) greater than 1.3 based on the average cell size h_c (mm). The properties of the coarse (3), medium (2), and fine (1) resolution meshes are detailed in Table 2.3.

Grid independence is assessed using Richardson extrapolation (RE) to estimate deviations between the numerical solutions f_1 - f_3 for lift C_L (-) and drag C_D (-) coefficients from asymptotic values f_{RE} , following the approach outlined for unstructured grids by Celik et al. (38). Although the GCI method is less rigorous for unstructured grids than for structured ones, it provides more comprehensive insights than simple error reporting (37) and has been successfully applied in similar automotive aerodynamics studies (39, 40, 41). Simulations for the grid convergence study are conducted on the *wTS* installation using the k - ω SST RANS turbulence model in ANSYS Fluent, executed on two AMD EPYC 7543 nodes utilising a total of 64 cores.

As presented in Table 2.4, the GCI values between the fine and medium meshes (GCI_{21}) are significantly lower than those between the medium and coarse meshes (GCI_{32}), indicating reduced grid dependency with increasing refinement.

Table 2.3: Overview of the mesh properties.

Property	Symbol	Unit	Coarse (3)	Medium (2)	Fine (1)
No. of cells	N	-	2,489,238	5,928,362	13,624,327
Average wall yplus	y^+	-	208	168	49
Average cell size	h_c	mm	36.95	27.67	20.97
Refinement factor	r_{32} & r_{21}	-	-	1.34	1.32
Computation time	t	h	0.36	1.22	3.08

Table 2.4: Grid convergence study results.

Variable	Coarse f_3 (-)	Medium f_2 (-)	Fine f_1 (-)	RE f_{RE} (-)	Accuracy order p (-)	GCI_{32} (%)	GCI_{21} (%)	$\frac{GCI_{32}}{r^p GCI_{21}}$ (-)
C_L	-0.8597	-0.8214	-0.8185	-0.8183	8.92	0.47	0.04	0.9966
C_D	0.2756	0.2575	0.2482	0.2374	2.23	9.68	5.39	0.9642

Consistent with findings from previous studies (39, 42), different convergence rates are observed for the force coefficients, with lift exhibiting a higher order of accuracy p (-) and lower GCI values than drag. Roache (37) noted that unstructured grids with non-integer refinement ratios can yield above-average orders of accuracy, depending on refinement levels in critical regions. On the *DrivAer hp-F* model, lift is primarily influenced by the pressure distribution along the roof and underbody, whereas drag is dominated by near-wake effects on its base pressure (43, 44, 45). Consequently, variations in refinement across these regions contribute to differing grid convergence rates.

Despite these variations, the asymptotic range condition (46), defined as $GCI_{32}/r^p GCI_{21}$, yields values close to one for both force coefficients, confirming that the solutions are within the asymptotic range. Considering grid dependency, computational cost, and the focus on lift as the key performance metric, the medium-resolution mesh is selected for all RANS simulations. This mesh exhibits a relative error of less than 0.5% for lift and just over 8% for drag when compared to the Richardson extrapolated asymptotic values.

2.2.3.2.3 DDES - High Near-Wall Resolution Mesh

The meshing strategy described above is extended to generate high near-wall resolution meshes for DDES simulations on the *WT* installation. These meshes incorporate a lower y^+ treatment of approximately 0.7 with 15 inflation layers, ensuring enhanced near-wall resolution. To improve transition smoothness between near-wall elements and the surrounding refinement zones, the element size in the refinement region around the vehicle is reduced from 6.25% to 2%. As a result, the high near-wall resolution meshes for DDES simulations on the *WT* installation contain approximately 22 million cells, whereas the corresponding RANS meshes consist of around 6 million cells.

2.2.3.3 Solve

2.2.3.3.1 RANS

Steady-state RANS simulations are conducted using the Spalart-Allmaras (47), k- ϵ Realizable (48), k- ω SST (49), and Reynolds Stress (50) turbulence models in ANSYS Fluent. Each model solves the continuity and momentum equations along with its respective transport equation(s). The solution methodology adheres to the recommendations by ANSYS Fluent (51). Accordingly, pressure-velocity coupling is conducted using the coupled scheme. For spatial discretisation, gradients are computed using the least-squares cell-based method, and pressure is discretised using a second-order scheme. The momentum equation is solved using a second-order upwind scheme, whereas turbulence equations are discretised using a first-order upwind scheme, except for the k- ω SST model, which employs second-order upwind discretisation as its default scheme.

A preliminary trade-off study between simulation duration and force convergence determined that 1000 iterations are sufficient for all RANS models. Statistical convergence, defined as the point where the monitored lift and drag forces stabilised, was typically reached within 400 iterations. Accordingly, force coefficients are averaged over the final 500 iterations. The simulations are executed on two AMD EPYC 7543 processors using a total of 64 cores, with average computation times of approximately 1 hour for the Spalart-Allmaras model, 1.2 hours for the k- ϵ Realizable and k- ω SST models, and 1.6 hours for the Reynolds Stress model.

2.2.3.3.2 DDES

Scale-resolving simulations on the WT installation are conducted using the DDES (52) model with the k- ω SST RANS background model. This hybrid RANS-LES model switches from the ‘RANS mode’ in wall boundary layers to the ‘LES mode’ in detached flow regions to resolve large-scale turbulent structures. The simulations are initialised using steady-state k- ω SST RANS solutions on the same high near-wall resolution mesh. The numerical setup follows the default ANSYS Fluent recommendations (51), maintaining the same pressure-velocity coupling and spatial discretisation as the RANS k- ω SST model, except for the momentum equation, which is solved using the suggested bounded central differencing scheme. Temporal discretisation of the governing equations is performed using a bounded second-order implicit formulation.

Time step selection follows the approach of Aultman et al. (53), whose findings indicate that a non-dimensional time step of 5χ provides sufficient resolution of turbulent structures without excessive computational cost. Accordingly, a physical time step of $t = 3.125 \times 10^{-4}$ s with 10 inner iterations is used. Each simulation is performed for 2 s of flow time, equivalent to 49.6 convective time units (CTUs), with the first 1 s (24.8 CTUs) dedicated to washing out the initial RANS solution and reaching statistical convergence, followed by the remaining 1 s (24.8 CTUs) used for flow field averaging and collecting unsteady statistics. The DDES simulations are conducted on four AMD EPYC 7543 processors using 128 cores, resulting in an average computational time of approximately 90 hours per simulation.

2.2.4 Wind Tunnel Correlation Process

CFD simulations and wind tunnel experiments are complementary tools for analysing vehicle aerodynamics. However, wind tunnel experiments are subject to physical constraints that can influence aerodynamic performance relative to open-road conditions. This section outlines a stepwise approach to numerically assess the blockage and measurement system interference effects, both of which impact the correlation between numerical and experimental results.

2.2.4.1 Blockage Effect and Correction Methods

The blockage effect is examined by comparing simulations in a computational wind tunnel domain (*WT* installation) with an open-road domain (*wTS* installation). Both cases include the vehicle with the overhead measurement system, differing only in the confinement imposed by the wind tunnel walls. This setup isolates blockage-induced effects, enabling a detailed analysis of flow field alterations and their impact on aerodynamic forces. Additionally, the methodology is used to assess the applicability of blockage correction methods by applying them to *WT* installation results and comparing the corrected values to results from the *wTS* installation.

Several blockage correction models are evaluated, including the Maskell, DNW Maskell, Extended Maskell III, Thom & Herriot, and Mercker methods (16, 20). The DNW Maskell and Extended Maskell III models build upon the Maskell approach, while the Thom & Herriot and Mercker methods explicitly differentiate between solid and wake blockage. Despite their varying complexity, all methods estimate the corrected dynamic pressure q_c (Pa) near the vehicle using measured force coefficients and geometric relationships. This corrected pressure is subsequently used, alongside the reference dynamic pressure q (Pa) that is typically measured at the wind tunnel inlet, to compute corrected lift C_{Lc} (-) and drag C_{Dc} (-) coefficients, as described by Eq. (2.1). These corrected force coefficients are estimations of the vehicle's aerodynamic performance in open-road conditions, free from wind tunnel blockage effects.

$$\frac{C_D}{C_{Dc}} = \frac{q_c}{q} \quad \text{and} \quad \frac{C_L}{C_{Lc}} = \frac{q_c}{q} \quad (2.1)$$

In addition to conventional correction methods, this study introduces a CFD-based blockage correction approach. Unlike the geometric relationships, it directly extracts dynamic pressure data from CFD simulations. The method compares average dynamic pressure values on a frontal plane within both the wind tunnel and open-road domains, as shown in Figure 2.6. To ensure accurate correction, the plane is longitudinally positioned where the blockage ratio is highest, preventing underestimation of the effects. Additionally, in the open-road domain, only the projected wind tunnel area is considered, avoiding far-field influence. The CFD-based approach provides a direct estimation of q_c and q near the vehicle and will serve as a benchmark for assessing the accuracy of conventional correction models in this study.

2.2.4.2 Overhead Measurement System Interference

Besides the blockage effect, interference effects of the overhead measurement system are numerically assessed by comparing simulations in the open-road domain with the top strut (*wTS* installation) and without the top strut (*woTS* installation). Since both configurations are free from wind tunnel blockage, any differences in aerodynamic forces and flow characteristics are solely attributed to the measurement system, allowing an assessment of its isolated interference effects.

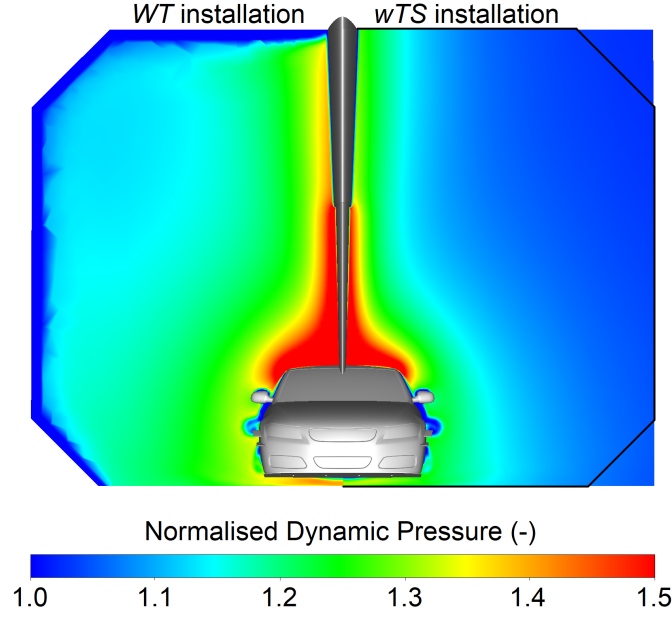


Figure 2.6: Comparison of dynamic pressure on a frontal cross-section ($x = 0.45\text{m}$) between the *WT* and *wTS* installations, with black borders indicating the integration area.

2.3 Results and Discussion

This section first evaluates the aerodynamic ride height dependency based on experimental force coefficient data. Additionally, the predictive capabilities of four practical RANS turbulence models are assessed through both a relative comparison with experimental data and an absolute comparison against high-fidelity DDES flow field predictions. The subsequent subsection investigates discrepancies between wind tunnel experiments and open-road simulations, including a numerical quantification of blockage effects, an evaluation of the applicability of correction methods, and an analysis of measurement system interference effects. The assessments are conducted within the context of the *DrivAer hp-F* experimental setup in the 8x6 Wind Tunnel at Cranfield University.

2.3.1 Turbulence Model Assessment

This study utilises experimental data from Soares et al. (26) on the *DrivAer hp-F* at varying ride heights for validation. While discrepancies between physical wind tunnel experiments and CFD simulations are anticipated, due to finite accuracy in replicating the exact operating conditions and inherent limitations in turbulence modelling, the dataset provides a robust benchmark for establishing reference aerodynamic performance trends.

However, the adopted dataset is limited to force measurements and other experimental data for high-performance vehicles is extremely scarce. To address this limitation, high-fidelity DDES simulations are conducted to serve as complementary reference data, enabling a more detailed examination of the vehicle's flow field. Accordingly, the predictive accuracy of various RANS models for high-performance vehicle aerodynamics is assessed by their relative agreement with experimental force data and their absolute correlation with DDES flow field data in the same computational wind tunnel domain.

Additionally, the experimental dataset corresponds to a vehicle without wheels, excluding known sources of unsteady flow along the underbody. This is expected to have a minimal impact on the assessment, since it focuses on the vehicle's entire flow field, which includes other regions dominated by highly unsteady flow, such as the wake. The analysis therefore still provides meaningful insights into the capabilities of RANS models in predicting the diverse and complex aerodynamic characteristics of high-performance vehicles.

2.3.1.1 Force Coefficients

Figure 2.7 illustrates the variation of the vehicle's lift and drag coefficients with ride height, where h_0 represents the normalised ride height of 45.5 mm. The experimental data reveals a force enhancement region within the $2h_0 - 0.33h_0$ range, characterised by an increase in downforce as ride height decreases. This behaviour is attributed to enhanced airflow acceleration beneath the vehicle. As a consequence of increased downforce, induced drag also rises across this range. However, a critical ride height of $0.33h_0$ is identified, below which significant downforce losses and a sharp reduction in drag occur. At such low ride heights, the underbody flow becomes increasingly unsteady, adversely affecting aerodynamic performance (54, 55).

The DDES model exhibits strong agreement with the experimental data, with an average downforce overestimation of 7% and a drag underestimation of 2%. Furthermore, the DDES results demonstrate high consistency, as reflected in the relatively stable errors shown in Figure 2.8, with a standard deviation of approximately 2% across the ride height range. Notably, given the interdependence of downforce and induced drag, a slight drag overestimation might have been anticipated. This discrepancy likely arises from the differing sensitivity of downforce and drag to numerical and experimental uncertainties, where downforce is predominantly influenced by near-wall flow characteristics and drag is more affected by unsteady wake dynamics (1, 45).

Overall, the DDES model accurately captures the trends observed in the experimental force coefficient data across the entire ride height range, reinforcing its reliability as a numerical reference for evaluating RANS model accuracy within the WT installation.

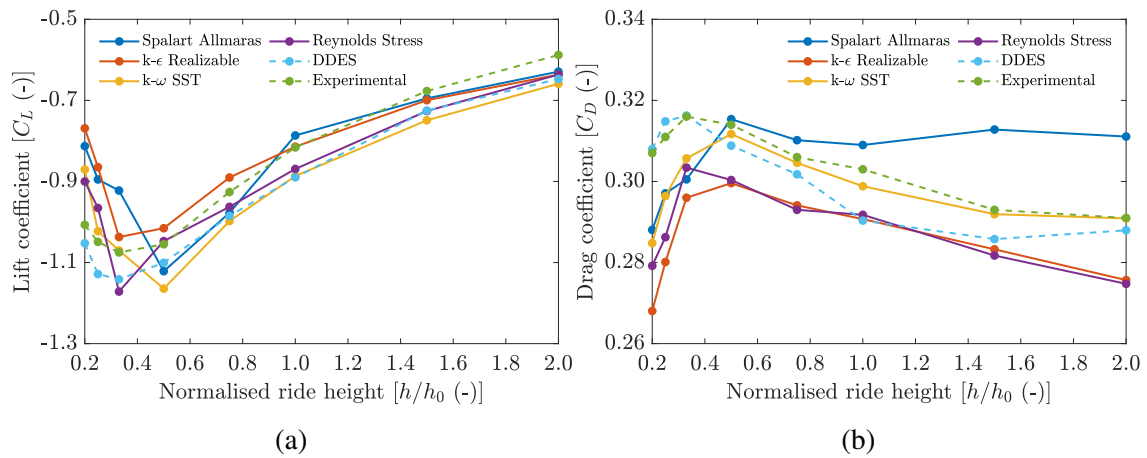


Figure 2.7: Comparison of (a) lift and (b) drag coefficients between the wind tunnel experiments and numerical simulations on the WT installation.

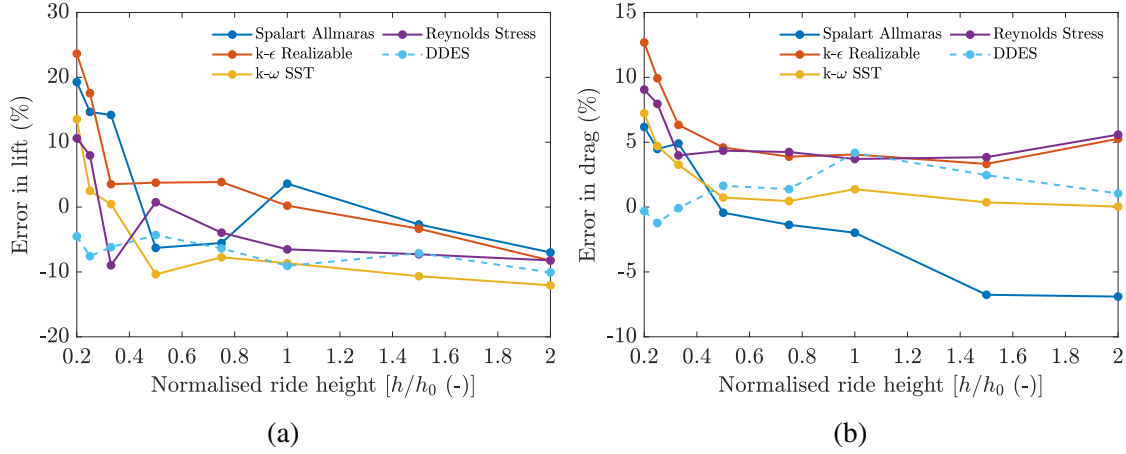


Figure 2.8: Error in (a) lift and (b) drag coefficients between the wind tunnel experiments and numerical simulations on the WT installation.

A well-known limitation of the averaging methods applied by RANS turbulence models is their inability to accurately predict highly unsteady flow phenomena (56). This limitation is also evident in Figure 2.7, where all RANS models fail to capture the aerodynamic behaviour of the vehicle below its critical ride height of $0.33h_0$. Consequently, the present assessment focuses on the high to medium ride height range of $2h_0 - 0.5h_0$, where aerodynamic performance is less critically influenced by highly unsteady flow effects.

Across this range, the downforce enhancement region is captured by all RANS models. However, variations in accuracy are observed. The Spalart-Allmaras, $k-\epsilon$ Realizable, and Reynolds Stress models underpredict downforce by approximately 4%, 8%, and 2% respectively, while the $k-\omega$ SST model overestimates downforce by around 2% relative to the DDES results. Among these models, the $k-\omega$ SST model provides the most consistent downforce predictions across the ride height range, with a relative deviation in lift error of less than 2%, as shown in Figure 2.8. In contrast, the other RANS models exhibit larger variations, with relative deviations in lift error of approximately 4–5%.

With respect to drag, the $k-\epsilon$ Realizable and Reynolds Stress models exhibit an average underestimation of approximately 2%, whereas the $k-\omega$ SST model overestimates drag by a similar magnitude relative to the DDES results. Notably, these three RANS models demonstrate consistent drag trend predictions across the evaluated ride height range, as evidenced by their drag error remaining within 1% relative deviation. Conversely, the Spalart-Allmaras model fails to adequately capture the drag trend, displaying a nearly ride height insensitive drag prediction across the $2h_0 - 0.5h_0$ range.

2.3.1.2 Pressure Distribution

The pressure distribution along the upper surface is illustrated in Figures 2.9 and 2.10. A high-pressure region is observed at the splitter and nose, followed by a low-pressure region as the airflow accelerates around the curvature of the nose. Along the bonnet, a pressure recovery occurs towards the base of the windscreen, which exhibits a high-pressure region. Airflow is subsequently accelerated over the roof, generating a low-pressure region that gradually recovers towards the roof's trailing edge.

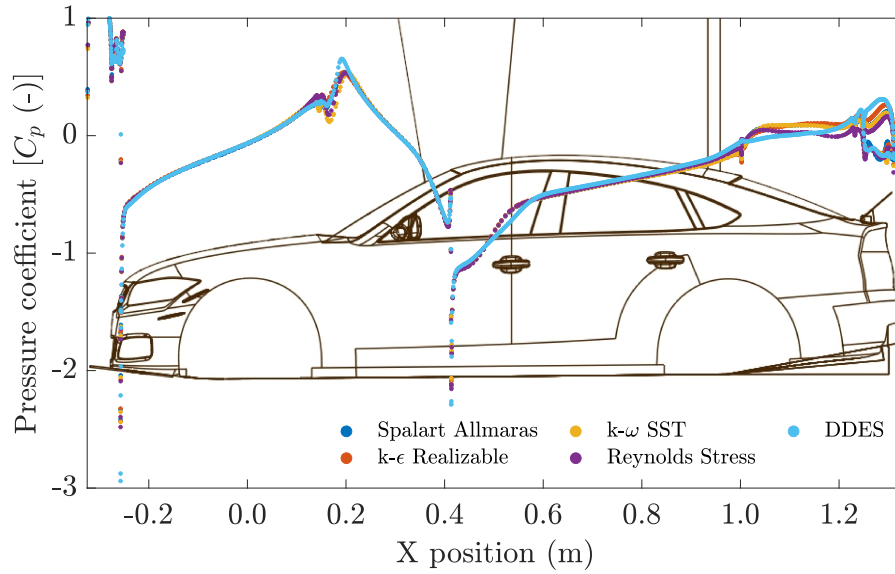


Figure 2.9: Pressure coefficient distributions along a near-symmetry line ($y = -50$ mm) on the upper body surface of the WT installation at $1h_0$.

Moderate pressures are observed on the slant due to minor flow separation behind the roof. Finally, a high-pressure region forms as the airflow interacts with the spoiler.

The RANS models generally exhibit good agreement with the DDES results on the upper surface. The $k-\omega$ SST and Reynolds Stress models predict pressures within 1% of the DDES results along the bonnet, while the $k-\epsilon$ Realizable and Spalart-Allmaras models deviate by approximately 2% and 4%, respectively. Downstream of the bonnet, the RANS models generally predict a larger recirculation zone ahead of the windscreen, leading to a higher redirection of airflow onto the windscreen relative to the DDES results. Consequently, a localised 18% reduction in the pressure peak is observed at the windscreen in the RANS models.

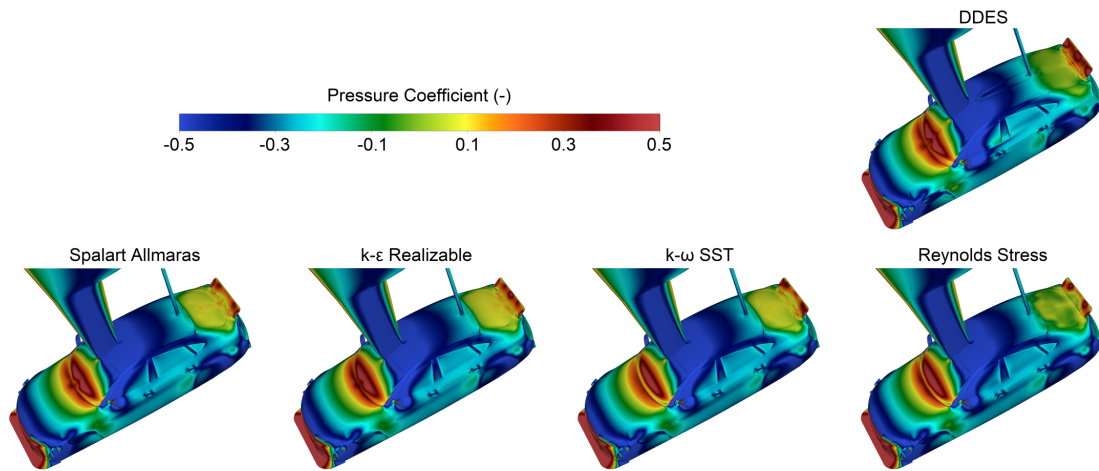


Figure 2.10: Pressure coefficient contour plots on the upper body surface of the WT installation at $1h_0$.

Along the roof, the RANS models depict a less gradual initial pressure recovery, resulting in discrepancies of approximately 10% towards the roof's trailing edge. Downstream at the slant, the RANS models underpredict flow separation, which subsequently affects the pressure distribution on the spoiler. The DDES model indicates more pronounced high-pressure concentrations on the spoiler, with peak pressures locally exceeding those predicted by the RANS models by up to 50%. Discrepancies in pressure distribution across the slant and spoiler due to flow separation predictions will be examined in more detail in Section 2.3.1.3.

As downforce is predominantly generated by the airflow beneath the vehicle, it is crucial that the pressure distribution along the underbody is accurately captured. Figures 2.11 and 2.12 depict the pressure distribution along the underbody. A low-pressure plateau followed by a steep pressure recovery is observed beneath the splitter, indicative of a separation bubble and subsequent flow reattachment (57). A small depression in pressure is present at the floor's leading edge as airflow accelerates between the floor and ground.

The pressure then recovers towards the central section of the floor before decreasing significantly toward the diffuser inlet due to flow extraction, forming a large low-pressure peak. Downstream of the inlet, pressure recovery occurs throughout the diffuser as airflow is gradually decelerated. Additionally, localised low-pressure regions appear at the edges of the diffuser channels, induced by strong vortex formation at the side plates. These vortices energise the airflow along the diffuser ramp, enhancing downforce generation.

While the RANS models generally capture the overall pressure distribution along the underbody, local discrepancies arise around the splitter, diffuser inlet, and diffuser ramp. The RANS models predict a smaller separation bubble and an earlier flow reattachment beneath the splitter compared to the DDES results. The $k-\omega$ SST model predicts the smallest separation bubble, resulting in an approximate 15% shorter reattachment length.

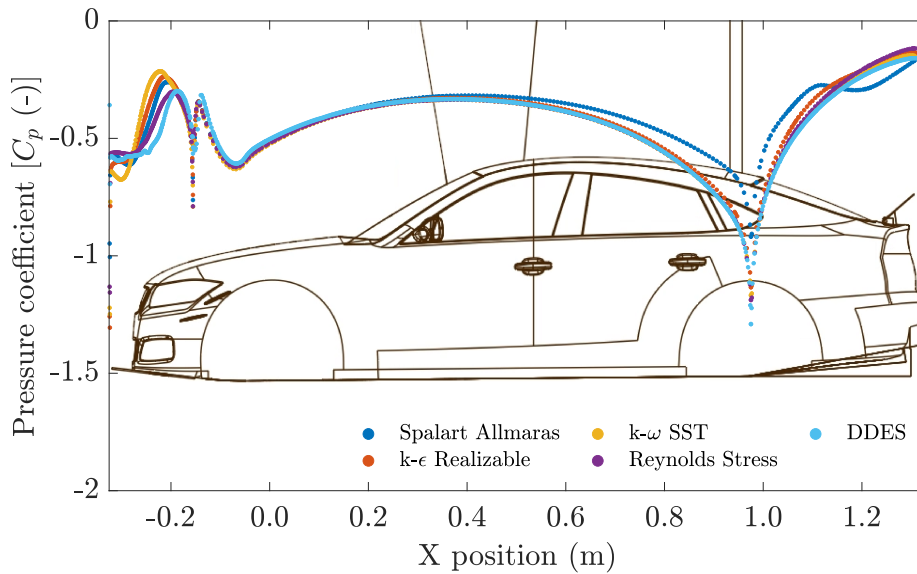


Figure 2.11: Pressure coefficient distributions along a near-symmetry line ($y = -50$ mm) on the underbody surface of the WT installation at $1h_0$.

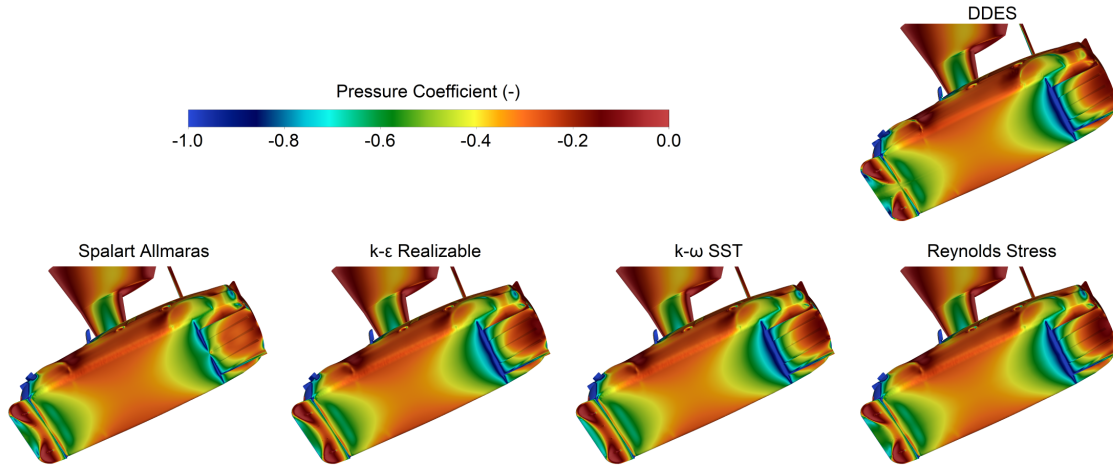


Figure 2.12: Pressure coefficient contour plots on the underbody surface of the WT installation at $1h_0$.

The Reynolds Stress model predicts the largest separation bubble among the RANS models but still underestimates the reattachment length by approximately 5%. Despite these variations, all RANS models provide pressure predictions within 4% of the DDES results around the suction peak and initial pressure recovery behind the floor's leading edge.

Further along the floor, the $k-\omega$ SST and Reynolds Stress models perform particularly well, with pressure deviations remaining within 1% of the DDES results. The $k-\epsilon$ Realizable model maintains an adequate level of accuracy, with deviations of up to 4%, whereas the Spalart-Allmaras model significantly underpredicts suction, with deviations of approximately 20%. The pressure distribution in this region is strongly influenced by the flow extraction induced by the expansion within the diffuser. The Spalart-Allmaras model is the only RANS model to exhibit large pressure fluctuations along the diffuser ramp, indicative of flow separation, which contributes to the observed underprediction of suction along the floor.

The other RANS models demonstrate better agreement with the DDES results along the diffuser ramp. In this region, pressure recovery primarily occurs along the central portion of the ramp, with low-pressure concentrations present at the edges of each diffuser channel. However, the $k-\epsilon$ Realizable and Reynolds Stress models exhibit a steeper pressure recovery across the diffuser, with surface pressures up to 20% higher than those predicted by the DDES model. The $k-\omega$ SST model shows the strongest agreement with the DDES data in this region, providing nearly identical pressure predictions downstream of the inlet, with discrepancies remaining within 10% towards the diffuser outlet.

2.3.1.3 Slant and Spoiler

The airflow along the slant and spoiler is primarily influenced by the prediction of vortex flow and flow separation along the C-pillars, roof trailing edge, and slant. The DDES results indicate a gradual airflow retardation along the slant, with higher velocity airflow concentrated along the centre and sides, as shown in Figure 2.13.

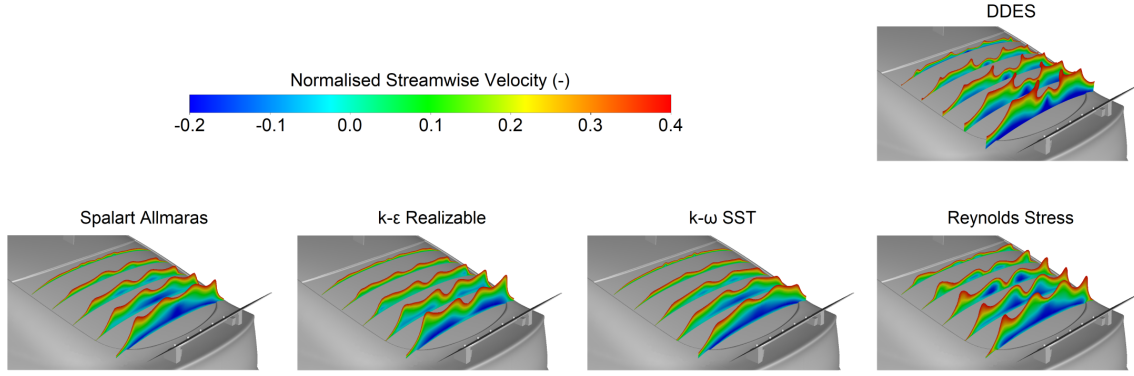


Figure 2.13: Normalised streamwise velocity contours at multiple frontal planes along the slant ($x = 1.02$ m, $x = 1.065$ m, $x = 1.11$ m, $x = 1.155$ m, and $x = 1.2$ m) of the WT installation at $1h_0$.

These high-velocity traces result from high-energy vortices originating from the overhead measurement system and C-pillars. As these vortices propagate downstream along the slant, they generate high-pressure concentrations on the spoiler, as previously identified in Section 2.3.1.2. In contrast, the RANS models predict significantly smaller vortices originating from the overhead measurement system, as evidenced by the smaller circular high-velocity traces at the most upstream plane. Additionally, the vortex structures are captured with lower fidelity, as indicated by the blending of two distinct circular traces into one as the vortices travel downstream.

Despite this limitation, the $k-\epsilon$ Realizable model predicts a comparable vortex trajectory along the centre of the slant. This is demonstrated by the height of the high-velocity airflow traces and the resulting pressure concentrations at the centre of the spoiler, previously demonstrated in Figure 2.10. Conversely, the Spalart-Allmaras, $k-\omega$ SST, and Reynolds Stress models predict a more upward vortex trajectory, evidenced by the increased distance between the high-velocity airflow traces and the slant surface, causing these models to exhibit reduced or even absent high-pressure concentrations around the centre of the spoiler. Further discrepancies arise in the size and trajectory of the C-pillar vortices, which leave wider and more inward directed high-velocity traces in the DDES results, leading to the larger high-pressure concentrations observed at the sides of the spoiler.

To examine separation predictions along the slant, velocity profiles at approximately one-quarter of the slant's width are extracted from each simulation, as depicted in Figure 2.14. A characteristic boundary layer response to an adverse pressure gradient is observed across all models, where near-wall flow deceleration leads to separation. However, variations in each model's sensitivity to adverse pressure gradients and flow predictions at the roof introduce notable discrepancies in the velocity profiles. The DDES results depict near-wall flow reversal at the most upstream velocity profile, indicating that airflow separates immediately at the roof's trailing edge. Along the slant, the airflow continues to decelerate, reaching a flow reversal velocity of approximately 25% of the freestream velocity at the most downstream velocity profile.

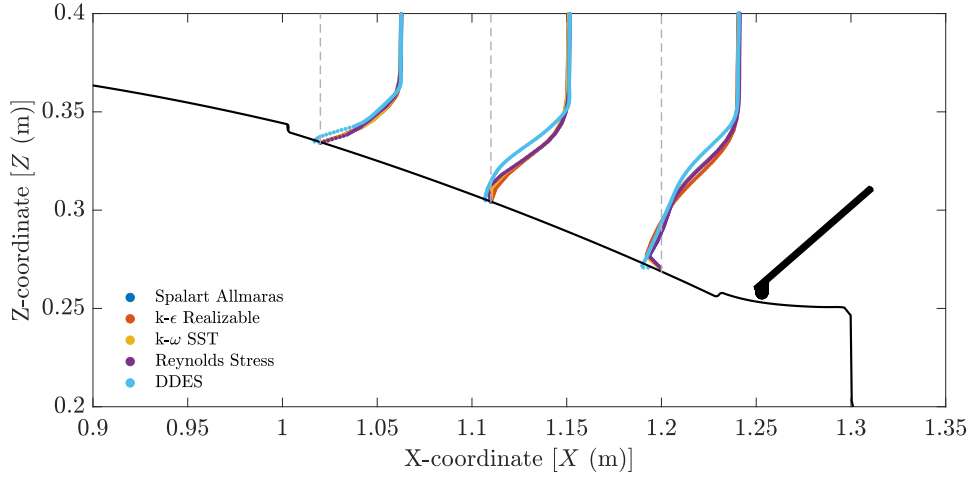


Figure 2.14: Streamwise velocity profiles at $x = 1.02$ m, $x = 1.11$ m, and $x = 1.2$ m along a longitudinal line ($y = -85$ mm) on the slant of the WT installation at $1h_0$.

The RANS models initially exhibit a more energised boundary layer that reattaches behind the roof, as demonstrated by the positive velocity in the most upstream velocity profile. Although all RANS models exhibit noticeable boundary layer deceleration in the subsequent velocity profile, the Reynolds Stress model is the only one to show near-wall flow reversal. At the most downstream velocity profile, better agreement is observed, with all RANS models predicting flow separation.

The $k-\epsilon$ Realizable and $k-\omega$ SST models predict a flow reversal velocity of approximately 20% of the freestream velocity and a nearly identical backflow height compared to the DDES results. The Spalart-Allmaras and Reynolds Stress models predict a lower flow reversal velocity of around 15% of the freestream velocity and underpredict the backflow height by approximately 2%. Additionally, higher streamwise velocities are observed towards the upper region of the boundary layer, likely influenced by the initially higher boundary layer momentum predictions further upstream on the slant.

2.3.1.4 Diffuser

The diffuser is a crucial aerodynamic component in high-performance vehicles, typically contributing over one-third of the total downforce. Therefore, accurate flow predictions in this region are essential, particularly when assessing the vehicle's sensitivity to ride height variations. As previously discussed in Section 2.3.1.2, a significant pressure depression forms at the diffuser inlet, followed by a gradual pressure recovery along the diverging diffuser ramp. To better understand the flow behaviour within the diffuser, Figure 2.15 illustrates the distribution of shear stress along the diffuser ramp and Figure 2.16 the streamwise velocity along several longitudinal positions in the diffuser.

The DDES results indicate high-shear-stress concentrations near the diffuser inlet, corresponding to areas of high near-wall flow velocity. Further downstream, a distinct region of low shear stress emerges along the centre of the diffuser, aligning with the noticeable flow deceleration observed in this region.

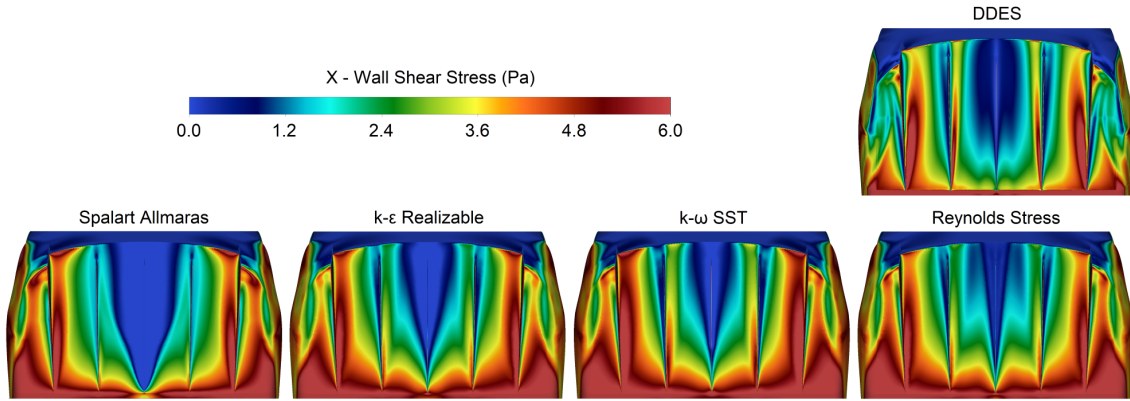


Figure 2.15: Streamwise shear stress contour plots on the diffuser surface of the *WT* installation at $1h_0$.

Vortices are generated at the diffuser side plates and separators due to pressure differentials between the underbody flow and the surrounding freestream air. These vortices induce high-energy airflow onto the diffuser ramp, enhancing flow attachment within the diffuser channels, evidenced by the localised high-shear-stress concentrations and elevated near-wall flow velocities. Furthermore, traces of moderate streamwise velocity are observed beneath the symmetry separator of the diffuser, indicative of local flow separation in this region.

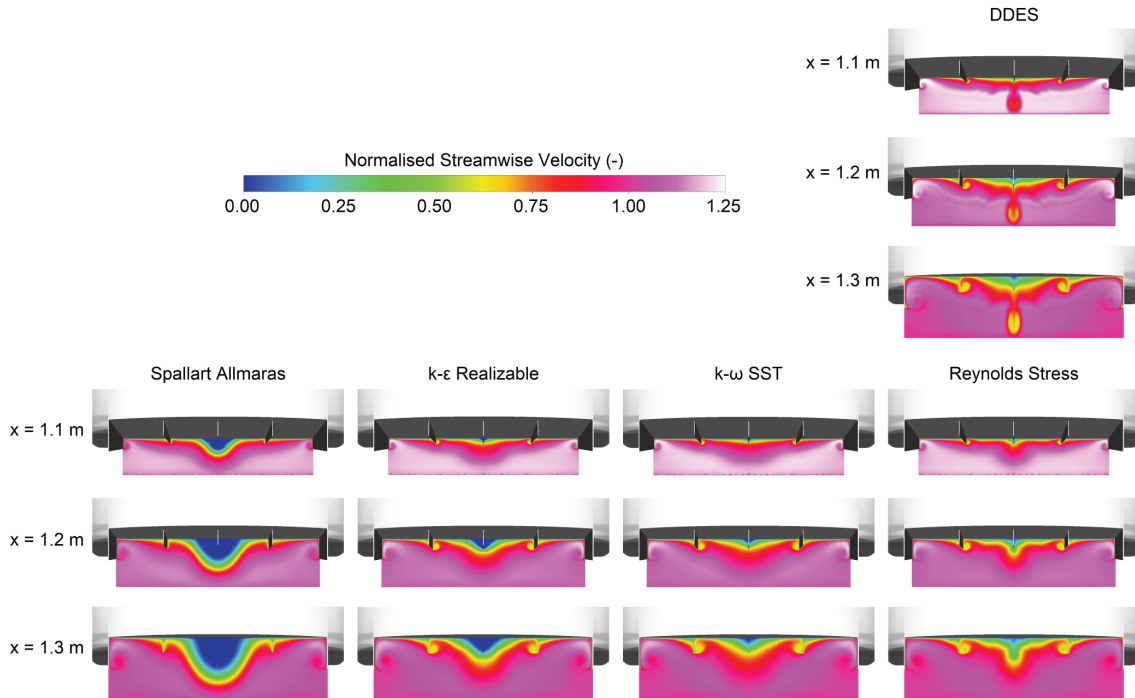


Figure 2.16: Normalised streamwise velocity contours at multiple frontal planes along the diffuser ($x = 1.1$ m, $x = 1.2$ m, and $x = 1.3$ m) of the *WT* installation at $1h_0$.

The RANS models exhibit varying levels of agreement with the DDES results in predicting diffuser flow characteristics. The Spalart-Allmaras model significantly overpredicts flow separation at the centre of the diffuser, as indicated by the extensive low-shear-stress and low-velocity regions. The $k-\epsilon$ Realizable model mitigates this overestimation but still predicts a broader low-shear-stress region compared to the DDES results. Better qualitative agreement is observed with the $k-\omega$ SST and Reynolds Stress models, which predict a narrower region of low shear stress and exhibit flow deceleration patterns more comparable to the DDES findings. However, none of the RANS models capture the traces of moderate streamwise velocity further from the diffuser ramp, suggesting limitations in their ability to predict flow separation behaviour in this area.

Despite these discrepancies, all RANS models capture flow entertainment from the side plate vortices within the outer diffuser channels. The $k-\omega$ SST model demonstrates the highest agreement with the DDES results regarding vortex-induced flow entertainment at the separators, followed by the $k-\epsilon$ Realizable and Reynolds Stress models. In contrast, the Spalart-Allmaras model underpredicts vortex-induced flow entertainment near the separators, likely due to its excessive flow separation predictions in this region. Overall, the $k-\omega$ SST model provides the closest match to the DDES diffuser flow predictions, estimating diffuser downforce within approximately 3% of the DDES results, whereas the other RANS models underpredict diffuser downforce, with discrepancies of up to 8%.

2.3.1.5 Wake

Airflow separation from the vehicle's body and spoiler generates a wake region of relatively low-energy airflow behind the vehicle, characterised by areas of low total pressure, as depicted in Figure 2.17. The DDES results reveal a squared near-wake region with significant pressure losses immediately behind the vehicle. Moderate-total-pressure traces are observed along the upper part of this wake, attributed to flow separation at the slant, spoiler, and overhead measurement system.

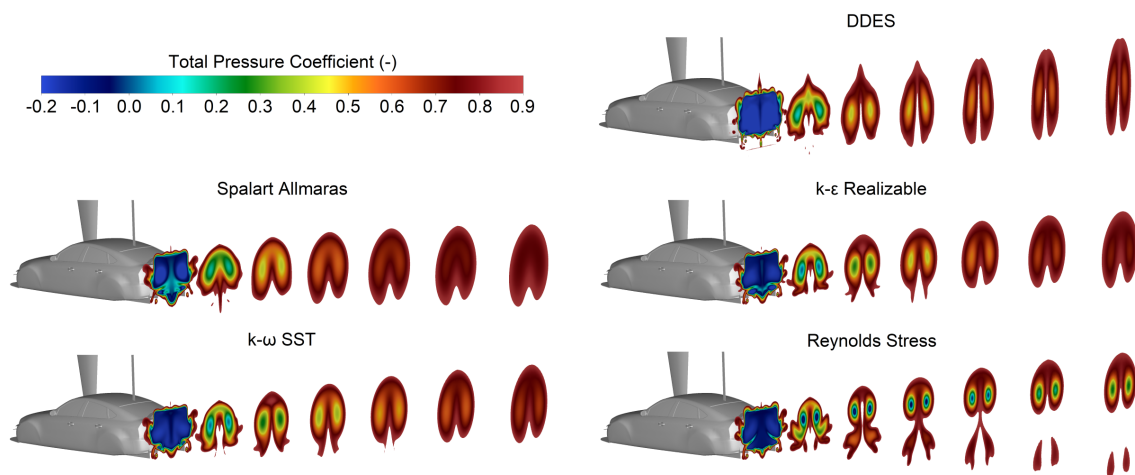


Figure 2.17: Total pressure coefficient contours at wake planes ($x = 1.35$ m, $x = 1.85$ m, $x = 2.35$ m, $x = 2.85$ m, $x = 3.35$ m, $x = 3.85$ m, and $x = 4.35$ m) of the WT installation at $1h_0$.

Wake traces from the mirrors and strakes appear along the periphery of the primary wake region, albeit with reduced intensity. Larger pressure losses occur in the middle portion of the near-wake, dominated by a pair of counter-rotating vortices originating from flow separation of the vehicle body, evident from the two distinct oval-shaped low-total-pressure zones which propagate downstream.

To further analyse wake characteristics, such as its dissipation and trajectory, the area-weighted average total pressure coefficient and total-pressure-loss-weighted vertical position are evaluated at multiple wake planes, as demonstrated in Figure 2.18. These results indicate that wake pressure losses decay exponentially with increasing distance from the vehicle. Moreover, the wake exhibits a predominantly upward trajectory, driven by the upwash effect from the interaction between the counter-rotating vortex pair.

The RANS models also predict a fairly squared near-wake region with pressure losses similar in distribution to the DDES results but with generally lower intensity. The Spalart-Allmaras and $k-\epsilon$ Realizable models demonstrate noticeably more pressure losses in the lower near-wake region due to excessive prediction of flow separation in the diffuser. The location and intensity of wake traces from the spoiler and strakes are reasonably well captured by all RANS models, whereas wake traces from the mirrors tend to be positioned further outboard and appear with reduced intensity. All RANS models predict the formation and propagation of two counter-rotating vortices, though their intensity and trajectory vary.

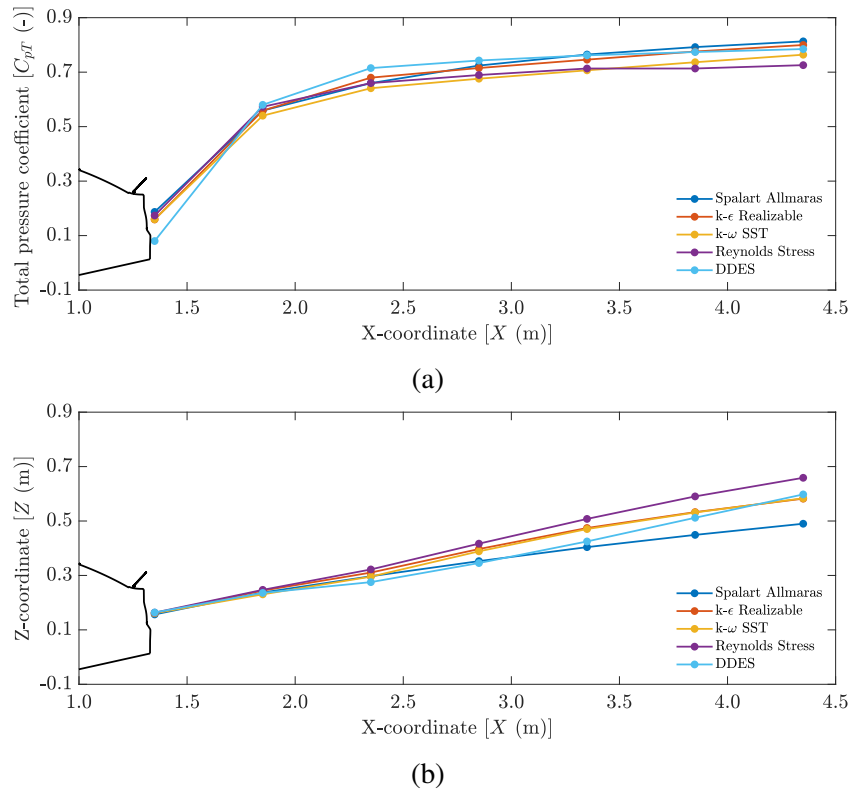


Figure 2.18: (a) Area-weighted average total pressure coefficient and (b) total-pressure-loss-weighted average vertical position of the wake on the WT installation at $1h_0$.

This is also evident in Figure 2.18, which reveals that the Spalart-Allmaras and $k-\epsilon$ Realizable models provide the most accurate representation of wake dissipation, particularly in the far-field, with deviations averaging around 3% from the DDES results. In contrast, the $k-\omega$ SST and Reynolds Stress models exhibit approximately 7% lower wake dissipation throughout most of the wake region. Despite achieving reasonable agreement in area-weighted total pressure across wake planes, the Reynolds Stress model uniquely displays minimal vortex diffusion and therefore depicts the lowest qualitative agreement with the DDES results.

Due to the minimal vortex diffusion, the Reynolds Stress model preserves stronger vortex interactions as the wake evolves downstream, enhancing upwashing effects and resulting in an approximately 10% higher vertical wake trajectory compared to the DDES results. The wake's vertical displacement is best captured by the $k-\epsilon$ Realizable and $k-\omega$ SST models, both achieving vertical position predictions within 2% of the DDES results at the furthest downstream plane. Conversely, the Spalart-Allmaras model struggles to capture vortex interactions accurately, leading to an 18% underprediction of the wake's vertical position at the most downstream plane.

2.3.1.6 Domain Sensitivity

Among key advantages of CFD simulations is the ability to control operating conditions, allowing for assessments of aerodynamic performance across different domains. An ideal turbulence model should produce robust and consistent results when applied to different computational setups, enabling reliable cross-domain analyses. To evaluate the sensitivity of each RANS model to variations in computational domain, simulations were conducted in both a wind tunnel (WT installation) and an open-road (wTS installation) domain.

These simulations provide a numerical estimation of wind tunnel blockage effects, which should remain largely unaffected by small ride height variations. As a result, a relatively stable offset between aerodynamic force coefficients from both domains is expected across the ride height range. Any significant deviation from this expected offset indicates increased sensitivity of a given RANS model to domain changes.

Figure 2.19 presents the force coefficients from the wTS installation normalised by those from the WT installation. The $k-\omega$ SST and Reynolds Stress models exhibit a nearly constant downforce offset across all ride heights, indicating strong cross-domain robustness. In contrast, the Spalart-Allmaras and $k-\epsilon$ Realizable models show fluctuations of approximately 4-5% in their downforce offset, highlighting a greater sensitivity to domain modifications. Additionally, the Spalart-Allmaras model presents deviations of around 2% in its drag offset, whereas the other RANS models demonstrate fluctuations of less than 1%.

2.3.1.7 Assessment Summary

Among the evaluated RANS models, the $k-\omega$ SST model has demonstrated the highest overall accuracy and robustness in predicting the aerodynamic performance of the *DriveAer hp-F*. Across the considered ride height range, it consistently estimated downforce and drag within approximately 2% of high-fidelity DDES results while maintaining comparable relative accuracy to experimental data.

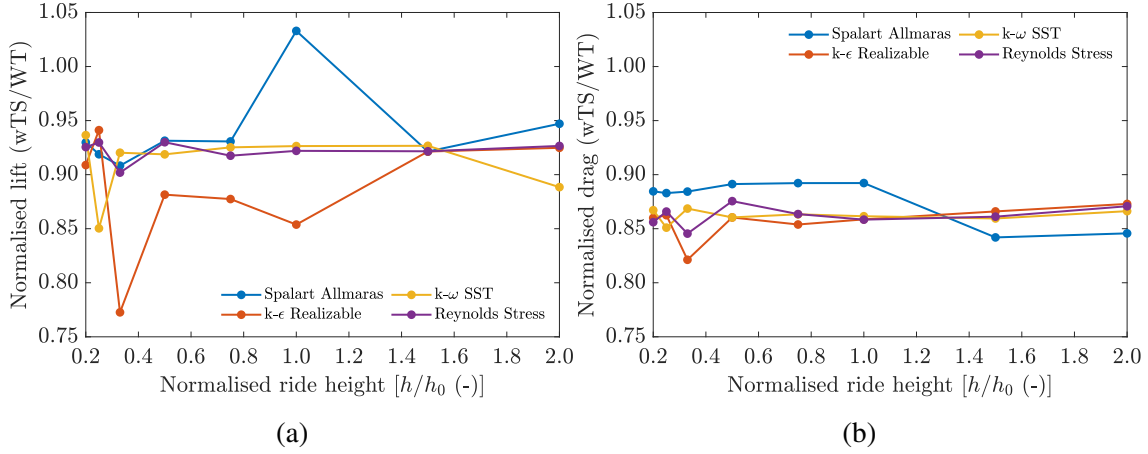


Figure 2.19: (a) Lift and (b) drag coefficients from the wTS installation normalised by results from the WT installation.

In terms of surface pressure distribution, the k- ω SST model effectively captured the key trends observed on upper surfaces. A discrepancy of approximately 10% was noted in the separated flow region over the slant and spoiler. Nonetheless, the k- ω SST model outperformed the other RANS approaches in this regard. For underbody flow, the k- ω SST model exhibited strong agreement with DDES predictions, with pressure distribution estimates generally within 4%. It accurately captured critical diffuser flow characteristics, including low-pressure concentrations, shear stress distribution, and streamwise velocity patterns. Consequently, diffuser-produced downforce was predicted with a deviation of just 3% from DDES results, reinforcing the model's reliability in predicting underbody aerodynamics.

Wake structure predictions further highlight the effectiveness of the k- ω SST model. It successfully captured dominant wake features, such as the counter-rotating vortex pair and wake traces from upstream components, though it slightly underpredicted wake dissipation by approximately 7%. Its wake trajectory predictions remained highly accurate, with deviations of less than 2% at the most downstream plane. Additionally, the k- ω SST model exhibited low sensitivity to variations in computational domain conditions, with relative deviations in force coefficient trends from the wind tunnel and open-road domains remaining below 1%, confirming its robustness for aerodynamic studies across different domains.

Overall, the k- ω SST model offers a good balance of accuracy, computational efficiency, and consistency for high-performance vehicle aerodynamics. It delivers adequate flow field predictions but requires 75 times less computational time while using half the computational resources compared to DDES simulations. Given its proven reliability across multiple aerodynamic aspects, the k- ω SST model is recommended as a practical and cost-effective approach for general aerodynamic performance evaluations and flow field characterisations of high-performance vehicles. Consequently, it is adopted for the subsequent analyses in this study.

2.3.2 Wind Tunnel Correlation Process

2.3.2.1 Blockage Effect

Test section wall confinement during wind tunnel experiments restricts airflow expansion, increasing local dynamic pressure near the vehicle model and altering its aerodynamics. To numerically assess this blockage effect, simulations are performed in both a computational wind tunnel domain (WT installation) and an open-road domain (wTS installation). As shown in Figure 2.23, the blockage effect causes a consistent increase of approximately 9% in downforce and 16% in drag across the ride height range when compared to open-road conditions.

To investigate blockage-induced aerodynamic alterations, the vehicle's surface pressure distributions in the wind tunnel and open-road domains are compared in Figure 2.20. The results reveal that low-pressure regions along the roof and underbody in the open-road domain experience further pressure reductions in the wind tunnel domain, indicating increased local airflow acceleration due to blockage, consistent with findings from prior studies (23, 58). The accelerated airflow along the roof increases local lift generation by approximately 8% compared to open-road conditions. Additionally, this accelerated flow influences downstream regions, enhancing spoiler downforce by around 3%. Similarly, higher underbody flow velocities in the wind tunnel domain result in approximately 7% and 9% greater downforce contributions from the vehicle's floor and diffuser, respectively.

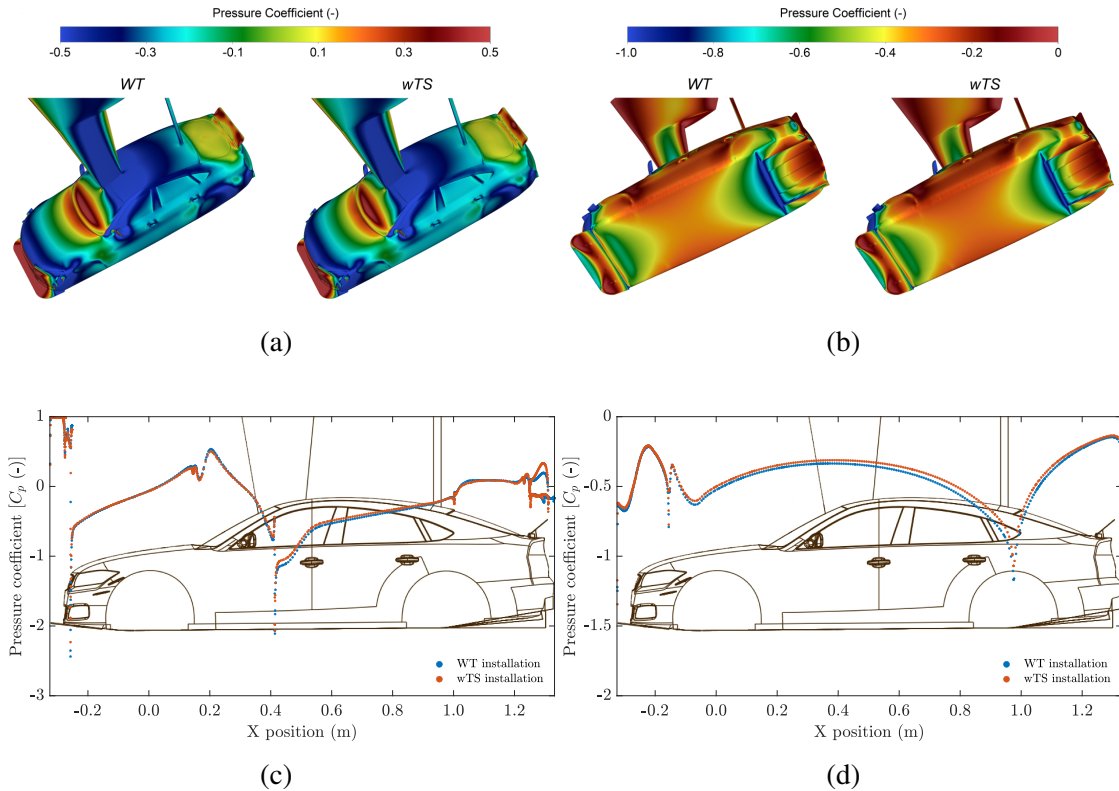


Figure 2.20: Comparison of surface pressure coefficients between the WT and wTS installations at $1h_0$, including (a) upper body and (b) underbody contour plots, as well as (c) upper body and (d) underbody distributions along a near-symmetry line ($y = -50$ mm).

In addition to its effects on downforce, blockage also induces a significant increase in drag, primarily due to increased pressure build-up at the vehicle's nose and enhanced flow separation at the rear. The former contributes to an approximate 11% increase in local drag at the front of the vehicle, while the latter leads to around 22% more pressure drag at the vehicle's base compared to open-road conditions. Furthermore, the elevated flow velocities in proximity to the vehicle intensify shear stress along the surface, increasing skin friction drag by over 7%.

A comparison of wake characteristics between the wind tunnel and open-road domains, shown in Figures 2.21 and 2.22, indicates that while the primary wake mechanisms remain largely unchanged, increased flow separation in the wind tunnel domain results in approximately 20% higher pressure losses in the near-wake at $x = 1.35$ m. However, enhanced wake dissipation is observed further downstream in the wind tunnel domain, leading to approximately 3% lower pressure losses at $x = 4.35$ m. This is attributed to a stronger counter-rotating vortex pair, which retains higher turbulent kinetic energy and promotes increased mixing with the high-energy freestream airflow. Additionally, as a consequence of intensified vortex interactions, the wake trajectory is approximately 5% higher in the wind tunnel domain relative to open-road conditions.

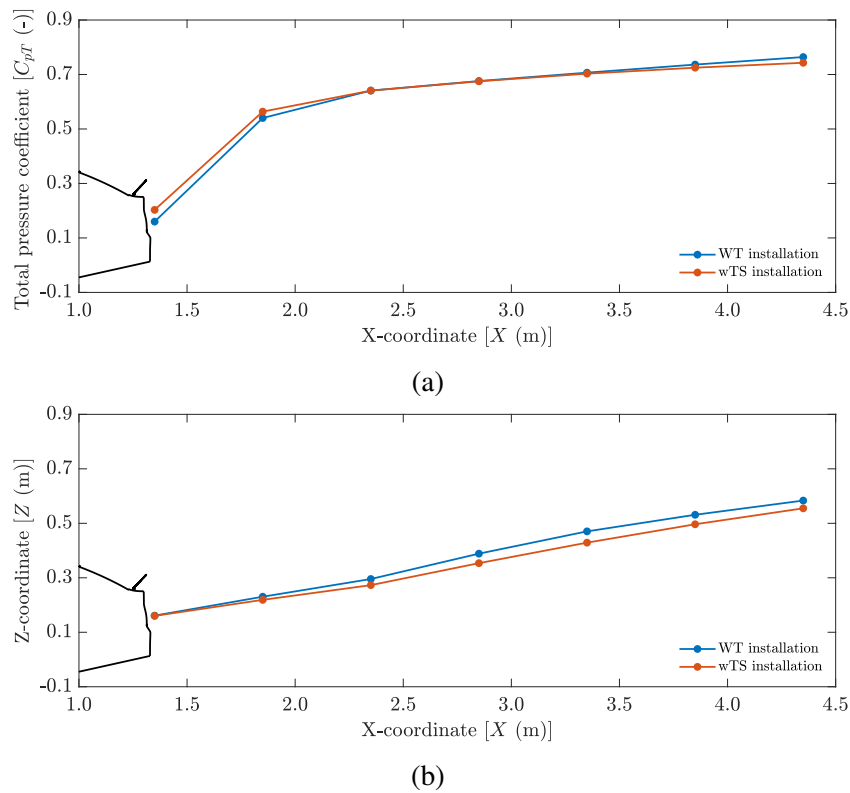


Figure 2.21: (a) Area-weighted average total pressure coefficient and (b) total-pressure-loss-weighted average vertical position of the wake on the WT and wTS installations at $1h_0$.

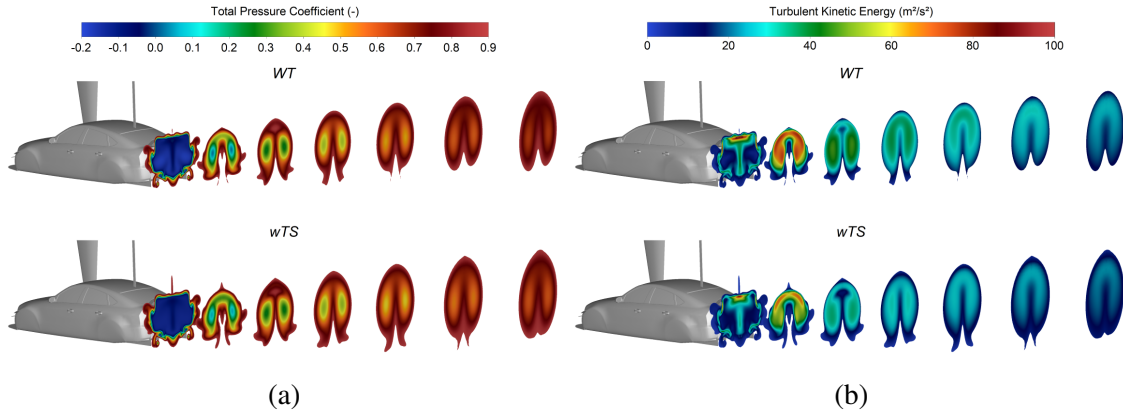


Figure 2.22: (a) Total pressure coefficient and (b) turbulent kinetic energy contour plots at wake planes ($x = 1.35$ m, $x = 1.85$ m, $x = 2.35$ m, $x = 3.35$ m, $x = 3.85$ m, and $x = 4.35$ m) on the WT and wTS installations at $1h_0$.

2.3.2.2 Blockage Correction Methods

To evaluate the accuracy of blockage correction methods, force coefficients obtained from the computational wind tunnel domain (WT installation) are corrected using various methods and compared against the numerical open-road reference (wTS installation). As shown in Figure 2.23, the corrected force coefficients exhibit a maximum relative spread of approximately 5% across different correction methods.

For the case under consideration, Maskell-based correction methods tend to undercorrect downforce by around 3% and drag by approximately 9%. The more complex Thom & Herriot and Mercker methods overcorrect downforce by approximately 2% and undercorrect drag by around 5%. The newly introduced CFD-based correction method provides adjusted force coefficients close to the average of the conventional methods, with an undercorrection of about 1% in downforce and 8% in drag.

Both the CFD and Maskell-based approaches estimate the dynamic pressure increase near the vehicle, which is then used to correct the force coefficients. While the CFD correction derives this information directly from representative simulation data, the Maskell-based methods rely on simplified geometric relationships. The agreement between these two approaches suggests that such geometric approximations provide a reasonable estimation of the blockage-induced dynamic pressure rise.

However, all examined correction methods apply a global correction, neglecting the distinct local effects on lift and drag previously discussed. As a result, conventional blockage correction methods should be applied with caution, particularly for drag, which may be significantly undercorrected. More detailed approaches, such as the numerical methodology used in this study to assess facility-induced effects, are therefore recommended for comprehensive aerodynamic analyses.

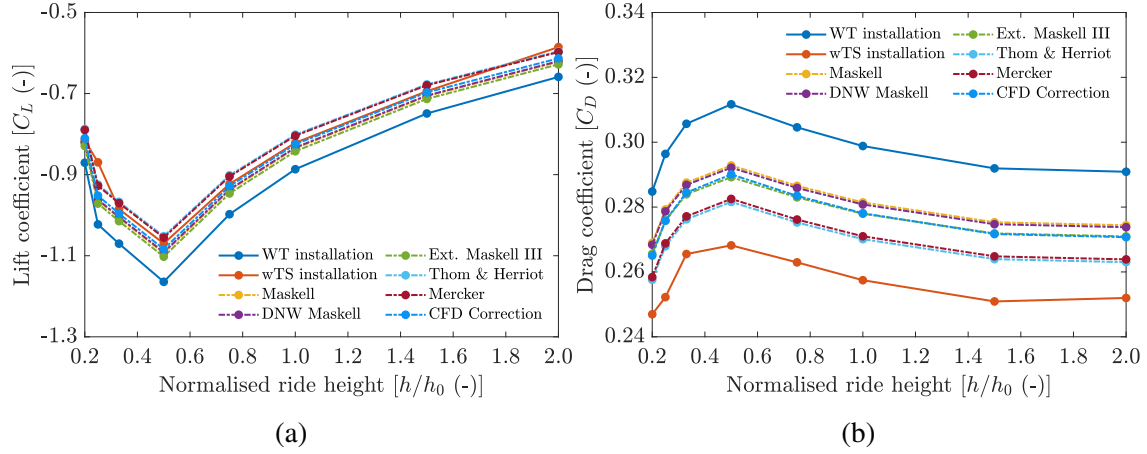


Figure 2.23: Correction methods applied to (a) lift and (b) drag coefficients from simulations on the WT installation and compared with results from the wTS installation.

2.3.2.3 Overhead Measurement System Interference

The aerodynamic interference effects of the overhead measurement system are numerically investigated by conducting simulations in the open-road domain with (wTS installation) and without (woTS installation) the overhead measurement system. The results indicate that its presence leads to a reduction in downforce of approximately 4% and a decrease in drag of around 8%, which aligns with previous findings on race car configurations (19).

Comparison of the surface pressure distribution for both vehicle configurations, shown in Figure 2.24a, indicates that stagnation at the leading edge of the overhead measurement system influences the recirculation zone ahead of the windscreen, increasing pressure on the windscreen itself. Additionally, the aerodynamic shape of the measurement system accelerates local airflow, leading to an approximate 6% increase in roof-generated lift. The system also produces relatively high-energy streamwise vortices, which energise airflow along the centre of the slant, as depicted in Figure 2.24b.

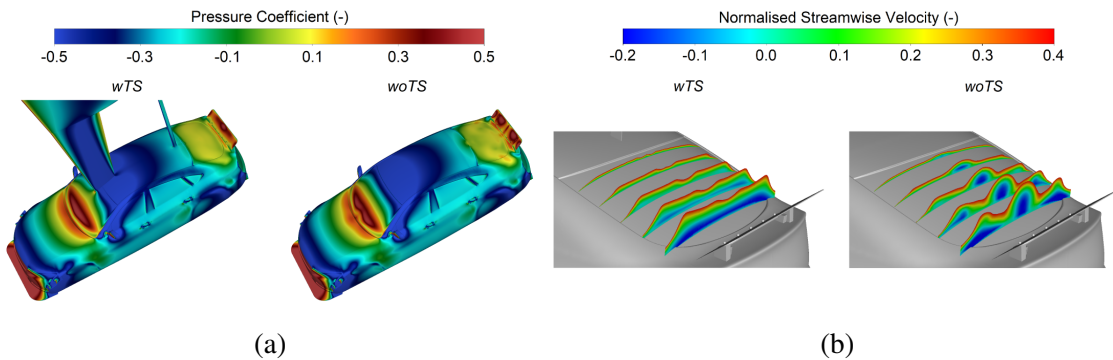


Figure 2.24: Comparison of (a) pressure coefficient contour plots on the upper body and (b) normalised streamwise velocity contour plots along vertical planes of the slant ($x = 1.02$ m, $x = 1.065$ m, $x = 1.11$ m, $x = 1.155$ m, and $x = 1.2$ m) between the wTS and woTS installations at $1h_0$.

Furthermore, the overhead measurement system alters the flow along the C-pillars, affecting the propagation of roofline vortices. Without the system, these vortices remain more inboard, as indicated by wider high-velocity airflow traces along the slant. This leads to a more uniform high-pressure distribution on the spoiler. In contrast, with the system in place, high-pressure concentrations become localised around the centre and edges of the spoiler, reducing spoiler-induced downforce by approximately 30%. The altered pressure distribution on the spoiler also impacts drag generation. The reduced pressure build-up with the overhead measurement system results in a similar 30% decrease in spoiler drag. Additionally, the diminished airflow interaction with the spoiler reduces local flow separation, contributing to an approximately 11% reduction in pressure drag around the vehicle's rear end.

2.4 Conclusion

This study systematically evaluated the predictive capabilities of four RANS turbulence models in capturing the aerodynamic characteristics of the *DrivAer hp-F*, benchmarking their performance against experimental force data and high-fidelity DDES simulation results. Additionally, a stepwise numerical correlation approach was employed to analyse wind tunnel blockage effects and interference from the overhead measurement system, while also assessing the applicability of various blockage correction methods.

The experimental force data showed that both downforce and drag increased as the vehicle was lowered from $2h_0$ to $0.33h_0$. However, below this critical ride height, aerodynamic performance deteriorated due to pronounced underbody flow unsteadiness. While the RANS models generally reproduced aerodynamic forces well at high to moderate ride heights, they failed to capture the force trends at lower ride heights. In contrast, the DDES approach accurately captured these trends across the entire ride height range.

Among the evaluated RANS models, the $k-\omega$ SST model exhibited the highest accuracy and consistency, predicting aerodynamic forces within approximately 2% of the DDES results and achieving comparable relative accuracy to experimental data. It also provided reliable estimations of the general surface pressure distribution and dominant wake characteristics. While all RANS models showed discrepancies in separated flow regions, particularly along the slant, the $k-\omega$ SST model demonstrated the best overall agreement with DDES predictions. Notably, it achieved this level of accuracy 75 times faster while using only half the computational resources required for the DDES simulations.

The numerical analysis of wind tunnel blockage effects revealed that the increased dynamic pressure near the vehicle model led to distinct localised alterations in the flow field. High-velocity regions, such as those on the roof and underbody, experienced further acceleration, resulting in an approximate 9% increase in total downforce. Simultaneously, increased pressure and skin friction drag contributed to an approximate 16% rise in total drag. Wake structure analyses indicated that, while the overall wake characteristics remained largely unchanged, increased separation in the wind tunnel domain led to higher pressure losses in the near-wake, with intensified wake dissipation observed further downstream.

The novel CFD-based correction method validated that conventional blockage correction approaches, based on geometric relationships, provided reasonable estimates of the blockage-induced dynamic pressure increase. However, their global corrections were more effective for downforce than for drag, where localised blockage effects were more pronounced. Additionally, the overhead measurement system altered the flow field by accelerating local airflow and inducing downstream interference, leading to a reduction of approximately 4% in downforce and 8% in drag.

Based on these findings, the $k-\omega$ SST model is recommended as a cost-effective approach for general aerodynamic studies and comparative design evaluations of high-performance vehicles at high to moderately low ride heights. For conditions dominated by highly unsteady flow behaviour, such as extremely low ride heights, high-fidelity DDES simulations are advised where computational resources permit. Furthermore, conventional blockage correction methods were found to be inadequate for high-performance vehicle applications, highlighting the need for a more comprehensive numerical quantification of facility-induced effects, as demonstrated in this study.

Overall, this research provided valuable insights into high-performance vehicle aerodynamics, the predictive accuracy of commonly used RANS models, and the impact of key wind tunnel-induced effects. The findings establish guidance on meshing strategies, turbulence model selection, and correlation processes, forming a robust foundation for integrated numerical and experimental aerodynamic investigations in both research and industry applications.

Data Availability Statement

All data supporting this study are included within the chapter.

References

1. Soares RF, de Souza FJ. Influence of CFD setup and brief analysis of flow over a 3D realistic car model. SAE Technical Paper 2015-36-0513. 2015. Available from: <https://doi.org/10.4271/2015-36-0513>.
2. Zhang C, Bounds CP, Foster L, Uddin M. Turbulence modeling effects on the CFD predictions of flow over a detailed full-scale sedan vehicle. *Fluids*. 2019;4(3):148. Available from: <https://doi.org/10.3390/fluids4030148>.
3. Igali D, Mukhmetov O, Zhao M, Fok SC, Teh L. Comparative analysis of turbulence models for automotive aerodynamic simulation and design. *International Journal of Automotive Technology*. 2019;20(6):1145-52. Available from: <https://doi.org/10.1007/s12239-019-0107-7>.
4. Fu C, Bounds CP, Selent C, Uddin M. Turbulence modeling effects on the aerodynamic characterizations of a NASCAR generation 6 racecar subject to yaw and pitch changes. *Proceedings of the Institution of Mechanical Engineers, Part D*:

- Journal of Automobile Engineering. 2019;233(14):3600-20. Available from: <https://doi.org/10.1177/0954407019826475>.
5. Heft AI, Indinger T, Adams NA. Experimental and numerical investigation of the DrivAer model. Proceedings of the ASME 2012 Fluids Engineering Division Summer Meeting collocated with the ASME 2012 Heat Transfer Summer Conference and the ASME 2012 10th International Conference on Nanochannels, Microchannels, and Minichannels. Volume 1: Symposia, Parts A and B. Rio Grande, Puerto Rico: The American Society of Mechanical Engineers; 2012. p. 41-51. Available from: <https://doi.org/10.1115/FEDSM2012-72272>.
 6. Guilmineau E. Numerical simulations of flow around a realistic generic car model. SAE International Journal of Passenger Cars - Mechanical Systems. 2014;7(2):646-53. Available from: <https://doi.org/10.4271/2014-01-0607>.
 7. Jakirlic S, Kutej L, Basara B, Tropea C. Computational study of the aerodynamics of a realistic car model by means of RANS and hybrid RANS/LES approaches. SAE International Journal of Passenger Cars - Mechanical Systems. 2014;7(2):559-74. Available from: <https://doi.org/10.4271/2014-01-0594>.
 8. Heft A, Indinger T, Adams N. Investigation of unsteady flow structures in the wake of a realistic generic car model. Proceedings of the 29th AIAA Applied Aerodynamics Conference. AIAA-2011-3669. Honolulu, Hawaii: AIAA; 2011. p. 14. Available from: <https://doi.org/10.2514/6.2011-3669>.
 9. Aljure DE, Calafell J, Baez A, Oliva A. Flow over a realistic car model: wall modeled large eddy simulations assessment and unsteady effects. Journal of Wind Engineering and Industrial Aerodynamics. 2018;174:225-40. Available from: <https://doi.org/10.1016/j.jweia.2017.12.027>.
 10. Jakirlic S, Kutej L, Unterlechner P, Tropea C. Critical assessment of some popular scale-resolving turbulence models for vehicle aerodynamics. SAE International Journal of Passenger Cars - Mechanical Systems. 2017;10(1):235-50. Available from: <https://doi.org/10.4271/2017-01-1532>.
 11. Ashton N, Revell A. Comparison of RANS and DES methods for the DrivAer automotive body. SAE Technical Paper 2015-01-1538. 2015. Available from: <https://doi.org/10.4271/2015-01-1538>.
 12. Ashton N, West A, Lardeau S, Revell A. Assessment of RANS and DDES methods for realistic automotive models. Computer and Fluids. 2016;128:1-15. Available from: <https://doi.org/10.1016/j.compfluid.2016.01.008>.
 13. Guilmineau E, Deng GB, Leroyer A, Queutey P, Visonneau M, Wackers J. Assessment of hybrid RANS-LES formulations for flow simulation around the Ahmed body. Computers & Fluids. 2018;176:302-19. Available from: <https://doi.org/10.1016/j.compfluid.2017.01.005>.

14. Mercker E. A blockage correction for automotive testing in a wind tunnel with closed test section. *Journal of Wind Engineering and Industrial Aerodynamics*. 1986;22(2):149-67. Available from: [https://doi.org/10.1016/0167-6105\(86\)90080-2](https://doi.org/10.1016/0167-6105(86)90080-2).
15. Wickern G. Wake related wind tunnel corrections for closed wall test sections. SAE Technical Paper 2006-01-0567. 2006. Available from: <https://doi.org/10.4271/2006-01-0567>.
16. Bin-Khalid U, Suda JM. Comparative study of wind tunnel blockage correction methods for bluff body aerodynamics (swimmer's hand model). MSc Thesis. Budapest University of Technology and Economics; 2020. Available from: <https://www.researchgate.net/publication/343127500>.
17. Hetherington B, Sims-Williams D. Wind tunnel model support strut interference. SAE Technical Paper 2004-01-0806. 2004. Available from: <https://doi.org/10.4271/2004-01-0806>.
18. Hetherington B. Interference of supports used for ground vehicle wind tunnel testing. PhD Thesis. Durham University; 2006. Available from: <http://etheses.dur.ac.uk/2671/>.
19. Hetherington B, Sims-Williams D. Support strut interference effects on passenger and racing car wind tunnel models. SAE Technical Paper 2006-01-0565. 2006. Available from: <https://doi.org/10.4271/2006-01-0565>.
20. Hyvärinen A. Investigation of blockage correction methods for full-scale wind tunnel testing of trucks. MSc Thesis. KTH, Royal Institute of Technology; 2015. Available from: <https://kth.diva-portal.org/smash/get/diva2:893809/FULLTEXT01.pdf>.
21. Yang Z, Schenkel M. Assessment of closed-wall wind tunnel blockage using CFD. SAE Technical Paper 2004-01-0672. 2004. Available from: <https://doi.org/10.4271/2004-01-0672>.
22. Yang Z, Nastov A, Schenkel M. Further assessment of closed-wall wind tunnel blockage using CFD. SAE Technical Paper 2005-01-0868. 2005. Available from: <https://doi.org/10.4271/2005-01-0868>.
23. Gupta S, Shah VJ. Numerical study on the effects of wind-tunnel conditions on a passenger vehicle in crosswinds. MSc Thesis. KTH Royal Institute of Technology; 2020. Available from: <https://kth.diva-portal.org/smash/get/diva2:1500220/FULLTEXT01.pdf>.
24. Carr G, Stapleford W. Blockage effects in automotive wind-tunnel testing. SAE Technical Paper 860093. 1986. Available from: <https://doi.org/10.4271/860093>.
25. Heft A, Indinger T, Adams N. Introduction of a new realistic generic car model for aerodynamic investigations. SAE Technical Paper 2012-01-0168. 2012. Available from: <https://doi.org/10.4271/2012-01-0168>.

26. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model. SAE Technical Paper 2018-01-0725. 2018. Available from: <https://doi.org/10.4271/2018-01-0725>.
27. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. DrivAer hp-F: the CAD geometry pack. Cranfield Online Research Data (CORD); 2018. Available from: <https://doi.org/10.17862/cranfield.rd.c.3969120>. (Accessed: 01 May 2022).
28. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. Dataset for SAE 2018-01-0725 “On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model”. Cranfield Online Research Data (CORD); 2018. Available from: <https://doi.org/10.17862/cranfield.rd.c.3969120>. (Accessed: 01 May 2022).
29. Ahmad NE, Abo-Serie E, Gaylard A. Mesh optimization for ground vehicle aerodynamics. CFD Letters. 2010;2(1):54-65. Available from: https://pure.coventry.ac.uk/ws/portalfiles/portal/12035851/63_210_1_PB.pdf.
30. Jowsey L, Passmore M. Experimental study of multiple-channel automotive underbody diffusers. Proceedings of the Institution of Mechanical Engineers, Part D: The Journal of Automobile Engineering. 2010;224(7):865-79. Available from: <https://doi.org/10.1243/09544070JAUTO1339>.
31. Jowsey L. An experimental study of automotive underbody diffusers. PhD Thesis. Loughborough University; 2013. Available from: <https://hdl.handle.net/2134/13646>.
32. Menter FR. Best practice: scale resolving simulations in Ansys CFD. ANSYS, Inc; 2021. Available from: <https://www.ansys.com/en-gb/resource-center/technical-paper/best-practice-scale-resolving-simulations-in-ansys-cfd>. (Accessed: 01 May 2022).
33. Lanfrit M. Best practice guidelines for handling automotive external aerodynamics with Fluent. University of Southampton; 2005. Available from: https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf. (Accessed: 01 May 2022).
34. Zore K, Caridi D, Lockley I. Fast and accurate prediction of vehicle aerodynamics using ANSYS mosaic mesh. SAE Technical Paper 2020-01-5011. 2020. Available from: <https://doi.org/10.4271/2020-01-5011>.
35. Modeling turbulent flows - introductory Fluent training. Ansys Inc; 2006. Available from: https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence_Notes_Fluent-v6.3.06.pdf. (Accessed: 01 May 2022).
36. Skaperdas V. Meshing for the 1st automotive CFD prediction workshop. BETA SIMULATION SOLUTIONS; 2019. Available from: <https://autocfd1.s3.eu-west-1.amazonaws.com/Presentations/BETA-CAE-Case2aMeshing.pdf>. (Accessed: 01 May 2022).

37. Roache PJ. Perspective: a method for uniform reporting of grid refinement studies. *Journal of Fluids Engineering*. 1994;116(3):405-13. Available from: <https://doi.org/10.1115/1.2910291>.
38. Celik I, Ghia U, Roache PJ, Freitas C, Coloman H, Raad P. Procedure of estimation and reporting of uncertainty due to discretization in CFD applications. *Journal of Fluids Engineering*. 2008;130(7):078001. Available from: <https://doi.org/10.1115/1.2960953>.
39. Haziqah N, Sukri M, Mat Ali MS, Mansor S, Muhamad S, Zaki A. Numerical study for flow over a realistic generic model, DrivAer, using URANS. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*. 2018;48(2):183-95. Available from: <https://api.semanticscholar.org/CorpusID:140121070>.
40. Kumar P, Guilmineau E. URANS simulation of a realistic road vehicle with $k-\omega$ SST and SSG/LRR- ω model as turbulence closure. *Proceedings of the Fourth International Conference in Numerical and Experimental Aerodynamics of Road Vehicles and Trains. Aerovehicles4*. Berlin, Germany: hal-03365255; 2020. Available from: <https://hal.science/hal-03365255/document>.
41. Guerrero A, Castilla R. Aerodynamic study of the wake effects on a Formula 1 car. *Energies*. 2020;13(19):5183. Available from: <https://doi.org/10.3390/en13195183>.
42. Lotz RD, Thompson BE, Konings CA. Numerical uncertainties in transonic flow calculations for airfoils with blunt trailing edges. *International Journal for Numerical Methods in Fluids*. 1997;24:355-73. Available from: [https://doi.org/10.1002/\(SICI\)1097-0363\(19970228\)24:4<355::AID-FLD498>3.0.CO;2-7](https://doi.org/10.1002/(SICI)1097-0363(19970228)24:4<355::AID-FLD498>3.0.CO;2-7).
43. Katz J. Aerodynamics of race cars. *The Annual Review of Fluid Mechanics*. 2006;38:27-63. Available from: <https://doi.org/10.1146/annurev.fluid.38.050304.092016>.
44. Himeno R, Fujitani K. Numerical analysis and visualization of flow in automobile aerodynamics development. *Journal of Wind Engineering and Industrial Aerodynamics*. 1993;46-47:785-90. Available from: [https://doi.org/10.1016/0167-6105\(93\)90354-Q](https://doi.org/10.1016/0167-6105(93)90354-Q).
45. Ahmed SR, Ramm G, Faitin G. Some salient features of the time-averaged ground vehicle wake. *SAE Technical Paper 840300*. 1984. Available from: <https://doi.org/10.4271/840300>.
46. Slater JW. Examining spatial (grid) convergence. NASA; 2021. Available from: <https://www.grc.nasa.gov/WWW/wind/valid/tutorial/spatconv.html>. (Accessed: 01 May 2022).
47. Spalart P, Allmaras S. A one-equation turbulence model for aerodynamic flows. *Proceedings of the 30th Aerospace Sciences Meeting and Exhibit*. Reno, USA: AIAA; 1992. Available from: <https://doi.org/10.2514/6.1992-439>.

48. Shih TH, Liou WW, Shabbir A, Yang Z, Zhu J. A new k- ϵ eddy viscosity model for high reynolds number turbulent flows. *Computers & Fluids*. 1995;24(3):227-38. Available from: [https://doi.org/10.1016/0045-7930\(94\)00032-T](https://doi.org/10.1016/0045-7930(94)00032-T).
49. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*. 1994;32(8):1598-605. Available from: <https://doi.org/10.2514/3.12149>.
50. Launder B, Reece G, Rodi W. Progress in the development of a reynolds stress turbulence closure. *Journal of Fluid Mechanics*. 1975;68(3):537-66. Available from: <https://doi.org/10.1017/S0022112075001814>.
51. Ansys Fluent theory guide - release 2022 R1. ANSYS, Inc; 2022.
52. Menter FR, Kuntz M, Langtry R. Ten years of industrial experience with the SST turbulence model. *Heat and Mass Transfer*. 2003;4:625-32. Available from: <https://api.semanticscholar.org/CorpusID:38105247>.
53. Aultman M, Wang Z, Duan L. Effect of time-step size on flow around generic car models. *Journal of Wind Engineering and Industrial Aerodynamics*. 2021;219:104764. Available from: <https://doi.org/10.1016/j.jweia.2021.104764>.
54. Porcar L, Toet W, Gamez-Montero J. Study of the effect of vertical airfoil endplates on diffusers in vehicle aerodynamics. *Designs*. 2021;5(3):45. Available from: <https://doi.org/10.3390/designs5030045>.
55. Zhang X, Senior A, Ruhrmann A. Vortices behind a bluff body with an upswept aft section in ground effect. *International Journal of Heat and Fluid Flow*. 2004;25(1):1-9. Available from: <https://doi.org/10.1016/j.ijheatfluidflow.2003.11.002>.
56. Breuer M, Jovičić N, Mazaev K. Comparison of DES, RANS and LES for the separated flow around a flat plate at high incidence. *International Journal for Numerical Methods in Fluids*. 2003;41:357-88. Available from: <https://doi.org/10.1002/fld.445>.
57. Crompton MJ, Barret RV. Investigation of the separation bubble formed behind the sharp leading edge of a flat plate at incidence. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*. 2000;214(3):157-76. Available from: <https://doi.org/10.1243/0954410001531980>.
58. Ljungskog E, Sebben S, Broniewicz A. Inclusion of the physical wind tunnel in vehicle CFD simulations for improved prediction quality. *Journal of Wind Engineering and Industrial Aerodynamics*. 2020;197:104055. Available from: <https://doi.org/10.1016/j.jweia.2019.104055>.

Chapter 3

Aerodynamic Analysis and Turbulence Model Evaluation at Varying Ride Heights

This chapter was originally published as:

1. Rijns S., Teschner TR, Blackburn K, Brighton J. Aerodynamic analysis of the ride height dependency of a high-performance vehicle equipped with a multichannel diffuser in ground effect. SAE Technical Paper 2023-01-5064. 2023. Available from: <https://doi.org/10.4271/2023-01-5064>.
2. Rijns S, Teschner TR, Blackburn K, Brighton J. Comparative analysis of RANS and DDES methods for aerodynamic performance predictions for high performance vehicles at low ground clearances. UKACM 2023 Conference. Coventry, UK: UKACM; 2023.

Abstract

The floor and diffuser of high-performance vehicles are key contributors to downforce generation, yet most research has focused on simplified bluff bodies with plane diffusers, leaving the flow physics of more complex multichannel diffusers less understood. This study investigates the aerodynamic performance of a high-performance vehicle equipped with a multichannel diffuser across various ride heights. A comparative assessment of Reynolds-Averaged Navier-Stokes (RANS) and Delayed Detached Eddy Simulation (DDES) models reveals that while both approaches effectively predict downforce and underbody flow characteristics at medium to high ride heights, only the DDES model captures the unsteady flow phenomena that dominate at lower ride heights. An in-depth aerodynamic analysis using the DDES results highlights the critical role of the diffuser's side plate vortices in driving flow attachment and thereby downforce generation. At extremely low ride heights, increased ground flow interactions destabilise diffuser vortex flow, weakening flow entrainment and reducing downforce. Overall, the findings deepen understanding of ride height effects on high-performance vehicles with multichannel diffusers, guiding underbody design considerations. The turbulence model evaluation further underscores the trade-off between cost-effective RANS models and high-fidelity DDES simulations in capturing unsteady flow, informing Computational Fluid Dynamics (CFD) methodology development for investigations on vehicle aerodynamics.

Keywords: Aerodynamics; CFD; RANS; DDES; Diffuser; Ground Effect; DrivAer

3.1 Introduction

The aerodynamic design of racing and high-performance vehicles is primarily focused on maximising downforce and minimising drag to enhance handling and overall performance. The vehicle's underbody, particularly the floor and diffuser, plays a crucial role in this process. In Formula One cars, for example, the underbody contributes to over 50% of the total downforce while generating only about 20% of the overall drag (1). The fundamental mechanism behind underbody downforce generation is based on the Venturi effect, where airflow is accelerated between the vehicle's floor and the ground, creating a low-pressure region underneath the vehicle. The diffuser expansion facilitates a controlled pressure recovery, increasing mass flow through the underfloor region and inducing suction at the diffuser inlet, thereby enhancing downforce generation (2).

Experimental studies on simplified bluff bodies equipped with plane diffusers have extensively characterised the influence of ride height on downforce and drag (2, 3, 4, 5, 6, 7). In addition to the Venturi effect, these studies highlighted the crucial role of vortex structures formed at the diffuser's edges, which aided in flow attachment and downforce generation. The relationship between downforce and ride height generally followed a trend in which decreasing ride height initially increased downforce until a critical point was reached, beyond which further reduction led to flow separation and downforce degradation. At low ride heights, the vortex structures in plane diffusers were found to become unstable, resulting in asymmetric flow patterns that negatively impact aerodynamic performance.

Experimental investigations on more complex multichannel diffusers have demonstrated superior aerodynamic performance compared to plane diffusers (8, 9). The additional channels were found to mitigate asymmetric flow conditions observed in plane diffusers at very low ride heights. Notably, four-channel diffusers were shown to exhibit particularly strong performance due to the formation of secondary vortices within the inner channels, which contributed to improved stability and downforce generation.

Numerical studies have also predominantly focused on plane diffusers, often at fixed ride heights (10, 11, 12). An exception is the study by Mayoral et al. (13), which explored ride height effects but was constrained to medium ride heights due to the limitations of Reynolds-Averaged Navier-Stokes (RANS) models in accurately capturing unsteady flow behaviour near the critical ride height for maximum downforce. However, research on multichannel diffusers remains limited. While RANS simulations have identified primary and secondary vortex structures in these configurations (14, 15), their ride height dependency and impact on aerodynamic performance have not been thoroughly examined yet.

Despite the advantages of multichannel diffusers, gaps remain in understanding their aerodynamic behaviour, particularly at varying ride heights. Most research has focused on simplified bluff bodies with plane diffusers, limiting the applicability of findings to realistic high-performance vehicles. Previous numerical studies were often restricted to specific operating conditions or relied on RANS simulations, which struggled to capture unsteady flow and vortex dynamics at low ride heights. These limitations highlight the need for high-fidelity simulations on realistic vehicle geometries with multichannel diffusers to assess aerodynamic performance across a range of ride heights.

This study addresses these gaps by conducting a detailed numerical investigation of a high-performance vehicle equipped with a four-channel diffuser. It examines the aerodynamic performance of this configuration across varying ride heights using Computational Fluid Dynamics (CFD) simulations with both RANS and Delayed Detached Eddy Simulation (DDES) models. The study benchmarks RANS predictions against experimental data and high-fidelity DDES results to assess its accuracy in capturing critical flow features. Additionally, the DDES simulations are used to provide an in-depth analysis of the unsteady flow phenomena governing diffuser performance, with particular attention to vortex formation and their role in diffuser aerodynamics. These findings provide insight into the applicability of RANS and DDES models for predicting ride height sensitivity, and enhance understanding of ride height effects on high-performance vehicles with multichannel diffusers.

3.2 Methodology

3.2.1 Vehicle Geometry

The DrivAer model is a widely used generic vehicle model designed for aerodynamic research (16). This study employs the high-performance variant of the DrivAer Fastback, known as the *DrivAer hp-F* model, developed by Soares et al. (17). As shown in Figure 3.1, the *DrivAer hp-F* is a 35% scale model featuring a smooth underbody, side mirrors, enclosed wheel cavities, and various aerodynamic add-ons, including a front splitter, forebody strakes, a rear spoiler, and an underbody diffuser. The key geometric parameters are summarised in Table 3.1, with further details available in (17). To support the analysis of diffuser aerodynamics, Figure 3.2 provides a detailed breakdown of its components.

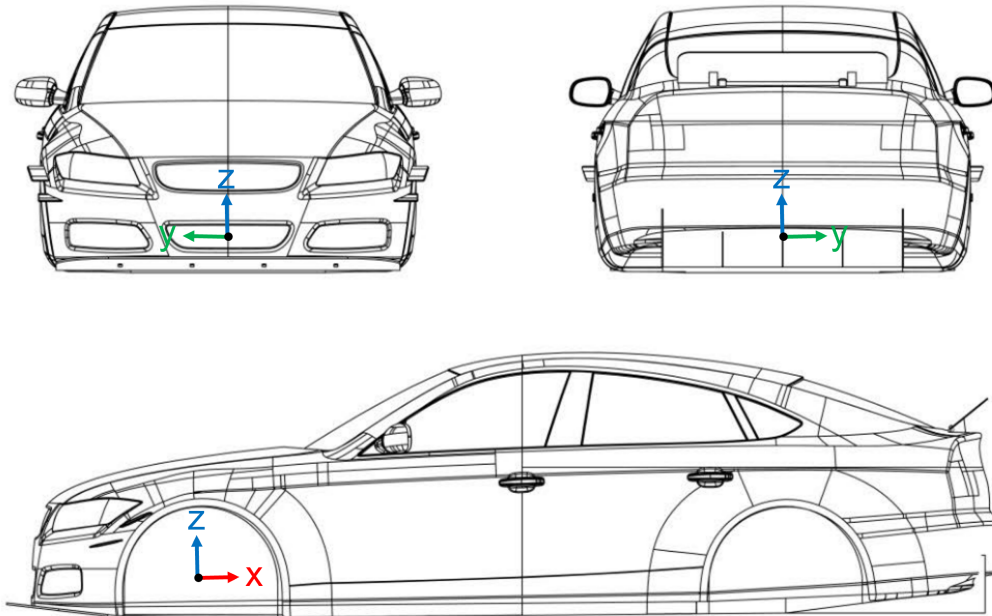


Figure 3.1: Illustration of the *DrivAer hp-F* model.

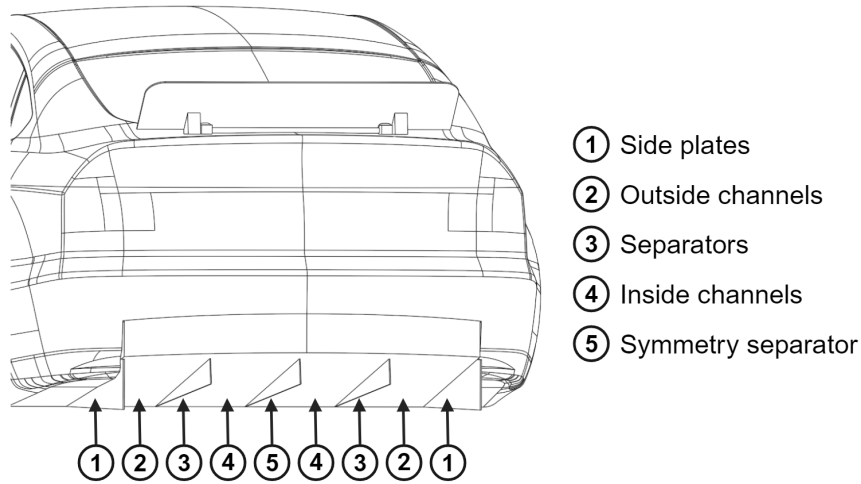


Figure 3.2: Description of the multichannel diffuser.

3.2.2 Experimental Validation Data

The numerical simulations in this study are validated against wind tunnel experiments conducted by Soares et al. (18) on the *DrivAer hp-F* model in the 8x6 Wind Tunnel at Cranfield University. The experimental setup incorporated a moving ground system and a two-stage boundary layer suction system. Aerodynamic forces were recorded across eight ride heights at a freestream velocity of 40 m/s using an overhead balance system with full-scale uncertainties of ± 0.90 N for lift and ± 0.18 N for drag. The measurement setup maintained a blockage ratio of approximately 10.2%.

3.2.3 Computational Fluid Dynamics Simulations

3.2.3.1 Domain and Boundary Conditions

Accurately reproducing wind tunnel conditions in CFD simulations is crucial for achieving strong correlation between numerical and experimental results, as demonstrated in Chapter 2 (19). Accordingly, the computational domain follows the methodology detailed in Section 2.2.3.1, ensuring close alignment with the testing conditions of the 8x6 Wind Tunnel at Cranfield University.

Table 3.1: Main dimensions of the *DrivAer hp-F* model.

Parameter	Symbol	Value	Unit
Car scale	CS	35	%
Length	L	1.6144	m
Width	W	0.6273	m
Diffuser length	L_D	0.3482	m
Ride height	h_0	0.0455	m
Area	A_{ref}	0.2556	m ²

In straight-line conditions, as in the experimental setup, the time-averaged flow field of a vehicle can be reasonably assumed to be symmetric. This assumption is further supported by prior research on multichannel diffusers, such as the one implemented on the *DrivAer hp-F*, which mitigate asymmetric flow structures typically observed in planar diffusers at low ride heights (8, 9). Leveraging this symmetry enables the use of a half-car model, which has been demonstrated to reduce mesh sizes by over 40% while maintaining force deviations below 1% compared to full-car simulations (20, 21).

Although the DDES simulations inherently resolve unsteady turbulence, which does not obey instantaneous symmetry (22), the focus of this study is on time-averaged flow properties, where turbulent fluctuations along the symmetry plane have a negligible influence. The strong agreement between DDES predictions and experimental force measurements, as will be shown in Section 3.3.1, further validates this modelling approach.

The computational wind tunnel, illustrated in Figure 3.3, is modelled as a straight test section with a height of 1.787 m and a width of 1.219 m, incorporating 431.05 mm long corner fillets. Following established best practices in automotive aerodynamics (23), the domain extends from a velocity inlet located three vehicle lengths (L) upstream to a pressure outlet positioned $5L$ downstream of the vehicle, ensuring proper flow development before and sufficient wake propagation behind the vehicle.

Boundary conditions are prescribed to replicate the wind tunnel setup. The ground plane is modelled as a no-slip moving wall to simulate the experimental moving ground system, while the top and side walls are treated as no-slip stationary surfaces. The inlet and moving ground velocity are set to match the freestream velocity of $U_\infty = 40$ m/s, with air properties corresponding to those in the experiments, as detailed in Table 3.2. To maintain consistency with the physical setup, the computational model explicitly incorporates both the vehicle and the overhead measurement system, preserving a blockage ratio of approximately 10.2%.

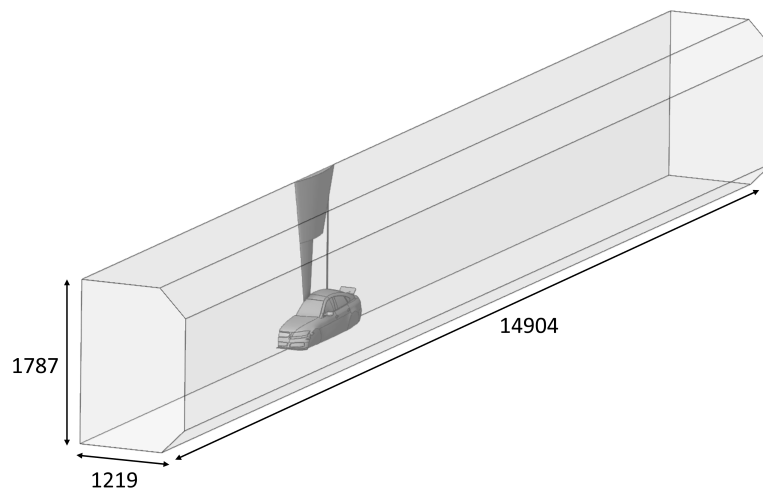


Figure 3.3: Illustration of the computational wind tunnel domain with dimensions in mm.

Table 3.2: Overview of the wind tunnel air properties.

Parameter	Symbol	Value	Unit
Air density	ρ	1.1678	kg/m ³
Air dynamic viscosity	μ	18.377e-6	kg/(m·s)
Atmospheric pressure	P_{atm}	99982	Pa
Dynamic pressure	P_{dyn}	934.18	Pa
Tunnel temperature	T	25.1	°C
Turbulent intensity	I_T	0.052	%

3.2.3.2 Mesh

Unstructured poly-hexcore meshes for the RANS and DDES simulations are generated in ANSYS Fluent Meshing, following the validated methodology in Section 2.2.3.2 (19), based on prior meshing strategies for automotive aerodynamics (21, 24, 25). Both meshes use a base mesh and vehicle surface mesh size of approximately 7.4% and 0.45% of the vehicle's length, respectively. To enhance geometric resolution, local surface refinement is applied to aerodynamic devices and small features, reducing element size in these areas by over 80%.

The RANS mesh adopts a practical near-wall treatment, with an average y^+ of approximately 150 and four inflation layers along the vehicle surface. In contrast, the DDES mesh employs a significantly more refined near-wall treatment, with an average y^+ of approximately 0.7 and 15 inflation layers, ensuring enhanced resolution of near-wall flow. The wind tunnel walls are treated with a y^+ of around 170, incorporating six inflation layers in both meshes to capture boundary layer development.

To balance computational efficiency with flow field resolution, four refinement zones progressively increase the element size from the near-field to the far-field domain, ensuring a smooth transition, as shown in Figure 3.4. In the DDES mesh, finer near-field resolution ensures a gradual transition between the inflation layers and surrounding cells, maintaining compatibility with the low y^+ treatment. The final grids contain approximately 6 million cells for the RANS simulations and 22 million cells for the DDES simulations.

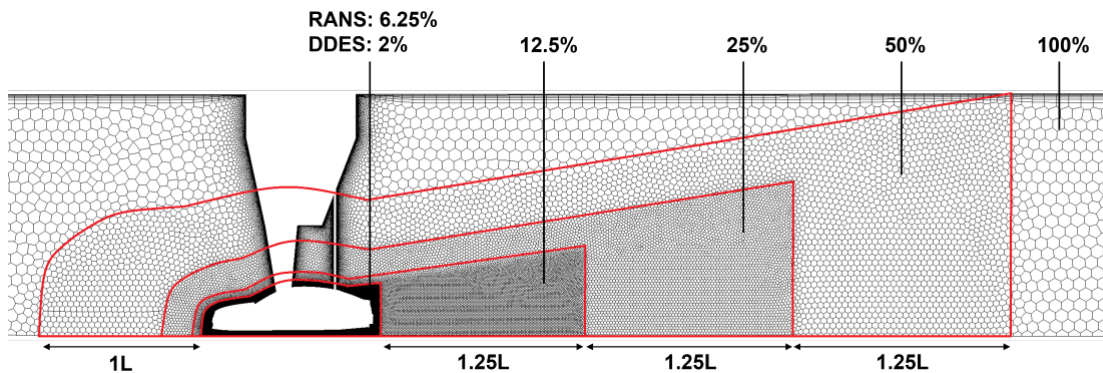


Figure 3.4: Close-up view of the mesh on the symmetry plane, highlighting refinement zones in red with element sizes expressed as a percentage of the base mesh size.

3.2.3.3 Solve

The RANS simulations employ the $k-\omega$ SST turbulence model (26) in ANSYS Fluent, selected based on the turbulence model evaluation in Section 2.3.1 (19), where it demonstrated the highest accuracy and consistency in predicting the aerodynamic performance of the *DrivAer hp-F*. The governing equations include continuity, momentum, and transport equations for turbulent kinetic energy and specific dissipation rate.

The numerical setup follows the methodology outlined in Section 2.2.3.3, with pressure-velocity coupling handled using the coupled scheme, gradients computed via the least-squares cell-based method, and pressure discretised using a second-order scheme. The momentum and transport equations are solved using a second-order upwind scheme. Each simulation runs for 1000 iterations, with force coefficients averaged over the final 500 iterations. Computations are performed on two AMD EPYC 7543 processors using 64 cores, with an average simulation time of approximately 1.2 hours per case.

The DDES simulations (27) also adopt the methodology from Section 2.2.3.3, utilising the $k-\omega$ SST RANS background model. These simulations are initialised using steady-state $k-\omega$ SST RANS solutions on the high near-wall resolution mesh. The numerical scheme mirrors that of the RANS setup except for the momentum equation, which is solved using the bounded central differencing scheme, as recommended by ANSYS Fluent (28). Furthermore, the governing equations are temporally discretised using a bounded second-order implicit formulation.

Time step selection follows the approach of Aultman et al. (29), where a non-dimensional time step of 5χ provides a balance between computational cost and resolution of turbulent structures. Accordingly, a physical time step of $t = 3.125 \times 10^{-4}$ s with 10 inner iterations per step is employed. Each simulation runs for 2 s of flow time, equivalent to 49.6 convective time units (CTUs), with the initial 1 s (24.8 CTUs) dedicated to washing out the RANS initialisation and achieving statistical convergence, followed by another 1 s (24.8 CTUs) for flow field averaging and collection of unsteady statistics. The DDES simulations are conducted on four AMD EPYC 7543 processors using 128 cores, requiring approximately 90 hours per case.

3.3 Results and Discussion

This section first examines the influence of ride height on the vehicle's aerodynamic performance using experimental force data. Subsequently, the predictive capabilities of the RANS and DDES turbulence models are evaluated, followed by a detailed analysis of the vehicle's aerodynamic behaviour and underbody flow structures as a function of ride height based on DDES results. Several figures within this section present flow field data on frontal planes positioned within the diffuser. These planes are normalised by the diffuser's length L_D (m) and referenced relative to the diffuser inlet, located at $x = 0.975$ m.

The experimental force coefficient data, presented in Figure 3.5, demonstrate a pronounced influence of ride height on aerodynamic performance. As the vehicle's ride height decreases from $2h_0$ to $0.33h_0$, downforce increases by more than 80%.

This enhancement is primarily driven by stronger flow acceleration beneath the vehicle, which generates a low-pressure region, thereby producing more downforce. However, this increase in downforce is accompanied by a rise in induced drag, leading to a nearly 10% increase in overall drag within this ride height range. At ride heights below $0.33h_0$, further reductions in ground clearance lead to a decline in downforce. Over the $0.33h_0 - 0.2h_0$ range, downforce decreases by approximately 6%, with a corresponding reduction in drag of around 3%. The mechanisms driving this deterioration in aerodynamic performance will be analysed in detail in Section 3.3.2.

It is important to note that the experimental data and subsequent analyses are based on a wheel-less vehicle configuration. Prior studies have shown that front wheels cause lower-energy airflow to enter the diffuser, while rear wheels suppress vortex formation along the sides of the diffuser and introduce wake disturbances into the diffuser region (30, 31, 32). As such, the presence of wheels is expected to reduce overall aerodynamic performance, limit downforce gains with decreasing ride height, and cause an earlier onset of critical ride height conditions. Nonetheless, the exclusion of wheels in this study enables focused analyses of the core flow mechanisms driving underbody and diffuser performance, providing meaningful insights and a robust basis for evaluating the predictive capabilities of RANS and DDES simulations.

3.3.1 RANS vs DDES

The comparison between numerical predictions and experimental data, presented in Figure 3.5, indicates that the DDES model consistently overestimates downforce by approximately 7% across the entire ride height range. Drag is slightly underpredicted by about 2% within the $2h_0 - 0.5h_0$ ride height range but converges more closely with experimental data at lower ride heights of $0.33h_0 - 0.2h_0$. The RANS model exhibits a slightly greater overestimation of downforce, averaging nearly 10% within the $2h_0 - 0.5h_0$ range. However, at critically low ride heights of $0.33h_0 - 0.2h_0$, the RANS model predicts a pronounced decline in downforce. Within the $2h_0 - 0.5h_0$ range, drag is underpredicted by about 1%, but at lower ride heights, the underprediction increases, reaching up to 5%.

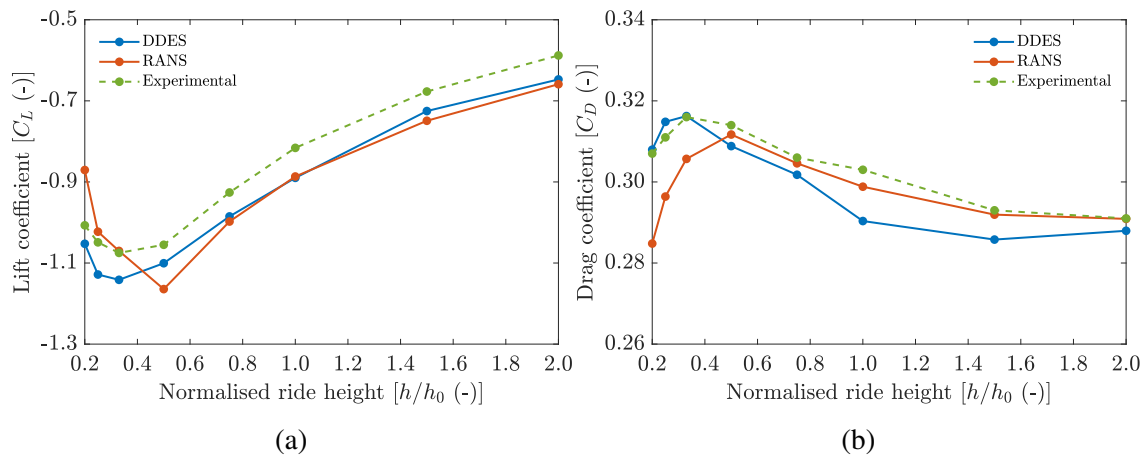


Figure 3.5: Comparison of (a) lift and (b) drag coefficients obtained from wind tunnel experiments and numerical simulations using RANS and DDES turbulence models.

Both turbulence models accurately capture the overall trend of increasing downforce and drag with decreasing ride height within the $2h_0 - 0.5h_0$ range. The RANS model provides downforce predictions that on average deviate by approximately 2% from the DDES results, while requiring around 75 times less computational time and utilising only half the number of processing cores. However, at very low ride heights of $0.33h_0 - 0.2h_0$, the RANS model diverges significantly from both experimental data and DDES predictions, failing to predict the correct force trends.

The systematic overestimation of downforce in both models may be attributed to simplifications in the computational wind tunnel domain, particularly in the contraction and diffuser sections. Hyvärinen (33) demonstrated that such simplifications can introduce minor flow field variations and reduce static pressure within the tunnel, consequently affecting the correlation of force predictions.

The limitations of the RANS model at low ride heights are further highlighted in the surface pressure distributions on the vehicle's underbody, presented in Figure 3.6a. At $1h_0$, both models predict similar pressure distributions, with only minor discrepancies near the symmetry separator, where localised flow separation is captured by the DDES model. The low-pressure regions generated by vortices along the diffuser side plates and separators are comparable in both size and magnitude at this ride height. However, at $0.33h_0$, the RANS model predicts significantly greater flow separation within the diffuser, leading to a higher overall surface pressure compared to the DDES results. Additionally, the low-pressure regions associated with the side plate and separator vortices are notably weaker and more confined in the RANS predictions, indicating reduced vortex strength.

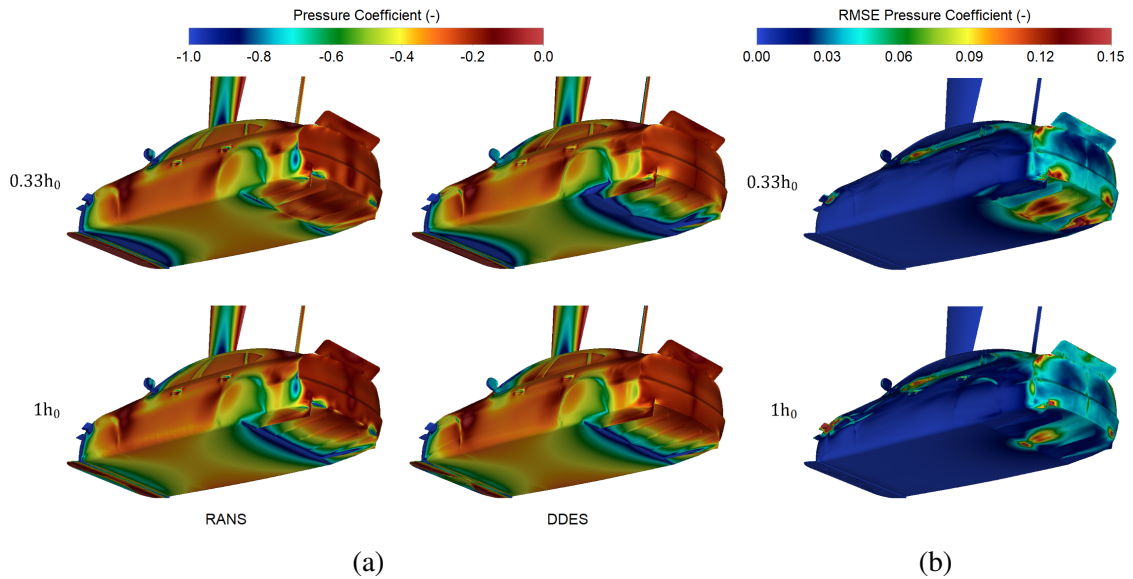


Figure 3.6: (a) Pressure coefficient contour plots from RANS and DDES simulations, and (b) contour plots of the Root Mean Square Error (RMSE) of the pressure coefficients from DDES simulations at ride heights of $0.33h_0$ and $1h_0$.

These discrepancies in surface pressure correlate with regions of elevated pressure fluctuations, as illustrated in Figure 3.6b. At $1h_0$, moderate pressure fluctuations are observed around the symmetry separator due to minor flow separation, while fluctuations near the side plate and separator vortices remain relatively small, indicative of steady vortex behaviour (34). In contrast, at $0.33h_0$, the diffuser exhibits significantly larger pressure fluctuations, particularly near the symmetry separator and along the side plates. This suggests a transition to highly unsteady vortex flow at low ride heights, a phenomenon that the RANS model fails to capture accurately.

The transition to unsteady vortex behaviour at low ride heights is further illustrated in the vorticity distributions shown in Figure 3.7. At $1h_0$, both models predict well-defined, circular vortex cores along the inside of the diffuser side plates and separators, indicative of stable vortex structures (6). While the DDES model provides greater flow details, the RANS model adequately captures the dominant vortex structures and their influence on diffuser performance at medium to high ride heights.

However, at $0.33h_0$, the DDES results reveal a significant transformation in vortex dynamics, with the concentrated vortex cores expanding into spiral-shaped structures characteristic of vortex instability (35). In contrast, the RANS model predicts substantially weaker vortices that dissipate rapidly as they propagate downstream. The model's inability to capture these unstable vortex structures explains its deteriorating prediction accuracy at low ride heights, where aerodynamic performance becomes dominated by unsteady flow phenomena. The transition to highly unsteady flow and its implications will be explored further below.

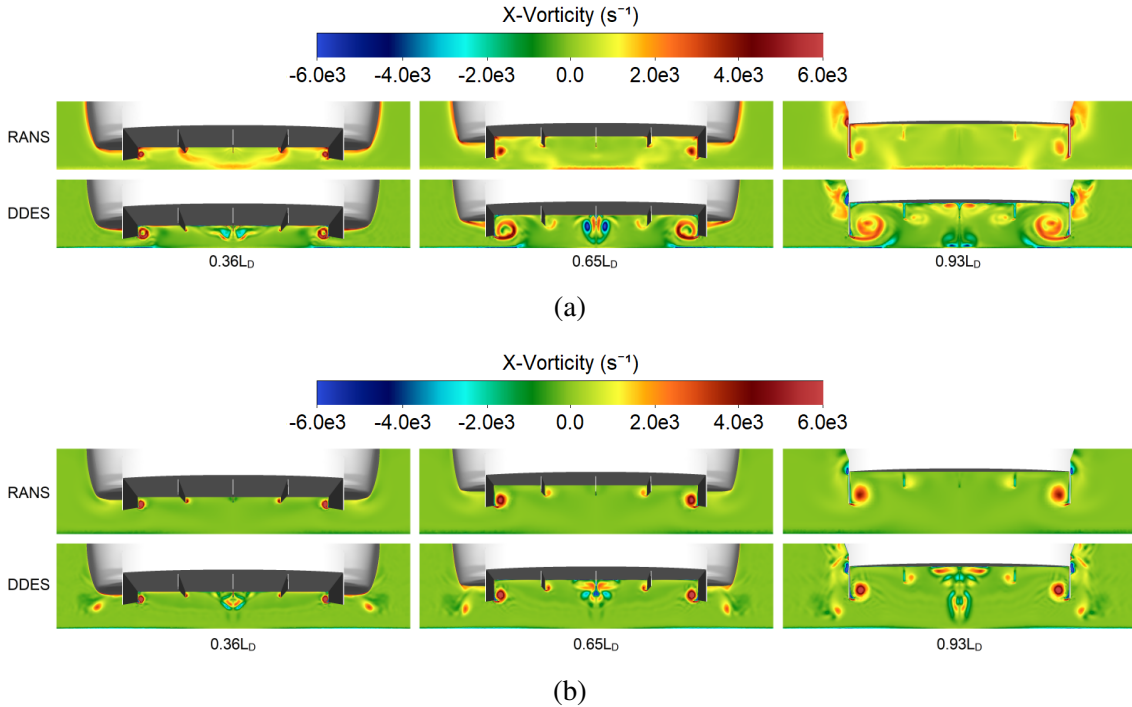


Figure 3.7: Streamwise vorticity contour plots on frontal planes at positions $0.36L_D$, $0.65L_D$, and $0.93L_D$ within the diffuser, from RANS and DDES simulations at ride heights of (a) $0.33h_0$ and (b) $1h_0$.

3.3.2 Ride Height Analysis

3.3.2.1 Underbody Pressure Distribution

The pressure distribution along the vehicle's underbody exhibits significant variation with ride height, as illustrated in Figure 3.8. At high ride heights, a low-pressure region forms beneath the front splitter due to flow separation induced by its sharp leading edge. Figure 3.9 shows that a characteristic low-pressure plateau is created along the separation bubble, followed by a pressure recovery region toward the point of flow reattachment, similar to the behaviour observed on a flat plate (36).

As the ride height decreases, the increased flow acceleration beneath the vehicle reduces the size of the separation bubble, resulting in a shorter low-pressure plateau and an earlier onset of pressure recovery. At very low ride heights, flow reattachment occurs so early that the local pressure beneath the splitter becomes marginally positive due to its slightly inclined orientation relative to the oncoming airflow.

Downstream of the reattachment point, a pronounced suction peak develops along the floor due to the acceleration of airflow beneath the vehicle. The magnitude of this suction peak increases nearly by a factor of four between the highest and lowest ride heights examined. Further along the underbody, the pressure gradually recovers towards the central section of the floor before the airflow accelerates again towards the diffuser inlet, generating a second suction peak.

Unlike the pressure depression observed near the front axle, the suction peak at the diffuser inlet does not continue to increase indefinitely as ride height decreases. Instead, it reaches a maximum around a ride height of $0.5h_0 - 0.33h_0$. The subsequent section will demonstrate that flow separation within the diffuser becomes more pronounced at lower ride heights, reducing the diffuser's capacity to extract airflow and thereby limiting the further increase in suction.

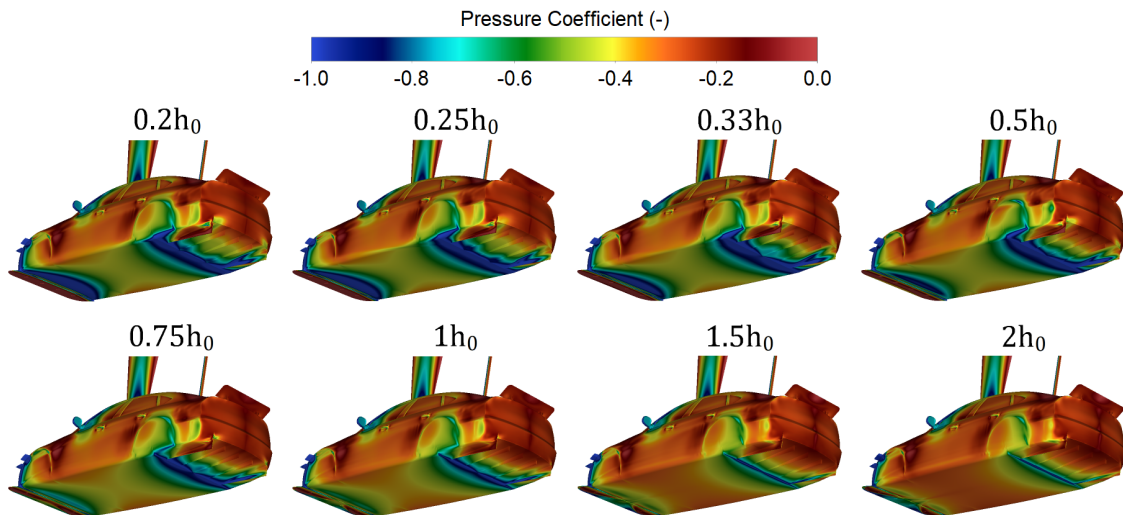


Figure 3.8: Pressure coefficient contour plots on the underbody surface at various ride heights.

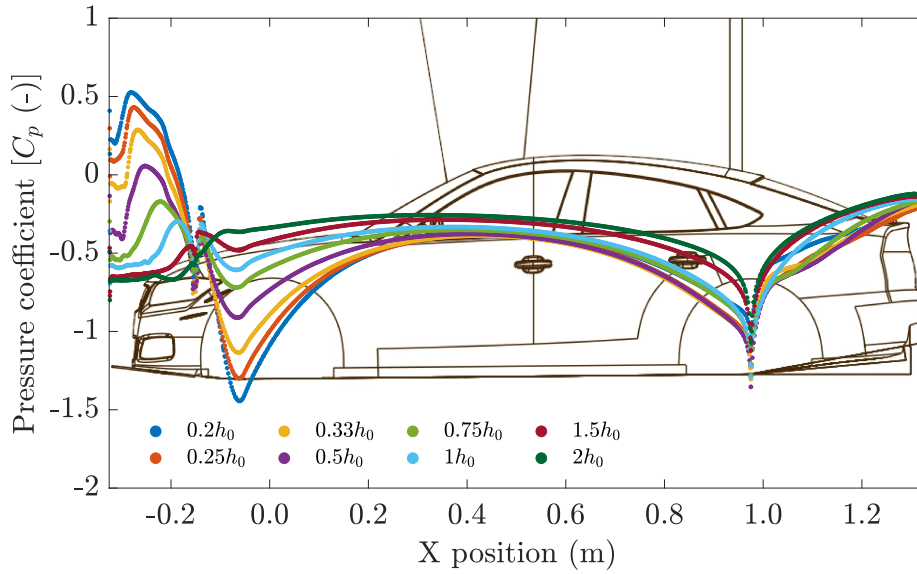


Figure 3.9: Pressure coefficient distributions along a near-symmetry line ($y = -50$ mm) on the underbody surface at various ride heights.

The suction peak at the diffuser inlet is followed by a gradual pressure recovery towards the diffuser exit. At high ride heights, this recovery occurs relatively uniformly, whereas at lower ride heights, minor pressure fluctuations emerge within the inner diffuser channels, indicative of localised flow separation. Additionally, the low-pressure regions in the outer diffuser channels grow in both size and magnitude at reduced ride heights, driven by intensified vortex flow. The influence of these vortex structures on diffuser performance will be examined in greater detail in the following discussion.

3.3.2.2 Multichannel Diffuser Flow

This section utilises streamwise vorticity, shear stress, and velocity magnitude results, presented in Figures 3.10, 3.11, and 3.12, to investigate the evolution of vortices within the multichannel diffuser and their influence on downforce generation as a function of ride height. Figure 3.10 depicts the development of these vortices, with downstream progression shown from left to right at three longitudinal positions, and variation with ride height illustrated from top to bottom.

Vortices originating from the front splitter can be identified on either side of the vehicle at high ride heights and progressively migrate inwards towards the increasing low-pressure region beneath the vehicle as ride height decreases. Additionally, localised vorticity fluctuations are observed around the symmetry separator, particularly at low ride heights, due to flow separation. However, these fluctuations do not resemble coherent vortex structures that promote flow attachment. This analysis primarily focuses on the dominant vortices generated at the side plates and secondary vortices formed at the separators.

The vortices at the side plates develop due to the pressure difference between the free-stream airflow along the vehicle's sides and the low-pressure region beneath the underbody. As the freestream flow is drawn inwards, it separates from the side plates and rolls up into vortices with high concentrations of vorticity, as shown in Figure 3.10.

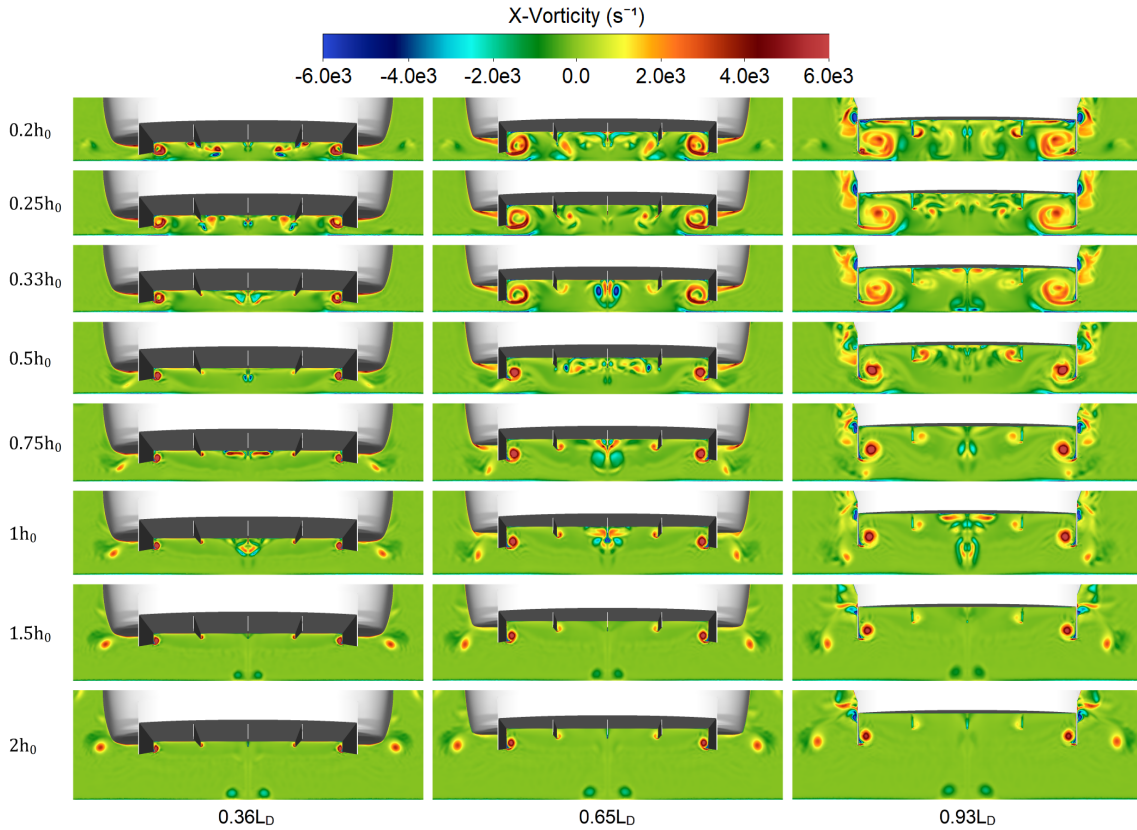


Figure 3.10: Streamwise vorticity contour plots on frontal planes at positions $0.36L_D$, $0.65L_D$, and $0.93L_D$ within the diffuser at various ride heights.

Additionally, as the airflow accelerates between the side plates and the ground plane, significant crossflow is induced, as depicted in Figure 3.12. The side plate vortices entrain this relatively high-energy airflow, along with a portion of the high-energy flow from underneath the vehicle, directing it upwards towards the diffuser ramp.

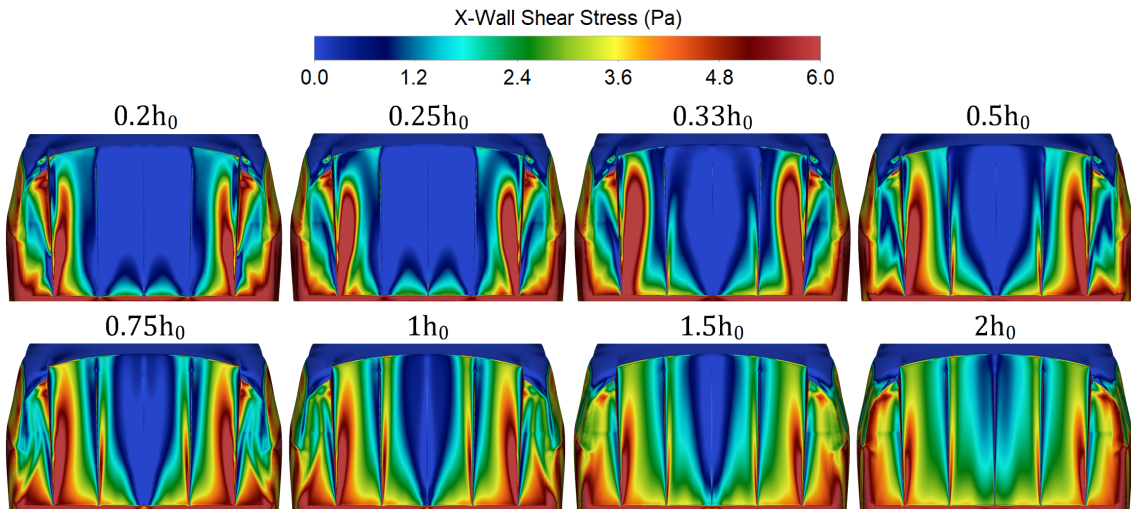


Figure 3.11: Streamwise shear stress contour plots on the diffuser at various ride heights.

This mechanism enhances flow attachment in the outer diffuser channels, evidenced by regions of high shear stress in Figure 3.11, and contributes to the formation of the low-pressure regions previously identified in Figure 3.8.

Within the ride height range of $2h_0 - 0.5h_0$, the side plate vortices remain relatively stable, forming circular cores of high vorticity. As these vortices progress through the adverse pressure gradient from $0.36L_D$ to $0.93L_D$, they increase in size but gradually weaken. Furthermore, as ride height decreases within this range, the vortices grow in strength, size, and streamwise velocity, thereby enhancing flow entertainment in the outer channels.

At lower ride heights of $0.33h_0 - 0.2h_0$, the vortex structures undergo a fundamental transition. Instead of stable circular cores, they form spiral-like distributions with peak vorticity concentrated in the lower regions. The vortices exhibit a greater radial expansion and their core velocities decrease or become negative, indicative of vortex breakdown (35). This transition is driven by stronger ground interaction, where the boundary layer formed by crossflow beneath the vehicle encounters a steep adverse pressure gradient due to high airflow acceleration beneath the side plate vortices.

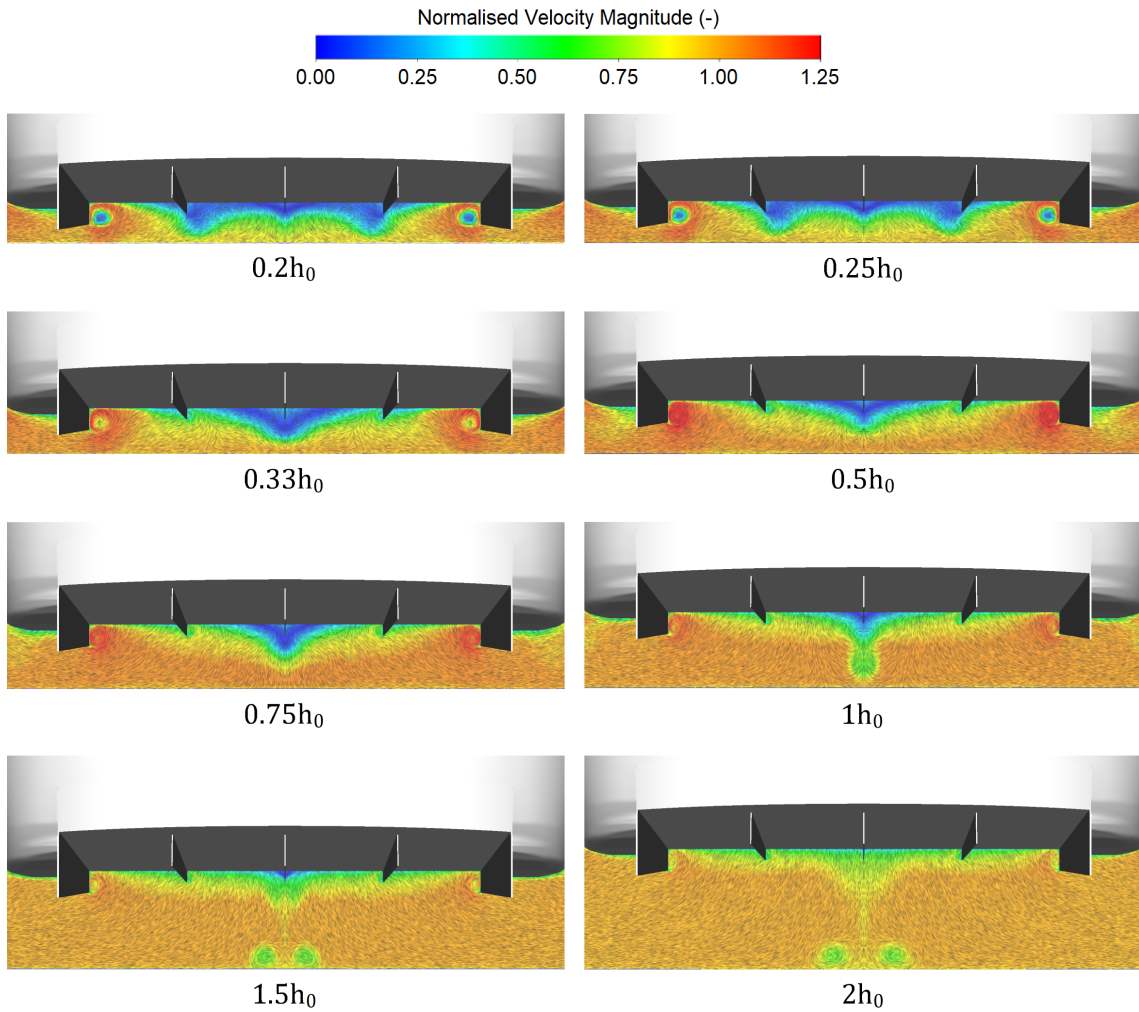


Figure 3.12: Normalised velocity magnitude (U/U_∞) contour plots on a frontal plane at position $0.36L_D$ within the diffuser at various ride heights.

In the $0.33h_0 - 0.2h_0$ ride height range, this adverse pressure gradient becomes strong enough to induce boundary layer separation, generating a layer of opposite vorticity that interacts with the side plate vortices. Studies on vortices in close ground proximity have demonstrated that this secondary layer of opposite vorticity induces an upward velocity on the primary vortex, commonly referred to as the rebound effect (37, 38, 39). However, within the confined geometry of the diffuser, this interaction inhibits the vortex from rolling into a concentrated core, instead generating an unstable spiral-like structure. At ride heights below $0.33h_0$, the intensified ground interaction leads to horizontal stretching of the side plate vortices, causing the freestream airflow to be induced predominantly horizontally rather than being directed upwards towards the diffuser ramp. Consequently, flow entertainment in the outer channels diminishes at these low ride heights.

The high-energy airflow induced by the side plate vortices not only enhances flow attachment in the outer channels but also facilitates the formation of vortices at the separators. These separator vortices are significantly smaller in scale, exhibiting less than half the peak vorticity and streamwise velocity of the side plate vortices. Despite their lower intensity, the separator vortices contribute to flow entertainment within the inner channels, mitigating large-scale separation that would otherwise dominate plane diffusers (9). However, the effectiveness of the separator vortices declines noticeably at ride heights below $0.75h_0$, whereas flow entertainment from side plate vortices only reduces below $0.33h_0$. This discrepancy arises from the above-mentioned altered induced airflow direction from the side plate vortices, which less effectively sustains vortex formation at the separators.

As ride height decreases further, the reduction in flow entertainment within the inner channels, combined with the increasingly severe adverse pressure gradient along the diffuser ramp, results in extensive flow separation. Consequently, the diffuser's ability to extract airflow from beneath the vehicle diminishes, a trend previously identified in Section 3.3.2.1, where the suction peak at the diffuser inlet was shown to reduce significantly at very low ride heights.

3.4 Conclusion

This study investigated the aerodynamic performance of the *DrivAer hp-F* model as a function of ride height, providing critical insights into the flow mechanisms governing downforce generation. In addition, it evaluated the predictive capabilities of both RANS and DDES turbulence models, with particular emphasis on their accuracy at critically low ride heights.

The turbulence model comparison showed that both models effectively captured overall trends in downforce and drag at medium to high ride heights. In this range, the RANS model predicted aerodynamic forces within approximately 2% of the DDES simulations but required 75 times less computation time while using only half the computational resources. It also closely replicated underbody pressure distributions and dominant vortex structures. However, at low ride heights, where unsteady flow phenomena governed aerodynamic performance, only the DDES model accurately reproduced the force trends. The RANS model failed to predict gradual flow separation and unstable vortex behaviour in the diffuser, leading to notable deviations from both the experimental and DDES results.

The ride height analysis demonstrated that downforce and drag increased by roughly 80% and 10%, respectively, as ground clearance decreased from $2h_0$ to $0.33h_0$. Under these conditions, the underbody pressure distribution exhibited a pronounced suction peak near the front axle, followed by a gradual recovery along the vehicle floor and a subsequent low-pressure peak at the diffuser inlet. The underbody pressure generally continued to decrease with lower ground clearance, due to the enhanced flow acceleration underneath the vehicle. Conversely, at critically low ride heights of $0.33h_0$ to $0.2h_0$, both downforce and drag declined as unstable vortex behaviour and extensive flow separation in the diffuser compromised performance.

Vortex dynamics within the diffuser played a pivotal role in aerodynamic performance. Dominant vortices formed at the diffuser's side plates, driven by the pressure differential between the freestream airflow and the low-pressure underbody flow. These vortices induced high-energy airflow onto the diffuser ramp, thereby promoting flow attachment and facilitating the formation of secondary vortices at the channel separators.

In the high to medium ride height range, the side plate vortices increased in strength and size with reducing ride height, thereby enhancing flow entertainment and diffuser downforce. However, at very low ride heights, intensified interaction between the side plate vortices and crossflow along the ground formed a layer of opposite vorticity. This interaction disrupted the formation of coherent vortex cores, causing the vortices to stretch and become unstable, which in turn diminished their ability to induce flow towards the diffuser ramp and reduced aerodynamic performance.

Overall, this study demonstrated the applicability of both RANS and DDES turbulence models across a wide range of ride heights and provided key insights into the aerodynamic effects of ride height on high-performance vehicles with multichannel diffusers. Findings on the turbulence models' prediction accuracy offer valuable guidance for model selection and assessment, supporting methodological advancements in both research and industry. Additionally, the insights into the influence of ride height on aerodynamic forces, surface pressure distributions, and vortex dynamics form a crucial foundation for developing more aerodynamically efficient underbody designs.

Data Availability Statement

All data supporting this study are included within the chapter.

References

1. Toet W. Aerodynamics and aerodynamic research in Formula 1. *The Aeronautical Journal*. 2013;117(1187):1-26. Available from: <https://doi.org/10.1017/S0001924000007739>.
2. Cooper KR, Bertenyi T, Dutil G, Syms J. The aerodynamic performance of automotive underbody diffusers. SAE Technical Paper 980030. 1998. Available from: <https://doi.org/10.4271/980030>.

3. George A, Donis JE. Flow patterns, pressures, and forces on the underside of idealized ground effect vehicles. *Proceedings of the ASME Fluids Engineering Division Symposium on Aerodynamics of Transportation II*. 1998;7:60-79. Available from: <https://www.researchgate.net/profile/Albert-George/publication/292650936>.
4. Zhang X, Senior A, Ruhrmann A. Vortices behind a bluff body with an upswept aft section in ground effect. *International Journal of Heat and Fluid Flow*. 2004;25(1):1-9. Available from: <https://doi.org/10.1016/j.ijheatfluidflow.2003.11.002>.
5. Senior AE, Zhang X. The force and pressure of a diffuser-equipped bluff body in ground effect. *Journal of Fluids Engineering*. 2001;123(1):105-11. Available from: <https://doi.org/10.1115/1.1340637>.
6. Senior AE. The aerodynamics of a diffuser equipped bluff body in ground effect. PhD Thesis. University of Southampton; 2002. Available from: <http://eprints.soton.ac.uk/id/eprint/47113>.
7. Ehirim OH. Aerodynamics and performance enhancement of a ground-effect diffuser. PhD Thesis. Cranfield University; 2017. Available from: <http://dspace.lib.cranfield.ac.uk/handle/1826/13211>.
8. Jowsey L, Passmore M. Experimental study of multiple-channel automotive underbody diffusers. *Proceedings of the Institution of Mechanical Engineers, Part D: The Journal of Automobile Engineering*. 2010;224(7):865-79. Available from: <https://doi.org/10.1243/09544070JAUTO1339>.
9. Jowsey L. An experimental study of automotive underbody diffusers. PhD Thesis. Loughborough University; 2013. Available from: <https://hdl.handle.net/2134/13646>.
10. Porcar L, Toet W, Gamez-Montero J. Study of the effect of vertical airfoil endplates on diffusers in vehicle aerodynamics. *Designs*. 2021;5(3):45. Available from: <https://doi.org/10.3390/designs5030045>.
11. Buscarlio FF, Assi GRS, Sherwin SJ. Computational study on an Ahmed body equipped with simplified underbody diffuser. *Journal of Wind Engineering & Industrial Aerodynamics*. 2019;209:104411. Available from: <https://doi.org/10.1016/j.jweia.2020.104411>.
12. Puglisevich LS. Large eddy simulation for automotive vortical flows in ground effect. PhD Thesis. Loughborough University; 2013. Available from: <https://hdl.handle.net/2134/12555>.
13. Mayoral S, Weiss H, Edirisinghe R. On the relationship between the vortices from an underbody diffuser in ground-effect and the resulting downforce. *SAE Technical Paper 2019-01-0650*. 2019. Available from: <https://doi.org/10.4271/2019-01-0650>.
14. Unni TPA. Numerical investigation on aerodynamic effects of vanes and flaps on automotive underbody diffusers. *SAE Technical Paper 2017-01-2163*. 2017. Available from: <https://doi.org/10.4271/2017-01-2163>.

15. Guerrero A, Castilla R, Eid G. A numerical aerodynamic analysis on the effect of rear underbody diffusers on road cars. *Applied Sciences*. 2022;12(8):3763. Available from: <https://doi.org/10.3390/app12083763>.
16. Heft A, Indinger T, Adams N. Introduction of a new realistic generic car model for aerodynamic investigations. SAE Technical Paper 2012-01-0168. 2012. Available from: <https://doi.org/10.4271/2012-01-0168>.
17. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model. SAE Technical Paper 2018-01-0725. 2018. Available from: <https://doi.org/10.4271/2018-01-0725>.
18. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. Dataset for SAE 2018-01-0725 “On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model”. Cranfield Online Research Data (CORD); 2018. Available from: <https://doi.org/10.17862/cranfield.rd.c.3969120>. (Accessed: 01 May 2022).
19. Rijns S, Teschner TR, Blackburn K, Brighton J. Integrated numerical and experimental workflow for high performance vehicle aerodynamics. SAE Technical Paper 2024-01-5016. 2024. Available from: <https://doi.org/10.4271/2024-01-5016>.
20. Soares RF, de Souza FJ. Influence of CFD setup and brief analysis of flow over a 3D realistic car model. SAE Technical Paper 2015-36-0513. 2015. Available from: <https://doi.org/10.4271/2015-36-0513>.
21. Ahmad NE, Abo-Serie E, Gaylard A. Mesh optimization for ground vehicle aerodynamics. *CFD Letters*. 2010;2(1):54-65. Available from: https://pure.coventry.ac.uk/ws/portalfiles/portal/12035851/63_210_1_PB.pdf.
22. Menter FR. Best practice: scale resolving simulations in Ansys CFD. ANSYS, Inc; 2021. Available from: <https://www.ansys.com/en-gb/resource-center/technical-paper/best-practice-scale-resolving-simulations-in-ansys-cfd>. (Accessed: 01 May 2022).
23. Lanfrit M. Best practice guidelines for handling automotive external aerodynamics with Fluent. University of Southampton; 2005. Available from: https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf. (Accessed: 01 May 2022).
24. Zore K, Caridi D, Lockley I. Fast and accurate prediction of vehicle aerodynamics using ANSYS mosaic mesh. SAE Technical Paper 2020-01-5011. 2020. Available from: <https://doi.org/10.4271/2020-01-5011>.
25. Skaperdas V. Meshing for the 1st automotive CFD prediction workshop. BETA SIMULATION SOLUTIONS; 2019. Available from: <https://autocfd1.s3.eu-west-1.amazonaws.com/Presentations/BETA-CAE-Case2aMeshing.pdf>. (Accessed: 01 May 2022).

26. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*. 1994;32(8):1598-605. Available from: <https://doi.org/10.2514/3.12149>.
27. Menter FR, Kuntz M, Langtry R. Ten years of industrial experience with the SST turbulence model. *Heat and Mass Transfer*. 2003;4:625-32. Available from: <https://api.semanticscholar.org/CorpusID:38105247>.
28. Ansys Fluent theory guide - release 2022 R1. ANSYS, Inc; 2022.
29. Aultman M, Wang Z, Duan L. Effect of time-step size on flow around generic car models. *Journal of Wind Engineering and Industrial Aerodynamics*. 2021;219:104764. Available from: <https://doi.org/10.1016/j.jweia.2021.104764>.
30. Breslouer O, George A. Exploratory experimental studies of forces and flow structure on a bluff body with variable diffuser and wheel configurations. *SAE Technical Paper 2008-01-0326*. 2008. Available from: <https://doi.org/10.4271/2008-01-0326>.
31. Desai S, Lo C, George A. A computational study of idealized bluff bodies, wheels, and vortex structures in ground effect. *SAE Technical Paper 2008-01-0327*. 2008. Available from: <https://doi.org/10.4271/2008-01-0327>.
32. Huminic A, Huminic G. Aerodynamic study of a generic car model with wheels and underbody diffuser. *International Journal of Automotive Technology*. 2017;18:397-404. Available from: <https://doi.org/10.1007/s12239-017-0040-6>.
33. Hyvärinen A. Investigation of blockage correction methods for full-scale wind tunnel testing of trucks. MSc Thesis. KTH, Royal Institute of Technology; 2015. Available from: <https://kth.diva-portal.org/smash/get/diva2:893809/FULLTEXT01.pdf>.
34. Lucca-Negro O, O'Doherty T. Vortex breakdown: a review. *Progress in Energy and Combustion Science*. 2001;27(4):431-81. Available from: [https://doi.org/10.1016/S0360-1285\(00\)00022-8](https://doi.org/10.1016/S0360-1285(00)00022-8).
35. Lambourne NC, Bryer DW. The bursting of leading-edge vortices - some observations and discussion of the phenomenon. *Aeronautical Research Council Reports and Memoranda*. 1961;3282. Available from: <https://reports.aerade.cranfield.ac.uk/handle/1826.2/3859>.
36. Crompton MJ, Barret RV. Investigation of the separation bubble formed behind the sharp leading edge of a flat plate at incidence. *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*. 2000;214(3):157-76. Available from: <https://doi.org/10.1243/0954410001531980>.
37. Harvey JK, Perry FJ. Flowfield produced by trailing vortices in the vicinity of the ground. *AIAA Journal*. 1971;9(8):1659-60. Available from: <https://doi.org/10.2514/3.6415>.
38. Wakim A, Brion V, Dolfi-Bouteyre A, Jacquin L. A vortex pair in ground effect, dynamics and optimal control. *Journal of Fluid Mechanics*. 2019;885:A26. Available from: <https://doi.org/10.1017/jfm.2019.1002>.

39. Harris DM, Williamson CHK. Instability of secondary vortices generated by a vortex pair in ground effect. *Journal of Fluid Mechanics*. 2012;700:148-86. Available from: <https://doi.org/10.1017/jfm.2012.108>.

Chapter 4

Aerodynamic Analysis of Yaw Conditions

This chapter was originally published as:

Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Experimental and numerical investigation of the aerodynamic characteristics of high-performance vehicle configurations under yaw conditions. *Physics of Fluids*. 2024;36(4):045112. Available from: <https://doi.org/10.1063/5.0196979>.

Abstract

This study examines the impact of yaw on the aerodynamic performance and flow field of three high-performance vehicle configurations using wind tunnel experiments and Unsteady Reynolds-Averaged Navier-Stokes (URANS) based Computational Fluid Dynamics (CFD) simulations. While yaw effects on automotive vehicles have been explored, their influence on the more complex flow fields of high-performance vehicles remains insufficiently researched. This study shows that yaw significantly degrades both downforce and drag performance, with lift on the standard vehicle nearly quadrupling and drag rising by 27% over a 20° yaw range. Spoiler and rear wing devices increase both downforce and drag, while also influencing the vehicle's yaw sensitivity compared to the baseline configuration. Depending on the device, variations in lift, drag, and side force coefficients across the yaw angle range are up to 29% higher, 31% lower, and 16% higher, respectively. Additionally, yaw conditions significantly alter vortex structures and local flow velocities, affecting downstream flow behaviour. Surface pressure measurements on the slant confirm these findings, highlighting notable yaw-induced effects and upstream pressurisation from the spoiler and rear wing devices. Wake analyses through total pressure measurements show that yaw induces a substantial deviation from straight-line wake characteristics, which become dominated by an inboard-rotating vehicle body vortex. Overall, this research advances the understanding of yaw effects on high-performance vehicle aerodynamics, providing comprehensive insights and experimental data for future analyses in real-world operating conditions.

Keywords: Aerodynamics; Yaw; Wind Tunnel; CFD; URANS; DrivAer

4.1 Introduction

Aerodynamic research plays a crucial role in optimising vehicle performance in both automotive and motorsport industries. Vehicles operate in a wide range of dynamic conditions, such as yaw and cornering, which can significantly influence their aerodynamic performance. This is particularly critical for high-performance vehicles, which rely on substantial downforce generation from their aerodynamic elements to enhance tractive forces. However, the majority of published research focuses on straight-line conditions, offering only limited insights into real-world dynamic driving scenarios.

Research on dynamic conditions has primarily focused on examining yaw effects on simplified automotive bodies, with findings generally indicating that drag increases exponentially with yaw angle (1, 2, 3, 4, 5, 6). While drag has been the primary focus, some studies reported a stronger exponential increase in lift with yaw angle (3, 6, 7, 8, 9). Additionally, variations in experimental setups and vehicle model mounting methods influenced yaw sensitivity, highlighting the significant role of underbody flow in aerodynamic performance under these conditions.

The impact of yaw conditions on aerodynamic performance has been attributed to variations in the flow field (2, 3, 7, 8, 10, 11, 12). These studies showed that yaw introduced a windward pressure build-up, resulting in a leeward-acting side force that increased almost linearly with yaw angle. Flow structures also altered, with enhanced vortex formation along the windward roofline, windward C-pillar, and leeward roofline. Additionally, yaw conditions increased separation along the vehicle's leeward side and generated a distinct vortex along the leeward edge of the underbody. Flow attachment typically enhanced along the slant, while strong low-pressure regions formed on the leeward side of the base due to wake recirculation, driven by interactions with the windward C-pillar vortex.

Research on more realistic passenger vehicles has revealed different trends compared to those observed for simplified automotive bodies (13, 14, 15, 16, 17). Findings showed that drag increased approximately linearly up to 10° yaw before the rate of increase began to plateau. Lift initially decreased at yaw angles below 4° but rose exponentially at larger angles. Side force increased fairly linearly with yaw, consistent with trends observed for simplified bodies.

Flow field investigations on passenger vehicles indicated a suppression of the windward A-pillar vortex and leeward C-pillar vortex under yaw conditions, while the leeward A-pillar vortex strengthened and shifted outward (14, 16, 17, 18, 19, 20, 21). The behaviour of the windward C-pillar vortex varied with rear-end geometry, though all configurations exhibited a dominant inboard-rotating wake vortex on the windward side. These discrepancies suggest that yaw effects observed for simplified bodies may not directly translate to more complex vehicle geometries.

Existing research has provided valuable insights into yaw-induced aerodynamic effects on simplified automotive bodies and passenger vehicles. However, high-performance vehicles feature aerodynamic elements such as splitters, spoilers, wings, and diffusers to generate downforce. These devices have been shown to significantly increase flow field complexity, affecting both on-body flow and wake characteristics (22, 23, 24, 25, 26, 27, 28).

Moreover, given the high sensitivity of isolated aerodynamic components to varying flow conditions (29, 30), high-performance vehicles are expected to exhibit greater yaw sensitivity than passenger vehicles.

The increased aerodynamic complexity of high-performance vehicles is evident in their wake behaviour. Research has shown that passenger vehicles typically exhibit a wake dominated by a counter-rotating vortex pair behind the base (31, 32, 33). In these studies, fastback and notchback configurations generated inboard-rotating vortex pairs, inducing minor inboard downwash and outboard outwash, while estates and SUVs produced outboard-rotating vortex pairs with weak inboard upwash and outboard inwash effects. Wake characteristics have also been found to depend on upper body design complexity and underbody flow rate (34, 35). In contrast, high-performance and racing vehicles have been demonstrated to exhibit wakes dominated by aerodynamic devices, particularly rear wings and spoilers, alongside strong upward flow from the diffuser, resulting in a highly turbulent wake with pronounced inboard upwash and outboard inwash (36, 37, 38, 39, 31).

In summary, existing research has demonstrated the significant impact of yaw conditions on the aerodynamics of simplified automotive bodies and passenger vehicles, with a primary focus on drag. However, studies have also highlighted that high-performance vehicles, equipped with aerodynamic elements, exhibit a far more complex flow field and potentially a heightened yaw sensitivity. As a result, the aerodynamic trends observed for passenger vehicles and simplified geometries do not directly translate to high-performance vehicles. The effects of yaw on high-performance vehicle aerodynamics remain largely unexplored, particularly in terms of detailed flow field characteristics and interactions with aerodynamic devices.

This study addresses this gap by investigating yaw-induced aerodynamic effects on a high-performance vehicle in standard, spoiler-equipped, and rear wing-equipped configurations. Wind tunnel measurements of aerodynamic forces, surface pressures, and wake total pressures are conducted in the 8x6 Wind Tunnel at Cranfield University, providing valuable experimental insights into alterations in their aerodynamic performance and flow field characteristics. Additionally, complementary Computational Fluid Dynamics (CFD) simulations are performed to support the interpretation of experimental findings and enable a more comprehensive discussion. The combined experimental and numerical findings enhance the understanding of yaw-induced effects on high-performance vehicle aerodynamics, laying the foundation for analyses in real-world driving scenarios.

4.2 Methodology

4.2.1 Vehicle Geometry

The DrivAer Fastback model (40) is a widely adopted generic vehicle model for aerodynamic research on automotive applications. In this study, a high-performance variant of this model called the *DrivAer hp-F* (41) is utilised, which serves as a reference model for aerodynamic research on high-performance vehicles. It is a 35% scale DrivAer Fastback model featuring a smooth underbody, side mirrors, and stationary wheels.

Table 4.1: Main dimensions of the *DrivAer hp-F* model.

Parameter	Symbol	Value	Unit
Car scale	CS	35	%
Length	L	1.6144	m
Width	W	0.6273	m
Height	H	0.4890	m
Ride height	h_0	0.0455	m
Area	A_{ref}	0.2647	m ²

The standard setup incorporates aerodynamic devices, including a 41 mm front splitter, forebody strakes, and a 10° underbody multichannel diffuser, as shown in Figure 4.1a. The spoiler-equipped configuration features an additional $350 \text{ mm} \times 80 \text{ mm}$ rear spoiler, mounted at a 40° angle of attack, as depicted in Figure 4.1b. The rear wing-equipped configuration is fitted with a NACA 6412 profile wing with a span of 420 mm and a chord length of 110 mm, positioned at a 15° angle of attack, as illustrated in Figure 4.1c. The key dimensions of the *DrivAer hp-F* are listed in Table 4.1, where the reference area corresponds to the rear wing configuration.

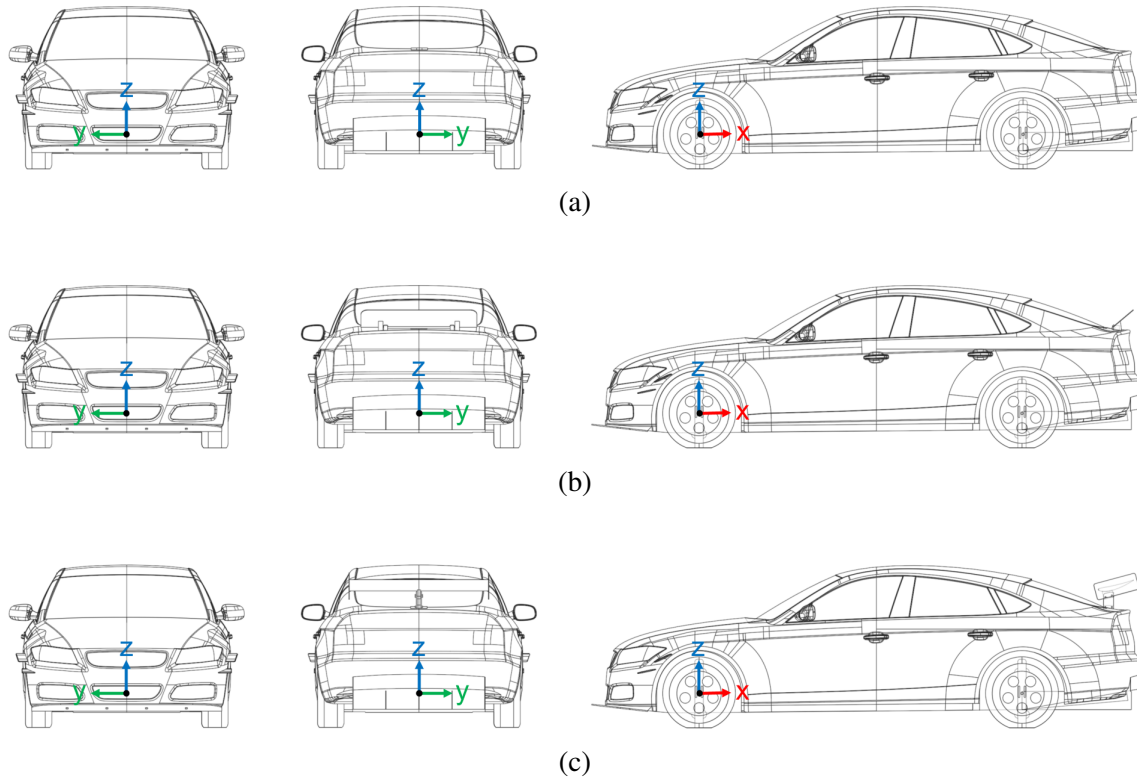


Figure 4.1: Illustration of the *DrivAer hp-F* model configurations: (a) standard, (b) spoiler and (c) rear wing.

4.2.2 Wind Tunnel Experiments

4.2.2.1 Cranfield University 8x6 Wind Tunnel

The wind tunnel campaign is conducted in the closed return 8x6 Wind Tunnel at Cranfield University, depicted in Figure 4.2. The standard test section is transformed using a 42 mm elevated floor with a turntable and two-stage boundary layer suction system, providing a 2.44 m by 1.79 m transformed test section. The first boundary layer suction system (BL1) consists of a scoop intake and is positioned 1.6 m upstream of the turntable. The second boundary layer suction system (BL2) operates by suction through a porous plate, which is located 0.85 m upstream of the turntable. The experiments were subject to a blockage ratio of approximately 6.3% with the vehicle at 0° yaw.

4.2.2.2 Aerodynamic Forces

The aerodynamic forces acting on the three high-performance vehicle configurations are measured over a yaw angle range of -20° to 20° in 5° increments at a wind speed of 40 m/s. Force measurements are acquired using four independent six-degree-of-freedom JR3 load cells, each positioned underneath a wheel of the vehicle model. The sensors have a nominal accuracy of $\pm 0.25\%$ of their full-scale measurement range, corresponding to an absolute measurement accuracy for each sensor of ± 1.11 N for drag and side force, and ± 2.22 N for lift force.

Data acquisition is conducted at a sampling rate of 1 kHz over a 20 s interval for each yaw angle. A constant fan rotor speed is maintained throughout the yaw sweep, resulting in a wind speed variation of ± 0.1 m/s within each acquisition interval. Due to increased blockage at higher yaw angles, the wind speed decreases by no more than 0.6 m/s between the straight-line condition and the maximum yaw angles of $\pm 20^\circ$.



Figure 4.2: Experimental setup in the 8x6 Wind Tunnel at Cranfield University.

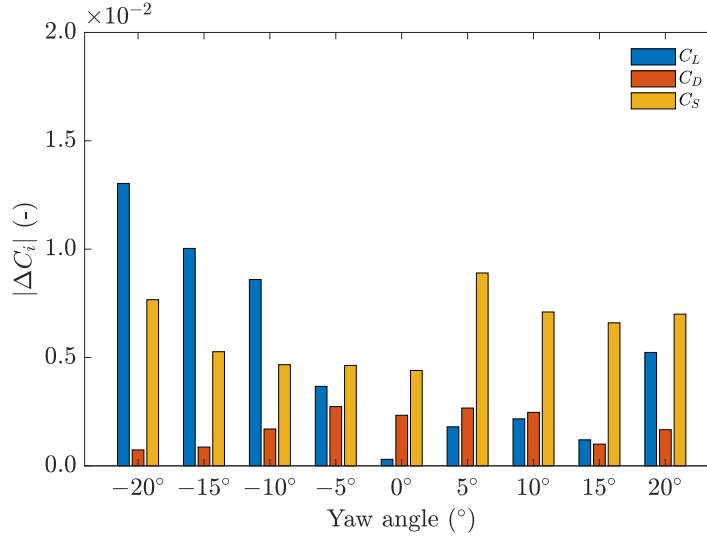


Figure 4.3: Repeatability assessment of the aerodynamic force measurements for the *DrivAer hp-F* with rear wing configuration across the yaw angle range, based on the maximum absolute deviation from the mean of repeated measurements.

The time-averaged force measurements are processed to compute the lift, drag, and side force coefficients using Eq. 4.1, where F_i (N) denotes the measured force and C_i (-) represents the corresponding non-dimensional coefficient, with i indicating lift (L), drag (D), or side (S) forces. The dynamic pressure measured near the test section's inlet is denoted by P_{dyn} (Pa), and the vehicle's reference area by A_{ref} (m²), which is listed in Table 4.1.

$$C_i = \frac{F_i}{P_{\text{dyn}} \cdot A_{\text{ref}}} \quad (4.1)$$

The reliability of the aerodynamic force data is evaluated through the repeatability of three complete yaw angle sweeps using the vehicle with rear wing configuration. The repeatability is quantified for the lift, drag, and side force coefficients by determining the maximum absolute deviation between individual measurement points and the corresponding mean force coefficients across the repeated runs for each yaw angle, as illustrated in Figure 4.3. The average of these maximum absolute deviations across the yaw angle range is $|\Delta C_L| \approx 5.1 \times 10^{-3}$, $|\Delta C_D| \approx 1.8 \times 10^{-3}$, and $|\Delta C_S| \approx 6.2 \times 10^{-3}$ for the lift, drag, and side force coefficients, respectively.

4.2.2.3 Wake Total Pressure

Total pressure measurements in the wake of the high-performance vehicle with rear wing configuration are conducted across a yaw angle range of -20° to 0° in 5° increments at a freestream velocity of 40 m/s. The wake is measured across a width of 1200 mm (1.91W) at three longitudinal planes, designated as P1, P2, and P3, positioned 400 mm (0.25L), 700 mm (0.43L), and 1000 mm (0.62L) downstream of the vehicle, as illustrated in Figure 4.4.

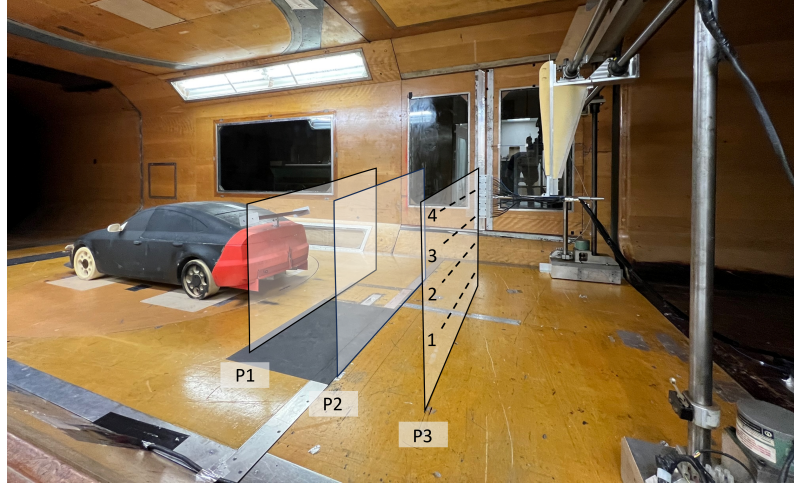


Figure 4.4: Experimental setup for the wake total pressure measurements.

The total pressure measurements are acquired using a pressure rake consisting of 39 vertically stacked single-hole probes, which are positioned by an automated traverse system. The rake is connected to two 20-channel differential pressure transducers that share a common atmospheric reference pressure. The 40th channel is allocated to a pressure tap installed on the wind tunnel wall at mid-height, aligned with the corresponding wake plane position. In this configuration, the rake probes measure total pressure within the wake, while the dedicated wall tap records the local static pressure in the wind tunnel, which is otherwise not directly available due to the differential measurement setup. This allows the wake pressures, originally referenced to atmospheric pressure, to be recalibrated relative to the local static pressure in the tunnel, ensuring consistency with the experimental operating conditions.

The pressure rake probes are arranged with a nominal vertical spacing of 4.77 mm, covering a total measurement height of 181.26 mm. A cost-effective horizontal resolution of 25 mm is employed, as established by Soares (31), resulting in 49 discrete measurement steps per horizontal sweep. To ensure full wake coverage, four horizontal sweeps are vertically stacked, with a minimum of five overlapping probes between successive stacks to maintain measurement continuity. Notably, due to minor surface irregularities in the wind tunnel floor, the ground clearance of the first vertical stack varies between 5 mm and 15 mm across the wake planes.

Data acquisition is conducted at a sampling rate of 1 kHz over a 10 s interval at each measurement position. To maintain consistency across measurements, the fan rotor speed is adjusted after each horizontal sweep, limiting wind speed fluctuations to ± 0.1 m/s within each acquisition interval and ensuring a maximum variance of ± 0.2 m/s across the yaw angle range. The time-averaged total pressure data is processed to compute total pressure coefficients using Eq. 4.2, where C_{pT} (-) represents the non-dimensional total pressure coefficient and P_t (Pa) is the measured total pressure, calibrated relative to the local static pressure in the wind tunnel. Complete wake maps are constructed by interpolating the time-averaged data in MATLAB, incorporating averaged values from overlapping probes of vertical stacks to ensure measurement continuity.

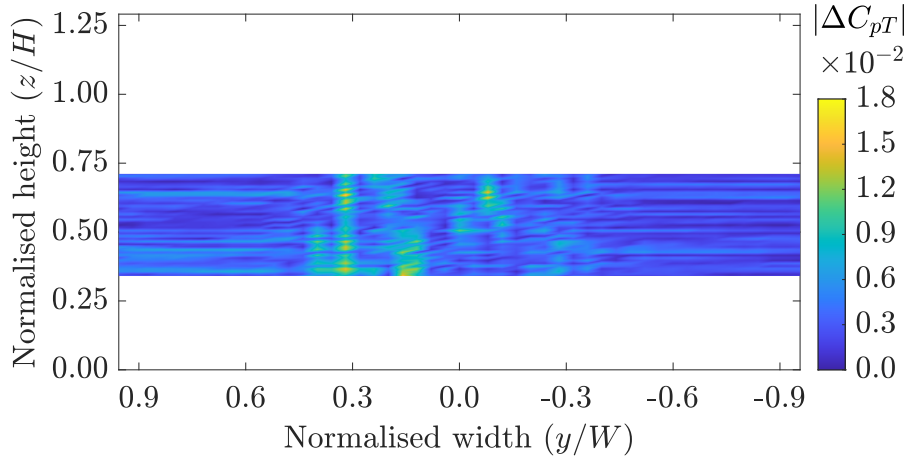


Figure 4.5: Repeatability assessment of the wake total pressure measurements for the *DrivAer hp-F* with rear wing configuration at 0° yaw on the second vertical stack of P3, based on the maximum absolute deviation from the mean of repeated measurements.

$$C_{pT} = \frac{P_t}{P_{\text{dyn}}} \quad (4.2)$$

Given the extensive time required to measure a complete wake plane, repeatability is assessed through three repeated horizontal sweeps at a fixed vertical position. These measurements are conducted on the second vertical stack of P3 at 0° yaw. This location, situated within a highly turbulent wake region, provides a conservative assessment of measurement repeatability. Figure 4.5 shows the maximum absolute deviation between individual measurements and the corresponding mean total pressure coefficient across the repeated tests. All recorded deviations remain within a tolerance band of $|\Delta C_{pT}| \approx 1.7 \times 10^{-2}$, with 90% of the data exhibiting deviations of less than $|\Delta C_{pT}| \approx 7.9 \times 10^{-3}$.

4.2.2.4 Surface Pressure

Surface pressure distributions along the slant of the three high-performance vehicle configurations are measured across a yaw range of -20° to 20° in 5° increments at a constant wind speed of 40 m/s, achieving a similar wind speed consistency as during aerodynamic force measurements. To maximise spatial resolution, 40 pressure taps are installed on one half of the slant, as illustrated in Figure 4.6, utilising all available channels from the two 20-channel differential pressure transducers. By measuring at both positive and negative yaw angles, pressure data from an opposite yaw condition is mirrored to the other side of the slant, ensuring full-surface coverage. Taps along the line of symmetry provide reference points for accurate data transposition across the slant.

The pressure transducers are both referenced to atmospheric pressure. Data acquisition is conducted at a sampling rate of 5 kHz over a 30 s interval for each yaw angle. The time-averaged surface pressure data is calibrated against the local static pressure in the wind tunnel using the averaged recording from the wall tap at the nearest P1 wake plane in the total wake pressure experiments.

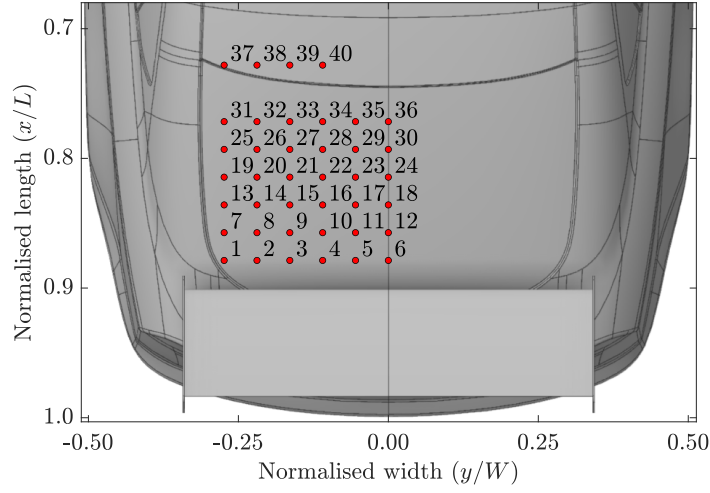


Figure 4.6: Locations of the surface pressure taps.

The calibrated measured pressures P_s (Pa) are successively converted into non-dimensional pressure coefficients C_p (-) using Eq. 4.3. The processed data is interpolated using a spline method in MATLAB to generate pressure distributions across the entire slant, with overlapping measurements along the vehicle's symmetry line averaged to maintain consistency across transposed data.

$$C_p = \frac{P_s}{P_{\text{dyn}}} \quad (4.3)$$

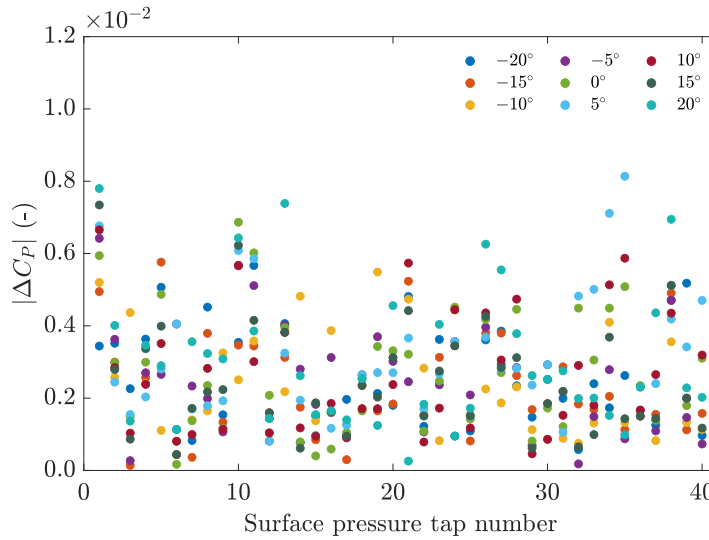


Figure 4.7: Repeatability assessment of the surface pressure measurements for the *DrivAer hp-F* with rear wing configuration across the yaw angle range, based on the maximum absolute deviation from the mean of repeated measurements.

The repeatability of the surface pressure measurements is evaluated through three yaw sweep experiments conducted on the rear wing configuration of the vehicle. Figure 4.7 presents the maximum absolute deviation between individual measurement points and the respective mean pressure coefficient across the three repeated runs for each surface pressure tap at every yaw angle. All recorded deviations fall within a tolerance band of $|\Delta C_p| \approx 8.0 \times 10^{-3}$, with no identified outliers.

4.2.3 Computational Fluid Dynamics Simulations

4.2.3.1 Domain and Boundary Conditions

The computational domain is designed to approximate the physical test conditions in the 8x6 Wind Tunnel at Cranfield University, following correlation practices previously demonstrated in Chapter 2 (22). To match the test section dimensions, the domain is 2438 mm wide, 1787 mm high, and has 431.05 mm long corner fillets, as illustrated in Figure 4.8. The velocity inlet is positioned 3 car lengths (L) in front, and the pressure outlet 5L behind the vehicle to provide sufficient space for the development of inlet flow and downstream wake characteristics (42), resulting in a total domain length of 14904 mm.

The simplified BL2 system extends across the full width of the tunnel and is modelled by prescribing a symmetry wall condition to the upstream floor section, preventing boundary layer growth. The wind tunnel floor downstream of the BL2 system and the remaining wind tunnel walls are modelled as no-slip stationary walls. The simulations are performed on the *DrivAer hp-F* with rear wing configuration across a yaw angle range of 0° to 20° with increments of 5° . Operating settings aimed at approximating the experimental conditions, listed in Table 4.2, are applied for improved correlation.

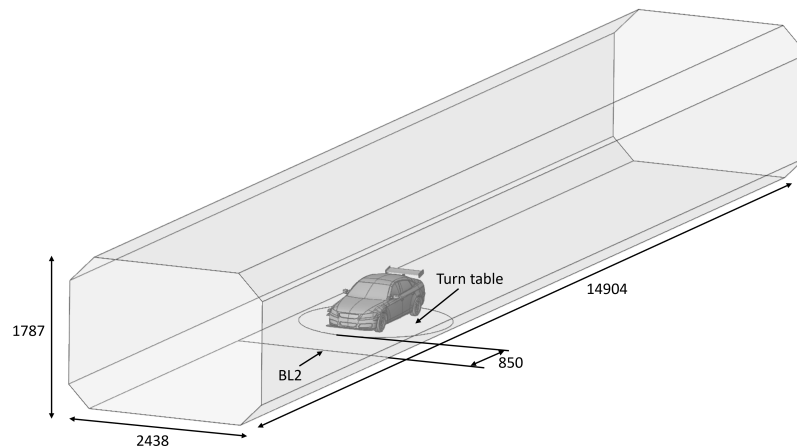


Figure 4.8: Illustration of the computational wind tunnel domain with dimensions in mm.

Table 4.2: Overview of the reference operating conditions for CFD simulations.

Parameter	Symbol	Unit	0° Yaw	-5° Yaw	-10° Yaw	-15° Yaw	-20° Yaw
Temperature	T	°C	30	30	30	30	30
Freestream velocity	U_∞	m/s	40.00	39.93	39.75	39.60	39.42
Dynamic pressure	P_{dyn}	Pa	917.30	913.80	906.01	898.32	889.94
Density	ρ	kg/m ³	1.1472	1.1464	1.1468	1.1457	1.1454
Dynamic viscosity	μ	kg/(m·s)	18.616e-6	18.616e-6	18.616e-6	18.616e-6	18.616e-6
Turbulent intensity	I_T	%	0.052	0.052	0.052	0.052	0.052

4.2.3.2 Mesh

The meshing strategy outlined in Chapter 2.2.3.2 (22) is employed to generate unstructured poly-hexcore meshes using ANSYS Fluent Meshing. To enhance numerical efficiency, a mesh interface approach is implemented, which enables yaw angle adjustments without requiring re-meshing. This is achieved by coupling a cylindrical volume mesh enclosing the vehicle to the wind tunnel domain via a mesh interface, as illustrated in Figure 4.9. Validation against a conventional single-domain meshing approach shows a negligible variation of approximately 0.1% in lift and drag coefficients, confirming that the mesh interface technique does not compromise simulation accuracy.

The meshing strategy applies a base element size and vehicle surface element size of approximately 7.4% and 0.45% of the vehicle's length, respectively. Vehicle surface element sizes are locally reduced by 80% on aerodynamic devices and small geometric features. A practical medium near-wall treatment with a $y^+ \approx 120$ -150 and about 4-6 inflation layers is used on the vehicle and wind tunnel walls to capture their boundary layers. Additionally, four refinement zones with gradually increasing element sizes are used to effectively capture the vehicle's near-field and wake region in more detail, as shown in Figure 4.9. The overall mesh contains approximately 13.6 million cells.

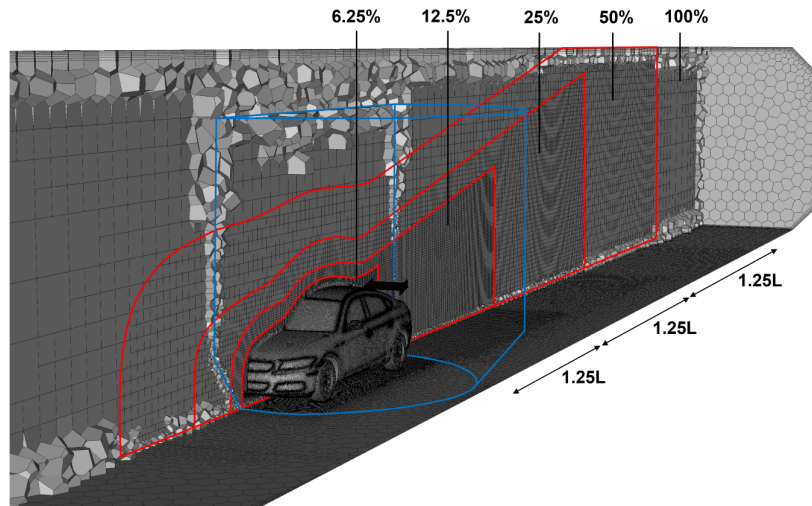


Figure 4.9: Close-up view of the volume mesh on the symmetry plane ($y = 0$), highlighting the mesh interface in blue and refinement zones in red with element sizes expressed as a percentage of the base mesh size.

4.2.3.3 Solve

Unsteady RANS simulations are conducted using the $k-\omega$ SST turbulence model (43) in ANSYS Fluent. This model is selected for its proven accuracy and computational efficiency in prior analyses, as demonstrated in Chapters 2 and 3 (22, 23). The governing equations include continuity, momentum, and transport equations for turbulent kinetic energy and specific dissipation rate. The simulation methodology follows the approach established in Section 2.2.3.3 (22), employing the coupled scheme for pressure-velocity coupling. Gradients are computed using a least-squares cell-based method, while pressure is discretised with a second-order scheme. Spatial discretisation of the governing equations is achieved through a second-order upwind scheme and temporal discretisation is performed using a bounded second-order implicit formulation.

Time step settings are derived from the methodologies established in the previous chapters and the work by Aultman et al. (44), ensuring a good balance between temporal resolution and computational efficiency. The simulations are performed with a fixed time step of $t = 3.125 \times 10^{-4}$ s and a maximum of 20 inner iterations over a total flow time of 1.2 s, corresponding to 29.8 convective time units (CTUs). The first 0.6 s (14.9 CTUs) is allocated for reaching statistical convergence of monitored lift and drag coefficients, after which time-averaged quantities and unsteady statistics are collected over the remaining 0.6 s (14.9 CTUs). Each yaw angle simulation is performed on eight AMD EPYC 7543 processors using 256 cores, resulting in an average computational time of approximately 50 hours.

4.3 Validation

This section evaluates the accuracy of the CFD simulations on the *DrivAer hp-F* with rear wing configuration by comparing them against experimental measurement data. The primary objective of the complementary numerical results is to support the interpretation of the experimental findings and enable a more in-depth analysis of yaw effects on the aerodynamics of high-performance vehicles. While some discrepancies between experimental and numerical results are anticipated due to simplifications in the computational setup, limitations of turbulence modelling, and challenges in predicting wind tunnel conditions in CFD simulations (45), the numerical solutions should ideally capture the dominant flow structures and general influence of yaw on aerodynamic performance.

4.3.1 Aerodynamic Forces

The lift, drag, and side force coefficients for the entire vehicle, along with the rear wing's contribution, are compared between experimental and numerical results in Figure 4.10. The experimental rear wing's contribution is determined as the difference in forces between the rear wing and standard vehicle configurations. Figure 4.10a indicates that the numerical simulations overpredict total downforce in the straight-line condition by approximately $\Delta C_L \approx -0.15$ and capture only around 30% of the increase in lift observed across the yaw angle range.

One possible contributor to this discrepancy may be a reduction in airflow momentum beneath the vehicle in the physical wind tunnel, potentially caused by imperfect suction systems and surface irregularities along the wind tunnel floor and turntable. These subtle floor details, which are infeasible to accurately replicate in the computational wind tunnel, could contribute to an overestimation of underbody downforce in the simulations. This interpretation is supported by the rear wing's downforce contribution, which is less affected by ground boundary layer flow, and shows closer agreement between experimental and numerical results, with comparable yaw sensitivity and an average deviation of only $\Delta C_L \approx 0.02$.

Figure 4.10b demonstrates that the numerical simulations capture the overall drag trend with reasonable accuracy. However, the overprediction in total downforce is accompanied by a corresponding overprediction in drag, with the numerical results exhibiting approximately $\Delta C_D \approx 0.04$ higher drag across the yaw angle range. The rear wing's drag contribution is captured relatively well, with both experimental and numerical results indicating that the drag coefficient remains fairly insensitive to yaw conditions, showing an average deviation of $\Delta C_D \approx 0.02$.

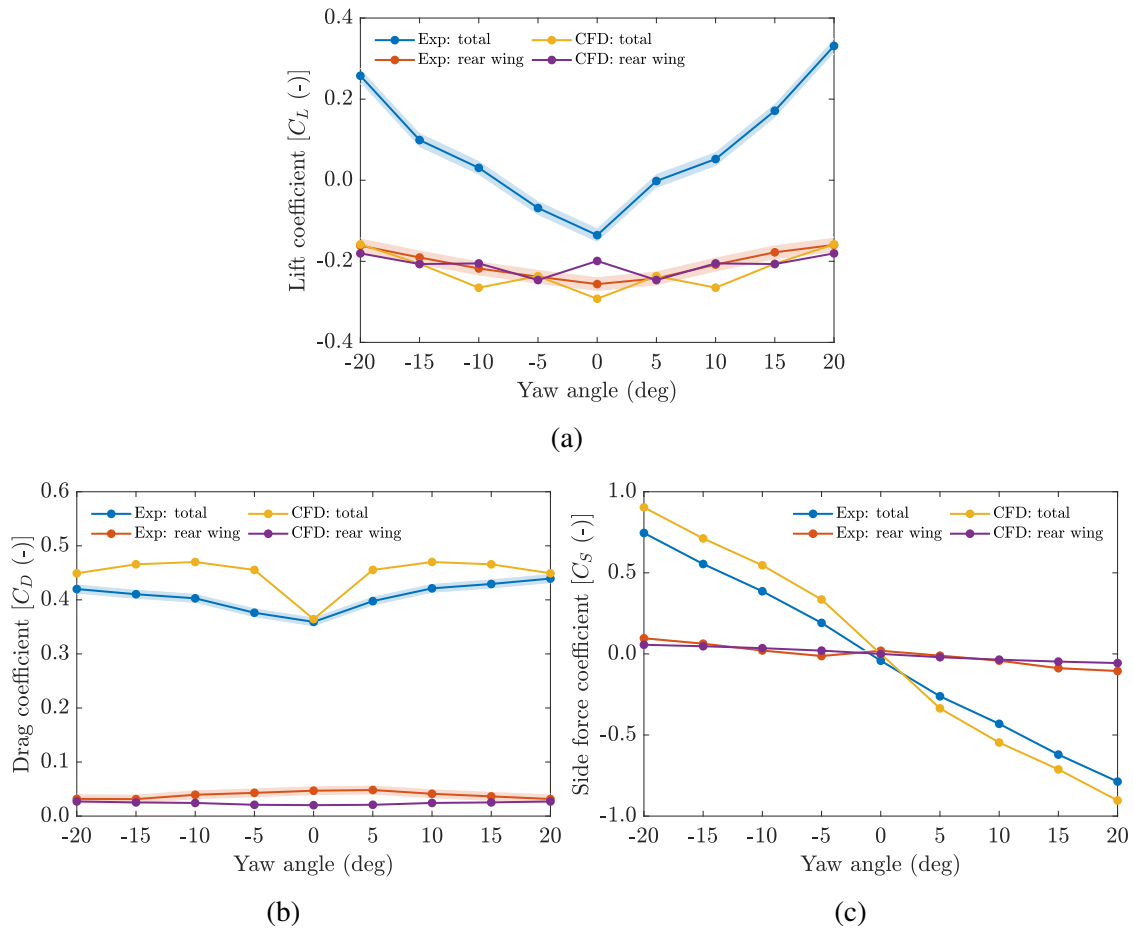


Figure 4.10: Comparison of (a) lift, (b) drag, and (c) side force coefficients of the entire vehicle and rear wing's contribution obtained from wind tunnel experiments and CFD simulations. The shaded bars indicate the accuracy of the measurement data.

Figure 4.10c presents the side force coefficients, showing that both experimental and numerical results exhibit a similar overall yaw sensitivity trend. However, the numerical simulations predict a steeper initial increase in side force at low yaw angles, after which the relative trend aligns more closely with the experimental data. Despite this, the CFD simulations consistently overpredict total side force by approximately $\Delta C_S \approx 0.12$ across the yaw range. In contrast, the rear wing's contribution to side force is more accurately captured, demonstrating an average deviation of only $\Delta C_S \approx 0.03$ relative to experimental measurements.

4.3.2 Surface Pressure

Figure 4.11 shows the experimental and numerical surface pressure coefficient distributions on the slant of the *DrivAer hp-F* with rear wing configuration. The simulation results exhibit stronger qualitative agreement in distribution trends than in absolute pressure magnitudes. Both datasets indicate that the pressure recovery along the slant gradually shifts towards the bottom-leeward corner as yaw angle increases, accompanied by the formation of a low-pressure region originating from the top windward corner. However, at low yaw angles, the simulations underpredict surface pressure, particularly near the slant's leading and trailing edges. While these discrepancies are smaller than those observed in the lift coefficient trends, they occur along the upper surface, where flow is unlikely to be affected by reduced underbody flow momentum. This indicates that additional modelling limitations may be contributing to the differences between the CFD simulations and experimental data. At larger yaw angles, agreement improves, with both the distribution trends and absolute magnitudes of surface pressure aligning more closely.

4.3.3 Wake Total Pressure

Figure 4.12 presents the experimental and numerical total pressure coefficient distributions on the P1 plane behind the *DrivAer hp-F* with rear wing configuration. The simulations show reasonable agreement with the experimental data in terms of overall wake shape, total pressure magnitude, and key wake structures. In the straight-line condition, both the numerical and experimental results capture dominant flow features, including the counter-rotating vortex pair generated by the rear wing in the upper wake region at $z/H > 0.75$ and the diffuser-induced upwash in the central wake region, characterised by a localised region of $C_{pT} \approx 0.30$.

Despite the general agreement, discrepancies are evident in certain regions. Low total pressure layers along the wind tunnel floor, outside the main wake, appear smaller in the simulation results than in the experiments. This suggests reduced boundary layer momentum along the wind tunnel floor, supporting the earlier interpretation that differences in underbody flow conditions may contribute to the numerical overprediction of downforce. At larger yaw angles, further discrepancies arise, with the simulations underpredicting total pressure losses along the leeward side outside the main wake ($y/W < -0.4$). These deviations may result from differences in flow separation behaviour and boundary layer control between the simulations and wind tunnel experiments. Nonetheless, the simulations capture the overall trend of progressive windward wake displacement and its increasing tilt angle towards the leeward side across the yaw angle range.

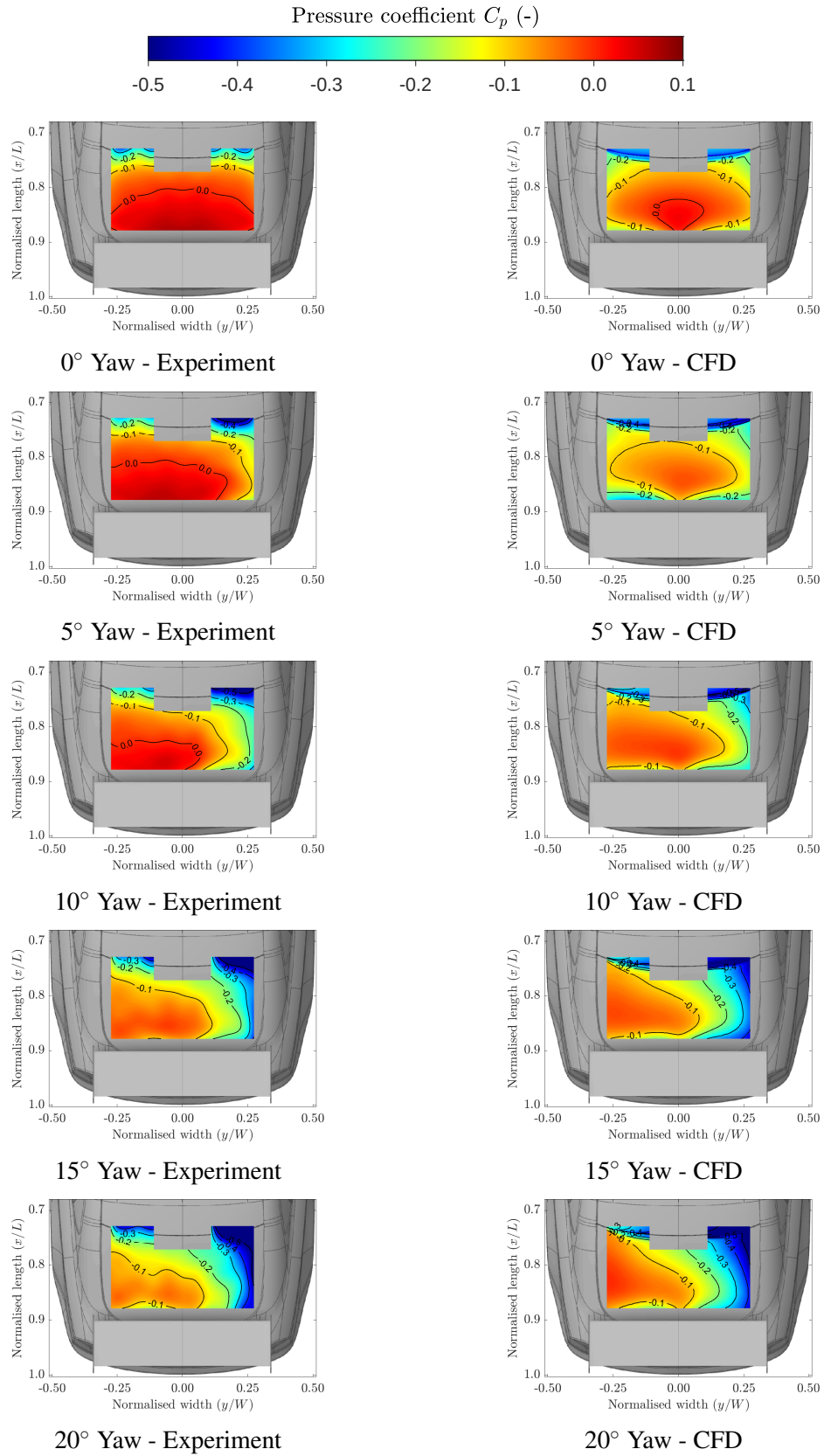


Figure 4.11: Comparison between experimental and numerical results of pressure coefficient distributions on the slant of the *DrivAer hp-F* with rear wing configuration.

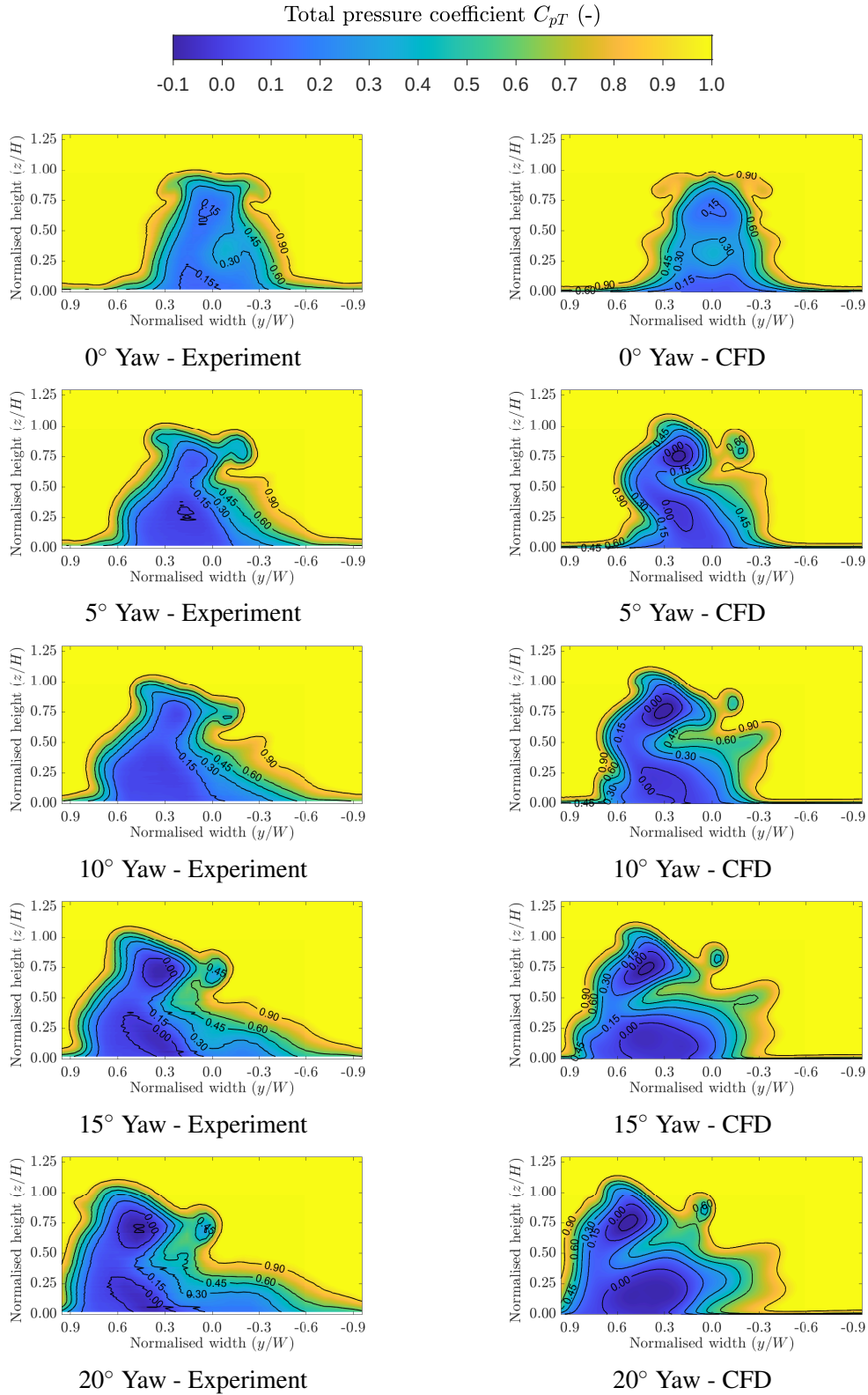


Figure 4.12: Comparison between experimental and numerical results of total pressure coefficient distributions on the P1 wake plane of the *DrivAer hp-F* with rear wing configuration.

4.3.4 Summary

The validation results show that the CFD simulations capture key aerodynamic trends and flow features with reasonable accuracy. However, notable discrepancies are observed, particularly in the magnitude and yaw sensitivity of the lift coefficients. One contributing factor may be reduced underbody flow momentum in the physical wind tunnel, evidenced by differences in total pressure levels along the floor, potentially influenced by floor irregularities and imperfect suction systems. Further discrepancies on the slant surface at low yaw angles, where underbody effects are minimal, indicate that additional modelling limitations are also likely involved. Nonetheless, the simulations reproduce slant pressure trends and dominant wake characteristics across the yaw range with acceptable fidelity. Overall, the CFD simulations provide a reliable complementary tool to the experimental data, enabling deeper analysis of yaw-induced aerodynamic effects.

4.4 Results and Discussion

This section presents experimental and numerical results to analyse the effects of yaw on the aerodynamic performance and flow field characteristics of the *DrivAer hp-F* model in its standard, spoiler, and rear wing configurations. Section 4.4.1 examines the aerodynamic forces across these configurations to evaluate performance trends under yaw. To illustrate the general effects on aerodynamic characteristics, Section 4.4.2 presents numerical visualisations of the rear wing configuration's flow field. Section 4.4.3 follows with a detailed analysis of surface pressure distributions, based on both experimental and numerical results. Finally, Section 4.4.4 discusses wake development and dominant flow structures using total pressure measurements behind the rear wing configuration. Throughout this study, positive yaw angles correspond to a counter-clockwise rotation of the vehicle model when viewed from above.

4.4.1 Aerodynamic Forces

Experimental measurement results of lift, drag, and side force coefficients on the standard, spoiler, and rear wing configurations of the *DrivAer hp-F* model are provided in this section, covering both positive and negative yaw angles. At positive yaw angles, the right side of the vehicle is on the windward side, while at negative yaw angles, it becomes the leeward side. Minor asymmetries are observed in the experimental data, indicative of physical limitations in load cells alignment, flow uniformity, and geometric symmetry of the vehicle model. However, these effects manifest as small systematic offsets in the force measurements and do not hinder the analysis of the vehicle's yaw sensitivity.

4.4.1.1 Lift Coefficients

The lift coefficients presented in Figure 4.13 indicate that the standard configuration achieves a minimum total lift coefficient of $C_L \approx 0.12$ at 0° yaw. In this condition, the front and rear axles exhibit lift coefficients of $C_L \approx -0.04$ and $C_L \approx 0.16$, respectively. This distribution suggests that aerodynamic elements such as the splitter and strakes, combined with initial airflow acceleration beneath the vehicle, contribute to minor front axle down-force generation.

Conversely, the absence of a moving ground, combined with wind tunnel floor irregularities, reduces airflow momentum beneath the vehicle during the experiments. This limits underbody and diffuser downforce generation relative to levels seen in Chapter 3 (23), contributing to the observed rear axle lift.

Lift generation by the standard configuration nearly quadruples across the yaw angle range, increasing by $\Delta C_L \approx 0.33$ to a maximum of $C_L \approx 0.45$ at $\pm 20^\circ$ yaw. Downforce on the leeward front wheel initially increases by $\Delta C_L \approx -0.02$ before stabilising beyond $\pm 5^\circ$ yaw. Similarly, lift on the leeward rear wheel gradually decreases at a rate of $\Delta C_L \approx -0.01$ per yaw increment up to $\pm 15^\circ$. In contrast, the windward front wheel experiences a steady lift increase of approximately $\Delta C_L \approx 0.05$ per 5° yaw increment. The windward rear wheel also exhibits increased lift, with an initial rise of $\Delta C_L \approx 0.07$ at 5° yaw, followed by smaller increments of $\Delta C_L \approx 0.02$ at higher yaw angles.

The spoiler configuration achieves a minimum total lift coefficient of $C_L \approx -0.05$ at 0° yaw, thereby generating downforce. In this straight-line condition, the addition of the spoiler increases rear axle downforce by $\Delta C_L \approx -0.19$ while slightly increasing front axle lift by $\Delta C_L \approx 0.02$, resulting in a net total downforce gain of $\Delta C_L \approx -0.17$ compared to the standard configuration.

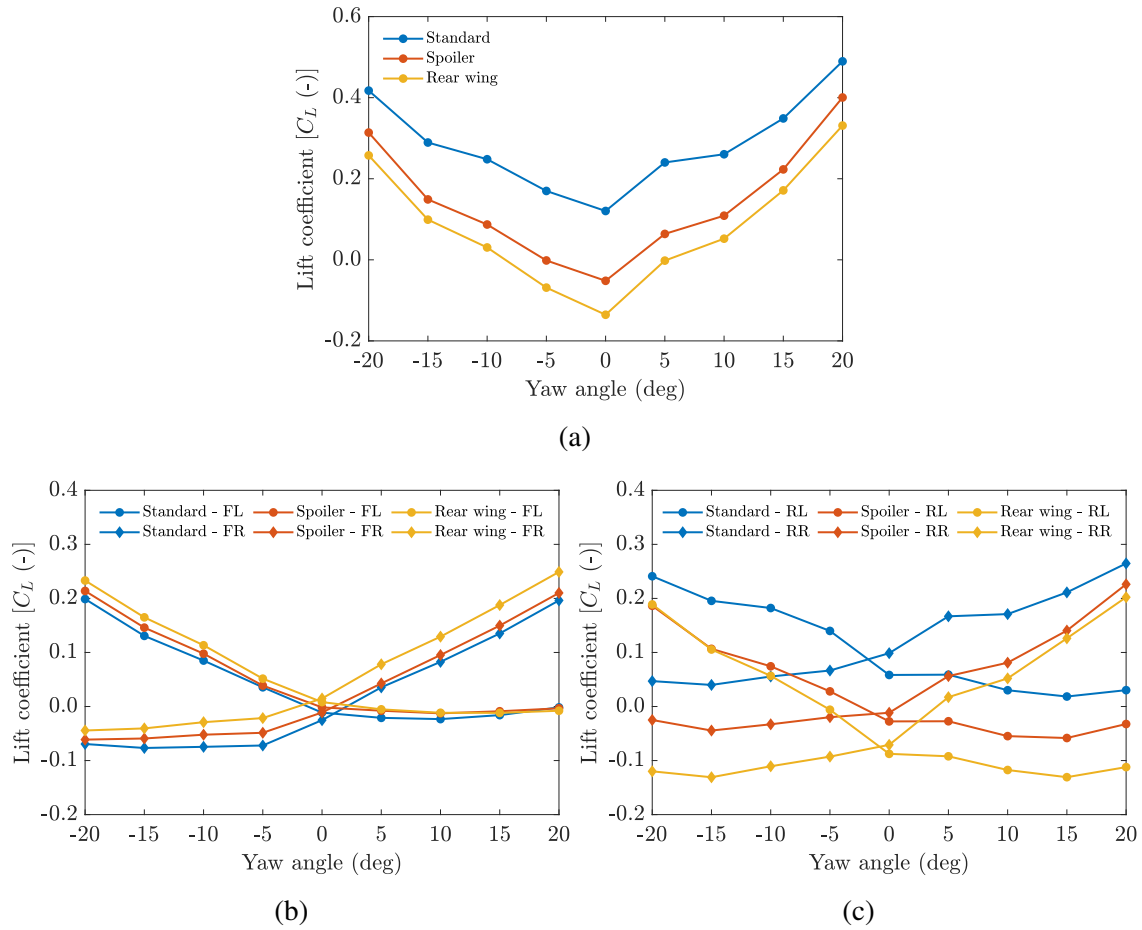


Figure 4.13: Experimental results of (a) total lift, (b) front axle lift, and (c) rear axle lift coefficients for the standard, spoiler, and rear wing *DrivAer hp-F* configurations.

Across the yaw range, the spoiler's influence on individual wheel loads remains largely consistent in trend with the standard configuration, except for the windward rear wheel. The diminishing lift coefficient offset between the configurations at higher yaw angles suggests a loss of downforce on the windward side of the spoiler. This yaw sensitivity is further evident in the total lift coefficient trend, where the spoiler configuration exhibits a $\Delta C_L \approx 0.41$ increase across the yaw angle range, 23% higher than the standard configuration, reaching a maximum lift coefficient of $C_L \approx 0.36$ at $\pm 20^\circ$ yaw.

The rear wing configuration produces the most downforce, reaching a minimum total lift coefficient of $C_L \approx -0.14$ at 0° yaw. This represents a $\Delta C_L \approx -0.26$ improvement over the standard configuration, generating nearly 50% more additional downforce than the spoiler. The rear wing increases rear axle downforce by $\Delta C_L \approx -0.32$ while causing a $\Delta C_L \approx 0.06$ rise in front axle lift at 0° yaw. Unlike the other configurations, the pronounced rearward shift in downforce distribution leads to minor net lift at the front axle.

The lift coefficient of the windward rear wheel exhibits a decreasing relative offset between the rear wing and standard configurations at higher yaw angles, indicating a progressive loss of rear wing effectiveness. This yaw sensitivity is further reflected in the total lift coefficients, where the rear wing configuration experiences a lift increase of $\Delta C_L \approx 0.43$ between 0° and $\pm 20^\circ$ yaw, approximately 29% greater than the standard configuration and 5% higher than the spoiler configuration, reaching a maximum of $C_L \approx 0.29$ at $\pm 20^\circ$ yaw. As will be discussed in Section 4.4.3.3, this increased yaw sensitivity is influenced by flow obstruction from the windward endplate.

4.4.1.2 Drag Coefficients

Figure 4.14a shows that yaw conditions induce a strongly non-linear effect on drag coefficients, with diminishing increments at larger yaw angles. The standard configuration attains a minimum drag coefficient of $C_D \approx 0.31$ at 0° yaw, which increases maximally by $\Delta C_D \approx 0.09$ to $C_D \approx 0.40$ (+27%) at $\pm 20^\circ$ yaw.

The spoiler configuration has a minimum drag coefficient of $C_D \approx 0.35$ at 0° yaw, which increases by a maximum of $\Delta C_D \approx 0.06$ to $C_D \approx 0.41$ (+17%) at $\pm 20^\circ$ yaw. The initial drag increase over the standard configuration decreases from $\Delta C_D \approx 0.04$ (+12%) at 0° to $\Delta C_D \approx 0.01$ (+3%) at $\pm 20^\circ$ yaw, experiencing an approximate 31% lower drag penalty across the yaw range. This trend indicates that the spoiler's additional drag contribution diminishes as the yaw angle increases, consistent with the yaw sensitivity observed in lift coefficients, where reduced downforce at high yaw angles leads to less associated induced drag.

The rear wing configuration reaches a minimum drag coefficient of $C_D \approx 0.36$ at 0° yaw, approximately 15% higher than the standard configuration and 3% higher than the spoiler configuration. Its drag increases by a maximum of $\Delta C_D \approx 0.07$ to $C_D \approx 0.43$ (+20%) at $\pm 20^\circ$ yaw. The absolute drag penalty across the yaw range is about 17% lower than for the standard configuration, indicating that the additional drag from the rear wing diminishes at higher yaw angles. However, its absolute drag increase is roughly 19% higher than that of the spoiler, suggesting spoiler-induced drag reduces more rapidly with yaw.

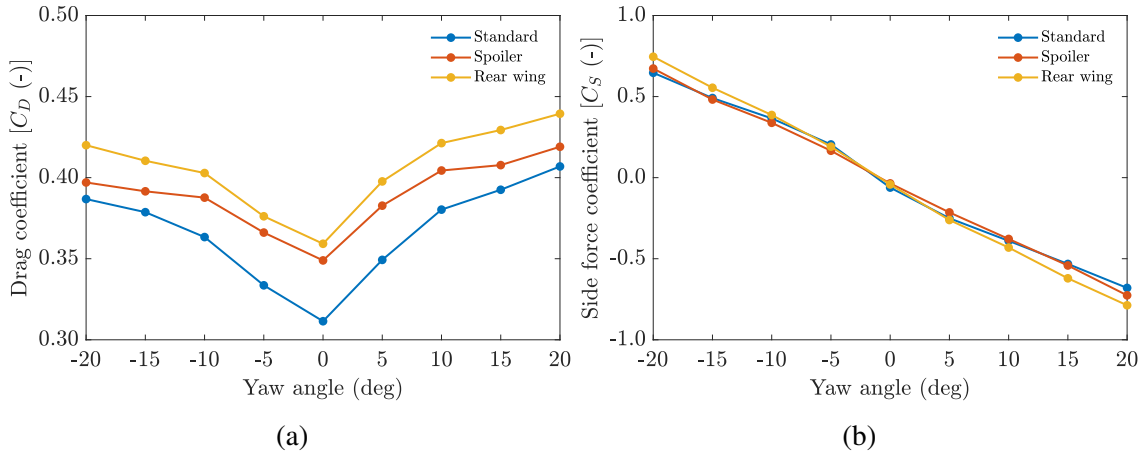


Figure 4.14: Experimental results of (a) drag and (b) side force coefficients for the standard, spoiler, and rear wing *DrivAer hp-F* configurations.

4.4.1.3 Side Force Coefficients

The side force coefficients for each configuration, shown in Figure 4.14b, increase nearly linearly with yaw angle. As will be discussed in Section 4.4.3.1, this trend corresponds to the progressive formation of a high-pressure region on the vehicle's windward side. The standard configuration reaches a maximum side force coefficient of $C_S \approx \pm 0.66$ at $\pm 20^\circ$ yaw. The spoiler moderately increases side force, reaching a 5% higher maximum of $C_S \approx \pm 0.70$ over the standard configuration. The rear wing configuration, with its largest side profile, generates the highest side force. It reaches a maximum of $C_S \approx \pm 0.77$, which is approximately 16% higher than the standard configuration and 10% higher than the spoiler configuration.

4.4.2 Flow Field

Numerical representations of the wake structures on the *DrivAer hp-F* model with rear wing configuration are presented in Figure 4.15a. These results reveal that at 0° yaw, most severe flow separation occurs at the wheels and behind the vehicle's base. Smaller separated flow regions are also observed at the splitter, mirrors and rear wing. When the vehicle is subject to yaw conditions, the flow separation underneath the splitter shifts leeward and becomes less severe. The wake from the windward mirror diminishes across the yaw angle range, while the wake from the leeward mirror increases in size.

The wheel wakes generally expand and develop with the freestream direction under yaw conditions. As a result, the windward wheel wakes propagate inboard, interfering with the underbody and diffuser flow, while the leeward wheel wakes move outboard, promoting additional flow separation along the vehicle's leeward side. Yaw conditions also amplify flow separation on the rear wing's lower surface and support structure, while vortical flow generated at the rear wing strengthens and grows in size across the yaw range. Additionally, a strong inboard-rotating vortex develops behind the windward side of the vehicle, becoming a dominant wake structure at larger yaw angles.

Figure 4.15b presents numerical representations of the turbulent structures on the rear wing configuration of the *DrivAer hp-F*. At 0° yaw, small streamwise vortices form along the A-pillars and strakes, while more distinct vortex structures develop around the rear wing due to the pressure difference with the freestream airflow outboard of the endplates. This pressure difference generates outboard-rotating vortices on the wing's upper surface and inboard-rotating vortices on its lower surface. The greater pressure differential beneath the rear wing produces noticeably larger inboard-rotating vortices.

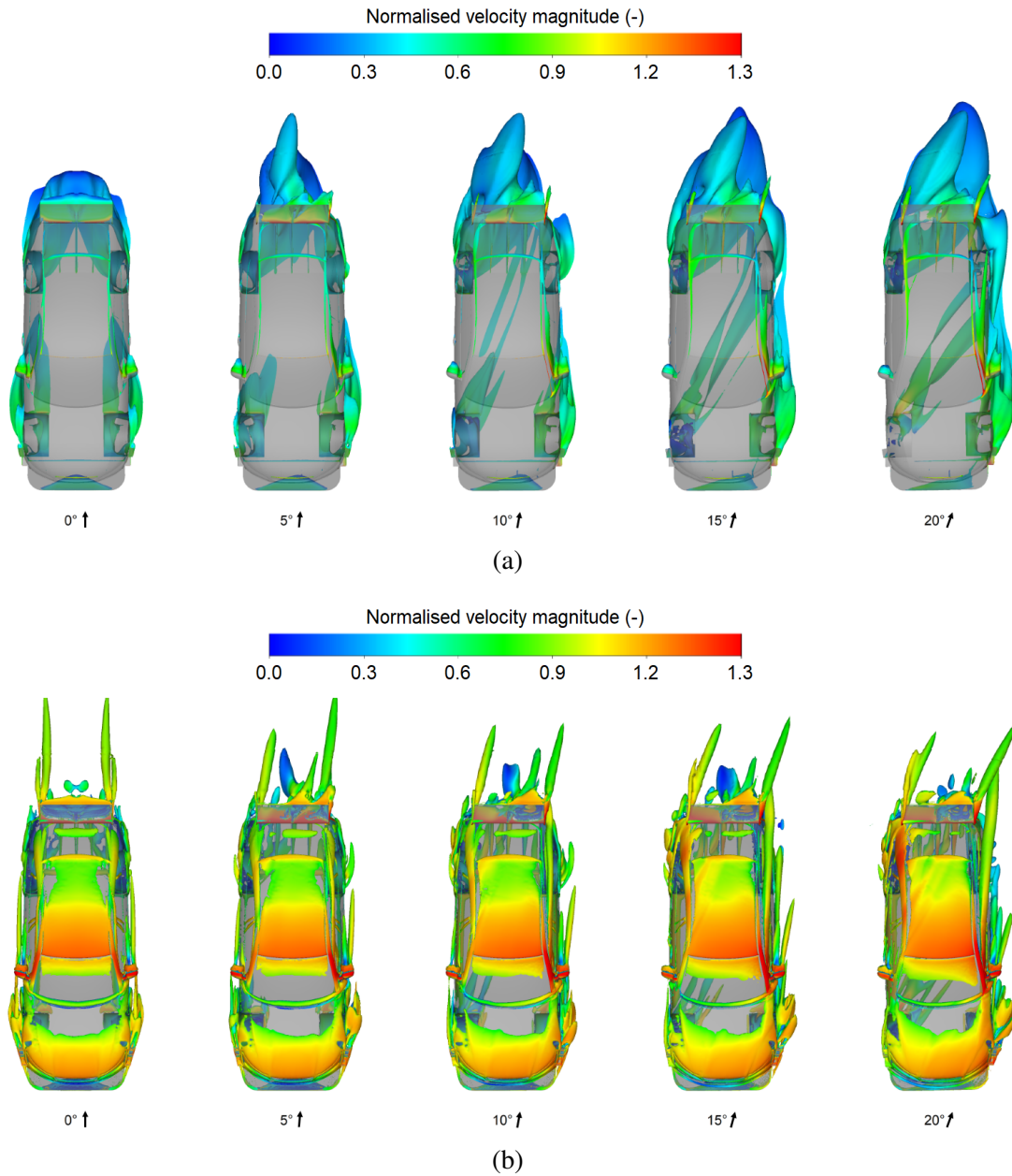


Figure 4.15: Numerical representations of the flow field of the *DrivAer hp-F* with rear wing configuration through iso-surfaces of (a) zero total pressure and (b) q-criterion ($QL^2/U_\infty^2 = 130$), coloured by normalised velocity magnitude.

Under yaw conditions, the windward A-pillar vortex diminishes in size, while the leeward A-pillar vortex expands significantly and shifts outboard. Section 4.4.3.3 will discuss that yaw conditions induce an increasing pressure build-up on the outboard side of the windward endplate and the inboard side of the leeward endplate. As a result, Figure 4.15b illustrates that the formation of an outboard-rotating vortex on the windward side of the rear wing's upper surface is suppressed, and at larger yaw angles, even an inboard-rotating vortex is formed. Additionally, the pressure build-up outboard of the windward endplate promotes the formation of an inboard-rotating vortex on the windward side of the rear wing's lower surface. Conversely, on the leeward side, the formation of an outboard-rotating vortex on the rear wing's upper surface is enhanced, while the inboard-rotating vortex on the lower surface is suppressed.

4.4.3 Pressure Distribution

4.4.3.1 Vehicle Body

Numerical results of time-averaged surface pressure coefficients on the *DrivAer hp-F* model with rear wing configuration are presented in Figure 4.16. The vehicle exhibits generally a symmetric pressure distribution at 0° yaw. The front view reveals that the airflow's frontal impact creates a stagnation zone at the nose, followed by a pressure reduction as airflow accelerates around the curvature of the nose. The top view shows a gradual pressure recovery along the bonnet towards the high-pressure concentration at the windscreen. Airflow accelerates again around the curvature of the A-pillars and roof's leading edge, resulting in low-pressure regions. A slight pressure increase along the roof is followed by a steeper pressure recovery region on the slant, which develops towards the rear wing support.

A single side view is presented at 0° yaw due to the symmetry of the time-averaged flow field. The side view reveals distinct low-pressure regions around the curvatures of the front wheel arch, A-pillar, roofline, and behind the rear wheel arch, while a relatively uniform moderately low pressure is observed along the door panels and windows. The bottom view highlights a localised low-pressure concentration at the centre of the splitter, resulting from flow separation at its sharp leading edge.

A larger low-pressure region forms between the front wheels due to the initial acceleration of airflow beneath the vehicle, followed by a pressure recovery along the floor. The diffuser exhibits a low-pressure region at the inlet, with a gradual pressure increase along the diffuser ramp. The rear view indicates a moderately low pressure across the vehicle's base, with more pronounced low-pressure concentrations at the upper and lower corners, attributed to the flow recirculation zones just behind the base.

The pronounced impact of yaw conditions on the pressure distribution is evident from the substantial shift in pressure coefficients across the yaw angle range. The front views illustrate that the stagnation zone displaces windward with the yaw flow impact angle. This shift is accompanied by larger low-pressure regions on the leeward side of the nose and bonnet due to enhanced airflow acceleration along their curvatures. Consequently, the localised drag build-up at the nose decreases by approximately 26% at 20° yaw compared to the straight-line condition.

The top views reveal that the high-pressure concentration on the windscreen shifts towards the windward side under yaw conditions. Additionally, increased airflow acceleration over the roof amplifies the extent of the low-pressure regions along the roof and windward roofline. As a result, local lift generation on the roof increases by approximately 19% at 20° yaw compared to 0° yaw. The airflow velocity along the windward C-pillar also increases with yaw angle, generating an expanding low-pressure region that extends leeward onto the slant. The resulting shifts in pressure distribution along the slant and rear wing will be examined in greater detail in Sections 4.4.3.2 and 4.4.3.3.

The bottom views illustrate the formation of an increasing high-pressure region beneath the splitter as the airflow impact angle changes with yaw. Flow separation at the splitter's leading edge, along with the associated low-pressure concentration, diminishes in size and shifts towards the leeward side. Additionally, the effective inlet area to the vehicle's underbody is reduced at larger yaw angles, leading to a weakening of the low-pressure region between the front wheels. This reduction results from the increasing flow obstruction by the windward front wheel, as evidenced by its expanding low-pressure wake.

Despite the decreased effective underbody inlet area at the front, additional airflow enters from the windward side at larger yaw angles, as indicated by the growing low-pressure region along the windward edge of the underbody. Consequently, the depression in pressure at the diffuser inlet shifts towards the windward side. Furthermore, the interaction between the yawed airflow and the diffuser separators induces vortex structures along the leeward side of the diffuser separators, indicated by distinct low-pressure concentrations.

The side views illustrate that the expanding low-pressure regions along the leeward side of the nose and bonnet extend towards the leeward front wheel, while the moderately low pressures along the leeward door panels remain fairly similar across the yaw angle range. In contrast, yaw conditions generate an increasing high-pressure region behind the windward front wheel and along the windward door panels.

As previously shown in Figure 4.15b, yaw promotes the formation of the leeward A-pillar vortex while suppressing the windward A-pillar vortex, a trend reflected in the respective low-pressure concentrations in these regions. The pressure distributions along the leeward roofline and C-pillar appear to be influenced by the trajectory of the strengthened leeward A-pillar vortex, exhibiting only a slight pressure rise at larger yaw angles. In contrast, the windward roofline and C-pillar experience a significant reduction in pressure across the yaw range due to enhanced airflow acceleration around these curvatures.

The windward rear quarter panel also develops an expanding low-pressure region, whereas the leeward rear quarter panel shows a gradual pressure increase just behind the rear wheel. However, the downstream edge of the leeward rear quarter panel exhibits an intensifying low-pressure concentration, attributed to airflow that rolls-up into a recirculation zone near the leeward side of the vehicle's base in yaw conditions.

The rear views illustrate significant changes in the pressure distribution across the base surface in yaw conditions. Moderately low pressures are concentrated on the windward side, attributed to the formation of a dominant streamwise vortex in this region, as previously identified in Figure 4.15a. This vehicle body vortex is formed by inboard-rotating high-velocity airflow originating from the windward C-pillar and rear quarter panel.

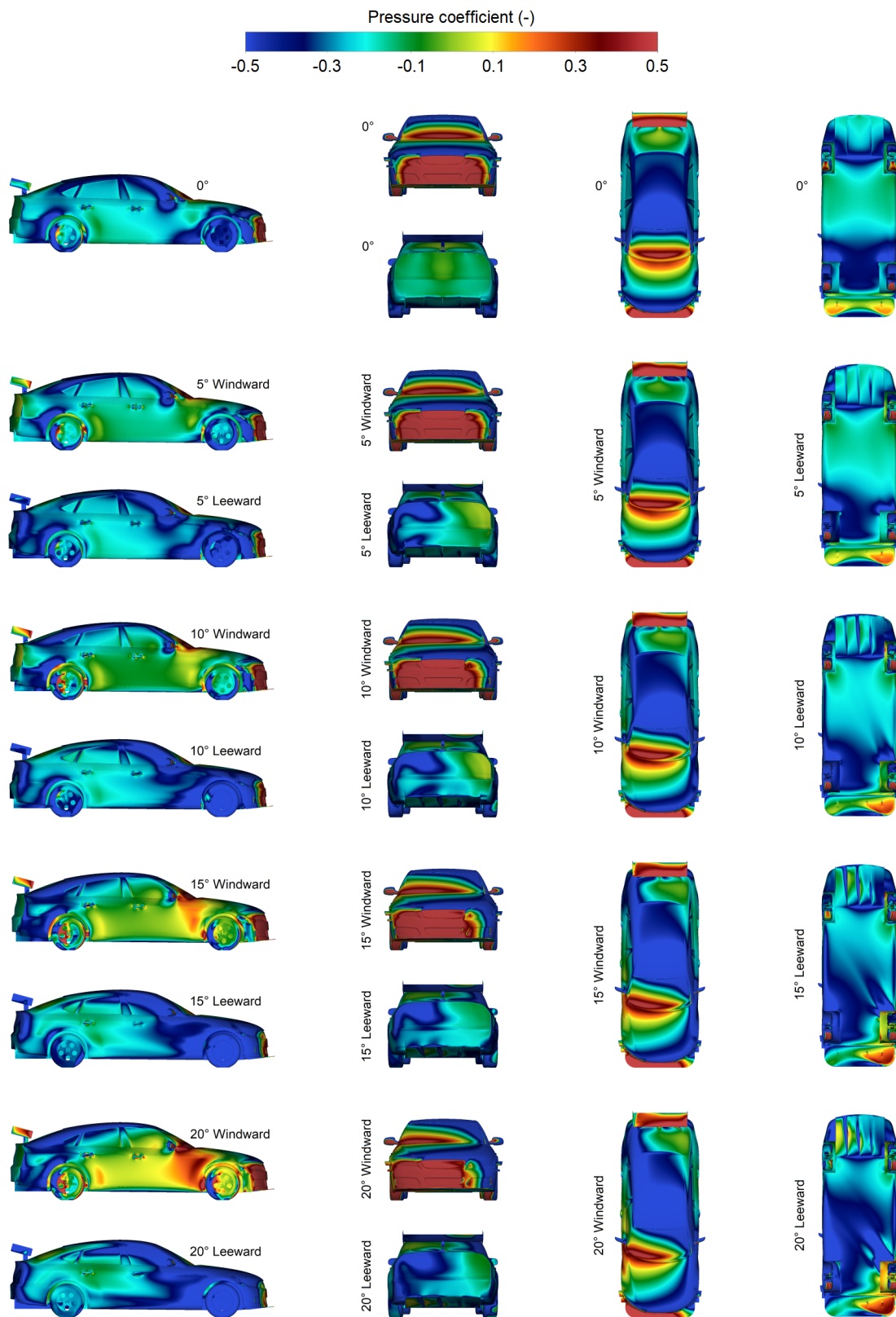


Figure 4.16: Numerical results of pressure coefficient distributions on the *DrivAer hp-F* with rear wing configuration.

In contrast, lower-velocity airflow along the leeward side of the slant and rear quarter panel moves around the periphery of the base and rolls up into recirculating wake vortices. These recirculation wake vortices extend in the spanwise direction as they are drawn towards the dominant vehicle body vortex, contributing to the formation of the pronounced low-pressure regions observed on the leeward side of the vehicle's base.

4.4.3.2 Slant

Experimental results of time-averaged surface pressure coefficients on the slant of the standard, spoiler, and rear wing configurations of the *DrivAer hp-F* model are presented in Figure 4.17. At 0° yaw, the standard configuration exhibits a pressure recovery from an average $C_p \approx -0.43$ near the roof's trailing edge to a maximum of $C_p \approx -0.02$ at the centre of the slant. A more gradual recovery is observed along the sides due to relatively high-velocity airflow along the C-pillars.

In yaw conditions, airflow accelerates around the windward C-pillar, forming a low-pressure region across the slant's top windward corner. The pressure recovery shifts in the direction of the freestream airflow, migrating the pressure maximum towards the bottom leeward corner. Across the yaw angle range, the average pressure on the slant decreases by approximately a factor of two, from $C_p \approx -0.13$ at 0° yaw to $C_p \approx -0.26$ at 20° yaw.

The spoiler configuration yields an average slant pressure of $C_p \approx 0.02$ at 0° yaw, an increase of $\Delta C_p \approx 0.15$ over the standard configuration. This rise results from the pressure build-up at the spoiler, which propagates upstream, affecting the slant and roof surfaces. Two local pressure maxima of $C_p \approx 0.08$ emerge at the slant's downstream corners due to interactions between the spoiler and high-velocity airflow from the C-pillars.

With increasing yaw angle, the pressure maxima shift with the altered flow direction. The leeward maximum disappears by 10° yaw, while the windward maximum migrates toward the bottom leeward corner at 20° yaw. A low-pressure region forms at the top windward corner, though spoiler-induced upstream pressurisation mitigates its severity. The pressure recovery aligns again with the freestream airflow direction, yielding an average slant pressure of $C_p \approx -0.18$ at 20° yaw, about 32% higher than for the standard configuration.

The rear wing configuration maintains a similar pressure distribution to the standard setup at 0° yaw but at higher pressures due to upstream pressurisation from the rear wing and its support. Unlike the spoiler, it does not induce significant pressure build-up along the slant's sides. The pressure recovers to a maximum of $C_p \approx 0.07$ at the slant's centre, with an average pressure of $C_p \approx -0.02$, an 82% increase over the standard configuration.

In yaw conditions, the pressure recovery aligns with the freestream airflow direction, forming an elongated pressure maximum along the slant's leeward side. As will be discussed in Section 4.4.3.3, this distribution results from the angled airflow interacting with the rear wing's leading edge, increasing pressure on the leeward upper surface. At 20° yaw, the slant reaches a maximum pressure of $C_p \approx -0.04$ with an average of $C_p \approx -0.19$, which is 26% higher than the standard configuration but 7% lower than the spoiler configuration.

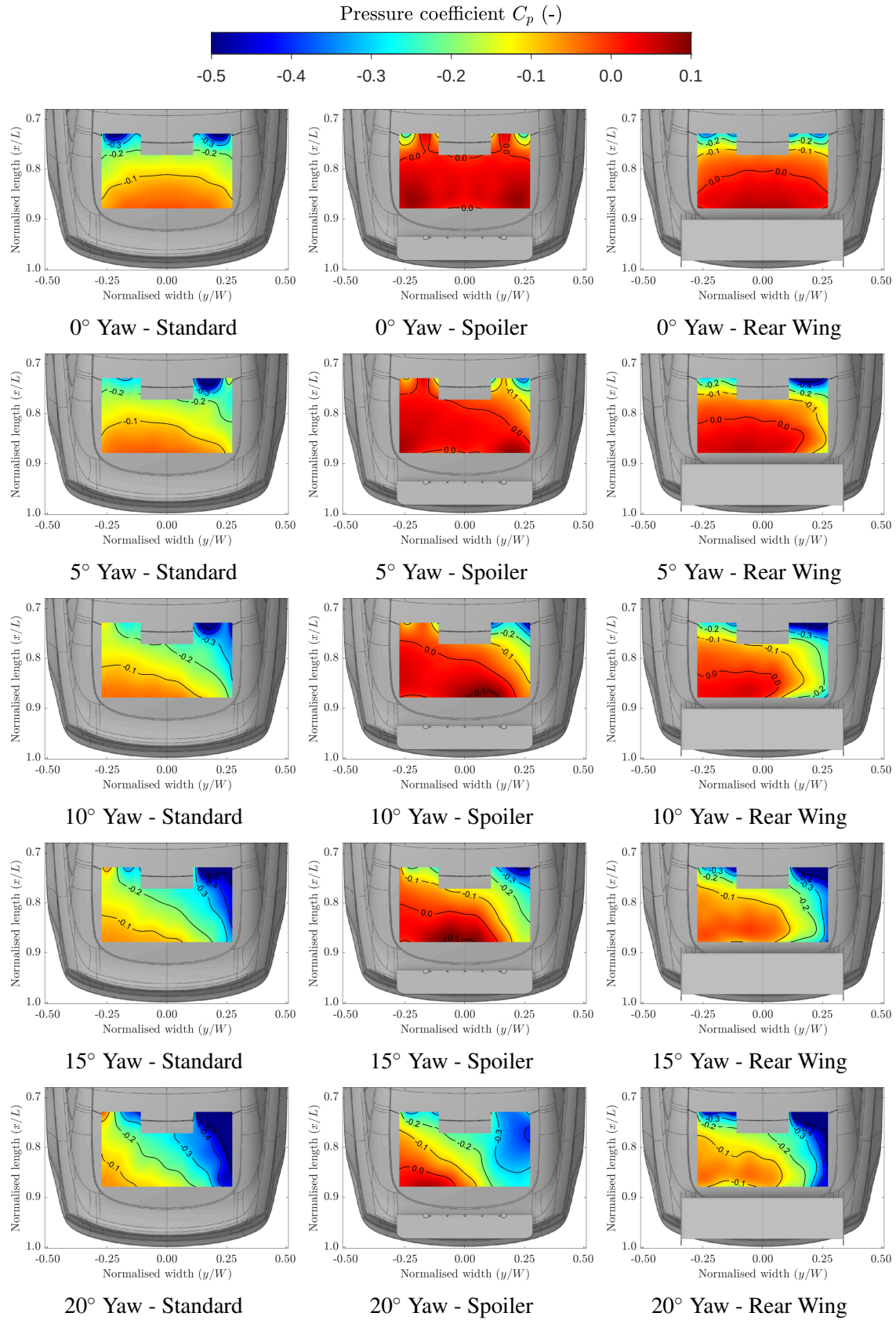


Figure 4.17: Experimental results of pressure coefficient distributions on the slant of the standard, spoiler, and rear wing *DrivAer hp-F* configurations.

4.4.3.3 Rear Wing

Numerical results of time-averaged pressure coefficients on the rear wing are presented in Figure 4.18. On the upper surface at 0° yaw, a high-pressure region forms along the leading edge, which gradually decreases towards the trailing edge. Steeper pressure gradients occur near the endplates, resulting from outboard-rotating vortices induced by the pressure differential with the freestream airflow. At higher yaw angles, windward endplate obstruction increases, reducing windward upper surface pressure while causing an outboard pressure build-up on the windward endplate, as previously shown in Figure 4.16.

At larger yaw angles, this build-up induces an inboard-rotating vortex, forming a distinct low-pressure region near the windward endplate. Conversely, the leeward side experiences less obstruction and retains higher pressures throughout the yaw range. The upstream pressurisation effects associated with this pressure distribution contribute to the leeward concentration of the pressure maximum on the slant, as previously observed in Figure 4.17. Additionally, the inboard pressure build-up at the leeward endplate leads to the more gradual pressure gradient near the leeward endplate in yaw conditions.

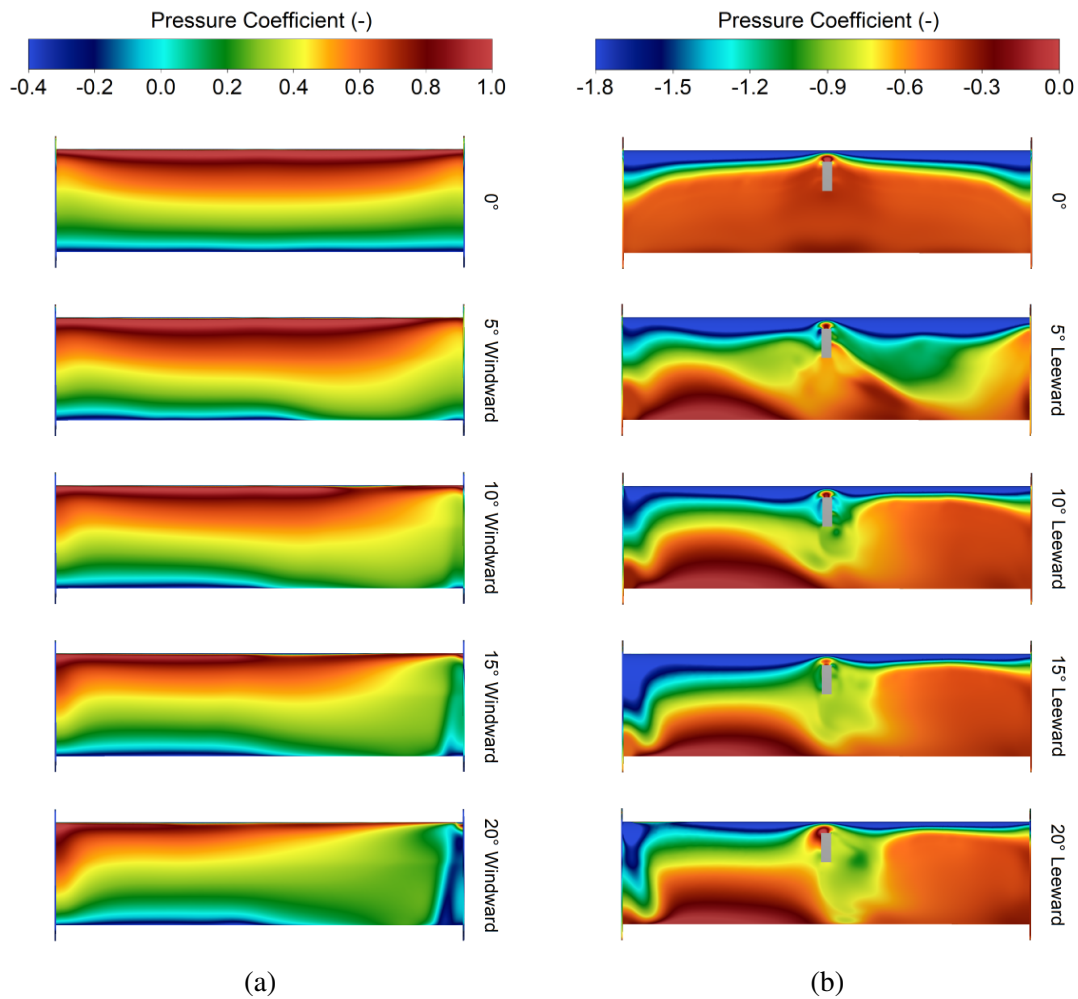


Figure 4.18: Numerical results of pressure coefficient distributions on the (a) upper and (b) lower surface of the rear wing of the *DrivAer hp-F*.

The lower surface exhibits a pronounced low-pressure region near the leading edge, followed by a sharp pressure recovery downstream at 0° yaw. Near the endplates, the pressure recovers more gradually due to the inboard-rotating vortices, induced by the pressure differential across the endplates, that direct airflow onto the lower surface. The rear wing support obstructs airflow near the leading edge, creating a small stagnation zone.

In yaw conditions, intensified outboard pressure build-up at the windward endplate strengthens the inboard-rotating vortex, further reducing pressure on the lower surface near the windward endplate. Increasing airflow velocity along the windward C-pillar and slant enhances the low-pressure region on the windward lower surface. Meanwhile, the leeward side is mainly affected by flow obstruction from the rear wing support. At small yaw angles, this has minimal impact, as seen at 5° yaw, but at larger yaw angles, it substantially influences pressure along the leeward lower surface. Additionally, the inboard pressure build-up at the leeward endplate suppresses the inboard-rotating vortex, eliminating the low-pressure region near the leeward endplate.

4.4.4 Wake Characteristics

This section analyses the wake characteristics of the *DrivAer hp-F* with rear wing configuration across the yaw angle range using experimental results of time-averaged total pressure coefficients, as presented in Figure 4.19. The wake is examined at three downstream planes, defined as P1, P2, and P3, which are located $0.25L$, $0.43L$, and $0.62L$ behind the vehicle model, respectively. The horizontal displacement of the main wake structure with increasing yaw angles results from the vehicle's rotation within the wind tunnel, altering its alignment relative to the measurement planes.

4.4.4.1 0° Yaw

The near-wake characteristics on the P1 plane at 0° yaw consist of a square-shaped and mushroom-shaped wake region. The square-shaped wake region, approximately $1W$ wide and $0.75H$ tall, results from flow separation behind the vehicle body. Above this, the mushroom-shaped region, spanning roughly $0.75W$ in width and $0.3H$ in height, is formed by vortical structures from the rear wing. Additional wake expansion, associated with the rear wheel wakes and wind tunnel floor boundary layer, occurs around $y/W \approx \pm 0.55$.

The dominant wake characteristics in straight-line conditions are governed by diffuser upwash and vortical flow from the rear wing. Flow exiting the diffuser is represented by a centralised circular region of $C_{pT} \approx 0.30$ at $z/H \approx 0.34$. The counter-rotating vortex pair shed from the rear wing is located at $y/W \approx \pm 0.28$, $z/H \approx 0.83$. The interaction between these outboard rotating vortices induces lateral inwash of high-energy freestream airflow and additional vertical upwash at the centre of the wake.

Downstream wake evolution on the P2 plane reveals a reduction in wake width of $0.08W$ due to lateral inwash. Simultaneously, vertical upwash and diffusion of the rear wing vortices increase the wake height by approximately $0.09H$. The inwash and upwash wake characteristics promote wake mixing, leading to an increase in the average total pressure coefficient of the main wake (defined by $C_{pT} < 0.90$) from $C_{pT} \approx 0.45$ on the P1 plane to $C_{pT} \approx 0.52$ on the P2 plane.

On the P3 plane, the wake width is further reduced by $0.12W$, resulting in an overall contraction of approximately 20% relative to the P1 plane. Concurrently, the wake height increases by another $0.04H$, amounting to a total growth of nearly 13% over the P1 plane. The dissipation of pressure losses, promoted by enhanced mixing of high-energy freestream airflow into the wake, raises the average total pressure coefficient of the main wake by approximately 32% compared to the P1, reaching $C_{pT} \approx 0.59$ on the P3 plane.

4.4.4.2 5° Yaw

The influence of yaw on the wake characteristics becomes immediately evident from the P1 plane at 5° yaw. A strong inboard-rotating vortex develops on the windward side, as identified in Section 4.4.2. This clockwise-rotating vehicle body vortex appears as a region of $C_{pT} \approx 0.08$ at $y/W \approx 0.12$, $z/H \approx 0.70$. Additionally, yaw-induced pressure build-ups on the rear wing endplates, discussed in Section 4.4.3.3, enhance the formation of stronger vortical structures at the rear wing. The rear wing's windward counter-clockwise-rotating vortex is positioned at $y/W \approx 0.36$, $z/H \approx 0.91$, while the leeward clockwise-rotating vortex is observed at $y/W \approx -0.20$, $z/H \approx 0.78$.

The diffuser upwash effect is eliminated due to the combined influence of angled oncoming airflow and the inward propagation of the windward wheel wakes across the diffuser ramp. As a result, the central wake region is dominated by low-energy separated flow, showing no noticeable traces of higher-energy airflow exiting the diffuser. Additionally, wake traces of an enlarged leeward A-pillar vortex emerge at $y/W \approx -0.32$, $z/H \approx 0.53$. Further wake expansion occurs beyond around $y/W < -0.40$ due to increased airflow separation along the leeward side of the vehicle.

The main wake at 5° yaw on the P1 plane exhibits an average total pressure coefficient of $C_{pT} \approx 0.39$, approximately 12% lower than at 0° yaw. At this yaw angle, wake characteristics are primarily governed by the dominant vehicle body vortex and rear wing vortices. The strong clockwise circulation of the vehicle body vortex induces a corresponding clockwise displacement of the rear wing's counter-rotating vortex pair. This motion weakens the lateral inwash generated by the windward rear wing vortex. However, the leeward rear wing vortex, reinforced by the co-rotation of the vehicle body vortex, produces a stronger inwash effect.

These wake characteristics are also evident from the wake development on the P2 plane, where the tilt angle of the upper wake region increases from around 8° at P1 to approximately 23° at P2. Stronger inwash on the leeward side reduces the local wake width by $0.08W$, while the wake width remains unchanged on the windward side. The inwash of high-energy freestream airflow facilitates pressure loss dissipation, raising the average total pressure by nearly 29% to $C_{pT} \approx 0.51$.

On the P3 plane, the upper wake region's tilt angle further increases to approximately 34°. The wake width narrows by an additional $0.07W$ compared to P2, resulting in a total reduction of 18% relative to P1. The continued dissipation of pressure losses leads to an average total pressure of $C_{pT} \approx 0.59$, representing an approximate 49% increase from the P1 plane.

4.4.4.3 10° Yaw

The vehicle body vortex strengthens further at 10° yaw, as indicated by the reduced total pressure trace of $C_{pT} \approx 0.03$ at $y/W \approx 0.22$, $z/H \approx 0.73$ on the P1 plane. The increased vortex strength is also evident from the more pronounced induced motion on the upper wake, now tilting at around 17°. Consequently, the windward and leeward rear wing vortices shift to around $y/W \approx 0.43$, $z/H \approx 0.94$ and $y/W \approx -0.12$, $z/H \approx 0.71$, respectively.

The wake trace of the leeward A-pillar vortex at $y/W \approx -0.30$, $z/H \approx 0.50$ becomes more distinct compared to the 5° yaw case. Additionally, the central wake region experiences greater pressure losses due to increased flow separation at the diffuser and vehicle body. Further flow separation along the leeward side of the vehicle leads to greater wake expansion at $y/W < -0.40$. Overall, the main wake on the P1 plane exhibits an average total pressure coefficient of $C_{pT} \approx 0.38$, marking a 3% decrease from 5° yaw.

The dominant influence of the vehicle body vortex on wake characteristics is further evident on the P2 plane. The upper wake region undergoes a stronger clockwise induction, increasing its tilt angle to approximately 35°. The interaction between the co-rotating vehicle body vortex and the leeward rear wing vortex not only intensifies lateral inwash on the leeward side but also introduces a local vertical downwash component.

The strengthened leeward inwash reduces wake width by approximately $0.08W$ compared to P1. However, this intensified one-sided inwash induces crossflow, leading to a minor windward outwash effect, which expands the windward wake width by $0.03W$. Simultaneously, the leeward downwash reduces wake expansion height at $y/W < -0.40$ by approximately $0.08H$. The enhanced wake mixing increases the average total pressure coefficient on the P2 plane to $C_{pT} \approx 0.49$, approximately 28% higher than on P1.

The strong circulation of the vehicle body vortex increases the upper wake region's tilt to approximately 53° on the P3 plane. The strong leeward inwash persists, reducing wake width by an additional $0.08W$ on the leeward side. Meanwhile, the windward outwash effect increases wake width on the windward side by $0.02W$. The leeward downwash further reduces wake expansion height at $y/W < -0.40$ by $0.06H$ compared to P2.

Altogether, the wake width on P3 reduces by approximately 9%, while the wake expansion height at $y/W < -0.40$ decreases by 36% relative to P1. These wake mixing effects enhance pressure loss dissipation, raising the average total pressure coefficient to $C_{pT} \approx 0.58$ on the P3 plane, representing a 52% increase from P1.

4.4.4.4 15° Yaw

The vehicle body vortex on the P1 plane at 15° yaw is identified by a low total pressure trace of $C_{pT} \approx -0.09$ at $y/W \approx 0.32$, $z/H \approx 0.72$. The further reduction in total pressure magnitude relative to the 10° yaw case indicates a continued increase in vortex strength. This intensification is also reflected in the displacement of the rear wing vortex pair, which has been induced further apart due to the stronger circulation of the vehicle body vortex. As a result, the leeward rear wing vortex is positioned at $y/W \approx 0.00$, $z/H \approx 0.70$, with its wake trace nearly detached from the main wake region.

The windward rear wing vortex shifts to $y/W \approx 0.55$, $z/H \approx 0.95$, resulting in an upper wake tilt angle of approximately 30° . The wake trace from the leeward A-pillar vortex at $y/W \approx -0.20$, $z/H \approx 0.43$ merges with the expanding wake region at $y/W < -0.30$, resulting from significant flow separation along the leeward side of the vehicle. The pressure losses in the middle wake region have intensified compared to the 10° yaw case due to increased flow separation at the diffuser and along the vehicle body. Overall, the main wake on the P1 plane exhibits an average total pressure coefficient of $C_{pT} \approx 0.37$, representing a 3% reduction compared to 10° yaw.

The wake characteristics at 10° yaw intensify at 15° . The strong circulation of the vehicle body vortex induces a pronounced clockwise motion on the upper wake region, increasing its tilt angle to approximately 43° on the P2 plane. This influence is particularly evident for the leeward rear wing vortex, which was nearly detached from the main wake on P1 but is now shifted downward and inboard, merging with the main wake trace on P2.

While no significant wake width reduction occurs on the leeward side due to the large declination of the main wake, the combined circulation of the vehicle body and leeward rear wing vortices generates a strong diagonal crossflow towards the wake's upper windward side. This results in a windward outwash effect, expanding the windward wake width by approximately $0.04W$ relative to the P1 plane. Additionally, leeward downwash reduces the wake expansion height at $y/W < -0.30$ by nearly $0.09H$. These dominant wake characteristics enhance wake mixing, increasing the average total pressure in the main wake on the P2 plane to $C_{pT} \approx 0.49$, approximately 32% higher than on the P1 plane.

At the P3 plane, the tilt angle of the upper wake region further increases to approximately 60° . Due to the continued declination of the main wake structure, no significant wake width reduction is observed on the leeward side. However, the intensified diagonal crossflow and resulting windward outwash increase the windward wake width by an additional $0.04W$ over the P2 plane, leading to a total windward wake width increase of approximately 14% compared to P1.

The leeward downwash further reduces the wake expansion height at $y/W < -0.30$ by an additional $0.10H$ relative to P2, resulting in a total wake expansion height reduction of approximately 60% compared to P1. The combined effects of wake inwash, outwash, and downwash promote enhanced dissipation of pressure losses across the downstream wake planes. Consequently, the main wake on the P3 plane exhibits an average total pressure coefficient of $C_{pT} \approx 0.58$, reflecting an approximate 56% increase relative to the P1 plane.

4.4.4.5 20° Yaw

The vehicle body vortex on the P1 plane at 20° yaw is identified by a wake trace of $C_{pT} \approx -0.10$ at $y/W \approx 0.48$, $z/H \approx 0.70$. The increased size and lower total pressure magnitude of this trace relative to the 15° yaw case indicate further strengthening of the vortex. The strong circulation induced by the vehicle body vortex is also reflected in the positioning of the rear wing vortices. The leeward rear wing vortex is positioned at $y/W \approx 0.04$, $z/H \approx 0.70$, with its wake trace further detached from the main wake compared to the 15° yaw case. The counter-clockwise rotating windward rear wing vortex is positioned at $y/W \approx 0.70$, $z/H \approx 0.95$.

An additional wake trace, attributed to a clockwise rotating vortex from the windward side of the rear wing's upper surface, is now distinctly observable at $y/W \approx 0.83$, $z/H \approx 0.90$. This secondary windward rear wing vortex forms due to the outboard pressure build-up at the windward endplate in yaw conditions, as discussed in Section 4.4.3.3. While this vortex is expected to form at lower yaw angles as well, it is likely weaker and merges quickly into the more dominant counter-clockwise rotating vortex from the rear wing's lower surface, preventing it from generating a clearly distinguishable wake trace at smaller yaw angles.

The pressure losses in the middle wake region have increased only slightly, indicating modest additional flow separation from the diffuser and vehicle body. The leeward A-pillar vortex no longer produces a distinct wake trace at 20° yaw, as it has fully merged into the expanding wake region at around $y/W < -0.15$. Compared to the 15° yaw case, the expansion of this region has increased, with greater pressure losses due to enhanced flow separation along the leeward side of the vehicle. Overall, the average total pressure coefficient of the main wake on the P1 plane is $C_{pT} \approx 0.35$, representing a 4% reduction relative to 15° yaw.

The P2 plane results show that the strong circulation from the vehicle body vortex remains the dominant wake characteristic at 20° yaw. The induced motion on the upper wake region increases its tilt angle from approximately 38° on the P1 plane to around 56° on the P2 plane. Although portions of the wake begin to extend beyond the measurement zone on the downstream planes, the crossflow-driven windward outwash effect is estimated to increase the windward wake width by approximately $0.04W$.

As in the 15° yaw case, no significant leeward wake width reduction is observed at the P2 plane due to the large declination of the main wake structure. However, the leeward downwash effect reduces the wake expansion height at $y/W < -0.15$ by nearly $0.09H$. These wake characteristics enhance wake dissipation, increasing the average total pressure coefficient of the main wake by approximately 30% to $C_{pT} \approx 0.46$.

At the P3 plane, the tilt angle of the upper wake region further increases to approximately 67° . The strong diagonal crossflow and resulting windward outwash increase the windward wake width by an additional estimated $0.04W$ over the P2 plane, leading to a total windward wake width increase of approximately 12% compared to the P1 plane. The leeward downwash reduces the wake expansion height at $y/W < -0.15$ by an additional $0.08H$ over the P2 plane, resulting in a total height reduction of approximately 33% relative to the P1 plane. Overall, the dissipation of pressure losses across the downstream planes increases the average total pressure coefficient of the main wake to $C_{pT} \approx 0.56$ on the P3 plane, representing an approximate 57% increase over the P1 plane.

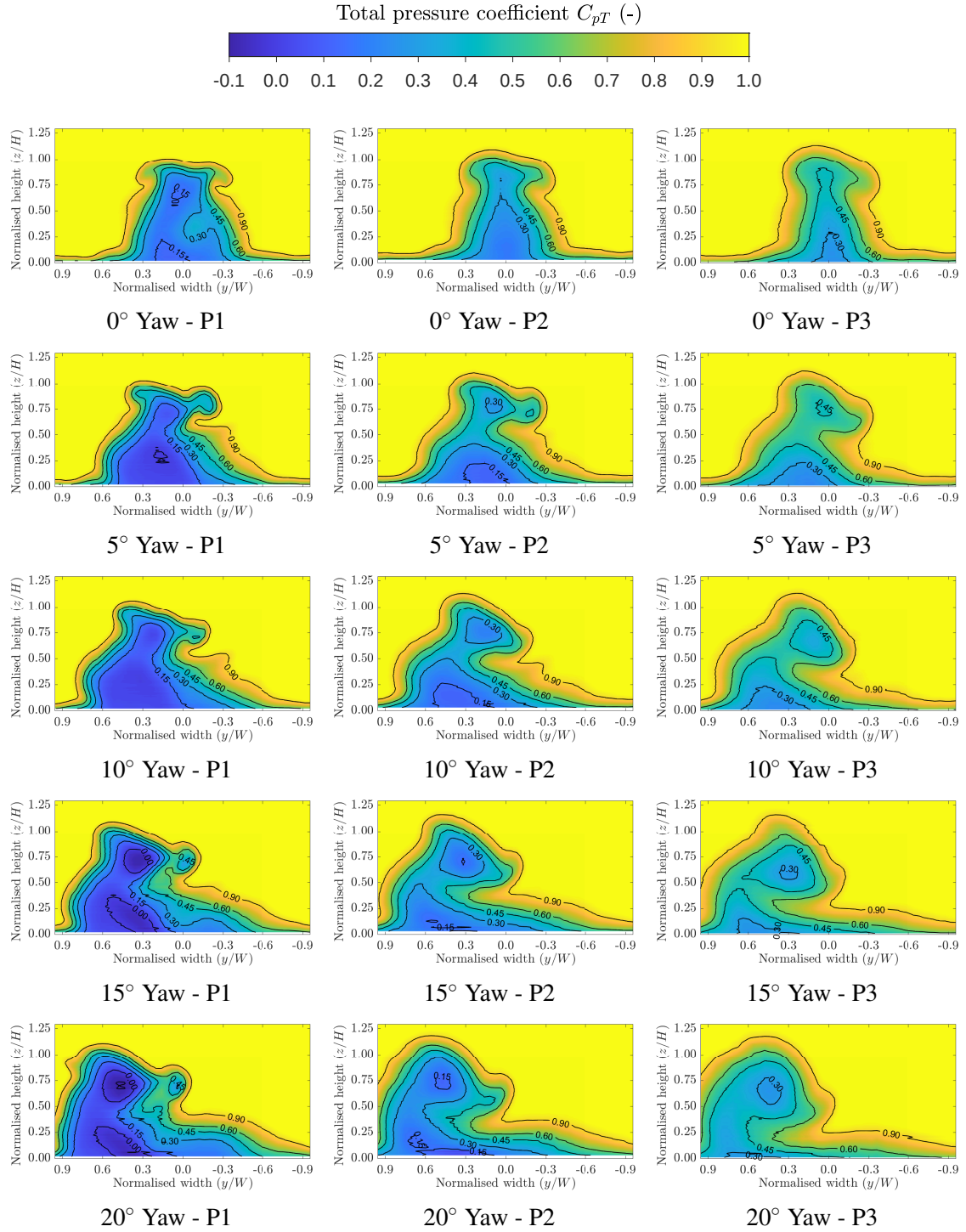


Figure 4.19: Experimental results of total pressure coefficient distributions on the P1, P2, and P3 wake planes behind the *DrivAer hp-F* with rear wing configuration.

4.5 Conclusion

This study investigated the effects of yaw conditions on the aerodynamic performance and flow field of high-performance vehicles through wind tunnel measurements of aerodynamic forces, slant surface pressures, and wake total pressures using the *DrivAer hp-F* model in the 8x6 Wind Tunnel at Cranfield University. Additionally, complementary CFD simulations were employed to support the interpretation of experimental data and facilitate a more comprehensive analysis of yaw-induced aerodynamic phenomena.

Aerodynamic force measurements on the standard, spoiler, and rear wing configurations of the *DrivAer hp-F* model demonstrated that, in straight-line conditions, the standard configuration generated slight front axle downforce and rear axle lift. The addition of a spoiler or rear wing significantly increased downforce, particularly at the rear axle. Across the yaw range, lift on the standard configuration increased nearly fourfold. The spoiler and rear wing configurations exhibited approximately 23% and 29% greater lift increases, respectively, due to their higher yaw sensitivity.

Yaw conditions induced a non-linear drag increase, with diminishing increments at higher yaw angles. The spoiler and rear wing configurations increased drag by 12% and 15%, respectively, compared to the standard configuration at 0° yaw. Across the yaw range, the standard configuration exhibited a 27% rise in drag, while the spoiler and rear wing configurations experienced approximately 31% and 17% lower drag penalties, respectively. Additionally, side force increased nearly linearly in the leeward direction, with the larger side profiles of the spoiler and rear wing contributing to a maximum side force increase of approximately 5% and 16% over the standard configuration at 20° yaw.

The vehicle's flow field and surface pressure distribution were also significantly affected by the yaw conditions. Wheel wakes propagated in the freestream direction, altering underbody flow and promoting additional outboard flow separation along the leeward side of the vehicle. Increased flow velocities around curvatures on the windward side, particularly along the roof and C-pillar, contributed substantially to lift augmentation across the yaw range. These flow field modifications also had a direct impact on the downstream pressure distribution on the slant, as confirmed by experimental surface pressure measurements.

Experimental surface pressure data indicated that the pressure recovery on the slant, along with the local pressure maximum, shifted in alignment with the freestream direction under yaw conditions. A low-pressure region developed at the top windward corner, while the average pressure across the slant decreased with increasing yaw. Furthermore, upstream pressurisation effects from the spoiler and rear wing significantly altered the pressure distribution on the slant, with these effects persisting up to the roof's trailing edge. The pressure distribution on the rear wing itself exhibited high sensitivity to yaw due to variations in vortical flow structures as well as flow obstruction from the endplates and rear wing support.

Total pressure measurements in the wake of the vehicle with rear wing configuration demonstrated that, in straight-line conditions, wake characteristics were primarily governed by diffuser upwash and vortical structures from the rear wing, generating a strong outboard inwash and central upwash effect. In yaw conditions, a dominant inboard-rotating vortex formed behind the windward side, increasing in strength and size with yaw angle. Meanwhile, on the leeward side, airflow rolled up into spanwise recirculating wake vortices, leading to substantial pressure reductions along the vehicle's base.

The interaction between the windward vehicle body vortex and the rear wing's vortical structures predominantly drove leeward inwash at small yaw angles. As yaw increased, the intensified circulation of the vehicle body vortex tilted the entire wake structure leeward, amplifying the inwash effect and generating a strong diagonal crossflow, which induced some windward outwash. The increased circulation also contributed to a leeward downwash over the wake expansion region. Collectively, these wake characteristics promoted enhanced wake mixing, accelerating the dissipation of pressure losses across the yaw range.

Overall, this study enhanced the understanding of yaw-induced effects on the aerodynamic performance and flow field of high-performance vehicles. The findings establish a strong foundation for future aerodynamic research into other real-world operating conditions and provide valuable wind tunnel data to expand the currently limited experimental database on high-performance vehicle aerodynamics.

Data Availability Statement

Data supporting this study are openly available from Cranfield Online Research Data.

- Experimental force data are available at: <https://dspace.lib.cranfield.ac.uk/handle/1826/22436>. (46)
- Experimental surface pressure data are available at: <https://dspace.lib.cranfield.ac.uk/handle/1826/23606>. (47)
- Experimental wake total pressure data are available at: <https://dspace.lib.cranfield.ac.uk/handle/1826/23609>. (48)
- CAD files for the *DrivAer hp-F* configurations are available at: <https://dspace.lib.cranfield.ac.uk/handle/1826/22433>. (49)

References

1. Bello-Millán FJ, Mäkelä T, Parras L, del Pino C, Ferrera C. Experimental study on Ahmed's body drag coefficient for different yaw angles. *Journal of Wind Engineering and Industrial Aerodynamics*. 2016;157:140-4. Available from: <https://doi.org/10.1016/j.jweia.2016.08.005>.
2. Nakamura Y, Nakashima T, Yan C, Shimizu K, Hiraoka T, Mutsuda H, et al. Identification of wake vortices in a simplified car model during significant aerodynamic drag increase under crosswind conditions. *Journal of Visualization*. 2022;25:983-97. Available from: <https://doi.org/10.1007/s12650-022-00837-8>.

3. The Hung T, Hijikuro M, Anyoji M, Uchida T, Nakashima T, Shimizu K. Deflector effect on flow behavior and drag of an Ahmed body under crosswind conditions. *Journal of Wind Engineering and Industrial Aerodynamics*. 2022;231:105238. Available from: <https://doi.org/10.1016/j.jweia.2022.105238>.
4. Grandemange M, Cadot O, Courbois A, Herbert V, Ricot D, Ruiz T, et al. A study of wake effects on the drag of Ahmed's squareback model at the industrial scale. *Journal of Wind Engineering and Industrial Aerodynamics*. 2015;145:282-91. Available from: <https://doi.org/10.1016/j.jweia.2015.03.004>.
5. Satheesh S, Cordier L, Kerhervé F, Spohn A. Yawed Ahmed body: impact of vertical flaps on flow structures. *Proceedings of the 12th International Symposium on Turbulence and Shear Flow Phenomena*. Osaka, Japan; 2022. Available from: <http://www.tsfp-conference.org/proceedings/2022/283.pdf>.
6. Cheng SY, Chin KY, Mansor S, Rahman ABA. Experimental study of yaw angle effect on the aerodynamic characteristics of a road vehicle fitted with a rear spoiler. *Journal of Wind Engineering and Industrial Aerodynamics*. 2019;184:305-12. Available from: <https://doi.org/10.1016/j.jweia.2018.11.033>.
7. Meile W, Ladinek T, Brenn G, Reppenhagen A, Fuchs A. Non-symmetric bi-stable flow around the Ahmed body. *International Journal of Heat and Fluid Flow*. 2016;57:34-47. Available from: <https://doi.org/10.1016/j.ijheatfluidflow.2015.11.002>.
8. Keogh J. The aerodynamic effects of the cornering flow conditions. PhD Thesis. University of New South Wales; 2016. Available from: <https://doi.org/10.26190/unsworks/19191>.
9. Viswanathan H. Aerodynamic performance of several passive vortex generator configurations on an Ahmed body subjected to yaw angles. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*. 2021;43:131. Available from: <https://doi.org/10.1007/s40430-021-02850-8>.
10. Booysen A, Das P, Ghaemi S. Large-scale 3D-PTV measurement of Ahmed-body wake in crossflow. *Experimental Thermal and Fluid Science*. 2022;132:110562. Available from: <https://doi.org/10.1016/j.expthermflusci.2021.110562>.
11. Tunay T, Firat E, Sahin B. Experimental investigation of the flow around a simplified ground vehicle under effects of the steady crosswind. *International Journal of Heat and Fluid Flow*. 2018;71:137-52. Available from: <https://doi.org/10.1016/j.ijheatfluidflow.2018.03.020>.
12. Lorite-Díez M, Jiménez-González JI, Pastur L, Cadot O, Martínez-Bazán C. Drag reduction on a three-dimensional blunt body with different rear cavities under cross-wind conditions. *Journal of Wind Engineering and Industrial Aerodynamics*. 2020;200:104145. Available from: <https://doi.org/10.1016/j.jweia.2020.104145>.

13. Nayeri CN, Wieser D, Schmidt HJ, Strangfeld C, Paschereit OC. Experiments on the influence of yaw on the aerodynamic behaviour of realistic car geometries. *Proceedings of The International Vehicle Aerodynamics Conference*. Woodhead Publishing; 2014. p. 43-51. Available from: <https://doi.org/10.1533/9780081002452.1.43>.
14. Yuan Z, Gu Z, Wang Y, Huang X. Numerical investigation for the influence of the car underbody on aerodynamic force and flow structure evolution in cross-wind. *Advances in Mechanical Engineering*. 2018;10(10). Available from: <https://doi.org/10.1177/1687814018797506>.
15. Strangfeld C, Wieser D, Schmidt H, Woszidlo R, Nayeri C, Paschereit C. Experimental study of baseline flow characteristics for the realistic car model DrivAer. *SAE Technical Paper 2013-01-1251*. 2013. Available from: <https://doi.org/10.4271/2013-01-1251>.
16. Wieser D, Schmidt HJ, Müller S, Strangfeld C, Nayeri C, Paschereit C. Experimental comparison of the aerodynamic behavior of fastback and notchback DrivAer models. *SAE International Journal of Passenger Cars — Mechanical Systems*. 2014;7(2):682-91. Available from: <https://doi.org/10.4271/2014-01-0613>.
17. Howell J. Aerodynamic drag of passenger cars at yaw. *SAE International Journal of Passenger Cars — Mechanical Systems*. 2015;8(1):306-16. Available from: <https://doi.org/10.4271/2015-01-1559>.
18. Wieser D, Nayeri CN, Paschereit CO. Wake structures and surface patterns of the DrivAer notchback car model under side wind conditions. *Energies*. 2020;13(2):320. Available from: <https://doi.org/10.3390/en13020320>.
19. Freeman CM, Gaylard AP. Correlation of CFD simulation for front side glass wall pressure fluctuations at yaw. *Proceedings of the 8th MIRA International Vehicle Aerodynamics Conference*; 2010. Available from: <https://www.researchgate.net/publication/245535402>.
20. Urquhart M. Vehicle wakes in side wind. PhD Thesis. Chalmers University of Technology; 2021. Available from: <https://research.chalmers.se/publication/526113>.
21. McAuliffe B, Sowmianarayanan B, Barber H. Near-to-far wake characteristics of road vehicles part 2: influence of cross winds and free-stream turbulence. *SAE International Journal of Advances and Current Practices in Mobility*. 2021;3(4):2009-24. Available from: <https://doi.org/10.4271/2021-01-0949>.
22. Rijns S, Teschner TR, Blackburn K, Brighton J. Integrated numerical and experimental workflow for high performance vehicle aerodynamics. *SAE Technical Paper 2024-01-5016*. 2024. Available from: <https://doi.org/10.4271/2024-01-5016>.
23. Rijns S, Teschner TR, Blackburn K, Brighton J. Aerodynamic analysis of the ride height dependency of a high-performance vehicle equipped with a multichannel diffuser in ground effect. *SAE Technical Paper 2023-01-5064*. 2023. Available from: <https://doi.org/10.4271/2023-01-5064>.

24. Szudarek M, Piechna J. CFD analysis of the influence of the front wing setup on a time attack sports car's aerodynamics. *Energies*. 2021;14(23):7907. Available from: <https://doi.org/10.3390/en14237907>.
25. Mathur A, Mahajan A, Aggarwal A, Mishra C, Roy A. Numerical study of swan neck rear wing for enhancing stability of ground vehicle bodies. *Emerging Trends in Mechanical Engineering. Lecture Notes in Mechanical Engineering*. Singapore: Springer; 2021. p. 199-208. Available from: https://doi.org/10.1007/978-981-15-8304-9_14.
26. Katz J, Largman R. Experimental study of the aerodynamic interaction between an enclosed-wheel racing-car and its rear wing. *Journal of Fluids Engineering*. 1989;111(2):154-9. Available from: <https://doi.org/10.1115/1.3243616>.
27. Garry KP, Le Good GM. An investigation of the sensitivity of rear wing orientation for saloon race cars. *SAE Technical Paper 2005-01-1018*. 2005. Available from: <https://doi.org/10.4271/2005-01-1018>.
28. Buljac A, Dzijan I, Korade I, Krizmanić S, Kozmar H. Automobile aerodynamics influenced by airfoil-shaped rear wing. *International Journal of Automotive Technology*. 2016;17(3):377-85. Available from: <https://doi.org/10.1007/s12239-016-0039-4>.
29. Keogh J, Doig G, Diasinos S, Barber T. The influence of cornering on the vortical wake structures of an inverted wing. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*. 2015;229(13):1817-29. Available from: <https://doi.org/10.1177/0954407015571673>.
30. Gogel D, Sakurai H. The effects of end plates on downforce in yaw. *SAE Technical Paper 2006-01-3647*. 2006. Available from: <https://doi.org/10.4271/2006-01-3647>.
31. Soares RF. Wake characterisation and simulation for ground vehicles. PhD Thesis. Cranfield University; 2020. Available from: <http://dspace.lib.cranfield.ac.uk/handle/1826/17293>.
32. McAuliffe B, Sowmianarayanan B, Barber H. Near-to-far wake characteristics of road vehicles part 1: influence of ground motion and vehicle shape. *SAE International Journal of Advances and Current Practices in Mobility*. 2021;3(4):2025-45. Available from: <https://doi.org/10.4271/2021-01-0957>.
33. Yang Y, Zheng L. Assessment of nonlinear-low reynolds number/detached eddy simulation turbulence model for wake flow field simulation of a realistic automotive model. *Physics of Fluids*. 2023;35(1):015133. Available from: <https://doi.org/10.1063/5.0129918>.
34. Su X, He K, Xu K, Gao G, Krajnović S. Comparison of flow characteristics behind squareback bluff-bodies with and without wheels. *Physics of Fluids*. 2023;35(3):035114. Available from: <https://doi.org/10.1063/5.0138305>.

35. Lai C, Zhu S, Feng S, Zhai G, Tan L, Obayashi S. Flow characteristics and wake topology of two-seat convertibles. *Physics of Fluids*. 2023;35(1):015144. Available from: <https://doi.org/10.1063/5.0134090>.
36. Watts M, Watkins S. Aerodynamic structure and development of Formula 1 racing car wakes. *SAE International Journal of Passenger Cars — Mechanical Systems*. 2014;7(3):1096-105. Available from: <https://doi.org/10.4271/2014-01-0600>.
37. Wilson MR, Dominy RG, Straker A. The aerodynamic characteristics of a race car wing operating in a wake. *SAE International Journal of Passenger Cars—Mechanical Systems*. 2009;1(1):552-9. Available from: <https://doi.org/10.4271/2008-01-0658>.
38. J Newbon J, R Dominy R, Sims-Williams D. CFD investigation of the effect of the salient flow features in the wake of a generic open-wheel race car. *SAE International Journal of Passenger Cars—Mechanical Systems*. 2015;8(1):217-32. Available from: <https://doi.org/10.4271/2015-01-1539>.
39. Newbon J, Sims-Williams D, Dominy R. Aerodynamic analysis of grand prix cars operating in wake flows. *SAE International Journal of Passenger Cars — Mechanical Systems*. 2017;10(1):318-29. Available from: <https://doi.org/10.4271/2017-01-1546>.
40. Heft A, Indinger T, Adams N. Introduction of a new realistic generic car model for aerodynamic investigations. *SAE Technical Paper 2012-01-0168*. 2012. Available from: <https://doi.org/10.4271/2012-01-0168>.
41. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model. *SAE Technical Paper 2018-01-0725*. 2018. Available from: <https://doi.org/10.4271/2018-01-0725>.
42. Lanfrit M. Best practice guidelines for handling automotive external aerodynamics with Fluent. University of Southampton; 2005. Available from: https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf. (Accessed: 01 May 2022).
43. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*. 1994;32(8):1598-605. Available from: <https://doi.org/10.2514/3.12149>.
44. Aultman M, Wang Z, Duan L. Effect of time-step size on flow around generic car models. *Journal of Wind Engineering and Industrial Aerodynamics*. 2021;219:104764. Available from: <https://doi.org/10.1016/j.jweia.2021.104764>.
45. Althaf AM, Bangaru YS, Fawcett R, Guled VM, Maloo TG, Gorgojo PN, et al. Numerical modeling and tunnel specific considerations for CFD model development of low-speed wind tunnels. *Proceedings of the AIAA AVIATION 2023 Forum*. AIAA-2023-3980. San Diego, USA: AIAA; 2023. Available from: <https://doi.org/10.2514/6.2023-3980>.

46. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Dataset DrivAer hp-F: force measurements in yaw conditions. Cranfield Online Research Data (CORD); 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/22436>.
47. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Dataset DrivAer hp-F: surface pressure measurements in yaw conditions. Cranfield Online Research Data (CORD); 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/23606>.
48. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Dataset DrivAer hp-F: wake total pressure measurements in yaw conditions. Cranfield Online Research Data (CORD); 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/23609>.
49. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. DrivAer hp-F: spoiler & rear wing configurations geometry pack. Cranfield Online Research Data (CORD); 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/22433>.

Chapter 5

Aerodynamic Analysis of Cornering Conditions

This chapter was originally published as:

Rijns S, Teschner TR, Blackburn K, Brighton J. Effects of cornering conditions on the aerodynamic characteristics of a high-performance vehicle and its rear wing. *Physics of Fluids*. 2024;36(4):045119. Available from: <https://doi.org/10.1063/5.0204204>.

Abstract

This study investigates the aerodynamic behaviour of a high-performance vehicle and the interaction with its rear wing in straight-line and steady-state cornering conditions. Reynolds-Averaged Navier-Stokes (RANS) based Computational Fluid Dynamics (CFD) simulations, employing a moving reference frame and overset mesh technique, are performed and validated against moving ground wind tunnel experiments. Results indicate an approximate 20% decrease in downforce and 35% increase in drag compared to straight-line conditions at the smallest considered corner radius of 2.9 car-lengths. Downforce losses primarily stem from performance deficits on the underbody and rear wing, alongside elevated upper body lift. Drag penalties mainly result from additional pressure drag induced by a recirculation wake vortex generated behind the vehicle's inboard side. The vehicle's lateral pressure distribution is also affected, introducing a centripetal force that increases with smaller corner radii. Additionally, analyses of the rear wing reveal alterations of its aerodynamic characteristics in cornering, particularly impacting vortical flow and suction on the lower surface. Throughout the operating conditions, the rear wing's individual downforce contribution diminishes beyond its stall angle. At higher angles of attack, the rear wing primarily generates downforce by pressurising the vehicle's upper surfaces, but its interaction with the near-wake leads to a substantially increased pressure drag. Overall, these findings provide crucial insights into the intricate aerodynamic interactions of high-performance vehicles in various cornering conditions, and form an essential foundation for future research on static and active aerodynamic designs in the pursuit of enhancing vehicle performance in dynamic driving scenarios.

Keywords: Aerodynamics; Cornering; CFD; RANS; Wind Tunnel; DrivAer

5.1 Introduction

Cornering conditions can have a substantial impact on the aerodynamic performance of ground vehicles. This is especially true for high-performance and racing vehicles, which are equipped with delicate aerodynamic elements that generate downforce to improve tractive forces. Yet, most aerodynamic design evaluations are conducted in straight-line conditions, leading to suboptimal vehicle designs in cornering conditions. The focus on straight-line conditions in vehicle aerodynamics research is primarily due to the challenges in replicating cornering conditions in wind tunnel experiments. Experimental techniques capable of recreating some aspects of cornering conditions exist, but they all have limitations in accuracy (1, 2). In contrast, numerical simulations offer complete control over operating conditions and are therefore preferred for cornering analyses.

Previous research on the influence of steady-state cornering on the aerodynamics of simplified automotive bodies has shown significant pressure distribution variations compared to straight-line conditions (3, 4, 5). Studies found that outboard pressure generally increased, whereas the inboard pressure decreased. The resultant centripetal force and outboard-directed yawing moment increased linearly with decreasing corner radii. Conversely, lift generation gradually decreased, while drag exhibited an exponential increase. Notably, the significant rise in drag was primarily attributed to heightened pressure drag, potentially linked to the formation of an asymmetric wake structure during cornering.

Research on the DrivAer model (6), a standard benchmark for automotive passenger vehicles, has demonstrated the significant impact of cornering conditions on its aerodynamic characteristics (7, 8, 9). In agreement with findings on simplified automotive bodies, these studies showed increased pressure on the vehicle's outboard side and decreased pressure on the inboard side compared to straight-line conditions. Additionally, an increase in centripetal force was observed during cornering, while the direction of the yawing moment was found to be influenced by the vehicle's sideslip angle, aligning with findings on dynamic vehicle motion (10, 11).

Studies on the DrivAer model also demonstrated an increase in drag and a slight reduction in lift as corner radii decreased. Drag primarily accumulated toward the vehicle's rear end, likely influenced by the reported asymmetric near-wake structure. Notably, elevated cornering sensitivity was observed for open-wheel race cars equipped with downforce-generating aerodynamic elements, as the performance of individual aerodynamic devices was also affected during cornering (12). This aligns with simulations of a Formula car navigating a sequence of corners, where downforce fluctuations of approximately 20% were reported (13).

Rear wings are commonly used aerodynamic elements on high-performance and racing vehicles due to their adaptability. Research on the effects of rear wings on simplified automotive vehicles in straight-line conditions has revealed not only their individual contributions to downforce and drag but also demonstrated secondary interactions with the vehicle and its near-wake, influencing overall aerodynamic performance (14, 15, 16). Analyses of open-wheel race cars have further shown that rear wing interactions with underbody airflow can enhance underbody downforce (17, 18).

Studies on aerodynamic elements have primarily focused on straight-line analyses. However, investigations on isolated inverted wings have indicated the significance of yaw and cornering flow conditions on their aerodynamic performance (19, 20). Additionally, the authors' previous research has highlighted the significant impact of yaw conditions on the aerodynamics of high-performance vehicles equipped with various aerodynamic elements (21). The findings showed that yaw not only affected individual component performance but also influenced interactions between aerodynamic elements and the vehicle body. Among the considered configurations, the rear wing was most effective but also exhibited the highest yaw sensitivity.

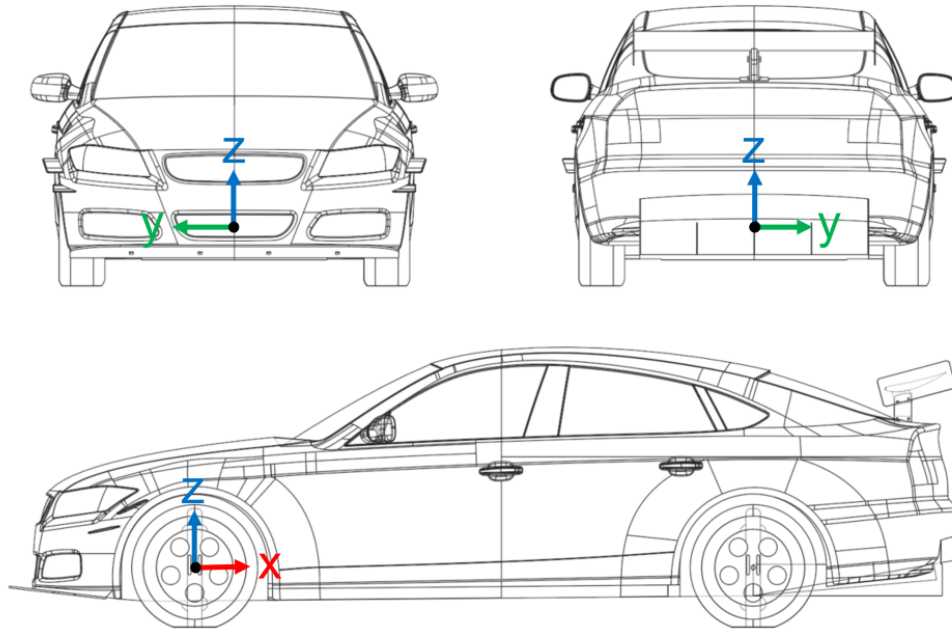
Overall, while some studies have underscored the crucial role of operating conditions on vehicle aerodynamics, most research primarily focused on straight-line conditions, leaving the aerodynamic effects in cornering conditions largely underexplored. Although studies on simplified automotive bodies and the DrivAer model have provided insights into certain flow field changes during cornering, research on high-performance vehicles with downforce-generating aerodynamic elements remains limited. Additionally, studies on isolated wings have suggested that yaw and cornering flow conditions significantly affect their aerodynamic performance, but little attention has been given to the aerodynamic interactions between rear wings and the vehicle body under cornering conditions.

This study addresses these gaps by analysing the aerodynamic characteristics of a high-performance vehicle and its interaction with a rear wing at varying angles of attack in both straight-line and cornering conditions. The investigation combines aerodynamic force measurements from moving ground experiments in the 8x6 Wind Tunnel at Cranfield University with numerical results from Reynolds-Averaged Navier-Stokes (RANS) based Computational Fluid Dynamics (CFD) simulations under varying operating conditions. The findings not only provide a foundation for future research on vehicles with static aerodynamic elements but also support the development of innovative active solutions with the potential to further enhance vehicle performance in dynamic operating conditions such as cornering (22, 23, 24, 25, 26).

5.2 Methodology

5.2.1 Vehicle Geometry

Vehicle aerodynamics research is commonly conducted on generic reference vehicle models such as the DrivAer model (6). This study uses the *DrivAer hp-F* model (27), a high-performance variant established on a 35% scale DrivAer Fastback configuration with smooth underbody and rotating wheels. The vehicle model is equipped with an additional 41 mm front bumper splitter, forebody strakes, a 10° underbody multichannel diffuser, and a NACA 6412 profile rear wing with a 0.11 m chord length and 0.42 m span. The *DrivAer hp-F* is illustrated in Figure 5.1 and its main dimensions are listed in Table 5.1.

Figure 5.1: Illustration of the *DrivAer hp-F* model.

5.2.2 Operating Conditions

This study analyses the vehicle's aerodynamic behaviour in straight-line and steady-state cornering conditions with radii of 5.75 and 2.9 car-lengths (L), corresponding respectively to relative airflow angles of 5° and 10° at the front of the vehicle. In the cornering conditions, the vehicle travels at a constant angular velocity ω (rad/s) about the corner's centre, as illustrated in Figure 5.2. The vehicle's position is assumed tangent to the curved path line through its centre, although this can vary substantially with driving styles (28).

The corner radius R (m) is defined as the distance between the centre of rotation (COR) and the centre of the vehicle. The velocity of the airflow relative to the vehicle, as well as the relative dynamic pressure, will increase with distance from the centre of rotation. Moreover, the angle of the relative airflow ψ ($^\circ$) will vary along the vehicle and can be expressed by Eq. (5.1). Here, x_c and y_c are the coordinates of the centre of rotation. This relationship provides a relative airflow angle at the front of the vehicle of $\psi \approx 5^\circ$ for the 5.75L radius corner and $\psi \approx 10^\circ$ for the 2.9L radius corner. To approximate steering angles, the front tyres match these respective relative airflow angles.

Table 5.1: Main dimensions of the *DrivAer hp-F* model.

Parameter	Symbol	Value	Unit
Car scale	CS	35	%
Length	L	1.6144	m
Width	W	0.6273	m
Height	H	0.4890	m
Ride height	h_0	0.0455	m
Area	A_{ref}	0.2647	m ²

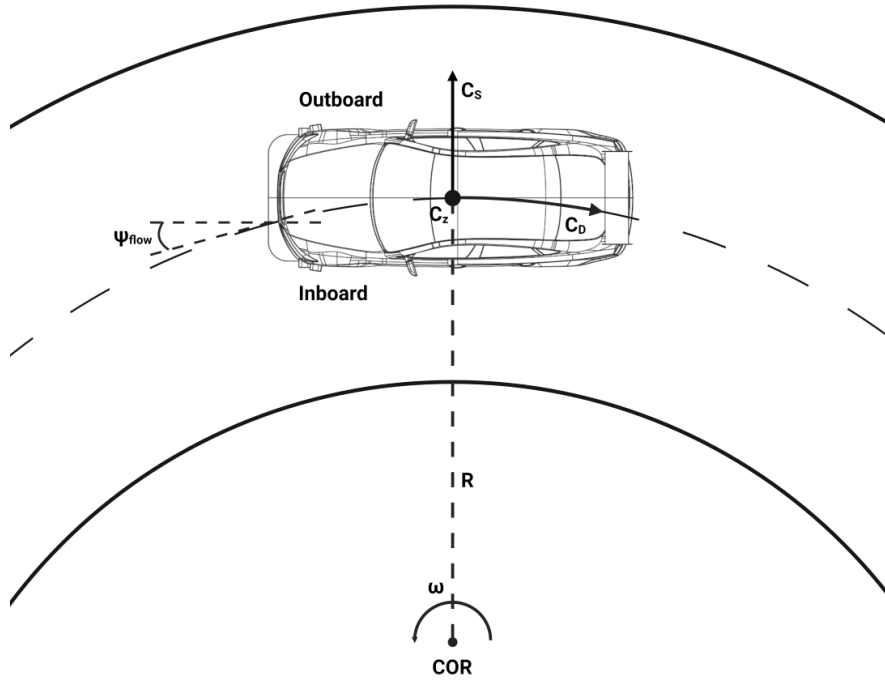


Figure 5.2: Illustration of a steady-state cornering condition with aerodynamic force directions.

$$\psi = \tan^{-1} \left(\frac{x - x_c}{y - y_c} \right) \quad (5.1)$$

The evaluation of aerodynamic lift and side forces remains in the z - and y -directions of the global coordinate system, respectively. However, aerodynamic drag is the resistance created by airflow in the direction of the vehicle's motion and therefore varies with the relative airflow angle along the vehicle during cornering, as detailed by Keogh et al. (1).

5.2.3 Wind Tunnel Experiments

Wind tunnel experiments on the *DrivAer hp-F* model are conducted in the closed return 8x6 Wind Tunnel at Cranfield University, illustrated in Figure 5.3. The wind tunnel has a 2.44 m by 1.83 m (8 ft x 6 ft) rectangular test section with corner fillets and is 5.18 m long. The test section is equipped with a 42 mm elevated moving ground facility to provide synchronous relative air and ground movement speed to prevent the formation of a ground plane boundary layer.

The vehicle model is suspended by an overhead strut system (OS) above a rolling belt (RB), which is 1.2 m wide and 2.75 m long. The wheels are independently supported by wheel supports (WS) and match the rolling belt speed. The facility is also equipped with a two-stage boundary layer suction system to optimise the ground plane boundary layer. The first stage system (BL1) consists of a scoop intake and is positioned 0.73 m upstream of the second stage system (BL2), which uses suction through a porous plate just upstream of the rolling belt system. The experimental setup for the *DrivAer hp-F* model using the moving ground facility provided a blockage ratio of approximately 9.6%.

Aerodynamic forces and moments are measured on the vehicle with varying rear wing angles of attack at a constant wind speed of 40 m/s, corresponding to a Reynolds number of 4.21×10^6 . The measurements are conducted with an Aerotech[®] six-component internal balance, positioned 800 mm downstream of the vehicle's nose and 120 mm above the vehicle's floor. The internal balance has a nominal accuracy of $\pm 0.06\%$ of the full scale measurement range, providing a measurement accuracy of ± 0.90 N for lift and ± 0.18 N for drag. The system does not measure forces on the independently supported wheels.

Data acquisition is conducted at a sampling rate of 200 Hz over a 10 s time interval. The constant fan rotor speed provides a wind speed variance of less than 0.1 m/s during each acquisition interval. Time-averaged quantities are obtained and converted into non-dimensional force coefficients using Eq. (5.2). The measured forces are indicated with F_i (N) and the non-dimensional force coefficients with C_i (-), where i is used to indicate lift (L) or drag (D). The freestream dynamic pressure measured near the test section inlet is indicated by P_{dyn} (Pa) and the vehicle model's reference area by A_{ref} (m²).

$$C_i = \frac{F_i}{P_{\text{dyn}} \cdot A_{\text{ref}}} \quad (5.2)$$

The data acquisition at each rear wing angle of attack is repeated three times to assess the repeatability of the experiments. The repeatability at each angle of attack is expressed as the maximum absolute deviation between a single measurement and the average of the three repeated runs at that angle. All deviations remain within a tolerance band of $|\Delta C_L| \approx 1.6 \times 10^{-3}$ for lift and $|\Delta C_D| \approx 0.9 \times 10^{-3}$ for drag, confirming a high level of experimental repeatability.

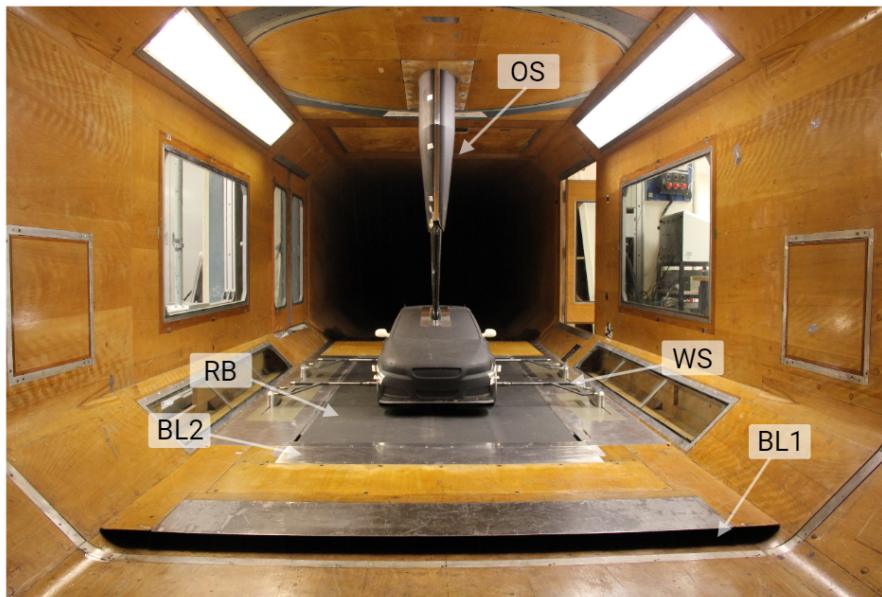


Figure 5.3: Moving ground setup in the 8x6 Wind Tunnel at Cranfield University.

5.2.4 Computational Fluid Dynamics Simulations

This study employs CFD simulations to analyse the aerodynamic behaviour of the *DriveAer hp-F* model with varying rear wing angles of attack in straight-line and cornering conditions within an open-road environment. Additional simulations replicating the experimental setup in the 8x6 Wind Tunnel at Cranfield University are performed, as Chapter 2 (29) demonstrated their value for validation and correlation practices. Both simulation sets are established on the same methodology but executed on different computational domains.

5.2.4.1 Domain and Boundary Conditions

The present study uses a rectangular domain with modified inlet and outlet positions in combination with a moving reference frame to simulate the straight-line and cornering operating conditions. Previous studies have indicated that this methodology improves the numerical workflow efficiency compared to using a curved computational domain, since it allows the investigation of various cornering conditions using only one domain (1, 4, 10, 11). This open-road computational domain, depicted in Figure 5.4, adheres to best practice guidelines for automotive aerodynamics simulations (30), with dimensions extending from 3L in front to 5L behind the vehicle. The side and top walls are positioned 1.5L away from the vehicle.

Cornering conditions are simulated with a rotational frame motion about an external point, which acts as the centre of rotation. Straight-line conditions are modelled with a constant translational frame motion to ensure consistency across all operating conditions. The side inlets and outlets are not required for straight-line conditions, which are therefore modelled as symmetry walls to prevent boundary layer growth. Additionally, the ground plane is prescribed to remain stationary relative to the moving reference frame in all operating conditions. Wheel rotation is modelled using moving wall boundary conditions, applying relative angular velocities to achieve the desired tangential velocity at each wheel.

An overview of the setup for each operating condition is provided in Table 5.2. All conditions are simulated at a (tangential) velocity of 40 m/s, corresponding to a Reynolds number of 4.29×10^6 . To replicate realistic road conditions, turbulence settings are based on on-road turbulence measurements for ground vehicles (31).

As previously stated, additional simulations are conducted on a numerical representation of the 8x6 Wind Tunnel at Cranfield University, as shown in Figure 5.5, to support validation and correlation. The computational domain extends 3L upstream and 5L downstream of the vehicle, resulting in a total length of 14904 mm. The cross-sectional dimensions match those of the physical wind tunnel, measuring 2438 mm in width and 1787 mm in height, with 431.05 mm long corner fillets.

The airflow conditions from the wind tunnel experiments are approximated using a velocity-inlet boundary condition, with a constant velocity of 40 m/s, a turbulent intensity of 0.05%, and a turbulent viscosity ratio of 10. Reference air properties are determined based on the averaged temperature and dynamic pressure recorded across all measurements, yielding a density of 1.178 kg/m^3 , a dynamic viscosity of $1.808 \times 10^{-5} \text{ kg/(m}\cdot\text{s)}$, and a temperature of 20°C.

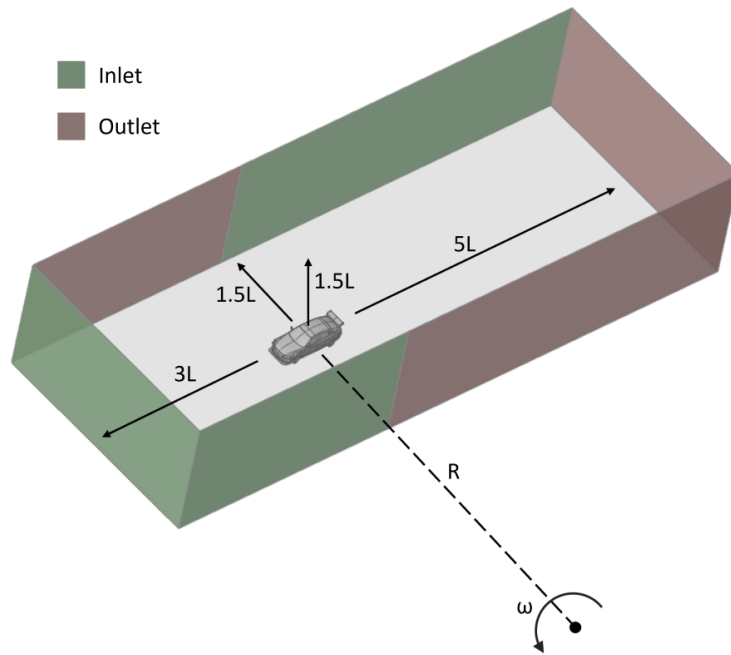


Figure 5.4: Illustration of the open-road computational domain with cornering boundary conditions.

To accurately represent the experimental setup, geometric features such as the overhead support system and wheel supports are incorporated, ensuring an approximate blockage ratio of 9.6%. The rolling belt is modelled using a translational moving wall condition of $v = 40$ m/s on the wall section depicted in black. Rotating wheels are modelled with a rotational moving wall condition of $\omega = 363.636$ rad/s to maintain consistency with the tangential velocity of the rolling belt. Lastly, the floor upstream of the two-stage boundary layer suction system, depicted in grey, is assigned a symmetry boundary condition to approximate the effects of boundary layer control.

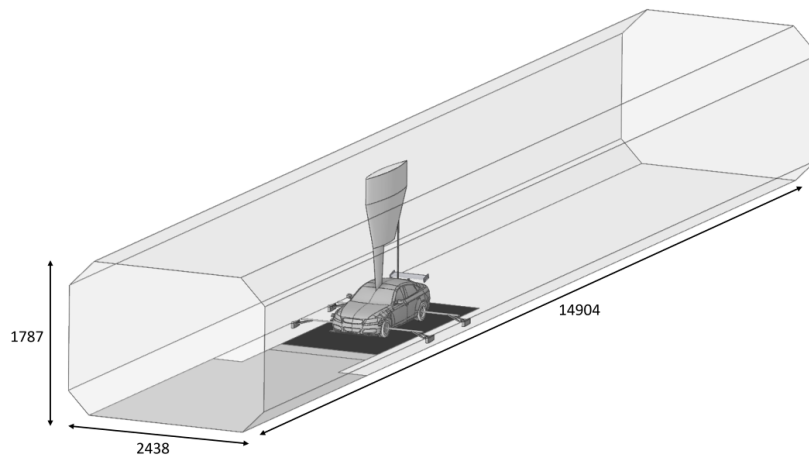


Figure 5.5: Illustration of the computational wind tunnel domain with dimensions in mm.

Table 5.2: Overview of the CFD simulation settings for each operating condition in the open-road domain.

Feature	Symbol	Straight-line	5.75L Radius corner	2.9L Radius corner
Impact flow angle	ψ	0°	5°	10°
Moving reference frame				
Frame motion		translational	rotational	rotational
Motion axis	x, y, z	1, 0, 0	0, 0, 1	0, 0, 1
Velocity	v / ω	40 m/s	4.309 rad/s	8.544 rad/s
Inlets & Outlets				
Velocity-inlet	v	0 m/s	0 m/s	0 m/s
Pressure-outlet	P	0 Pa	0 Pa	0 Pa
Turbulent intensity	I_T	4%	4%	4%
Turbulent length scale	l_T	2.5 m	2.5 m	2.5 m
Wheels rotation				
Rotation axis front	x, y, z	0, -1, 0	$\sin(\psi)$, -1, 0	$\sin(\psi)$, -1, 0
Rotation axis rear	x, y, z	0, -1, 0	0, -1, 0	0, -1, 0
Velocity inboard	ω	363.64 rad/s	353.73 rad/s	345.05 rad/s
Velocity outboard	ω	363.64 rad/s	374.55 rad/s	386.16 rad/s
Reference air properties				
Density	ρ	1.204 kg/m ³	1.204 kg/m ³	1.204 kg/m ³
Dynamic viscosity	μ	1.813e-5 kg/(m·s)	1.813e-5 kg/(m·s)	1.813e-5 kg/(m·s)
Temperature	T	20°C	20°C	20°C

5.2.4.2 Mesh

The computational domain is constructed with five unstructured poly-hexcore meshes using an overset mesh technique in ANSYS Fluent Meshing. This technique interpolates cell data in overlapping mesh regions, enabling targeted mesh transformation without the need to re-mesh the entire domain. This technique has demonstrated effectiveness in automotive applications (32) and is therefore deployed to improve the numerical workflow efficiency in this study, which involves the simulation of 48 vehicle configurations.

The overset mesh strategy employs four component mesh zones, illustrated in Figure 5.6, overlaid on a background mesh which covers the entire domain including refinement zones. The grey overset interface contains the vehicle model and rear wheels. The blue overset interfaces include the front wheels and enable steering input adjustments based on corner radius. The green overset interface confines the rear wing, facilitating targeted adjustments of its angle of attack.

The mesh settings are based on established guidelines from Chapter 2 (29), where grid convergence levels and near-wall modelling approaches for practical RANS simulations on high-performance vehicles were evaluated against experimental data and high-fidelity Delayed Detached Eddy Simulations (DDES) on a wall-resolved mesh with a $y^+ < 1$. Based on the Grid Convergence Index (GCI) results of $GCI_{32} = 0.47\%$ and $GCI_{21} = 0.04\%$ for lift coefficients, obtained from coarse (3), medium (2) and fine (1) grids, the study recommended the medium-resolution mesh strategy.

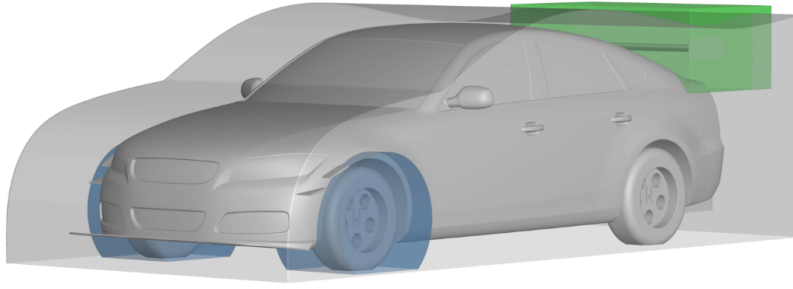


Figure 5.6: Illustration of the overset component mesh interfaces.

Accordingly, this study uses base element and vehicle surface element sizes of approximately 7.4% and 0.45% of the vehicle's length, respectively. Vehicle surface element sizes are locally reduced by over 80% for refinement on aerodynamic elements and small geometric features. The vehicle model is prepared with a medium near-wall treatment, targeting a $y^+ \approx 120$ -150 with around 4-6 inflation layers. Additionally, refinement zones with gradually increasing element sizes are implemented for increased resolution in the vehicle's near-field and wake region for each operating condition, as shown in Figure 5.7.

Overall, this mesh strategy yields a total cell count of approximately 15.1 million cells for the open-road computational domain, with zero orphan cells and 99% of the cells with an orthogonal quality above 0.66. The same strategy is applied to the wind tunnel computational domain, resulting in a total cell count of around 19.2 million cells with zero orphan cells and 99% of the cells with an orthogonal quality above 0.61.

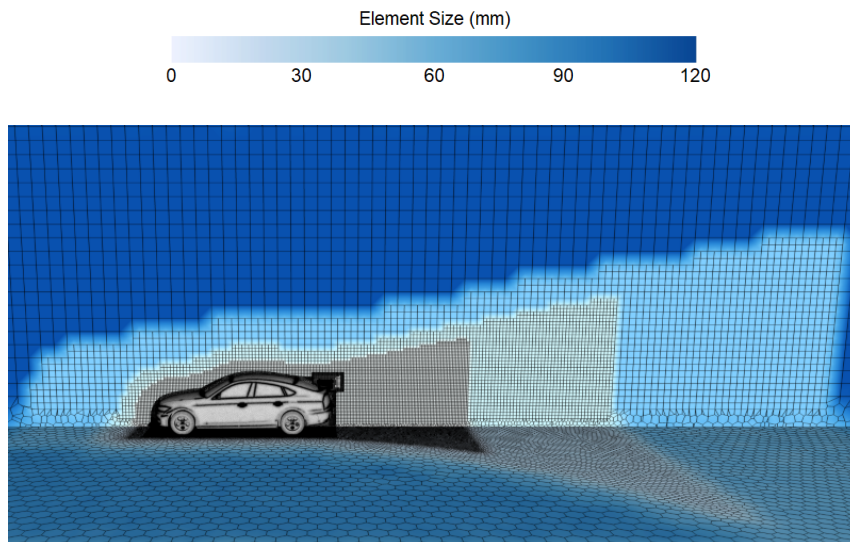


Figure 5.7: Contour plot of element sizes in proximity of the vehicle in the open-road domain, using a volume slice at $y = 0$ (symmetry).

5.2.4.3 Solve

Steady-state RANS simulations using the $k-\omega$ SST turbulence model (33) in ANSYS Fluent are conducted in this study. The $k-\omega$ SST model is selected for its proven cost-effectiveness for aerodynamic simulations on the *DrivAer hp-F* model in Chapter 2 (29), which assessed the predictive capabilities of various RANS models against high-fidelity DDES simulations and experimental data.

Along with the continuity and momentum equations, the transport equations for the turbulent kinetic energy and specific dissipation rate are solved. The setup follows the methodology outlined in Section 2.2.3.3, with pressure-velocity coupling handled using the coupled scheme, gradients computed via the least-squares cell-based method, and pressure discretised using a second-order scheme. The momentum and transport equations are solved using a second-order upwind scheme.

Initial trade-off efforts between simulation duration and statistical convergence of the vehicle's lift and drag coefficients determined a total duration of 750 iterations. Statistical convergence, defined as the point at which the moving mean of the monitored force coefficients stabilised, was typically reached within 400 iterations. The presented force coefficients are therefore averaged over the last 250 iterations. The simulations were performed on two AMD EPYC 7543 processors using a total of 64 cores. On average, the simulations required approximately 5 hours and 30 minutes of computation time on the open-road domain, and around 4 hours on the wind tunnel domain.

5.2.4.4 Post-Processing

Throughout the analyses in this study, numerical results of aerodynamic force coefficients, static pressure coefficients and total pressure coefficients are used. The force coefficients are expressed by Eq. (5.2), and use a dynamic pressure based on the reference air density ρ (kg/m³) and velocity v (m/s), as described in Eq. (5.3).

$$P_{\text{dyn}} = \frac{1}{2} \cdot \rho \cdot v^2 \quad (5.3)$$

Static pressure coefficients C_p (Pa) and total pressure coefficients C_{pT} (Pa) are expressed by Eq. (5.4), where P_s (Pa) represents the static pressure, P_t (Pa) the total pressure, and P_{op} (Pa) the operating pressure of 101325 Pa.

$$C_p = \frac{P_s - P_{\text{op}}}{P_{\text{dyn}}} \quad \text{and} \quad C_{pT} = \frac{P_t - P_{\text{op}}}{P_{\text{dyn}}} \quad (5.4)$$

5.3 Validation

This section validates the numerical methodology against experimental data and analyses differences between wind tunnel and open-road conditions for correlation. Validation is based on agreement between force data on the vehicle body from the experiments (Exp) and simulations on the computational wind tunnel domain (WT), presented in Figure 5.8, whereas detailed analyses of the results will be provided in Section 5.4.

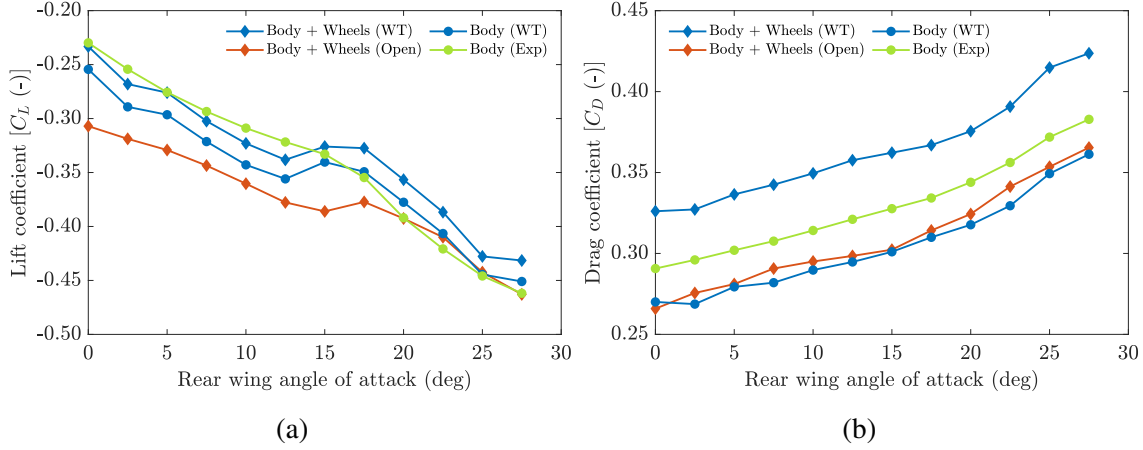


Figure 5.8: Comparison of (a) lift and (b) drag coefficients from experiments (Exp), wind tunnel simulations (WT), and open-road simulations (Open).

The simulations overpredict downforce by $\Delta C_L \approx -0.028$ (+10%) in the 0° – 12.5° angle of attack range, with an average relative deviation of $|\Delta C_L| \approx 0.004$ (2%). Beyond the rear wing's stall angle of approximately 12.5° , most downforce from the rear wing is generated by its interaction with the vehicle body. While some discrepancies are present in the transition just beyond the stall angle, the simulations provide a steady downforce underprediction of $\Delta C_L \approx 0.009$ (-2%) along the 17.5° – 27.5° range, with an average relative deviation of $|\Delta C_L| \approx 0.005$ (1%). Furthermore, the simulations underpredict drag on average by $\Delta C_D \approx -0.025$ (-8%), with an average relative deviation of $|\Delta C_D| \approx 0.002$ (<1%) across the angle of attack range.

Aerodynamic forces on the wheels were excluded in the wind tunnel experiments, but the simulation results from the computational wind tunnel domain indicate that their inclusion reduces total downforce by $\Delta C_L \approx 0.019$ (-6%) and increases total drag by $\Delta C_D \approx 0.06$ (+20%). The inclusion of wheel forces manifests as nearly invariant offsets, with average relative deviations of < 1% for both forces across the angle of attack range, suggesting they do not influence experimental force trend analyses in this study.

The simulated force data from the open-road and wind tunnel domains show expected absolute force offsets attributable to wind tunnel conditions such as blockage and interference effects. However, there are also noticeable relative gradient variations, particularly across the 17.5° – 27.5° angle of attack range. Figure 5.9 illustrates that these force offsets are primarily influenced by reductions in upper surface pressure associated with locally increased airflow velocities in the wind tunnel domain, consistent with prior research (29). Moreover, the rate at which the rear wing's pressurisation affects the vehicle's upper surfaces is shown to be influenced, contributing to the variations in force trend gradients.

Overall, strong agreement between numerical and experimental force trends provides confidence in the accuracy of the numerical methodology. The correlation results provide insights into the effects of wheel forces and experimental conditions on the vehicle's aerodynamic characteristics, supporting interpretation of the experimental data.

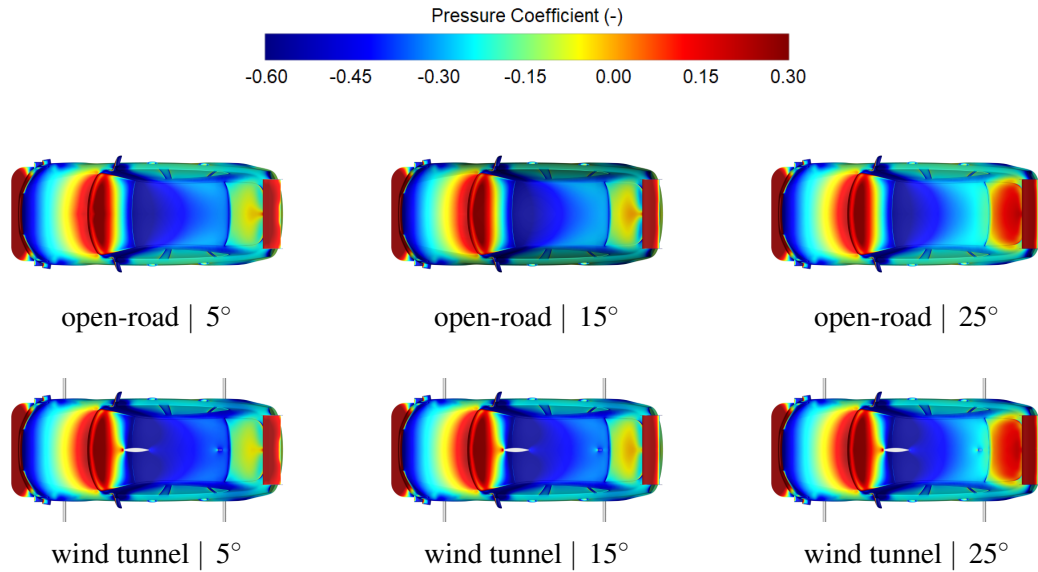


Figure 5.9: Comparison of pressure coefficient distribution on the vehicle between open-road and wind tunnel simulations at varying rear wing angles of attack. The overhead balance system in the wind tunnel domain is omitted for clarity.

5.4 Results and Discussion

This section analyses the aerodynamic behaviour of the vehicle with various rear wing angle of attack settings in the straight-line, 5.75L radius corner, and 2.9L radius corner conditions. Initially, the overarching influence of these operating conditions on downforce and drag performance is discussed. Subsequently, detailed analyses of surface pressure and wake characteristics are provided to further clarify the observed force trends. Finally, aerodynamic characteristics and interactions involving the rear wing are investigated. Throughout the presentation of results, left-handed corners are considered, with the outboard side referring to the vehicle's right side. Figure descriptions provide additional clarification where necessary, ensuring the correct interpretation of presented views.

5.4.1 Aerodynamic Forces

The downforce performance of the entire vehicle and the rear wing individually are presented in Figure 5.10. In straight-line conditions, the vehicle's downforce generally increases at a rate of $\Delta C_L \approx -0.014$ per 2.5° angle of attack increment within the initial 0° – 12.5° downforce enhancement region. Following a minor force plateau between 12.5° and 17.5° , the downforce gain intensifies by approximately 47%, reaching $\Delta C_L \approx -0.020$ per 2.5° increment in the subsequent 17.5° – 27.5° downforce enhancement region. Downforce generated by the rear wing significantly contributes to the vehicle's initial total downforce enhancement region, but rapidly decreases after its stall angle of approximately 12.5° . However, the vehicle's total downforce continues to increase, suggesting the rear wing has additional effects on the vehicle's aerodynamic performance, which will be explored later.

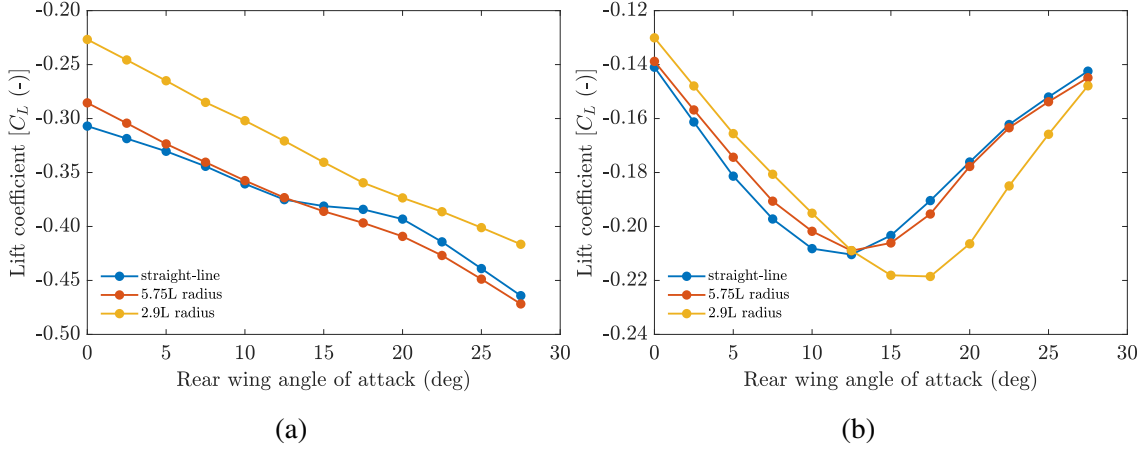


Figure 5.10: Lift coefficients for (a) the entire vehicle and (b) the rear wing as a function of angle of attack in straight-line and cornering conditions.

In the 5.75L radius corner condition, the vehicle experiences a drop of about $\Delta C_L \approx 0.022$ (-7%) in downforce performance at low angles of attack compared to straight-line conditions. However, at higher angles of attack, downforce losses diminish and the vehicle generates on average $\Delta C_L \approx -0.012$ (+3%) more downforce than in straight-line conditions. The rear wing exhibits a noticeable corner sensitivity, with a $\Delta C_L \approx 0.007$ (-4%) drop in downforce performance before its stall angle. The corner sensitivity diminishes beyond the stall angle, where downforce performance is primarily influenced by the pressure build-up on the rear wing's upper surface.

In the 2.9L radius corner condition, the vehicle's total downforce within the 0° – 12.5° range decreases by approximately $\Delta C_L \approx 0.065$ (-19%) compared to the straight-line condition and by $\Delta C_L \approx 0.050$ (-14%) compared to the 5.75L radius corner. Along the 17.5° – 27.5° range, the downforce performance depicts a reduced sensitivity to angle of attack increments. The rear wing's individual downforce performance decreases by about $\Delta C_L \approx 0.016$ (-9%) compared to the straight-line condition and by $\Delta C_L \approx 0.009$ (-5%) compared to the 5.75L radius corner condition. However, its stall angle is delayed by approximately 5° , which consequently also defers the downforce drop-off at higher angles.

Regarding the vehicle's drag performance, presented in Figure 5.11, total drag in straight-line conditions increases by $\Delta C_D \approx 0.006$ per 2.5° angle of attack increment within the 0° – 12.5° range. This drag penalty grows by over 90% to $\Delta C_D \approx 0.012$ at higher angles of attack. The rear wing's drag production increases at an increasing rate throughout the angle of attack range. Yet, its contribution to the vehicle's total drag penalty only reaches about 70%. This indicates that the rear wing has additional interactions with the vehicle's flow field that produce drag, which will be investigated later.

In the 5.75L radius corner condition, the vehicle generates on average $\Delta C_D \approx 0.057$ (+19%) more drag compared to straight-line conditions. The initial $\Delta C_D \approx 0.012$ drag penalty per 2.5° angle of attack increment within the 0° – 10° range reduces by about 32% to $\Delta C_D \approx 0.008$ at higher angles of attack after a small force plateau. The rear wing's drag production has steadily increased by approximately $\Delta C_D \approx 0.036$ throughout the angle of attack range relative to straight-line conditions.

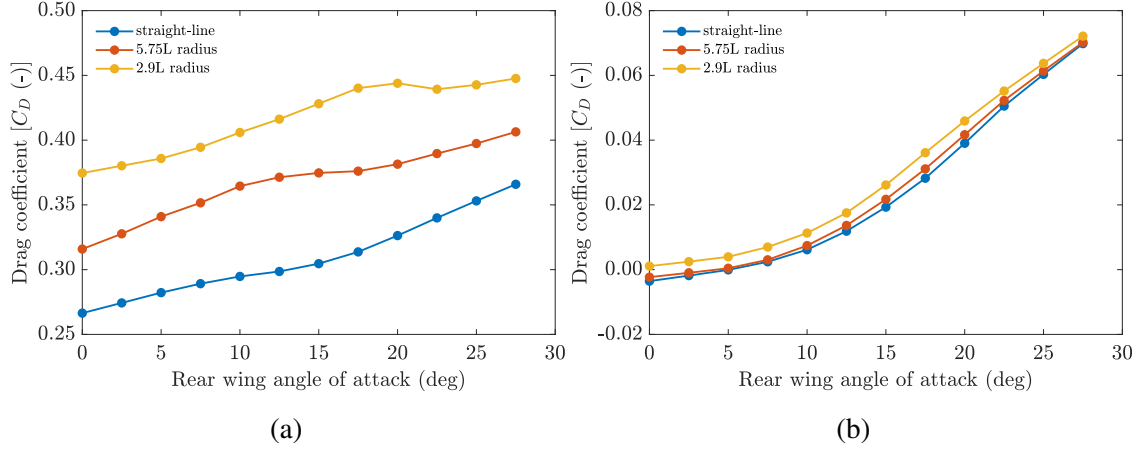


Figure 5.11: Drag coefficients for (a) the entire vehicle and (b) the rear wing as a function of angle of attack in straight-line and cornering conditions.

In the 2.9L radius corner condition, the vehicle's total drag increases non-linearly within the 0° – 17.5° range and subsequently plateaus at higher angles of attack. On average, total drag increases by approximately $\Delta C_D \approx 0.108$ (+35%) compared to straight-line conditions and by $\Delta C_D \approx 0.050$ (+14%) compared to the 5.75L radius corner condition. The rear wing's drag production also increases by around $\Delta C_D \approx 0.007$ compared to straight-line conditions and by $\Delta C_D \approx 0.004$ compared to the 5.75L radius corner condition.

5.4.2 Vehicle Aerodynamics

This section analyses the vehicle's aerodynamic characteristics across each operating condition to elaborate on the observed force coefficient trends. A comparison of surface pressure distributions is provided in Figures 5.12 to 5.14. In straight-line conditions, a stagnation zone forms at the nose, followed by a low-pressure region as airflow accelerates along its curvature. Pressure recovers along the bonnet toward a high-pressure region at the windscreen, with additional low-pressure areas along the A-pillars and roof's leading edge. Pressure recovery along the roof is gradual, but becomes steeper along the slant due to upstream pressurisation from the rear wing.

Cornering conditions cause the high-pressure regions on the vehicle's nose and windscreen to concentrate progressively toward the inboard side with decreasing radii. The pressure across the A-pillars and roof's leading edge increases gradually on the outboard side but decreases on the inboard side. Conversely, as the angle of the relative airflow reverses downstream of the vehicle's centre, pressure along the rooflines and C-pillars decreases on the outboard side and increases on the inboard side. Additionally, the increasing relative airflow angle at the rear of the vehicle shifts the pressure maximum along the slant progressively outboard.

The pressure distribution on the vehicle's underbody in straight-line conditions indicates the formation of a separation bubble behind the splitter's sharp leading edge, accompanied by high-pressure regions on either side due to airflow interaction with the inclined splitter surface. Ground clearance is at a minimum between the front wheels, providing a strong venturi effect, evident from the significant low-pressure region around the front axle.

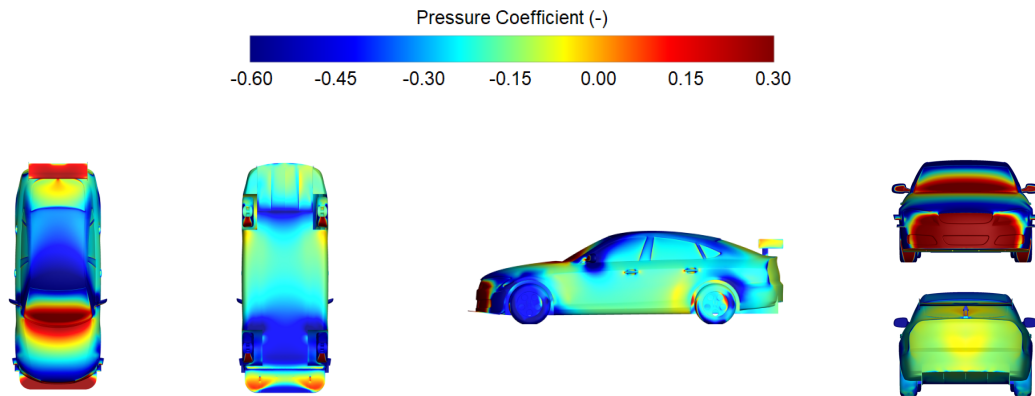


Figure 5.12: Pressure coefficient distribution on the vehicle with a 0° rear wing angle of attack in straight-line conditions.

The pressure recovers along the centre of the underbody, before airflow is accelerated toward the diffuser inlet, generating a strong low-pressure region. The diffuser provides a gradual pressure recovery, with localised low-pressure concentrations resulting from vortex flow near the separators between the diffuser channels.

More airflow interacts with the inboard side of the splitter in cornering conditions, resulting in a localised increase in pressure. The curved path also causes the splitter's separation bubble to shift with the relative airflow direction. In the 5.75L radius corner condition, a modest inward propagation of the inboard front wheel is evident alongside the low-pressure region around the front axle. This wheel wake becomes more prominent in the 2.9L radius corner condition and the inboard front wheel itself starts to impose interference effects, hindering front axle downforce generation.

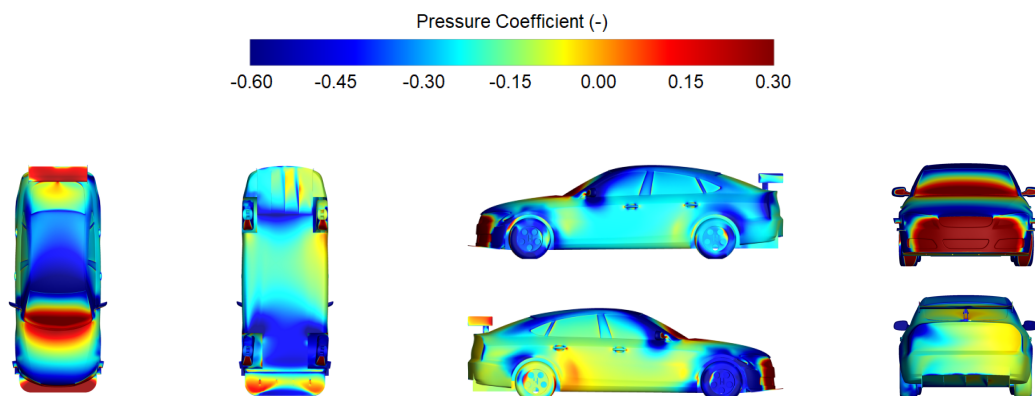


Figure 5.13: Pressure coefficient distribution on the vehicle with a 0° rear wing angle of attack in a 5.75L radius corner.

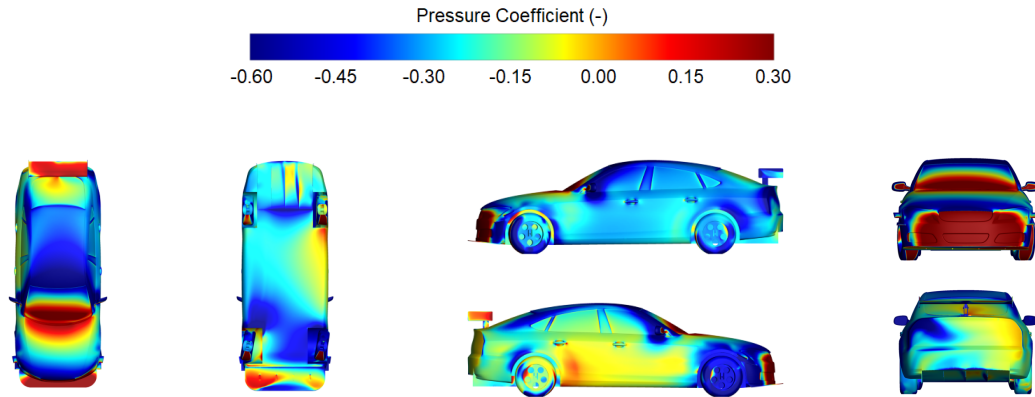


Figure 5.14: Pressure coefficient distribution on the vehicle with a 0° rear wing angle of attack in a 2.9L radius corner.

Downstream at the underbody, cornering conditions lead to a progressive pressure decrease ahead of the inboard rear wheel and a pressure increase ahead of the outboard rear wheel. In the 5.75L radius corner, there is a noticeable pressure rise on the outboard side of the diffuser, attributed to interference from the outboard front and rear wheel wakes extending into the diffuser. In the 2.9L radius corner, the larger steering input and path curvature prevent the outboard front wheel wake from reaching the diffuser, resulting in a more prominent low-pressure region at the outboard side of the diffuser inlet.

However, the curved relative airflow still induces the outboard rear wheel wake across the diffuser, causing downforce losses. Additionally, cornering conditions enhance vortex formation at the diffuser's separators, as indicated by increased low-pressure concentrations in the diffuser channels. Management of the inboard rear wheel wake is also improved, evident from the reduced pressure on the inboard side of the diffuser compared to the straight-line condition.

The vehicle's side panels in straight-line conditions exhibit low-pressure regions extending from the nose to the front wheels, with additional low-pressure regions along the A-pillars, rooflines and C-pillars. The pressure remains relatively consistent along the doors and windows, but reduces around the rear quarter panels. The low-pressure concentrations on the downstream edges of the rear quarter panels are caused by airflow rolling into recirculation zones behind the vehicle, also evident from the associated low-pressure regions on either side of the vehicle's base.

The pressure across the outboard door panels and windows gradually increases in cornering conditions with reducing radii, while the pressure on the outboard roofline and C-pillar noticeably decreases. Conversely, the pressure across the inboard door panels and windows progressively decreases, while the pressure on the inboard C-pillar slightly increases. The inboard rear quarter panel experiences a significant pressure reduction compared to straight-line conditions, especially around its downstream edge. Overall, these variations induce an inboard acting side force of $C_S \approx 0.107$ in the 5.75L radius corner condition and $C_S \approx 0.138$ in the 2.9L radius corner condition.

The pressure across the inboard side of the vehicle's base reduces significantly in cornering conditions, leading to increased pressure drag. This pressure reduction is associated with the intensified formation of inboard recirculation wake vortices, also suggested by the previously identified reduced pressure across the inboard rear quarter panel. These low-pressure traces extend further inward with decreasing corner radii due to enhanced wake circulation promoted at high relative airflow curvatures.

Investigation of the wake structures, presented in Figure 5.15, reveals substantial differences between straight-line and cornering conditions. In the straight-line condition, a distinct recirculation zone forms behind the vehicle's body, accompanied by rear wheel wakes. In cornering conditions, the wake becomes increasingly asymmetric with decreasing radii. The inboard rear wheel wake slightly reduces in size and shifts outward, lessening its impact on the diffuser as previously observed. Conversely, the outboard rear wheel wake significantly enlarges and extends inward, leading to downforce losses on the diffuser's outboard side.

The recirculation wake vortex structure originating behind the inboard rear quarter panel is evident in the upper-wake region in cornering conditions. This structure initially develops predominantly in the spanwise direction near the vehicle's base, but subsequently merges with higher energy airflow from the vehicle's outboard side, forming a dominant streamwise wake vortex. The circulation from this wake structure creates an inboard inwash of high-energy freestream airflow, aiding in the dissipation of pressure losses.

A closer examination of the wake characteristics in cornering conditions, depicted in Figure 5.16, indicates that the strength of the recirculation wake vortex intensifies with decreasing corner radii. This is evident from the concentration of lower total pressure behind the inboard side of the vehicle's base at the $z = 210$ mm plane. Smaller corner radii also position the subsequent streamwise wake vortex structure further toward the inboard side, shown by the location of the downstream low total pressure traces. Overall, the magnitude and downstream development of pressure losses on the $z = 210$ mm plane are most pronounced for smaller corner radii.

At the $z = 130$ mm plane, a notable inboard inwash effect from the streamwise wake vortex is observed. The resultant crossflow from the inboard to the outboard side of the wake strengthens with decreasing corner radii. This inwash effect energises the wake with high-energy freestream airflow and accordingly mitigates some of the pressure losses from the outboard rear wheel wake at this plane.

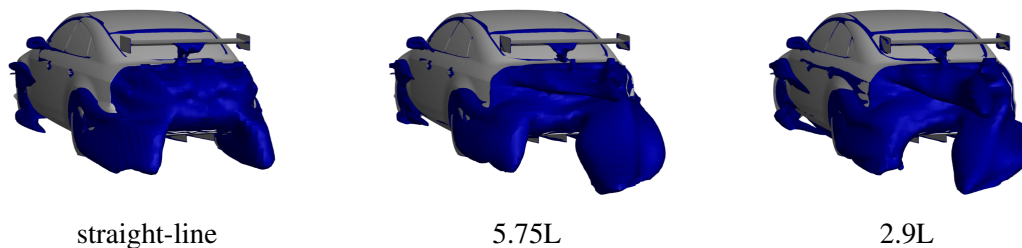


Figure 5.15: Wake structure visualisations using iso-surfaces of zero total pressure for the vehicle with a 0° rear wing angle of attack in straight-line and cornering conditions.

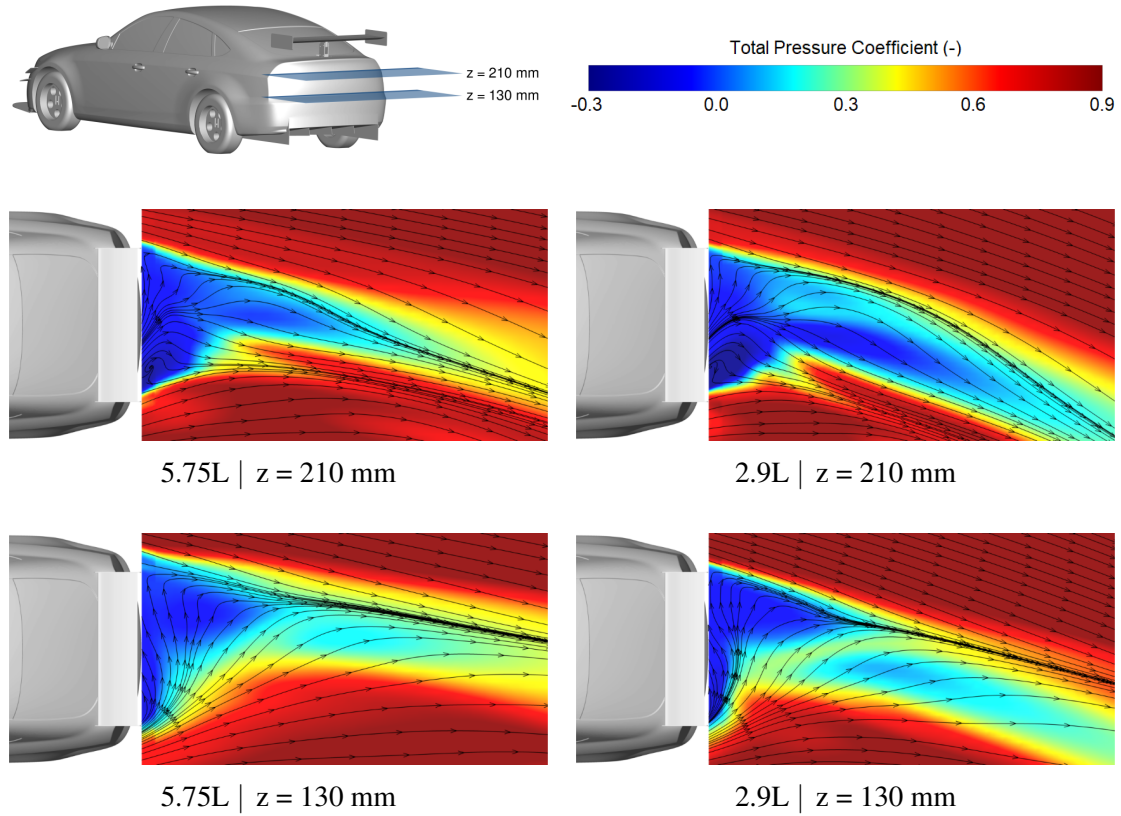


Figure 5.16: Total pressure coefficient distribution with streamlines on wake planes behind the vehicle in cornering conditions. Wake plane positions are shown in the top left corner.

5.4.3 Rear Wing Aerodynamics

This section investigates the aerodynamic characteristics of the rear wing across a wide angle of attack range in each operating condition. The effects of operating conditions on the vortical flow are first considered to support subsequent analyses of surface pressure data. Pressure differentials across the endplates drive vortical flow at the rear wing's upper and lower surfaces, as shown in Figure 5.17. Vortices labelled O_U and I_U form on the outboard and inboard sides of the upper surface, respectively. Similarly, vortices labelled O_L and I_L form on the outboard and inboard sides of the lower surface, respectively.

In straight-line conditions, the pressure differential induces outward-rotating O_U and I_U vortices, and inward-rotating O_L and I_L vortices. A larger pressure differential is present at the lower surface, creating more pronounced O_L and I_L vortices compared to the O_U and I_U vortices at the depicted 0° angle of attack.

Cornering conditions create a pressure build-up on the outside of the outboard endplate, which increases with decreasing corner radii. The altered pressure differential induces an inward-rotating O_U vortex and enhances the O_L vortex. The pressure on the inside of the inboard endplate is also increased, strengthening the I_U vortex. Additionally, the pressure differential driving the I_L vortex is reduced due to the low-pressure region emerging on the inboard rear quarter panel in cornering conditions.

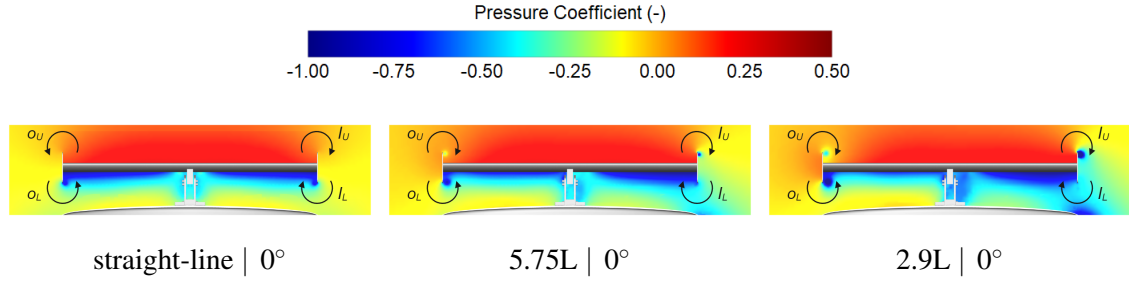


Figure 5.17: Pressure coefficient distribution on a frontal plane ($x = 1.25$ m) at the rear wing with a 0° angle of attack in straight-line and cornering conditions, with labelled vortices. The left side represents the outboard side in cornering.

The pressure distribution on the rear wing's upper surface, illustrated in Figure 5.18, demonstrates a high-pressure region behind the leading edge that gradually diminishes toward the trailing edge. In straight-line conditions, the distribution is symmetrical and pressure increases with angle of attack. However, local pressure reductions occur near the endplates due to airflow spillage from the O_U and I_U vortices.

In cornering conditions, a noticeable pressure reduction is initially observed near the outboard endplate at low angles of attack compared to straight-line conditions, attributed to the inward rotation of the O_U vortex. As the angle of attack increases, the pressure rise across the upper surface mitigates the inward rotation of the O_U vortex and its associated pressure reduction. Nonetheless, flow obstruction from the outboard endplate continues to cause relatively lower pressures behind it at higher angles of attack.

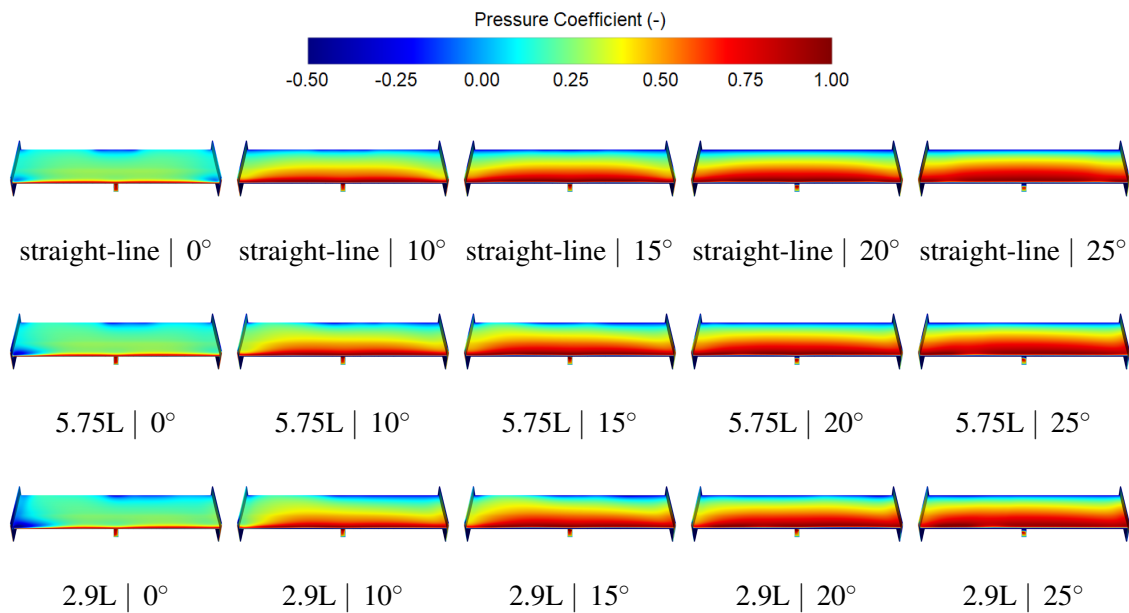


Figure 5.18: Pressure coefficient distribution on the upper surface of the rear wing at various angles of attack in straight-line and cornering conditions. The left side represents the outboard side in cornering.

The rear wing's lower surface exhibits a significant low-pressure region near the leading edge curvature which gradually recovers toward the trailing edge, as depicted in Figure 5.19. In straight-line conditions, the low-pressure region near the leading edge curvature is initially most substantial across the centre section at 0° , indicating a locally increased effective angle of attack due to downwards induced airflow along the vehicle's body.

Suction near the leading edge, as well as the pressure recovery gradient, increases with angle of attack up to the stall angle around 12.5° - 15° , beyond which severe flow separation negatively impacts downforce performance. The pressure recovery remains most gradual near the endplates, attributed to the low pressure generated by the O_L and I_L vortices, which is observable even beyond the stall angle.

In cornering conditions, suction near the leading edge curvature initially increases on the inboard side but decreases on the outboard side at 0° , indicating a favourable inboard relative airflow alignment. However, suction across the outboard side noticeably increases at higher angles of attack, while suction on the inboard side starts to decay due to interference from the rear wing support.

The increased suction is attributed to the enhanced interaction of the rear wing with faster moving outboard relative airflow as its leading edge nears the vehicle body, along with the increasing strength and flow inducement effects of the O_L vortex at higher angles of attack. Additionally, the strengthening of the O_L vortex and suppression of the I_L vortex with reducing corner radii are evident from the dominant low-pressure traces near the outboard endplate and absence of low-pressure traces near the inboard endplate across the angle of attack range.

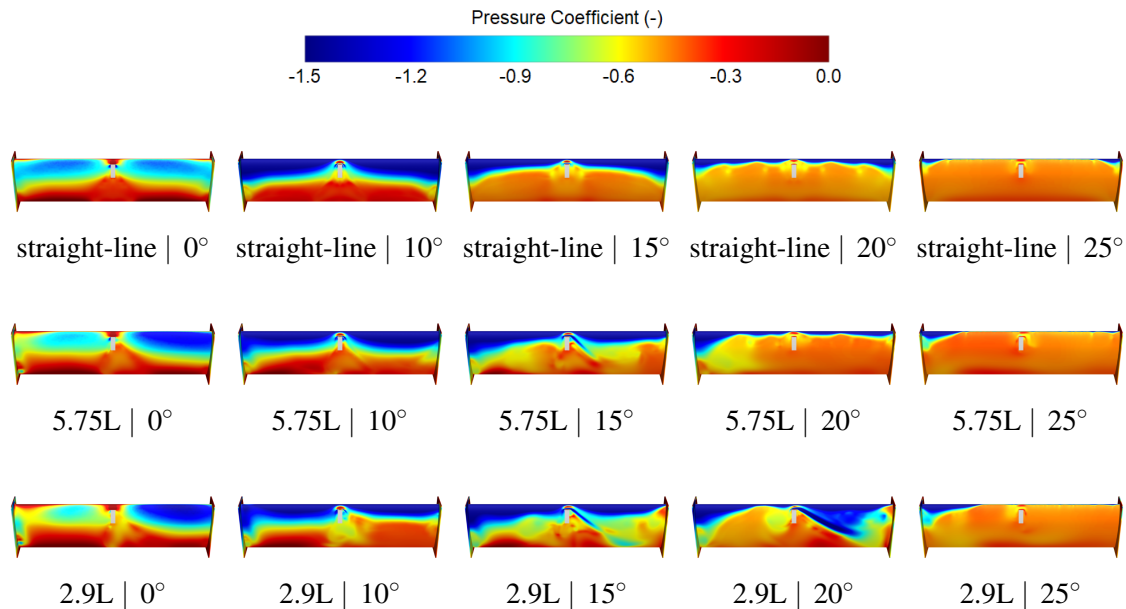


Figure 5.19: Pressure coefficient distribution on the lower surface of the rear wing at various angles of attack in straight-line and cornering conditions. The left side represents the outboard side in cornering.

5.4.4 Rear Wing Interaction

This section investigates the aerodynamic interaction between the rear wing and the vehicle's flow field in each operating condition, building upon previous discussions highlighting the rear wing's significant influence on the vehicle's overall aerodynamic performance, even at angles of attack beyond its stall angle.

The rear wing initially creates a modest pressurisation onto the slant at 0° , which shifts further outboard at smaller corner radii, as shown in Figure 5.20. The pressurisation magnitude and subsequent pressure rise along the roof increase gradually with angle of attack increments before the stall angle. However, suction beneath the rear wing at these angles of attack casts a low-pressure region onto the vehicle's body. This enhances airflow velocity along both C-pillars in straight-line conditions and along the outboard C-pillar in cornering conditions, reducing local pressure. These aerodynamic interactions collectively result in a maximum average increase of $\Delta C_L \approx 0.007$ (+1.5%) in lift along the slant and roof when operating just before the stall angle compared to at 0° .

Beyond the rear wing's stall angle, suction on the lower surface diminishes while pressure on the upper surface keeps increasing, as previously observed. The emitted low-pressure region from the rear wing onto the vehicle's body therefore disappears whereas pressurisation onto the slant and roof significantly increases. Altogether, the aerodynamic interactions at higher angles of attack decrease lift along the slant and roof surface compared to at 0° , with an averaged maximum of $\Delta C_L \approx -0.105$ (-24%) reached at the highest angle of attack of 27.5° in each operating condition.

Interaction between the rear wing and airflow around the vehicle's rear-end and near-wake is also identified. In straight-line conditions, the rear wing's enhanced suction and vortical flow at angles before the stall angle augment recirculation behind the vehicle. This is indicated by reduced pressure traces on the vehicle's base surface, increasing its local drag by $\Delta C_D \approx 0.017$ (+30%) right before the stall angle compared to 0° . Beyond the rear wing's stall angle, strong airflow recirculation and enhanced O_U and I_U vortices further amplify recirculation behind the vehicle, causing a maximum $\Delta C_D \approx 0.047$ (+84%) drag increase on the vehicle's base surface at the maximum 27.5° angle compared to 0° .

In cornering conditions, the rear wing's suction and I_U vortex strength progressively increase with angles of attack before the stall angle, enhancing the inboard recirculation wake vortex and its associated pressure losses. Consequently, drag on the vehicle's base surface increases by $\Delta C_D \approx 0.027$ (+35%) in the 5.75L radius corner condition and by $\Delta C_D \approx 0.023$ (+23%) in the 2.9L radius corner condition right before their respective stall angles compared to 0° .

Beyond the stall angle, dominant airflow separation behind the rear wing and the further intensified I_U vortex feed the recirculation wake vortex. Consequently, drag on the vehicle's base surface increases by $\Delta C_D \approx 0.037$ (+48%) in the 5.75L radius corner condition and by $\Delta C_D \approx 0.026$ (+25%) in the 2.9L radius corner condition at the maximum 27.5° angle of attack compared to 0° .

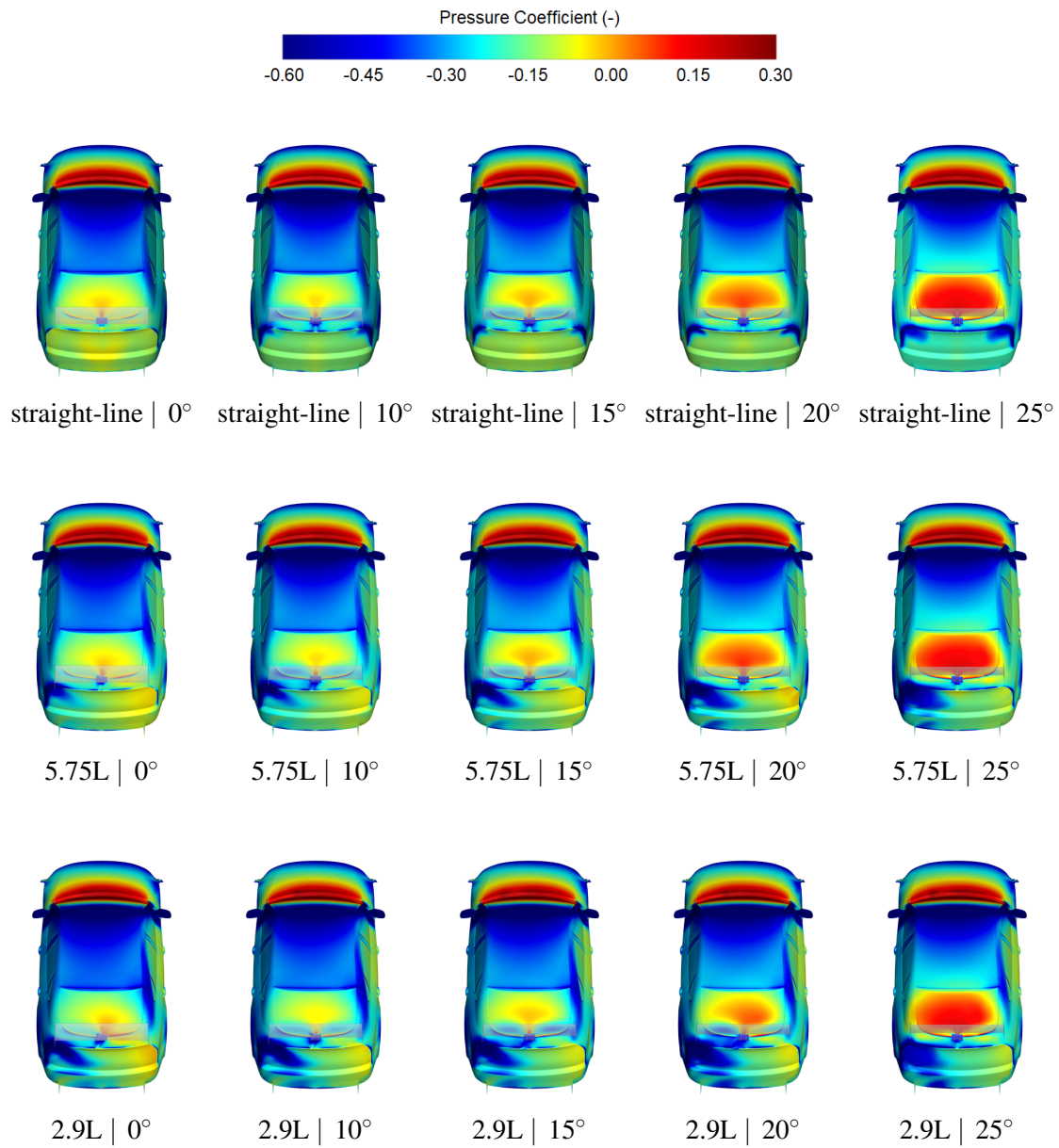


Figure 5.20: Pressure coefficient distribution on the vehicle with varying rear wing angles of attack in straight-line and cornering conditions.

5.5 Conclusion

This study investigated the aerodynamic behaviour of a high-performance vehicle with varying rear wing angles of attack in straight-line and cornering conditions. The analyses were performed on the *DrivAer hp-F*, a high-performance variant of the standard *DrivAer* Fastback configuration. The CFD methodology was validated against aerodynamic force measurement results from moving ground experiments in the 8x6 Wind Tunnel at Cranfield University.

Cornering conditions adversely impacted both the vehicle's overall downforce and drag performance, as well as the rear wing's individual performance, with smaller corner radii increasing the detriments. For each operating condition, two downforce enhancement regions were identified along the rear wing's angle of attack range, transitioning around the stall angle. Despite adverse cornering effects, the smallest corner radius marginally extended the rear wing's stall angle.

Downforce losses primarily stemmed from deficits on the underbody and rear wing, alongside increased lift generation on the vehicle's upper surfaces in cornering conditions. Drag penalties mainly resulted from substantial pressure reductions across the vehicle's base surface. Additionally, cornering conditions introduced lateral pressure distribution variations, inducing a centripetal force that increased with reducing corner radii.

The significant pressure reduction on the vehicle's base was due to the formation of a dominant recirculation wake vortex behind the vehicle's inboard side, which intensified at reducing corner radii. Additionally, strong circulation generated by the streamwise wake structure created an inboard inwash of relatively high-energy freestream airflow into the wake region, promoting wake dissipation.

Cornering conditions influenced the rear wing's vortical flow, notably affecting the pressure distribution near the endplates. Most significant pressure distribution variations were found on the rear wing's lower surface before reaching its stall angle. This included an initial increased suction on the inboard side at low angles of attack, which transitioned to increased suction levels on the outboard side at higher angles of attack due to intensified vortical flow and enhanced interaction with faster-moving relative airflow.

In each operating condition, the vehicle's initial total downforce enhancement region was mainly due to the rear wing's individual downforce contribution. However, beyond its stall angle, the rear wing's performance declined significantly. At high angles of attack, its pressurisation effect on the vehicle's upper surfaces became the primary mechanism for generating downforce. Nonetheless, at such high angles, the rear wing's interaction with the near-wake resulted in notably higher pressure drag.

Overall, this research highlighted the significance of operating conditions on the aerodynamic intricacies of high-performance vehicles. The findings offer crucial insights into the complex aerodynamic interactions between a high-performance vehicle and its rear wing under varying operating conditions. The analysis of such characteristics does not only support the design of static rear wing configurations, but also lays the groundwork for the development of active designs. Such advancements offer promising opportunities to further optimise vehicle aerodynamics and push performance boundaries.

Data Availability Statement

Experimental data supporting this study are openly available from Cranfield Online Research Data at: <https://dspace.lib.cranfield.ac.uk/handle/1826/22438> (34) and the CAD files for the *DrivAer hp-F* with rear wing at: <https://dspace.lib.cranfield.ac.uk/handle/1826/22433> (35).

References

1. Keogh J, Barber T, Diasinos S, Doig G. Techniques for aerodynamic analysis of cornering vehicles. SAE Technical Paper 2015-01-0022. 2015. Available from: <https://doi.org/10.4271/2015-01-0022>.
2. Toet W. Aerodynamics and aerodynamic research in Formula 1. The Aeronautical Journal. 2013;117(1187):1-26. Available from: <https://doi.org/10.1017/S0001924000007739>.
3. Keogh J. The aerodynamic effects of the cornering flow conditions. PhD Thesis. University of New South Wales; 2016. Available from: <https://doi.org/10.26190/unsworks/19191>.
4. Keogh J, Barber T, Diasinos S, Doig G. The aerodynamic effects on a cornering Ahmed body. Journal of Wind Engineering and Industrial Aerodynamics. 2016;154:34-46. Available from: <https://doi.org/10.1016/j.jweia.2016.04.002>.
5. Nakashima T, Mutsuda H, Kanehira T, Tsubokura M. Fluid-dynamic force measurement of Ahmed model in steady-state cornering. Energies. 2020;13(24):6592. Available from: <https://doi.org/10.3390/en13246592>.
6. Heft A, Indinger T, Adams N. Introduction of a new realistic generic car model for aerodynamic investigations. SAE Technical Paper 2012-01-0168. 2012. Available from: <https://doi.org/10.4271/2012-01-0168>.
7. Josefsson E, Hagvall R, Urquhart M, Sebben S. Numerical analysis of aerodynamic impact on passenger vehicles during cornering. SAE Technical Paper 2018-37-0014. 2018. Available from: <https://doi.org/10.4271/2018-37-0014>.
8. Nakashima T, Tsubokura M, Okada Y, Nouzawa T, Kono R, Doi Y. Aerodynamic characteristics of a road vehicle in steady-state cornering. Proceedings of the ASME/JSME/KSME 2015 Joint Fluids Engineering Conference. Volume 1A: Symposia, Part 2. Seoul, South Korea: The American Society of Mechanical Engineers; 2015. Available from: <https://doi.org/10.1115/AJKFluids2015-17553>.
9. Kono R, Nakashima T, Tsubokura M, Okada Y, Nouzawa T. Aerodynamics simulation of a sedan-type road vehicle in cornering motion with roll angle. Proceedings of the 34th AIAA Applied Aerodynamics Conference. AIAA 2016-3726. Washington, USA: AIAA; 2016. Available from: <https://doi.org/10.2514/6.2016-3726>.

10. Okada Y, Nouzawa T, Okamoto S, Fujita T, Kamioka T, Tsubokura M. Unsteady vehicle aerodynamics during a dynamic steering action: 1st report, on-road analysis. SAE Technical Paper 2012-01-0446. 2012. Available from: <https://doi.org/10.4271/2012-01-0446>.
11. Tsubokura M, Ikawa Y, Nakashima T, Okada Y, Kamioka T, Nouzawa T. Unsteady vehicle aerodynamics during a dynamic steering action: 2nd report, numerical analysis. SAE International Journal of Passenger Cars - Mechanical Systems. 2012;5(1):340-57. Available from: <https://doi.org/10.4271/2012-01-0448>.
12. Albukrek C, Doddegowda P, Ivaldi A, Amodeo J, Bardoscia E. Unsteady flow analysis of a Formula type open wheel race car in cornering. SAE Technical Paper 2006-01-3661. 2006. Available from: <https://doi.org/10.4271/2006-01-3661>.
13. Nara K, Tsubokura M, Ikeda J. A numerical analysis of unsteady aerodynamics of Formula car during dynamic cornering motion. Proceedings of the 32nd AIAA Applied Aerodynamics Conference. AIAA 2014-3138. Atlanta, USA: AIAA; 2014. Available from: <https://doi.org/10.2514/6.2014-3138>.
14. Garry KP, Le Good GM. An investigation of the sensitivity of rear wing orientation for saloon race cars. SAE Technical Paper 2005-01-1018. 2005. Available from: <https://doi.org/10.4271/2005-01-1018>.
15. Mathur A, Mahajan A, Aggarwal A, Mishra C, Roy A. Numerical study of swan neck rear wing for enhancing stability of ground vehicle bodies. Emerging Trends in Mechanical Engineering. Lecture Notes in Mechanical Engineering. Singapore: Springer; 2021. p. 199-208. Available from: https://doi.org/10.1007/978-981-15-8304-9_14.
16. Buljac A, Dzijan I, Korade I, Krizmanić S, Kozmar H. Automobile aerodynamics influenced by airfoil-shaped rear wing. International Journal of Automotive Technology. 2016;17(3):377-85. Available from: <https://doi.org/10.1007/s12239-016-0039-4>.
17. Katz J, Dykstra L. Study of an open-wheel racing-car's rear-wing aerodynamics. SAE Technical Paper 890600. 1989. Available from: <https://doi.org/10.4271/890600>.
18. Katz J, Largman R. Experimental study of the aerodynamic interaction between an enclosed-wheel racing-car and its rear wing. Journal of Fluids Engineering. 1989;111(2):154-9. Available from: <https://doi.org/10.1115/1.3243616>.
19. Keogh J, Doig G, Diasinos S, Barber T. The influence of cornering on the vortical wake structures of an inverted wing. Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering. 2015;229(13):1817-29. Available from: <https://doi.org/10.1177/0954407015571673>.
20. Gogel D, Sakurai H. The effects of end plates on downforce in yaw. SAE Technical Paper 2006-01-3647. 2006. Available from: <https://doi.org/10.4271/2006-01-3647>.

21. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Experimental and numerical investigation of the aerodynamic characteristics of high-performance vehicle configurations under yaw conditions. *Physics of Fluids*. 2024;36(4):045112. Available from: <https://doi.org/10.1063/5.0196979>.
22. Kurec K, Remer M, Mayer T, Tudruj S, Piechna J. Flow control for a car-mounted rear wing. *International Journal of Mechanical Sciences*. 2019;152:384-99. Available from: <https://doi.org/10.1016/j.ijmecsci.2018.12.034>.
23. Broniszewski J, Piechna J. A fully coupled analysis of unsteady aerodynamics impact on vehicle dynamics during braking. *Engineering Applications of Computational Fluid Mechanics*. 2019;13(1):623-41. Available from: <https://doi.org/10.1080/19942060.2019.1616326>.
24. Broniszewski J, Piechna JR. Fluid-structure interaction analysis of a competitive car during brake-in-turn manoeuvre. *Energies*. 2022;15(8):2917. Available from: <https://doi.org/10.3390/en15082917>.
25. Piechna JR, Kurec K, Broniszewski J, Remer M, Piechna A, Kamieniecki K, et al. Influence of the car movable aerodynamic elements on fast road car cornering. *Energies*. 2022;15(3):689. Available from: <https://doi.org/10.3390/en15030689>.
26. Zavala D, Vicente W, Esquivel R, Salinas-Vazquez M. Numerical simulation of the effect of a centripetal wing spoiler on the flow aerodynamics of a sports car. *SAE Technical Paper 2023-01-5032*. 2023. Available from: <https://doi.org/10.4271/2023-01-5032>.
27. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model. *SAE Technical Paper 2018-01-0725*. 2018. Available from: <https://doi.org/10.4271/2018-01-0725>.
28. Milliken DL, Kasprzak EM, Metz LD, Milliken WF. *Race car vehicle dynamics - problems, answers and experiments*. Warrendale, USA: SAE International; 1995.
29. Rijns S, Teschner TR, Blackburn K, Brighton J. Integrated numerical and experimental workflow for high performance vehicle aerodynamics. *SAE Technical Paper 2024-01-5016*. 2024. Available from: <https://doi.org/10.4271/2024-01-5016>.
30. Lanfrit M. Best practice guidelines for handling automotive external aerodynamics with Fluent. University of Southampton; 2005. Available from: https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf. (Accessed: 01 May 2022).
31. Saunders JW, Mansour RB. On-road and wind tunnel turbulence and its measurement using a four-hole dynamic probe ahead of several cars. *SAE Technical Paper 2000-01-0350*. 2000. Available from: <https://doi.org/10.4271/2000-01-0350>.
32. Szudarek M, Piechna J. CFD analysis of the influence of the front wing setup on a time attack sports car's aerodynamics. *Energies*. 2021;14(23):7907. Available from: <https://doi.org/10.3390/en14237907>.

33. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*. 1994;32(8):1598-605. Available from: <https://doi.org/10.2514/3.12149>.
34. Rijns S, Teschner TR, Blackburn K, Brighton J. Dataset DrivAer hp-F: force measurements at various rear wing angles of attack. Cranfield Online Research Data (CORD); 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/22438>.
35. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. DrivAer hp-F: spoiler & rear wing configurations geometry pack. Cranfield Online Research Data (CORD); 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/22433>.

Chapter 6

Aerodynamic and Steady-State Performance Analysis of Active Aerodynamics During Cornering

This chapter was originally published as:

Rijns S, Teschner TR, Blackburn K, Brighton J. Performance analyses of active aerodynamic load balancing designs on high-performance vehicles in cornering conditions. *Physics of Fluids*. 2024;36(8):085199. Available from: <https://doi.org/10.1063/5.0222780>.

Abstract

This study presents a comprehensive investigation into the impact of Active Aerodynamic Load Balancing (AALB) on the cornering performance of high-performance vehicles. The research explores the use of 2D active aerodynamic devices, specifically split and tilt wing concepts, capable of influencing aerodynamic balance both longitudinally and laterally. The performance potential of AALB is assessed through quasi-steady static coupling of aerodynamic data with a detailed vehicle dynamics model. The findings show that inside bias operating states of the split and tilt rear wing concepts, which favour loads on the inside tyres, can improve cornering velocities by up to 0.5% and 2% compared to high symmetric operating states, respectively. Notably, through effective distribution of aerodynamic loads, the inside bias operating states produce less downforce and drag, thereby reducing the propulsion power required to overcome drag by 15%-20%, depending on the cornering condition. The tilt rear wing concept demonstrates the highest AALB capability and most consistent response to its control strategy, attributed to its ability to generate vertical and horizontal aerodynamic force components. It can therefore achieve over 1% higher maximum cornering velocities compared to the split rear wing, while also offering efficiency benefits. Overall, the research highlights the effectiveness of AALB in improving cornering performance and efficiency, offering valuable insights for the development of active aerodynamic systems for high-performance automotive applications and paving the way for future advancements in the field.

Keywords: Active Aerodynamics; Vehicle Dynamics; Cornering; CFD; RANS

6.1 Introduction

Aerodynamics is a critical factor in the design of high-performance and motorsport vehicles, significantly impacting performance, efficiency, and handling in dynamic operating conditions such as cornering. During cornering, an outward-directed centrifugal force acts on the vehicle, instigating a load transfer from the inside to the outside tyres. Due to the non-linear vertical load sensitivity of tyres, this lateral load transfer reduces the potential combined lateral force produced by the tyres on each axle. Excessive load transfer can even overload the tyres, thereby drastically reducing grip and compromising cornering performance (1, 2).

The objectives of this study stem from the hypothesis that counteracting lateral load transfer from the inside to the outside tyres during cornering can enhance overall tyre grip. To achieve this, Active Aerodynamic Load Balancing (AALB) is explored. This refers to modulating aerodynamic balance both longitudinally and laterally to favour vertical forces on the inside tyres while inducing an aerodynamic roll moment that opposes the overturning moment caused by the centrifugal force. Aerodynamic systems capable of AALB require asymmetrical actuation to influence both longitudinal and lateral aerodynamic balance, referred to as ‘2D active aerodynamic systems’. Compared to conventional methods such as suspension adjustments, tyre design modifications, or centre of gravity repositioning, which primarily redistribute or partially mitigate load transfer within inherent design constraints, 2D active aerodynamic systems offer a more adaptable and potentially more effective solution.

Research has demonstrated that cornering conditions have a significant impact on vehicle aerodynamics. Studies on automotive applications revealed notable changes in surface pressure distributions and wake characteristics, which in turn affected aerodynamic lift, drag, and side forces (3, 4, 5, 6, 7, 8, 9). Findings on high-performance and motorsport vehicles demonstrated heightened sensitivity to these conditions, primarily due to their aerodynamic devices (10, 11, 12, 13). Downforce and drag penalties ranging from 20% to 30% were reported for such vehicles in tight corners and yaw conditions compared to straight-line conditions. These findings highlight the importance of analysing a diverse range of representative operating conditions to assess a vehicle’s aerodynamic performance.

To thoroughly investigate overall vehicle performance, it is essential to capture the interrelation between its aerodynamics and dynamics. This interplay can be examined using coupling methods of varying complexity (14), namely quasi-steady static coupling, unsteady static coupling, and unsteady dynamic coupling. Quasi-steady static coupling employs steady-state Computational Fluid Dynamics (CFD) simulations, while unsteady static coupling utilises transient CFD simulations on a vehicle with a fixed position as input to a vehicle dynamics model. Unsteady dynamic coupling, the most complex method, iterates between CFD simulations and a vehicle dynamics model to continuously update changes in the vehicle’s position and aerodynamic loads. However, the increased complexity of these methods results in a significant trade-off in computational time and resources.

Various coupling methods have been employed for automotive applications to investigate the stability and handling of passenger vehicles and large trucks during dynamic events such as crosswind gusts (15, 16, 17, 18, 19). Comparative studies have highlighted the trade-off between computational efficiency and accuracy among these methods (14, 20, 19, 21). While results varied, fully coupled methods generally benefited large vehicles in severe conditions despite higher computational costs, whereas one-way coupling methods sufficed for smaller passenger vehicles.

Quasi-steady static coupling has predominantly been used to investigate the impact of aerodynamic devices on vehicle performance. Most studies employed aerodynamic data from isolated devices (22, 23, 24, 25), while others took the aerodynamics of the entire vehicle into account (26, 27). Findings among these studies generally indicated that aerodynamic devices contributed to enhancements in cornering performance and stability. More intricate fully coupled analyses involving a vehicle equipped with various aerodynamic add-on devices also reported improvements in cornering performance, a finding corroborated by road tests (28). Subsequent investigations on the same vehicle demonstrated a reduction in braking distance and an improvement in stability during straight-line braking and brake-in-turn events using an active rear wing (29, 30).

More promising results for the impact of active aerodynamic systems on vehicle performance have been demonstrated. Studies showed that active small spoiler plates underneath larger spoilers enhanced downforce generation and benefited from faster actuation (31, 32). Findings on 2D active aerodynamic systems demonstrated that split rear wings were able to vary downforce across the inside and outside tyres, but primarily affected the rear axle. Tilt rear wings showed the capability to generate both downforce and side force, influencing the lateral distribution of vertical tyre loads (32, 33). Furthermore, road tests of a vehicle equipped with a tilt rear wing demonstrated an approximate 13% increase in cornering velocity compared to a configuration without aerodynamic devices (32).

Despite these positive findings, most considered active aerodynamic devices primarily generated rear axle downforce, leading to aerodynamically unbalanced setups. While studies indicated that splitters contributed to front axle downforce, front wings proved significantly more effective (34). Given the importance of distributing aerodynamic loads across both axles, front wings are expected to provide better balance when paired with large active rear wings compared to splitters.

In general, the reviewed studies have underscored the effectiveness of quasi-steady static coupled simulations in analysing the complex interaction between vehicle aerodynamics and dynamics while highlighting some potential benefits of active aerodynamic devices. However, existing research predominantly focused on symmetric aerodynamic devices, often relying on data from isolated elements in straight-line conditions. As a result, little attention has been given to 2D active aerodynamic devices capable of AALB. The ability of these systems to prioritise aerodynamic loads on the inside tyres during cornering remains largely underexplored, limiting the understanding of their role in enhancing performance.

This study addresses this gap by investigating the performance potential of 2D active aerodynamic systems in cornering. It evaluates the AALB capabilities of active split and tilt rear wings, combined with an active split front wing, assessing their influence on lateral aerodynamic balance while maintaining a stable longitudinal load distribution. Using quasi-steady static coupling of extensive CFD-obtained aerodynamic data across representative cornering conditions with a detailed vehicle dynamics model, the study systematically examines the impact of these active aerodynamic systems on cornering performance. The findings offer detailed insights into each system's AALB capability and impact on vehicle performance, establishing a foundation for further exploration of 2D active aerodynamic systems for high-performance vehicles.

6.2 Methodology

This study hypothesises that aerodynamic loads which prioritise downforce on the inside tyres and oppose lateral load transfer improve overall tyre grip during cornering. This AALB approach is explored through two high-performance vehicle concepts with 2D active aerodynamic systems that have been identified as promising designs in literature.

The impact of these systems on cornering performance is investigated using quasi-steady static coupling of the vehicle's aerodynamics and dynamics. This method employs a one-way coupling of aerodynamic data as input for a vehicle dynamics model. Conventionally, aerodynamic data is often generated in straight-line conditions due to the absence of prior knowledge on vehicle performance. However, Chapters 4 and 5 (12, 13) underscored the significant influence of operating conditions on the aerodynamic performance of high-performance vehicles, indicating that straight-line aerodynamic data may not accurately reflect the aerodynamic performance in dynamic events such as cornering.

Therefore, a key aspect of the methodology is generating aerodynamic data across representative operating conditions. As outlined in Figure 6.1, the study begins with a performance assessment using wind tunnel data of an existing high-performance vehicle, which serves as the baseline geometry. This initial performance assessment is used to determine representative operating conditions for the CFD simulations on the active high-performance vehicle concepts. Successively, the vehicle dynamics model uses the simulated aerodynamic data to examine the cornering performance of the active high-performance vehicle concepts. The subsequent sections of the methodology will detail the vehicle geometry, wind tunnel experiments, CFD simulations, and vehicle dynamics model employed throughout the study.

6.2.1 Vehicle Geometry

The *DrivAer hp-F* vehicle model (35), a high-performance variant of the well-established *DrivAer* model (6), serves as the baseline geometry for this study. Specifically developed for aerodynamic research on high-performance vehicles, the *DrivAer hp-F* model features a smooth underbody and is equipped with additional aerodynamic elements. These elements include a 41 mm front bumper splitter, forebody strakes, a 10° underbody multichannel diffuser, and a NACA 6412 profile rear wing, as depicted in Figure 6.2a.

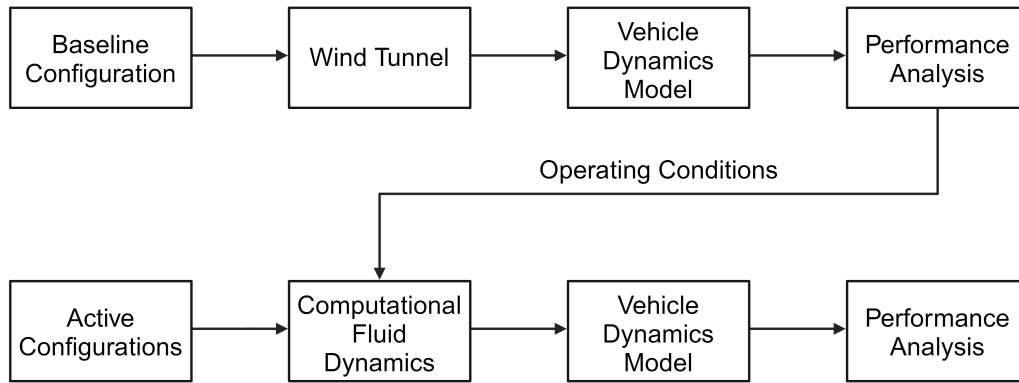


Figure 6.1: Flowchart of the research process applied in this chapter.

This study introduces two high-performance vehicle concepts based on the *DrivAer hp-F*, each incorporating 2D active aerodynamic devices. The first concept features a split rear wing, with left and right elements capable of independently adjusting their angle of attack from 0° to 30° . The second concept utilises a rear wing that can adjust both its angle of attack between 0° and 30° , and roll angle between -12° and $+12^\circ$, referred to as the tilt rear wing concept. To maintain aerodynamic balance, both active high-performance vehicle concepts feature a split front wing, consisting of left and right elements that can independently vary their angle of attack from 0° to 10° . The active high-performance vehicle concepts are illustrated in Figure 6.2 and their main dimensions are listed alongside those of the baseline *DrivAer hp-F* in Table 6.1.

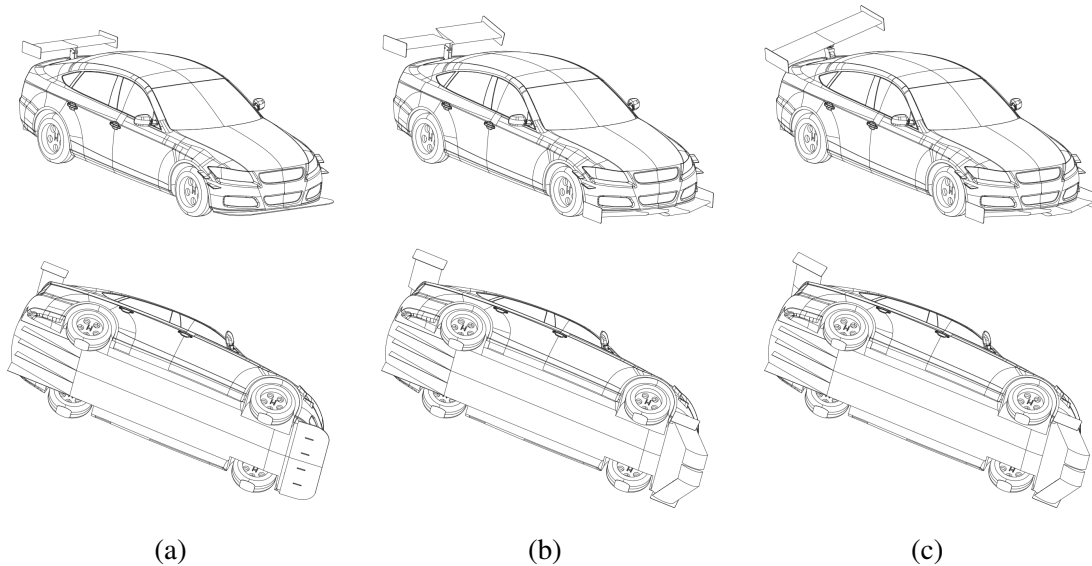


Figure 6.2: Illustration of the *DrivAer hp-F* configurations, including (a) the baseline vehicle with standard rear wing, (b) the split wing concept featuring an active split front wing and split rear wing, and (c) the tilt wing concept featuring an active split front wing and tilt rear wing. The split and tilt wing vehicle concepts are shown in an asymmetric operating state to illustrate their aerodynamic functionality.

Table 6.1: Main dimensions of the baseline, active aerodynamic concepts, and 35% scale wind tunnel model of the *DrivAer hp-F*.

Parameter	Symbol	Unit	Baseline	Active	35% scale
Length	L	m	4.6126	4.6126	1.6144
Width	W	m	1.7923	1.7923	0.6273
Height	H	m	1.3971	1.3971	0.4890
Ride height	h_0	m	0.1300	0.1300	0.0455
Area	A_{ref}	m ²	2.1596	2.1596	0.2647
Rear wing chord	c_{RW}	m	0.3143	0.3929	0.1100
Rear wing span	b_{RW}	m	1.200	1.500	0.4200

6.2.2 Wind Tunnel Experiments

This study utilises experimental data from previously conducted wind tunnel tests on the *DrivAer hp-F* model in the closed-return 8x6 Wind Tunnel at Cranfield University from Chapter 5 (13). The aerodynamic performance data is used for the initial performance assessment, as detailed in Section 6.2, to establish representative operating conditions for the CFD simulations on the active high-performance vehicle concepts. The experiments were performed on a 35% scale model of the baseline *DrivAer hp-F* configuration with a rear wing. The main model dimensions are provided in Table 6.1 and the full-scale vehicle geometry is illustrated in Figure 6.2a.

The wind tunnel features a rectangular test section measuring 2.44 m in width, 1.83 m in height, and 5.18 m in length. The experimental setup, shown in Figure 6.3, included a moving ground and a two-stage boundary layer suction system to optimise the ground plane boundary layer. The vehicle model was suspended by an overhead strut system (OS) above a rolling belt (RB), which measured 1.2 m in width and 2.75 m in length.

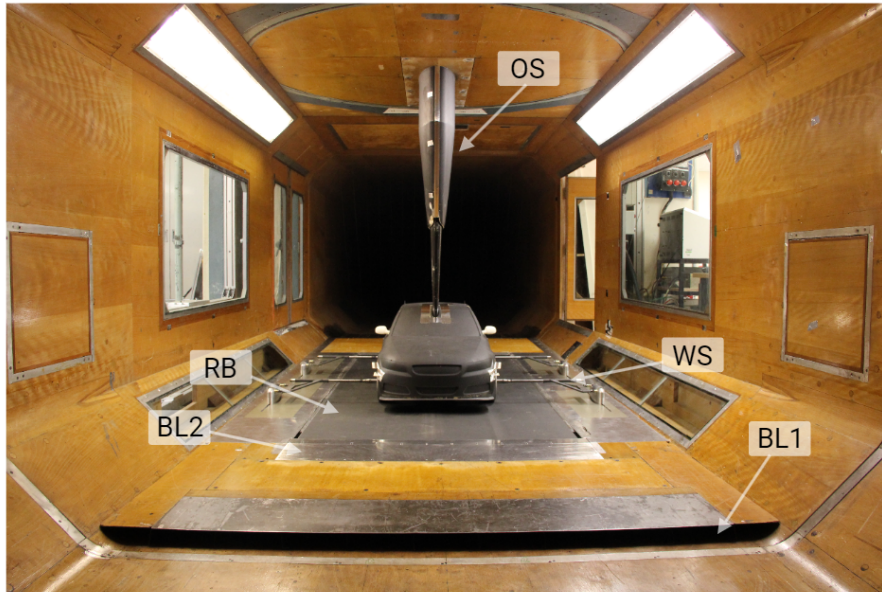


Figure 6.3: Moving ground setup in the 8x6 Wind Tunnel at Cranfield University.

The wheels were independently supported by wheel supports (WS) and matched the rolling belt speed. The boundary layer control consisted of two suction systems. The first stage (BL1) featured a scoop intake positioned 0.73 m upstream of the second stage (BL2), which utilised suction through a porous plate just upstream of the rolling belt. The complete experimental setup resulted in a blockage ratio of approximately 9.6%.

The dataset includes aerodynamic force and moment measurements for varying rear wing angles of attack. The experiments were conducted at a constant freestream velocity of 40 m/s, corresponding to a Reynolds number of 4.21×10^6 . Forces and moments were recorded using an Aerotech[®] six-component internal balance with a nominal accuracy of $\pm 0.06\%$ of the full-scale measurement range, providing a measurement accuracy of ± 0.90 N for lift and ± 0.18 N for drag. Further details on the experimental methodology and setup can be found in Section 5.2.3 (13).

6.2.3 Operating Conditions

A variety of cornering conditions are considered for the performance analyses to formulate a comprehensive understanding of the potential benefits and limitations of AALB provided by the 2D active aerodynamic systems on the high-performance vehicle concepts. Corners with radii of 15, 30, and 50 car-lengths (L) are selected to cover various driving scenarios, ranging from tight corners to high-speed bends. As previously presented in Figure 6.1, an initial performance assessment of the baseline high-performance vehicle is used to determine representative operating conditions for the CFD simulations on the active high-performance vehicle concepts in each cornering condition. Following the assessment that will be discussed in Section 6.3.1, the 15L, 30L, and 50L radius corners are simulated at 29 m/s, 42 m/s and 54 m/s, respectively.

6.2.4 Computational Fluid Dynamics Simulations

This study employs CFD simulations to obtain aerodynamic data for the *DrivAer hp-F* concepts featuring 2D active aerodynamic systems under the previously specified operating conditions. The methodology follows the validated approach from Chapter 5 (13), which builds on the extensive methodological work in Chapter 2 (36), including a grid convergence study, turbulence model assessment, and comprehensive validation. An overview of the methodology is provided here, while further details are available in the prior chapters.

6.2.4.1 Domain and Boundary Conditions

The steady-state cornering conditions are simulated using a rectangular domain with modified inlet and outlet positions. The computational domain, illustrated in Figure 6.4, in combination with a moving reference frame, allows for the simulation of various corner radii. This approach eliminates the need for re-meshing the entire domain for each operating condition, thereby improving the numerical workflow efficiency. The domain dimensions adhere to best practice guidelines for automotive aerodynamic simulations (37), extending three car lengths (L) upstream, 5L downstream, and 1.5L above and to the sides of the vehicle.

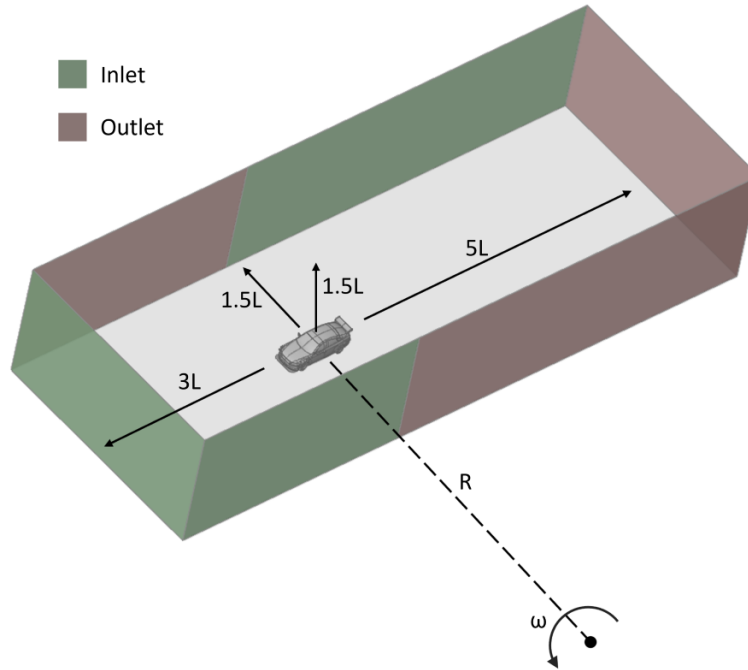


Figure 6.4: Illustration of the computational domain with cornering boundary conditions.

The moving reference frame approach defines a rotational frame motion about an external point, referred to as the centre of rotation, to simulate the curved path during cornering. Since the domain is prescribed with an angular velocity, the ground plane is modelled to remain stationary. Wheel rotation is simulated with rotational moving wall boundary conditions, which prescribe an angular velocity to match the required tangential velocity at each wheel.

As detailed in Section 6.2.3, representative operating states are selected based on an initial performance assessment of the baseline *DrivAer hp-F*. Additionally, turbulence conditions are derived from on-road turbulence measurements for ground vehicles (38) to replicate realistic driving conditions. An overview of the simulation settings for each operating condition is provided in Table 6.2.

6.2.4.2 Mesh

The computational domain is constructed using an overset mesh approach, which utilises individual unstructured poly-hexcore meshes created in ANSYS Fluent Meshing. The overset mesh technique applies data interpolation within overlaying mesh regions, thereby allowing specific component mesh transformations without needing to re-mesh the entire domain. Supported by its demonstrated effectiveness in prior studies (13, 34), the overset mesh methodology is a particularly beneficial approach to improve the workflow efficiency for the substantial amount of CFD simulations in this study.

Figure 6.5 shows the overset interfaces of the component meshes, where the interfaces of the vehicle body, front wing elements, and individual wheels are shown in grey, orange and blue, respectively. The tilt rear wing consists of a single component mesh, depicted in green, while the split rear wing elements comprise two individual component meshes.

Table 6.2: Overview of the CFD simulation settings for each cornering condition.

Feature	Symbol	15L Radius corner	30L Radius corner	50L Radius corner
Velocity (tangential)	v	29 m/s	42 m/s	54 m/s
Moving reference frame				
Frame motion		rotational	rotational	rotational
Motion axis	x, y, z	0, 0, 1	0, 0, 1	0, 0, 1
Velocity	ω	0.4191 rad/s	0.3035 rad/s	0.2341 rad/s
Inlets & Outlets				
Velocity-inlet	v	0 m/s	0 m/s	0 m/s
Pressure-outlet	P	0 Pa	0 Pa	0 Pa
Turbulent intensity	I_T	4%	4%	4%
Turbulent length scale	l_T	2.5 m	2.5 m	2.5 m
Wheels				
Steering angle	δ	2.2°	1.5°	1.1°
Rotation axis				
Front	x, y, z	$\sin(\delta), -1, 0$	$\sin(\delta), -1, 0$	$\sin(\delta), -1, 0$
Rear	x, y, z	0, -1, 0	0, -1, 0	0, -1, 0
Velocity				
Front inboard	ω	91.272 rad/s	132.903 rad/s	171.225 rad/s
Front outboard	ω	93.299 rad/s	134.371 rad/s	172.358 rad/s
Rear inboard	ω	91.267 rad/s	132.901 rad/s	171.225 rad/s
Rear outboard	ω	93.294 rad/s	134.370 rad/s	172.357 rad/s
Reference air properties				
Density	ρ	1.204 kg/m ³	1.204 kg/m ³	1.204 kg/m ³
Dynamic viscosity	μ	1.813e-5 kg/(m·s)	1.813e-5 kg/(m·s)	1.813e-5 kg/(m·s)
Temperature	T	20°C	20°C	20°C

These meshes, superimposed on the background mesh inclusive of refinement zones, form the complete domain. Utilising this approach enables independent adjustments of the wheels' steering angles and operating settings of the active aerodynamic devices.

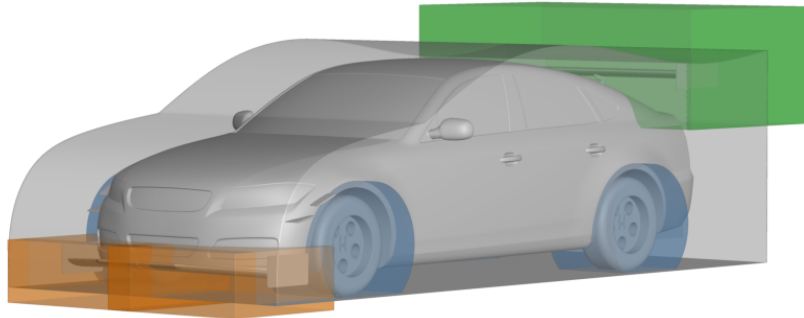


Figure 6.5: Illustration of the overset component mesh interfaces.

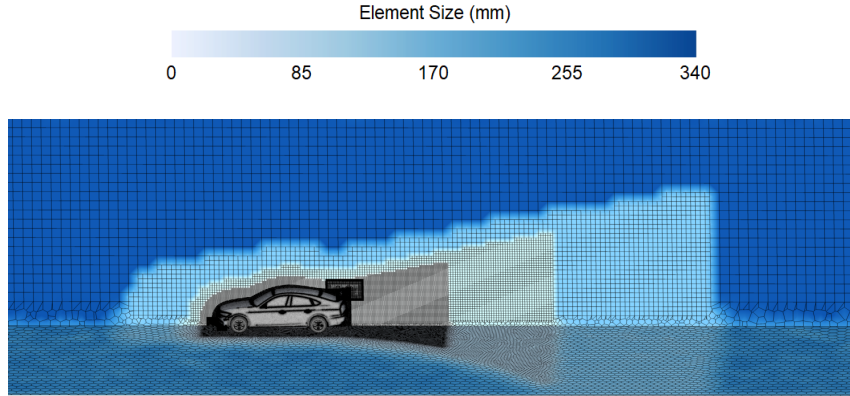


Figure 6.6: Contour plot of element sizes in proximity of the vehicle using a volume slice at $y = 0$ (symmetry).

The mesh settings in this study are derived from the authors' previous research on the *DrivAer hp-F* model in Chapter 2 (36), which evaluated grid convergence and near-wall modelling approaches for cost-effective Reynolds-Averaged Navier-Stokes (RANS) simulations against experimental data and high-fidelity Delayed Detached Eddy Simulations (DDES) on a wall-resolved mesh with a $y^+ < 1$. Accordingly, the base element and vehicle surface element sizes are approximately 7.4% and 0.45% of the vehicle's length, respectively. Vehicle surface element sizes are locally reduced by over 80% for refinement on aerodynamic elements and small geometric features.

A medium near-wall treatment with a $y^+ \approx 120$ -150 and 4-6 inflation layers is targeted for each operating condition. Additionally, refinement zones for improved resolution in the vehicle's near-field and wake region are incorporated, as depicted in Figure 6.6. Altogether, the meshes for the split rear wing concept have an average cell count of approximately 19.6 million, while those for the tilt rear wing concept have around 23.9 million. All meshes generally contain an orthogonal quality above 0.7 for 99% of the cells.

6.2.4.3 Solve

Steady-state RANS simulations are performed using the $k-\omega$ SST turbulence model (39) in ANSYS Fluent. The selection of this model is based on its demonstrated accuracy and computational efficiency for aerodynamic simulations on the *DrivAer hp-F* model in Chapter 2 (36), which compared various RANS models against high-fidelity DDES simulations and experimental data.

The governing equations include the continuity, momentum, and transport equations for turbulent kinetic energy and specific dissipation rate. Pressure-velocity coupling is performed using the coupled scheme, with gradients computed via the least-squares cell-based method. Pressure discretisation employs a second-order scheme, while momentum and turbulence transport equations are solved using a second-order upwind scheme.

Simulation duration is determined based on the statistical convergence of lift and drag coefficients, monitored throughout the iterations. The moving mean of these coefficients typically stabilises within 200 iterations. To ensure convergence, simulations are carried out for 750 iterations, with force coefficients averaged over the final 450 iterations. The simulations are executed on two nodes consisting of AMD EPYC 7543 processors, utilising a total of 128 cores. The computational time varies between the high-performance vehicle configurations, averaging approximately 4 hours for the split rear wing concept and 5 hours for the tilt rear wing concept.

6.2.4.4 Post-Processing

The aerodynamic analyses in this study use non-dimensional coefficients of forces and pressures obtained from the CFD simulations. Force coefficients are expressed by Eq. (6.1), where F_i (N) denotes the numerical force results and C_i (-) the non-dimensional force coefficients, with i indicating lift (L) or drag (D). Similarly, pressure coefficients are given by Eq. (6.2), where P_s (Pa) represents the numerical static pressure results, P_{op} (Pa) the operating pressure of 101325 Pa, and C_p (-) the non-dimensional pressure coefficient. Reference quantities of the air density ρ (kg/m³), velocity v (m/s), and frontal area A_{ref} (m²) are listed in Tables 6.1 and 6.2.

$$C_i = \frac{F_i}{\frac{1}{2} \rho v^2 A_{ref}} \quad (6.1)$$

$$C_p = \frac{P_s - P_{op}}{\frac{1}{2} \rho v^2} \quad (6.2)$$

6.2.5 Vehicle Dynamics Model

This research utilises a vehicle dynamics model which incorporates aerodynamic forces, lateral load transfer, and non-linear tyre characteristics to evaluate vehicle performance in steady-state cornering conditions. A general overview of the model, illustrated in Figure 6.7, is first provided to support the interpretation of detailed formulations.

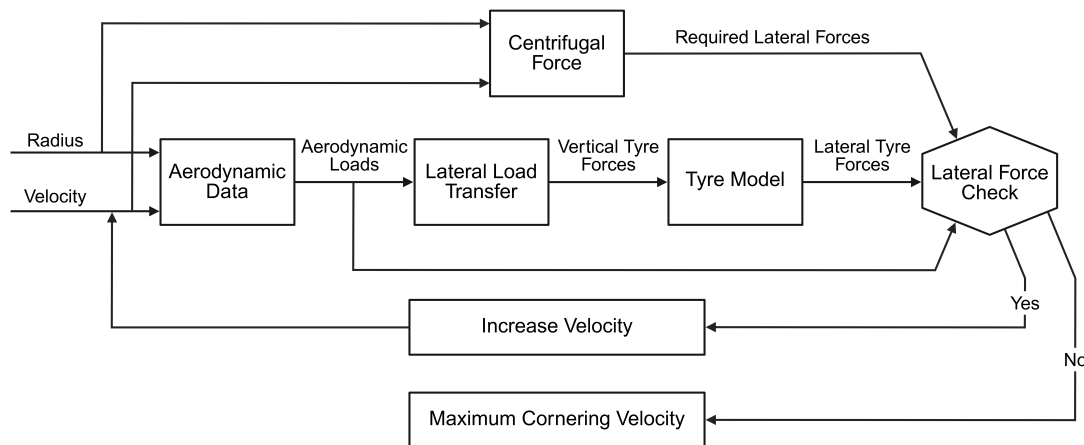


Figure 6.7: Vehicle dynamics model workflow.

The model operates iteratively, cycling through velocities for a specified corner radius. At each velocity, the required lateral force for the vehicle to maintain its cornering trajectory is calculated for both the front and rear axles. To ensure that aerodynamic performance is accurately represented, aerodynamic maps are created for the specified cornering conditions and incorporated into the model. These maps, combined with the effects of lateral load transfer, are used to calculate the vertical forces acting on each tyre.

These vertical forces are then input into the tyre model, which iterates through front and rear slip angles until the required lateral force for each axle is achieved. This iterative process continues, with the velocity incrementally increasing until the required lateral force at either the front or rear axle can no longer be produced. The final velocity at this point represents the maximum achievable velocity for the given corner radius. The model assumes traction-limited states, implying that the vehicle's propulsion system can generate the necessary power to achieve the maximum velocity. The vehicle parameters used in the model, listed in Table 6.3, are based on indicative specifications of sports and racing cars (2).

Table 6.3: Overview of the vehicle parameters.

Parameter	Symbol	Unit	Value
Mass			
Sprung vehicle body mass	m_b	kg	1300
Driver mass	m_d	kg	75
Front unsprung mass	m_{uf}	kg	90
Rear unsprung mass	m_{ur}	kg	100
Total sprung mass	m_s	kg	1375
Total mass	m	kg	1565
Dimensions			
Wheelbase length	l	m	2.786
Distance front axle to COG	l_f	m	1.4766
Distance rear axle to COG	l_r	m	1.3094
Height COG	h_{COG}	m	0.5
Height front unsprung mass	h_{uf}	m	0.2969
Height rear unsprung mass	h_{ur}	m	0.2969
Height roll centre at COG position	h	m	0.0912
Height front roll centre	h_f	m	0.2969
Height rear roll centre	h_r	m	0.2969
Distance roll axis to COG	d	m	0.4088
Width front track	t_f	m	1.5204
Width rear track	t_r	m	1.5204
Suspension			
Roll stiffness	k_ϕ	Nm/rad	1.1459×10^5
Front axle roll stiffness	$k_{\phi f}$	Nm/rad	6.3025×10^4
Rear axle roll stiffness	$k_{\phi r}$	Nm/rad	5.1566×10^4

6.2.5.1 Vehicle Model

The vehicle model is based on a bicycle model, which has been demonstrated to adequately simulate dominant vehicle dynamics effects even in extreme operating conditions (40), making it a suitable approach for this study.

The vehicle model, depicted in Figure 6.8, uses combined slip angles α_f (rad) and α_r (rad) for the front and rear tyres, respectively. Applying a small angle approximation, the steering angle δ (rad) and side slip angle β (rad) are expressed by Eqs. (6.3) and (6.4). Small angle approximations employed in this study have a maximum error of around 0.2%, which is considered acceptable for the purpose of the analyses.

$$\delta = \frac{l}{R} + \alpha_f - \alpha_r \quad (6.3)$$

$$\beta = \frac{l_r}{R} - \alpha_r \quad (6.4)$$

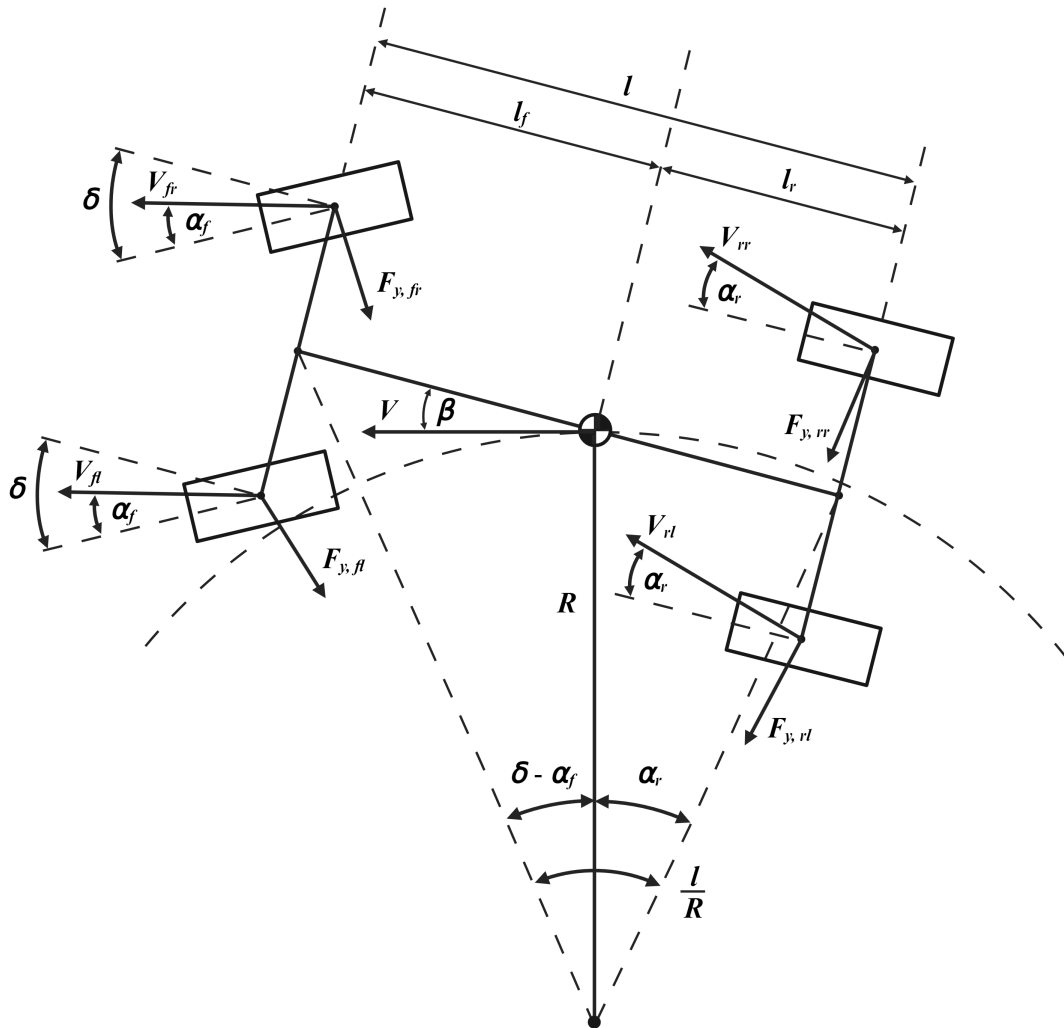


Figure 6.8: Schematic illustration of the vehicle model.

6.2.5.2 Centrifugal Force

In order for a vehicle to maintain its trajectory in a steady-state cornering condition, it must generate a lateral cornering force equal to the centrifugal force F_c (N), formulated by Eq. (6.5). Here, m (kg) represents the total vehicle mass, V (m/s) the vehicle's velocity, R (m) the corner radius, and a (m/s²) the vehicle's lateral acceleration.

$$F_c = m \frac{V^2}{R} = m a \quad (6.5)$$

The moment equilibrium about the vehicle's centre of gravity (COG) determines the required lateral forces which must be produced at the front $F_{yf,req}$ (N) and rear $F_{yr,req}$ (N) axles to maintain the cornering trajectory, as expressed by Eqs. (6.6) and (6.7). In these equations, l (m) denotes the wheelbase length, and l_f (m) and l_r (m) represent the distances between COG and the front and rear axles, respectively.

$$F_{yf,req} = F_c \frac{l_r}{l} \quad (6.6)$$

$$F_{yr,req} = F_c \frac{l_f}{l} \quad (6.7)$$

6.2.5.3 Vertical Tyre Forces

The actual lateral forces produced by the tyres are dependent on the individual vertical tyre forces. In steady-state cornering, the loads are influenced by aerodynamic forces and lateral load transfer. A schematic view of lateral load transfer during cornering is provided in Figure 6.9, serving as a visual aid for the formulations of vertical wheel loads.

The front and rear static wheel loads, denoted with W_f (N) and W_r (N), are defined by Eqs. (6.8) and (6.9). Here, m_s (kg) represents the sprung mass, m_{uf} (kg) the front unsprung mass, m_{ur} (kg) the rear unsprung mass, and g (m/s²) the gravitational acceleration.

$$W_f = \frac{1}{2} g (m_{uf} + m_s \frac{l_r}{l}) \quad (6.8)$$

$$W_r = \frac{1}{2} g (m_{ur} + m_s \frac{l_f}{l}) \quad (6.9)$$

The lateral load transfers resulting from the centrifugal force on the front and rear unsprung masses, indicated with $F_{Tf,1}$ (N) and $F_{Tr,1}$ (N) respectively, are expressed by Eqs. (6.10) and (6.11). In these equations, h_{uf} (m) and h_{ur} (m) denote the heights of the front and rear unsprung masses, and t_f (m) and t_r (m) the front and rear track widths.

$$F_{Tf,1} = m_{uf} a \left(\frac{h_{uf}}{t_f} \right) \quad (6.10)$$

$$F_{Tr,1} = m_{ur} a \left(\frac{h_{ur}}{t_r} \right) \quad (6.11)$$

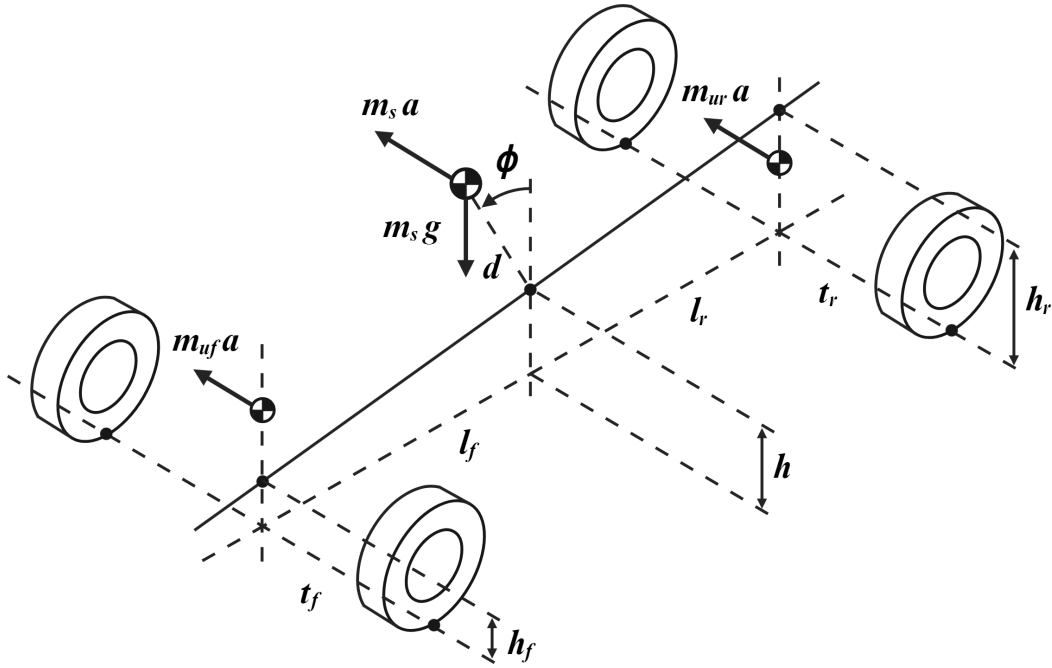


Figure 6.9: Schematic illustration of lateral load transfer.

The centrifugal force on the sprung mass can be expressed as a force acting on the roll axis, transmitted via the suspension links, and an additional roll couple due to the distance between the COG and the roll axis, transmitted through the springs. Similarly, the aerodynamic loads on the sprung mass can be conceptualised as an aerodynamic side force acting on the roll axis, and an aerodynamic roll moment about the roll axis.

Considering the forces transmitted through the suspension links first, the load transfer resulting from the centrifugal force is distributed between the front and rear axles by the vehicle's COG position. The magnitudes of aerodynamic side force components acting on the front and rear roll centres, represented by F_{Sf} (N) and F_{Sr} (N), impact the lateral load transfer resulting from aerodynamic loads at the respective axles. Together, the forces transmitted via the suspension links induce a lateral load transfer at the front and rear axle, denoted by $F_{Tf,2}$ (N) and $F_{Tr,2}$ (N), which are formulated by Eqs. (6.12) and (6.13). Here, h_f (m) and h_r (m) represent the front and rear roll centre heights. The aerodynamic side force components are defined in the same direction as the centrifugal force.

$$F_{Tf,2} = m_s a \left(\frac{l_r}{l} \right) \left(\frac{h_f}{t_f} \right) + F_{Sf} \left(\frac{h_f}{t_f} \right) \quad (6.12)$$

$$F_{Tr,2} = m_s a \left(\frac{l_f}{l} \right) \left(\frac{h_r}{t_r} \right) + F_{Sr} \left(\frac{h_r}{t_r} \right) \quad (6.13)$$

The roll moment of the sprung mass about the vehicle's roll axis, generated by the centrifugal force and aerodynamic loads, is transmitted through the suspension springs. By applying a small angle approximation, the resultant roll angle of the sprung mass ϕ (rad) is expressed by Eq. (6.14).

Here, $M_{A,rc}$ (Nm) denotes the aerodynamic roll moment about the vehicle's roll centre at the COG position, defined in the opposite direction of the moment created by the centrifugal force. The distance between the COG and the roll axis is represented by d (m), and the total roll stiffness by k_ϕ (Nm/rad).

$$\phi = \frac{m_s a d - M_{A,rc}}{k_\phi - m_s g d} \quad (6.14)$$

The roll moments reacted at front and rear axles, denoted by M_f (Nm) and M_r (Nm), are proportional to the roll stiffnesses of the axles. The roll moments are formulated by Eqs. (6.15) and (6.16), where the roll stiffnesses of the front and rear axles are represented by $k_{\phi f}$ (Nm/rad) and $k_{\phi r}$ (Nm/rad), respectively.

$$M_f = k_{\phi f} \phi \quad (6.15)$$

$$M_r = k_{\phi r} \phi \quad (6.16)$$

The lateral load transfers at the front $F_{Tf,3}$ and rear $F_{Tr,3}$ axles, resulting from the roll moments, are subsequently expressed by Eqs. (6.17) and (6.18).

$$F_{Tf,3} = \frac{M_f}{t_f} \quad (6.17)$$

$$F_{Tr,3} = \frac{M_r}{t_r} \quad (6.18)$$

During cornering, lateral load transfer occurs from the inside to the outside wheels. Given that this study considers left-hand corners, the load transfers from the left to the right tyres. Additionally, vertical wheel loads are affected by aerodynamic downforce on each wheel, as formulated by Eqs. (6.19)-(6.22). The downforce on the front left, front right, rear left, and rear right tyres is denoted by $F_{DF,fl}$ (N), $F_{DF,fr}$ (N), $F_{DF,rl}$ (N), and $F_{DF,rr}$ (N). The overall vertical forces acting on the respective tyres are represented by $F_{Z,fl}$ (N), $F_{Z,fr}$ (N), $F_{Z,rl}$ (N), and $F_{Z,rr}$ (N).

$$F_{Z,fl} = W_f - F_{Tf,1} - F_{Tf,2} - F_{Tf,3} + F_{DF,fl} \quad (6.19)$$

$$F_{Z,fr} = W_f + F_{Tf,1} + F_{Tf,2} + F_{Tf,3} + F_{DF,fr} \quad (6.20)$$

$$F_{Z,rl} = W_r - F_{Tr,1} - F_{Tr,2} - F_{Tr,3} + F_{DF,rl} \quad (6.21)$$

$$F_{Z,rr} = W_r + F_{Tr,1} + F_{Tr,2} + F_{Tr,3} + F_{DF,rr} \quad (6.22)$$

6.2.5.4 Tyre Model

The actual lateral force produced by each individual tyre is modelled as a function of slip angle and vertical load, using the widely deployed Pacejka '96 model (41). The lateral tyre force F_y (N) is described by Eq. (6.23), where α_i (rad) is the slip angle, with subscript i indicating the front (f) or rear (r) tyres.

$$F_y = D_y \sin[C_y \tan^{-1}\{B_y \alpha_i - E_y (B_y \alpha_i - \tan^{-1}(B_y \alpha_i))\}] \quad (6.23)$$

The peak value D_y , shape factor C_y , curvature factor E_y , stiffness factor B_y , and normalised change in vertical load df_z are described by Eqs. (6.24)-(6.28). Here, the nominal vertical wheel load is denoted by F_{Z0} (N), the actual vertical wheel load by F_Z (N), and the wheel camber by γ (rad). Furthermore, the Pacejka coefficients are represented by $p_{CY1}...p_{KY3}$ and the scaling factors by $\lambda_{\mu y}... \lambda_{Ey}$.

$$D_y = F_Z (p_{DY1} + p_{DY2} df_z)(1 - p_{DY3} \gamma^2) \lambda_{\mu y} \quad (6.24)$$

$$C_y = p_{CY1} \lambda_{Cy} \quad (6.25)$$

$$B_y = \frac{p_{KY1} F_{Z0} \sin \left[2 \tan^{-1} \left(\frac{F_Z}{p_{KY2} F_{Z0} \lambda_{Z0}} \right) \right] (1 - p_{KY3} |\gamma|) \lambda_{FZ0} \lambda_{Kya}}{C_y D_y} \quad (6.26)$$

$$E_y = (p_{EY1} + p_{EY2} df_z)(1 - (p_{EY3} + p_{EY4} \gamma) \operatorname{sgn}(\alpha_i)) \lambda_{Ey} \quad (6.27)$$

$$df_z = \frac{F_Z - F_{Z0}}{F_{Z0}} \quad (6.28)$$

Following the procedure of Seward (1), Pacejka coefficients are derived which approximate lateral tyre force data for a 250/570R13 Avon British F3 tyre with 0° camber (42). The tyre model coefficients are listed in Table 6.4 and the corresponding lateral tyre force characteristics are illustrated in Figure 6.10.

Table 6.4: Overview of the tyre model coefficients.

Coefficient	Value	Coefficient	Value
F_{Z0}	4905 N	p_{KY1}	-35
p_{CY1}	2.5	p_{KY2}	3
p_{DY1}	-1.41	p_{KY3}	0
p_{DY2}	0.6	$\lambda_{\mu y}$	1
p_{DY3}	0	λ_{Cy}	1
p_{EY1}	0.5	λ_{FZ0}	1
p_{EY2}	0	λ_{Kya}	1
p_{EY3}	0	λ_{Ey}	1
p_{EY4}	0	γ	0°

6.2.5.5 Tyre Characteristics

The tyre's ability to generate lateral force depends on both slip angle and vertical load, as shown in Figure 6.10a. At low slip angles, lateral force increases approximately linearly, but the relationship becomes non-linear near the peak, beyond which grip decreases with increasing slip angle. Additionally, lateral force increases with vertical load, but at a diminishing rate.

This vertical tyre load sensitivity is further illustrated in Figure 6.10b, which plots lateral force against vertical load at constant slip angles. The rate of lateral force gains is greatest at low to moderate vertical loads but declines significantly as the tyre approaches its saturation point, beyond which more vertical load can even reduce grip. This behaviour is most pronounced near the tyre's peak operating range, which for this 250/570R13 Avon British F3 tyre is at slip angles of around 6° - 8° , depending on the vertical load.

Tyre characteristics have significant implications for vehicle performance during cornering. Lateral load transfer increases vertical load on the outside tyres while reducing it on the inside tyres. For the vehicle considered in this study, individual vertical tyre loads can range from nearly 8000 N on the outside tyres to as little as 700 N on the inside tyres. Due to the tyre's non-linear vertical load sensitivity, this imbalance reduces the total lateral grip generated by the axle. The lightly loaded inside tyres lose lateral force capacity more rapidly than the heavily loaded outside tyres gain, resulting in a net reduction in overall grip compared to a more balanced load distribution.

Conventional symmetric aerodynamic devices generate downforce equally on both sides of the vehicle, increasing vertical tyre loads without reducing lateral load transfer. While this enhances grip, it does not leverage the tyre's load sensitivity to optimise total lateral force. In contrast, 2D active aerodynamic systems capable of AALB enable asymmetric actuation, allowing downforce to be prioritised on the underloaded inside tyres while generating an aerodynamic roll moment that opposes lateral load transfer. Although this AALB approach may slightly reduce total downforce due to lower loading on the outside tyres, the aerodynamic roll moment can reduce lateral load transfer across the axle.

The resulting decrease in load on the outside tyres, which already operate near saturation, leads to only a minor loss in lateral force. Meanwhile, the inside tyres benefit from both high local downforce and reduced load transfer away from them. Since the inside tyres operate in the more linear, high-sensitivity region of the tyre load curve, the net effect is an increase in total lateral grip across the axle. This mechanism forms the theoretical basis for the two high-performance vehicle concepts featuring 2D active aerodynamic systems capable of AALB evaluated in this study.

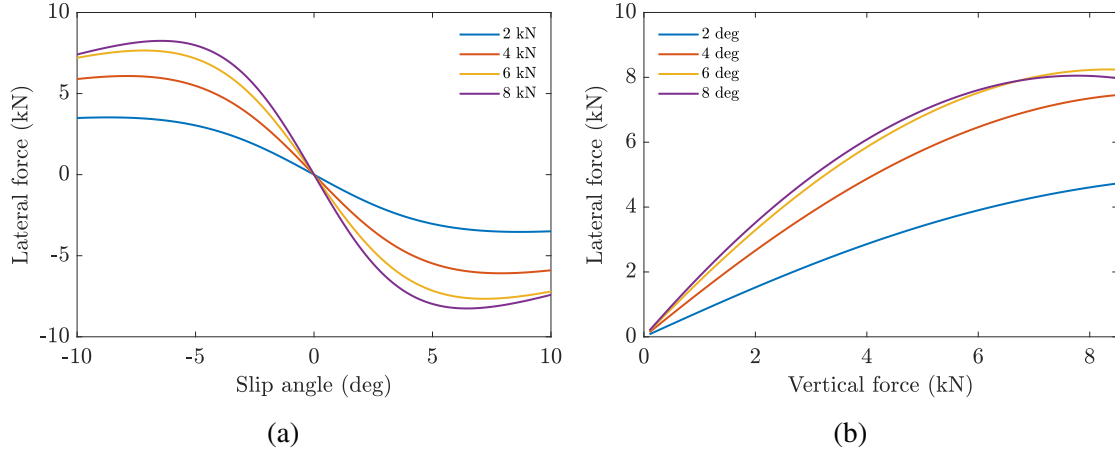


Figure 6.10: (a) Lateral tyre force as a function of slip angle at constant vertical loads and (b) lateral tyre force as a function of vertical load at constant slip angles.

6.3 Results and Discussion

This section begins by evaluating the performance of the baseline *DrivAer hp-F* configuration to establish a reference for subsequent analyses. As outlined in Section 6.2, these results are used to define the operating states for the CFD simulations of the active high-performance vehicle concepts. Building on this foundation, the impact of AALB on cornering performance is analysed for both the split rear wing and tilt rear wing concepts. The section concludes with a comparative assessment of the AALB capabilities of the two configurations.

6.3.1 Baseline Vehicle Configuration

The experimental aerodynamic force data for the baseline *DrivAer hp-F*, presented in Figure 6.11, shows two downforce enhancement regions across the rear wing's angle of attack range. In the initial 0° - 15° range, downforce increases by approximately $\Delta C_L \approx -0.103$ until the rear wing stalls at around 15° . Beyond the stall angle, downforce increases by another $\Delta C_L \approx -0.129$ along the 15° - 27.5° range. Prior analyses have shown that downforce generation from the rear wing itself diminishes after its stall angle, but that high angles of attack create substantial upstream pressurisation, reducing lift generation along the vehicle's upper surfaces (12, 13).

High rear wing angles of attack also produce substantially more drag. The average drag penalty per 2.5° angle of attack increment rises by nearly 80%, from $\Delta C_D \approx 0.006$ in the 0° - 15° range to $\Delta C_D \approx 0.011$ in the 15° - 27.5° range. The downforce produced by the rear wing primarily acts on the rear axle, while slightly reducing front axle downforce by $\Delta C_L \approx 0.027$ across the 0° - 27.5° range. Consequently, the vehicle's aerodynamic balance shifts from around 88% front axle downforce at 0° to approximately 38% at 27.5° .

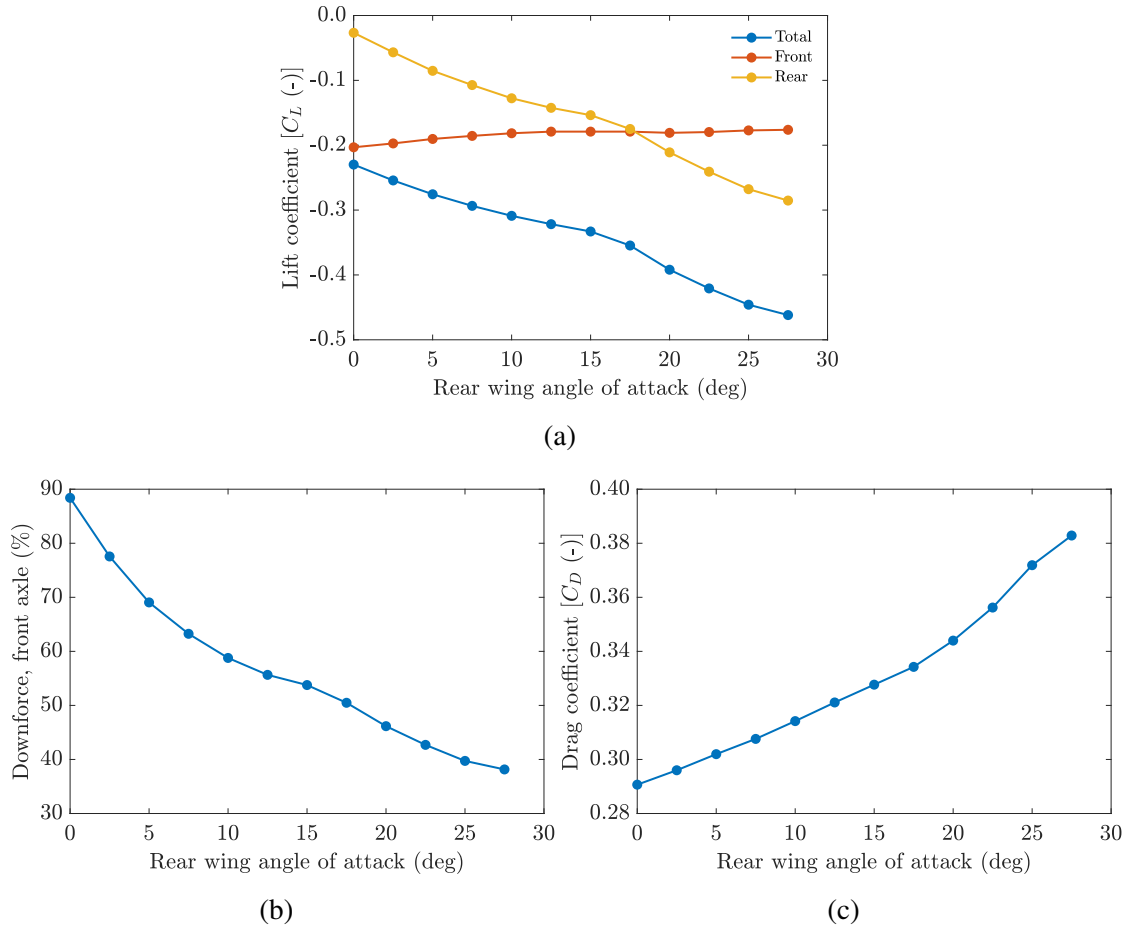


Figure 6.11: Experimental results of (a) lift coefficients, (b) downforce distribution, and (c) drag coefficients for the baseline *DrivAer hp-F* with varying rear wing angles of attack.

Performance results derived from this aerodynamic data, presented in Figure 6.12, indicate that the maximum cornering velocity initially increases with the rear wing's angle of attack before reaching an optimal point and subsequently declines. Compared to the 0° rear wing angle of attack, the maximum cornering velocity improves by approximately $\Delta v \approx 0.14$ m/s (0.5%) at 15° in the 15L radius corner, $\Delta v \approx 0.48$ m/s (1.2%) at 20° in the 30L radius corner, and $\Delta v \approx 1.09$ m/s (2.1%) at 20° in the 50L radius corner. These velocity enhancements correspond to peak lateral acceleration improvements of approximately 1.0%, 2.4%, and 4.2% over the 0° rear wing angle of attack cases in the 15L, 30L, and 50L radius corners, respectively.

Notably, peak performance across the cornering conditions is achieved at front axle downforce distributions ranging from approximately 45% to 55%. Furthermore, it is evident that the rear wing's impact on the vehicle's cornering performance becomes more substantial at higher velocities. This is a result of the absolute magnitude of aerodynamic downforce, which quadratically increases with the vehicle's velocity. Figure 6.12 also shows that the vehicle's performance is initially limited by the grip generated by the rear tyres, transitioning to being front grip limited at higher rear wing angles of attack.

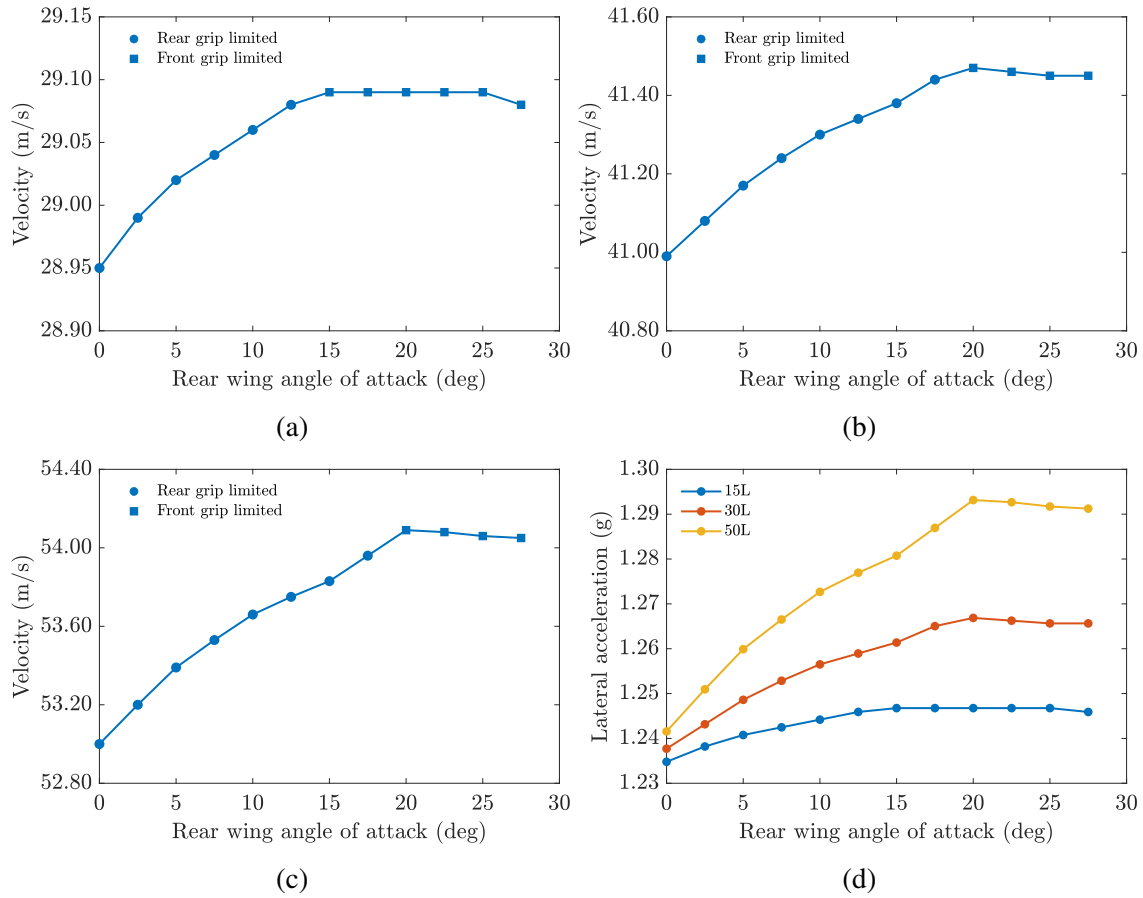


Figure 6.12: Maximum velocity for the baseline *DrivAer hp-F* in the (a) 15L, (b) 30L, and (c) 50L radius corners, along with (d) the corresponding lateral accelerations for each cornering condition, as a function of the rear wing angle of attack.

To further elaborate on the vehicle's limit behaviour, the lateral tyre forces for front and rear grip limited states in the 50L radius corner are presented in Figure 6.13. It has been demonstrated that front axle downforce slightly diminishes with increasing rear wing angle of attack, whereas rear axle downforce noticeably increases. The reduced front axle downforce together with the increased lateral load transfer at a 20° rear wing angle of attack cause a reduction in the potential grip production of the front inside tyre compared to that at a 0° rear wing angle of attack. Conversely, despite the larger lateral load transfer, the enhanced rear axle downforce at the 20° rear wing angle of attack improves the potential grip production of both the rear inside and outside tyres.

Consequently, the combined lateral tyre forces, depicted in Figures 6.13c and 6.13d, demonstrate that if the velocity of the vehicle with a 0° rear wing angle of attack continues to increase, the rear tyres would not be able to generate the required lateral force, leading to oversteering limit behaviour. Contrarily, at a 20° rear wing angle of attack, the front tyres would produce an insufficient lateral force, resulting in understeering limit behaviour. Additionally, both figures indicate that the rear axle must generate a larger lateral force to maintain the moment equilibrium about the vehicle's COG, given that the COG is positioned closer to the rear axle.

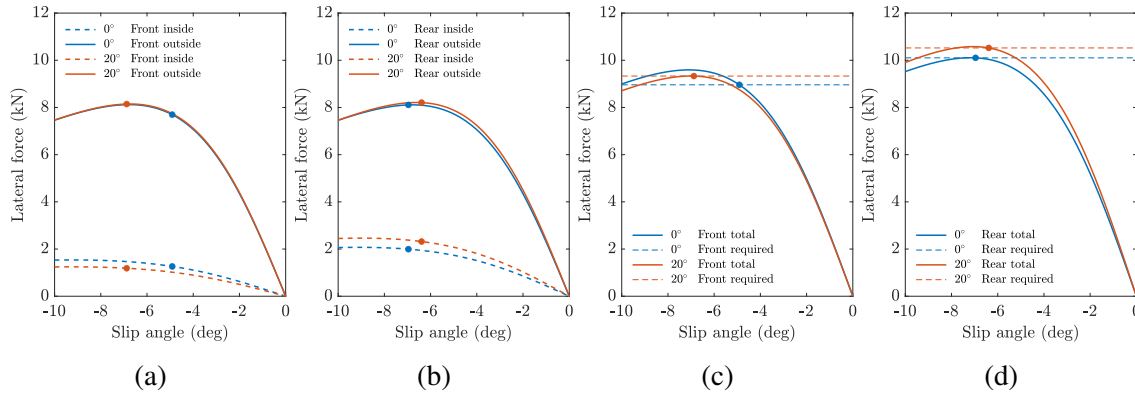


Figure 6.13: Comparison of lateral tyre forces from the baseline *DrivAer hp-F* at 0° and 20° rear wing angles of attack in the 50L radius corner, showing forces for (a) individual front tyres, (b) individual rear tyres, (c) combined front tyres, and (d) combined rear tyres.

6.3.2 Active Aerodynamic Load Balancing Vehicle Concepts

The split rear wing and tilt rear wing high-performance vehicle concepts, detailed in Section 6.2.1, are designed to influence the distribution of aerodynamic loads to improve cornering performance. This section first discusses the control strategy developed for the active aerodynamic devices on these high-performance vehicle concepts, followed by analyses of the aerodynamic characteristics and cornering performance. The section concludes with a comparison between the AALB capabilities of both high-performance vehicle concepts.

6.3.2.1 Control Strategy

The control strategy in this study is designed to operate the active aerodynamic devices of both high-performance vehicle concepts at comparable operating settings to ensure a consistent comparison between the concepts. The control strategy leverages findings from the initial performance assessment of the baseline vehicle, which demonstrated that a balanced front-to-rear downforce distribution is essential for achieving high cornering velocities. The settings for the front and rear located active aerodynamic devices are therefore coupled on either side of the high-performance vehicle concepts to maintain this balance.

The operating settings are derived from aerodynamic data obtained by varying the front and rear wing angles of attack under straight-line conditions, depicted in Figure 6.14a. A viable operating space of front and rear wing angle of attack combinations is determined using a 40%-60% front axle downforce distribution constraint, shown in Figure 6.14b. Within the defined space, this study opted to use an operating line representing linear combinations of front and rear wing angles of attack, sampled at five evenly spaced operating settings that are simulated to map the aerodynamic performance of the high-performance vehicle concepts.

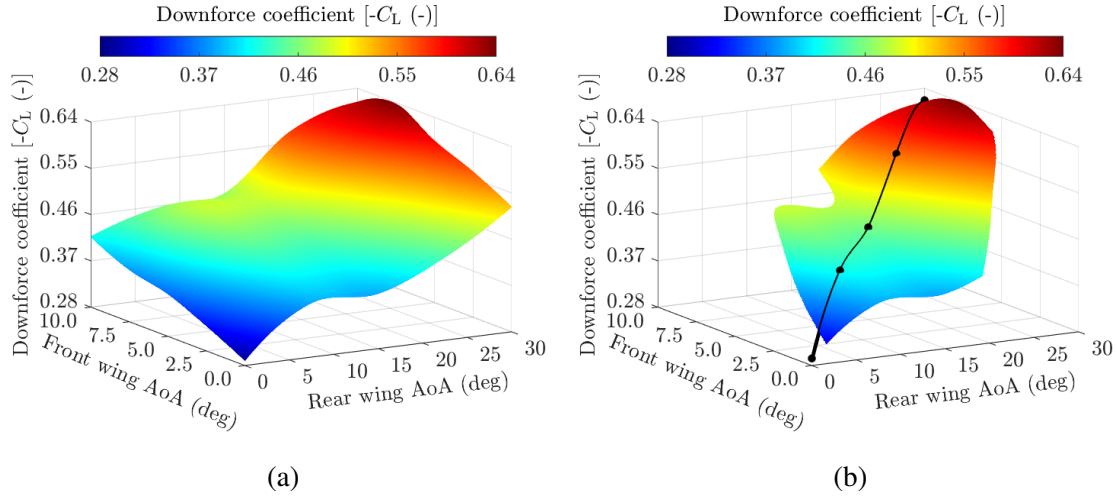


Figure 6.14: Downforce coefficients on the vehicle body (a) across the entire range of simulated front and rear wing angles of attack, and (b) within the 40%–60% front axle downforce distribution range, including the determined operating line and settings.

Accordingly, operating settings for the split rear wing concept pair a 2.5° front wing element angle of attack increment with a 7.5° rear wing element angle of attack increment on the same side of the vehicle, as detailed in Tables 6.5 and 6.6. To establish comparable operating settings for the tilt rear wing concept, a 2.5° front wing element angle of attack increment is coupled with a 3.75° rear wing angle of attack increment and 3° upward tilt of the rear wing on the same side of the vehicle, as outlined in Tables 6.5 and 6.7.

By adjusting the operating settings for the active aerodynamic devices located at the inside and outside of the high-performance vehicle concepts, the distribution of aerodynamic loads across the inside and outside tyres can be influenced. Given that the control strategies for both concepts apply similar operating settings, a consistent comparison between the two concepts can be conducted in terms of their performance and AALB capabilities.

Table 6.5: Split front wing: angles of attack for the inside (α_I) and outside (α_O) located elements for each operating setting combination.

	O1		O2		O3		O4		O5	
	α_I	α_O	α_I	α_O	α_I	α_O	α_I	α_O	α_I	α_O
I1	0°	0°	0°	2.5°	0°	5°	0°	7.5°	0°	10°
I2	2.5°	0°	2.5°	2.5°	2.5°	5°	2.5°	7.5°	2.5°	10°
I3	5°	0°	5°	2.5°	5°	5°	5°	7.5°	5°	10°
I4	7.5°	0°	7.5°	2.5°	7.5°	5°	7.5°	7.5°	7.5°	10°
I5	10°	0°	10°	2.5°	10°	5°	10°	7.5°	10°	10°

Table 6.6: Split rear wing: angles of attack for the inside (α_I) and outside (α_O) located elements for each operating setting combination.

	O1		O2		O3		O4		O5	
	α_I	α_O	α_I	α_O	α_I	α_O	α_I	α_O	α_I	α_O
I1	0°	0°	0°	7.5°	0°	15°	0°	22.5°	0°	30°
I2	7.5°	0°	7.5°	7.5°	7.5°	15°	7.5°	22.5°	7.5°	30°
I3	15°	0°	15°	7.5°	15°	15°	15°	22.5°	15°	30°
I4	22.5°	0°	22.5°	7.5°	22.5°	15°	22.5°	22.5°	22.5°	30°
I5	30°	0°	30°	7.5°	30°	15°	30°	22.5°	30°	30°

Table 6.7: Tilt rear wing: angles of attack (α_A) and tilt angles (α_T) for each operating setting combination.

	O1		O2		O3		O4		O5	
	α_A	α_T	α_A	α_T	α_A	α_T	α_A	α_T	α_A	α_T
I1	0°	0°	3.75°	3°	7.5°	6°	11.25°	9°	15°	12°
I2	3.75°	-3°	7.5°	0°	11.25°	3°	15°	6°	18.75°	9°
I3	7.5°	-6°	11.25°	-3°	15°	0°	18.75°	3°	22.5°	6°
I4	11.25°	-9°	15°	-6°	18.75°	-3°	22.5°	0°	26.25°	3°
I5	15°	-12°	18.75°	-9°	22.5°	-6°	26.25°	-3°	30°	0°

6.3.2.2 Split Rear Wing

Analyses of the aerodynamic performance of the split rear wing high-performance vehicle concept reveal that downforce and drag coefficients generally increase with higher operating settings for the inside and outside active aerodynamic devices throughout the different cornering conditions, as observed in Figure 6.15. The rate at which downforce coefficients increase is highest in the 15L radius corner and lowest in the 50L radius corner, indicating that the response to operating settings becomes slightly more gradual with increasing corner radii.

Inside dominant operating states mainly exhibit higher downforce and lower drag coefficients than outside dominant operating states. This observation is especially evident in the 15L radius corner, suggesting a favourable flow alignment on the inside of the vehicle in tighter corners. The maximum downforce coefficient in the 15L radius corner also exceeds that of the 30L and 50L radius corners by 3.5% and 5%, respectively, whereas the maximum drag coefficient is only 1% higher. Across the operating states, the front axle downforce distribution varies from approximately 49% to 57%, thereby remaining within the intended range of the control strategy.

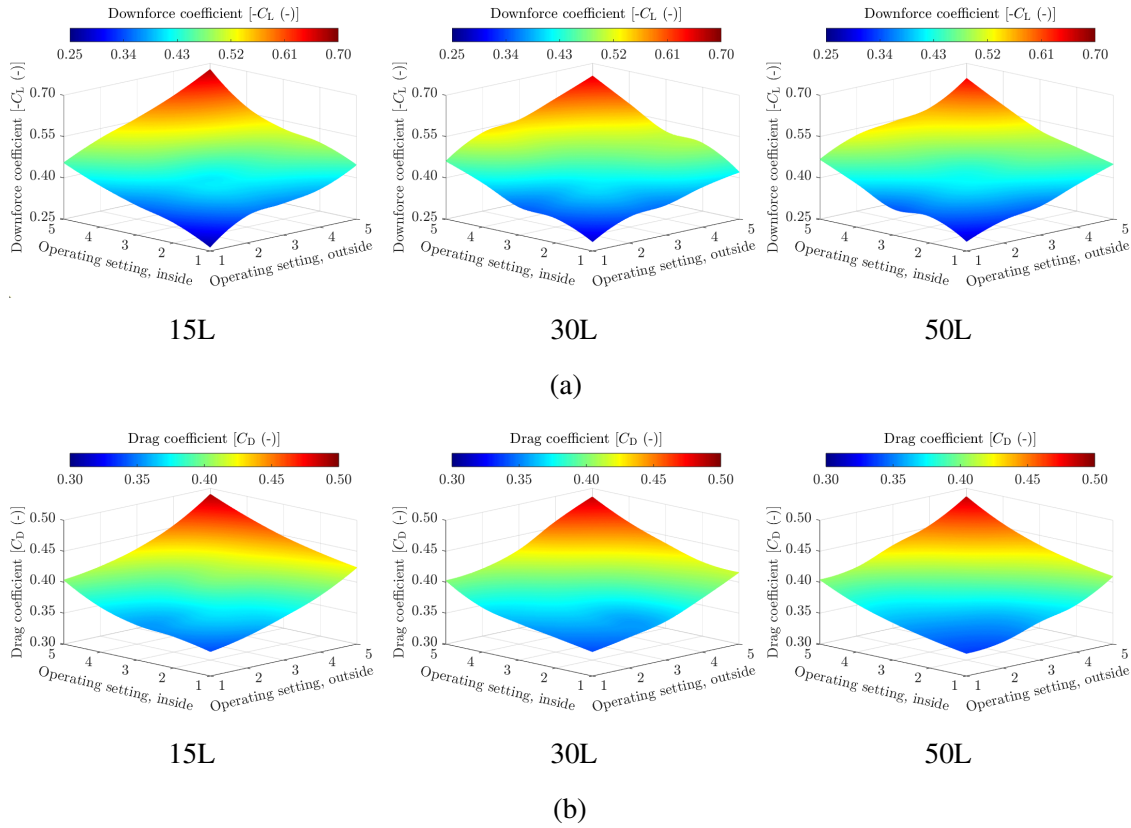


Figure 6.15: Downforce (a) and drag (b) coefficients for the split rear wing high-performance vehicle concept across various cornering conditions, as a function of the inside and outside operating settings of the active aerodynamic systems.

Figure 6.16 demonstrates that aerodynamic performance variations between maximum symmetric operating states in different cornering conditions primarily occur on the underbody. Upper body surfaces depict comparable pressure coefficient distributions, where the uniform high-pressure region on top of the split rear wing elements produces a noticeable upstream pressurisation along the roof.

On the underbody, high symmetric front wing settings result in substantial low-pressure regions on both elements. Another pronounced low-pressure area is generated around the front axle, which develops further downstream with reducing corner radii. Flow conditions in tight corners also favour low-pressure generation on the diffuser section on the inside of the vehicle, whereas the pressure coefficient distribution becomes more symmetric in larger corners that approach symmetric straight-line conditions.

Maximum inside bias operating states generate a more substantial high-pressure region on the upper surface of the inside element than on the outside element of the split rear wing in each cornering condition, as shown in Figure 6.17. The pressure difference between the split rear wing elements induces a vortex which adversely affects the high-pressure region on the outside element. Furthermore, the upstream pressurisation at this operating state is primarily generated by the inside element and is therefore considerably less substantial than at the maximum symmetric operating state.

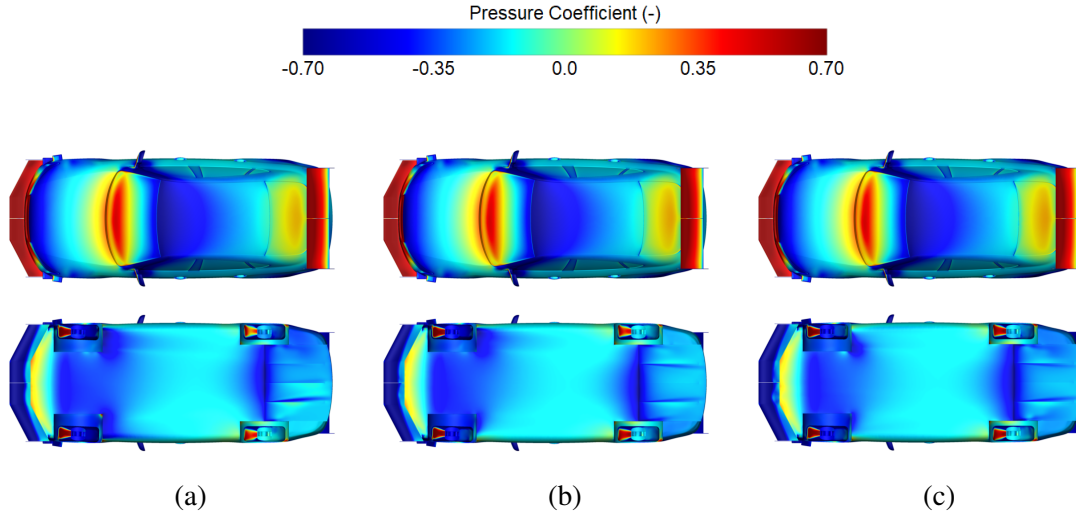


Figure 6.16: Pressure coefficient distribution on the split rear wing high-performance vehicle concept at its maximum symmetric state (5,5) in the (a) 15L, (b) 30L, and (c) 50L radius corners.

The pressure coefficient distributions on the vehicle's underbody with a maximum inside bias operating state show that the inside split front wing element generates a stronger low-pressure region than the outside element. The pressure difference between these elements drives the formation of a vortex, which propagates along the floor and leaves a low-pressure trace. Additionally, the front wing settings influence downstream flow conditions, where high angles support low-pressure generation in the diffuser on the same side of the vehicle. This effect appears more pronounced in smaller corners, likely augmented by the flow alignment effects discussed for the maximum symmetric state.

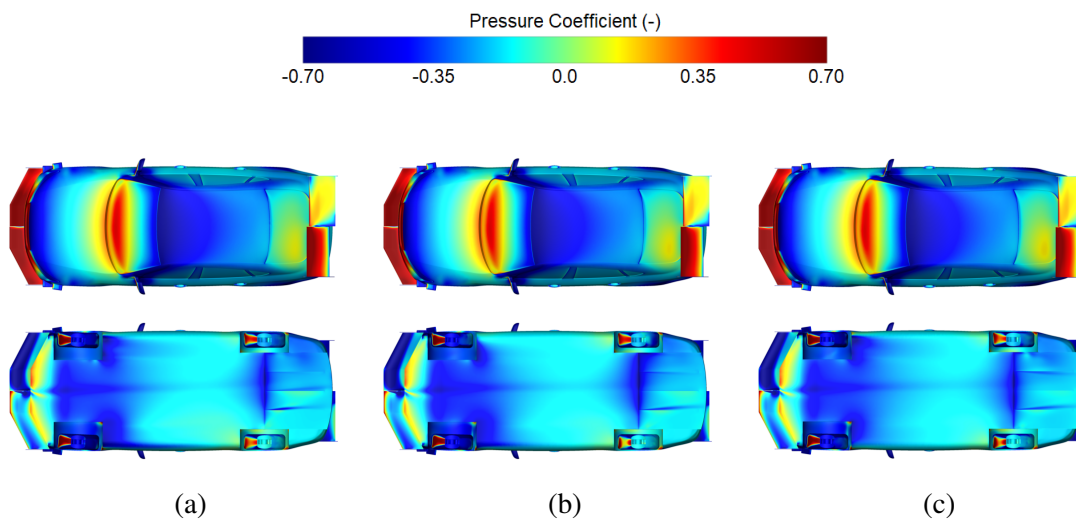


Figure 6.17: Pressure coefficient distribution on the split rear wing high-performance vehicle concept at its maximum inside bias operating state (1,5) in the (a) 15L, (b) 30L, and (c) 50L radius corners.

Having established an understanding of the functioning behind the active aerodynamic devices of the split rear wing high-performance vehicle concept, Figure 6.18 presents the influence of the operating states on the downforce acting on the inside and outside tyres in each cornering condition. As previously identified, flow conditions in tight corners have been shown to benefit downforce generation on the inside of the vehicle. Accordingly, the highest inside and lowest outside downforce coefficients are generally observed in the 15L radius corner. Conversely, the nearly symmetric flow conditions in the 50L radius corner result in more balanced magnitudes of inside and outside downforce coefficients.

Throughout the cornering conditions, the inside downforce coefficients increase with higher operating settings for the aerodynamic devices on the same side of the vehicle. However, for a given inside operating setting, the inside downforce coefficients also increase with increasing outside operating settings. The same response to operating settings for the aerodynamic devices on the opposite side of the vehicle is observed for the outside downforce coefficients, indicating that an amplifying aerodynamic interaction between aerodynamic devices on either side of the vehicle occurs.

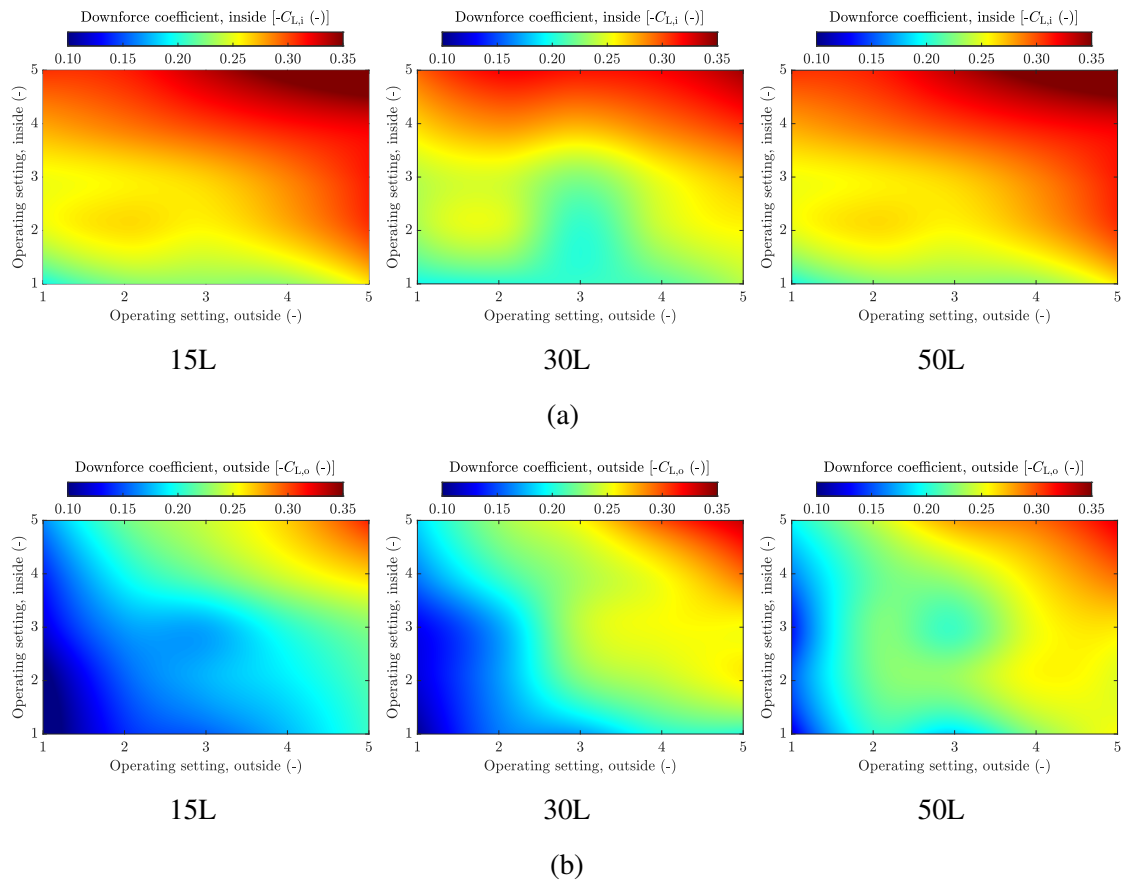


Figure 6.18: Downforce coefficients on the (a) inside and (b) outside tyres for the split rear wing high-performance vehicle concept in various cornering conditions, as a function of the inside and outside operating settings of the active aerodynamic systems.

The distribution of aerodynamic loads, influenced by the operating states of the active aerodynamic devices, induces an aerodynamic roll moment about the vehicle's roll axis. The aerodynamic roll moment coefficients, depicted in Figure 6.19, therefore provide an indication of the vehicle's AALB capability. In this analysis, positive coefficients refer to an aerodynamic roll moment toward the inside tyres and negative coefficients to an aerodynamic roll moment toward the outside tyres. Hence, a desirable AALB concept in symmetric straight-line conditions would not generate an aerodynamic roll moment at symmetric operating states, whereas inside and outside bias operating states would respectively generate positive and negative aerodynamic roll moments.

The 50L radius corner results show most resemblance to this described straight-line behaviour, where inside and outside bias operating states generally result in positive and negative aerodynamic roll moment coefficients. However, as previously identified, the asymmetric cornering conditions favour downforce on the inside of the vehicle. Consequently, positive aerodynamic roll moment coefficients are superior, especially in the high operating settings range, as evidenced by the contour line of zero aerodynamic roll. Positive aerodynamic roll moment coefficients become increasingly more dominant with reducing corner radii, corresponding to the high disparity previously seen between the downforce generated on the inside and outside tyres of the vehicle.

The influence of AALB on the maximum cornering velocity of the split rear wing high-performance vehicle concept is shown in Figure 6.20. Higher cornering velocities are generally achieved with inside bias compared to outside bias operating states, confirming their respective positive and negative impacts on tyre load distribution. Prior discussions focused on aerodynamic force and moment coefficients to compare aerodynamics in low- and high-speed cornering. However, as aerodynamic forces scale quadratically with velocity, subsequent analyses will show that AALB has a greater absolute impact in high-speed than in low-speed corners.

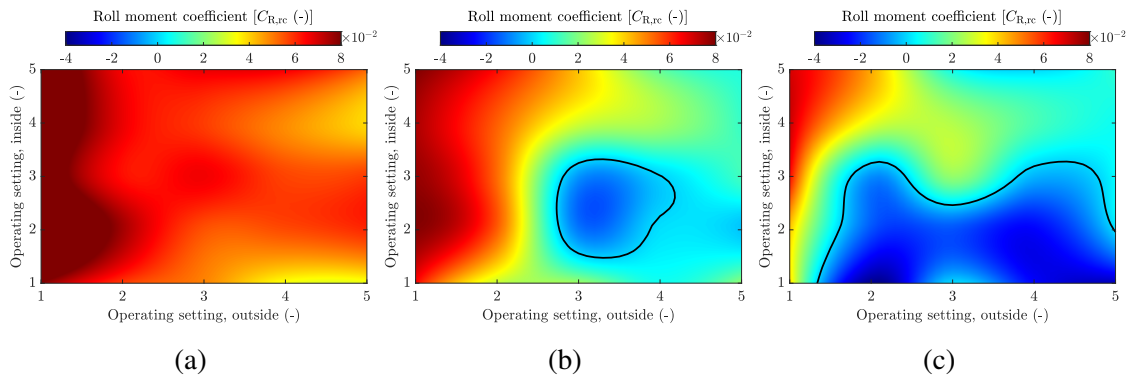


Figure 6.19: Aerodynamic roll moment coefficients about the vehicle's roll axis for the split rear wing high-performance vehicle concept in the (a) 15L, (b) 30L, and (c) 50L radius corners, as a function of the inside and outside operating settings of the active aerodynamic systems. Black contour lines indicate a zero aerodynamic roll moment.

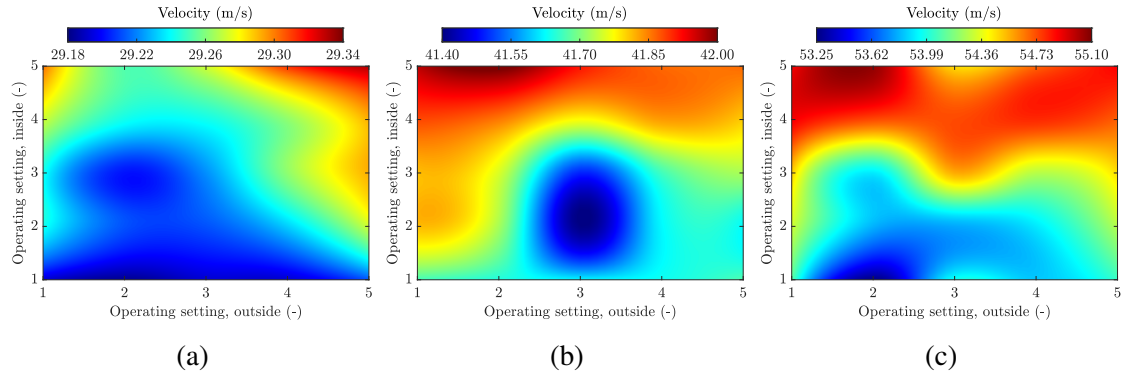


Figure 6.20: Maximum velocity for the split rear wing high-performance vehicle concept in the (a) 15L, (b) 30L, and (c) 50L radius corners, as a function of the inside and outside operating settings of the active aerodynamic systems.

In the 50L radius corner, the peak velocity of 55.09 m/s is reached at the (1.7,5) inside bias operating state, representing a 3.5% improvement over the minimum velocity of 53.25 m/s at operating state (2,1) and a 0.4% increase over the velocity of 54.87 m/s at the maximum symmetric operating state (5,5). Regarding lateral acceleration, this corresponds to a 7% improvement over operating state (2,1) and 0.8% over (5,5). The inside bias configuration achieves this peak performance while generating approximately 20% less downforce and 16% less drag, reducing the power required to overcome aerodynamic resistance by about 15% compared to the maximum symmetric operating state.

The peak velocity of 42.01 m/s in the 30L radius corner is achieved at the (1.8,5) inside bias operating state, providing an approximate 1.5% improvement over the minimum velocity of 41.40 m/s at operating state (3,2) and 0.4% over the velocity of 41.86 m/s at the maximum symmetric operating state (5,5). In terms of lateral acceleration, this corresponds to around 3.0% and 0.7% improvements over the (3,2) and (5,5) operating states, respectively. The inside bias configuration achieves this while generating approximately 19% less downforce and 16% less drag, requiring around 15% less power to overcome aerodynamic resistance.

Performance in the 15L radius corner exhibits the lowest sensitivity to AALB. While outside bias operating states result in the lowest cornering velocity and inside bias states enhance performance, the vehicle primarily benefits from a high downforce state, showing limited sensitivity to lateral load distribution. Consequently, a peak velocity of 29.33 m/s is achieved at the maximum symmetric operating state (5,5), increasing the minimum velocity of 29.18 m/s at operating state (2,1) by approximately 0.5%, corresponding to a 1% increase in lateral acceleration.

The performance differences between operating states are driven by the aerodynamic load distribution and its influence on tyre force generation. This is shown in Figure 6.21, which depicts the lateral tyre forces for the maximum symmetric (5,5) and optimal inside bias (1.7,5) operating states in the 50L radius corner. The inside bias state generates only a modest aerodynamic roll moment, reducing lateral load transfer by approximately 1.2% relative to the symmetric state. Combined with the high inside downforce, this reduces the vertical load disparity between the inside and outside tyres by about 2.9%.

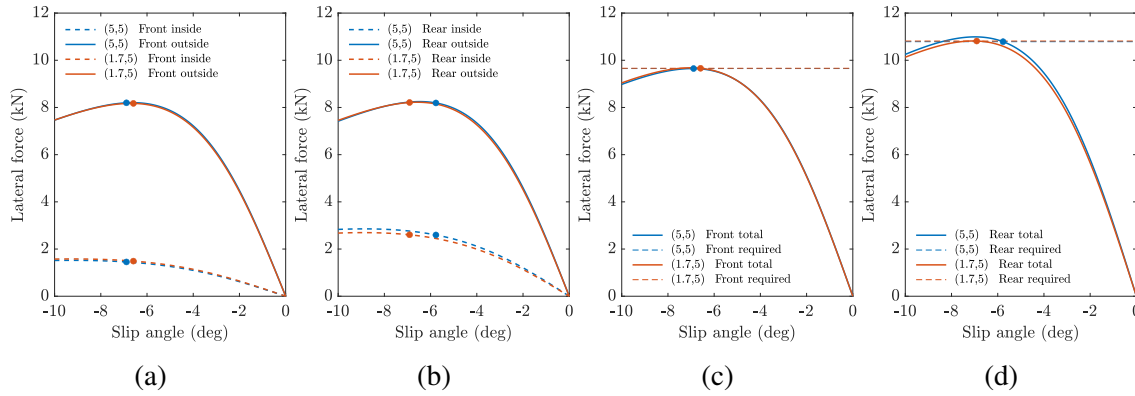


Figure 6.21: Comparison of lateral tyre forces from the split rear wing high-performance vehicle concept in its maximum symmetric (5,5) and optimal inside bias (1.7,5) operating states in the 50L radius corner, showing forces for (a) individual front tyres, (b) individual rear tyres, (c) combined front tyres, and (d) combined rear tyres.

The inside tyres, experiencing vertical loads of 800-1600 N, operate in the steep, high-sensitivity region of the vertical load curve, whereas the outside tyres, loaded at 7400-8000 N, operate near saturation, as previously discussed in Section 6.2.5.5. Prioritising load on the inside tyres therefore yields a net gain in lateral grip, which in this scenario predominantly benefits the otherwise grip-limited front axle under symmetric loading. However, the shifted aerodynamic balance and reduced total rear downforce require the rear tyres to operate at higher slip angles to sustain sufficient lateral force, becoming the grip-limited axle for the inside bias state. Consequently, while the inside bias state marginally enhances overall grip through effective load distribution, the modest aerodynamic roll moment limits its potential to deliver more significant cornering velocity gains.

6.3.2.3 Tilt Rear Wing

The downforce and drag coefficients of the tilt rear wing high-performance vehicle concept, presented in Figure 6.22, mainly increase with higher operating settings for the inside and outside active aerodynamic devices in each cornering condition. The rate at which downforce coefficients increase at low to medium operating settings is highest in the 50L radius corner, whereas the response to high operating settings is most substantial in the 15L radius corner.

The 15L radius corner results also demonstrate noticeable performance fluctuations at high inside and outside bias operating states. These aerodynamic performance variabilities are attributed to the interaction between the tilt rear wing and recirculation wake vortices. Prior work has demonstrated that these vortex structures arise in cornering conditions and intensify with reducing corner radii (13), thereby only manifesting as pronounced performance variations in the 15L radius corner. The interaction seems to be triggered by critical distances between the tilt rear wing and the vehicle body, influenced by the angle of attack and tilt angle settings. The interaction slightly reduces the low-pressure generation underneath the tilt rear wing and increases pressure drag on the vehicle's base, therefore resulting in relatively low-downforce and high-drag states.

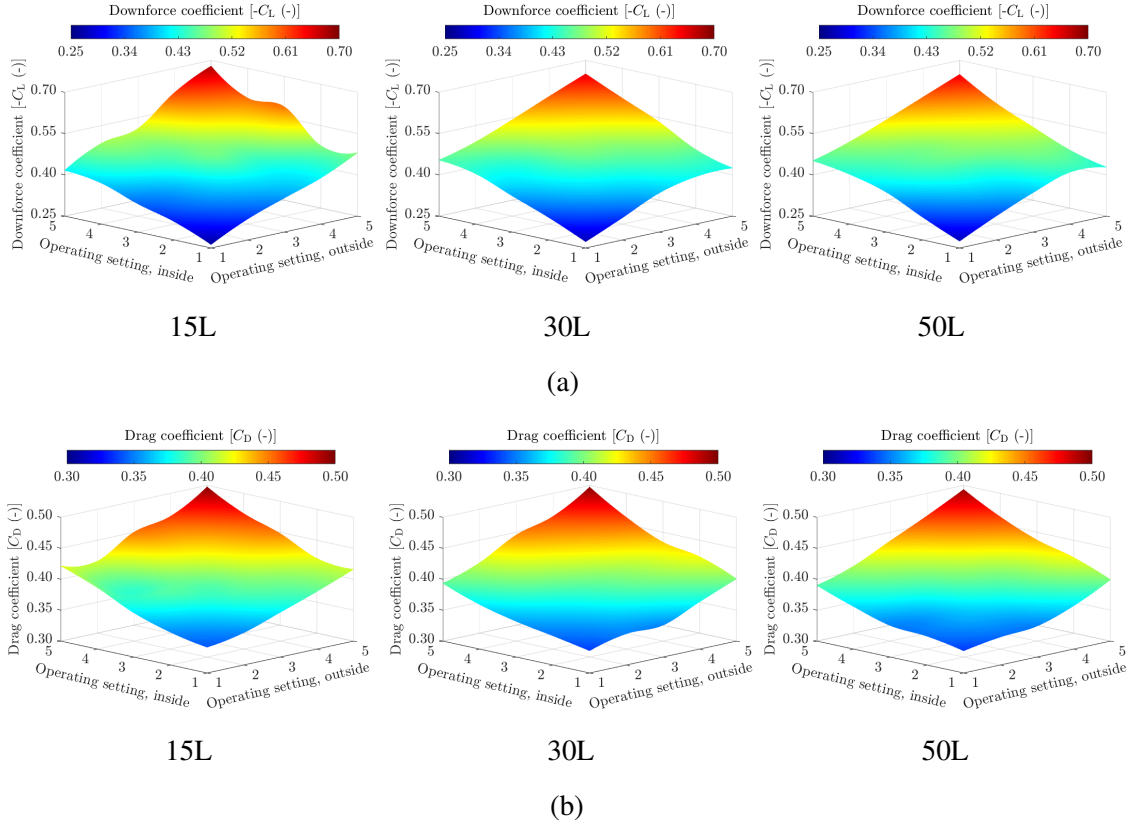


Figure 6.22: Downforce (a) and drag (b) coefficients for the tilt rear wing high-performance vehicle concept across various cornering conditions, as a function of the inside and outside operating settings of the active aerodynamic systems.

The front axle downforce distribution varies between approximately 50% to 56% throughout the operating states in each cornering condition, thereby remaining within the intended range of the control strategy. Quantitatively, the maximum downforce coefficient in the 15L radius corner is about 4.5% higher than in the 30L and 50L radius corners, while the maximum drag coefficient is less than 1% higher.

Similar to the split rear wing concept, pressure coefficient distributions on the maximum symmetric operating state reveal that performance variations across cornering conditions primarily occur on the underbody, as shown in Figure 6.23. Comparable features like the upstream pressurisation of the rear wing and low-pressure generation on the front wing elements are observable. Furthermore, the downstream development of the low-pressure region near the front axle and the asymmetric pressure distribution in the diffuser again indicate the variation in flow alignment between small and large radius corners.

Pressure coefficient distributions on the maximum inside bias state, presented in Figure 6.24, depict a relatively symmetric distribution across the tilt rear wing's span, despite its high tilt angle. However, the upstream pressurisation effect is primarily generated by the wing's outside section, which is closest to the vehicle body, and consequently less pronounced than for the maximum symmetric state. The increased proximity between the outside wing section and the vehicle body also induces local airflow acceleration, slightly promoting flow attachment on the lower surface of the downward-tilted wing section.

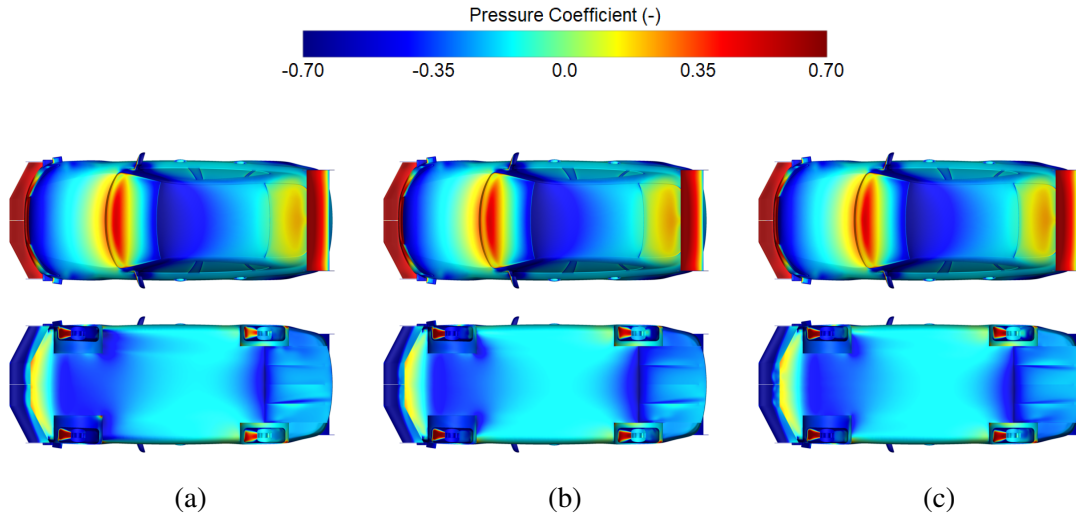


Figure 6.23: Pressure coefficient distribution on the tilt rear wing high-performance vehicle concept at its maximum symmetric state (5,5) in the (a) 15L, (b) 30L, and (c) 50L radius corners.

Similar findings to those of the split rear wing concept's maximum inside bias state are observed along the underbody. The low-pressure region on the inside front wing element is more substantial than on the outside element, driving the formation of a vortex which propagates along the floor. Furthermore, front wing operating settings influence downstream flow conditions, where high angles promote low-pressure generation in the diffuser section on the same side of the vehicle. This behaviour appears again more pronounced in smaller corners, likely amplified by the previously discussed flow alignment effects.

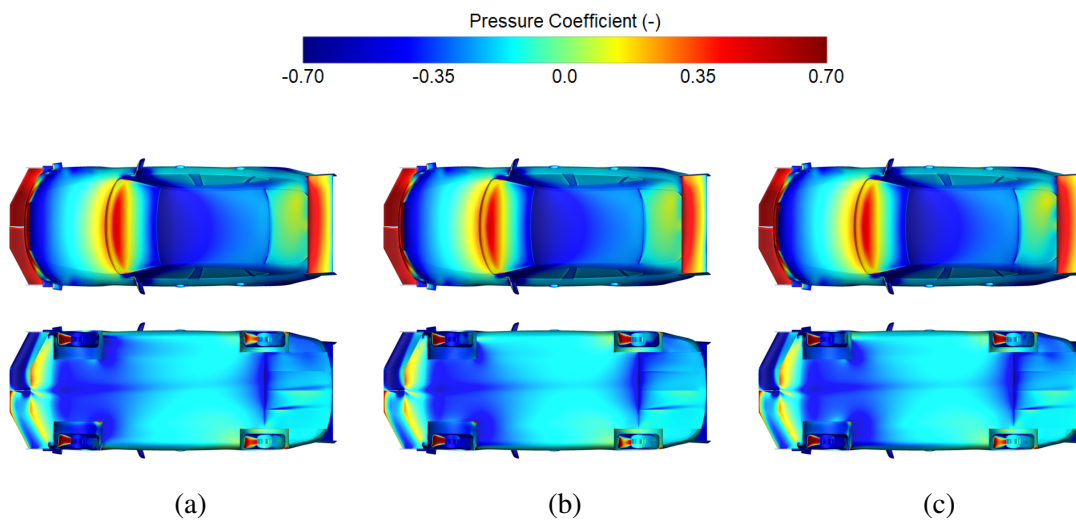


Figure 6.24: Pressure coefficient distribution on the tilt rear wing high-performance vehicle concept at its maximum inside bias operating state (1,5) in the (a) 15L, (b) 30L, and (c) 50L radius corners.

With the operational principles of the active aerodynamic devices of the tilt rear wing high-performance vehicle concept now established, Figure 6.25 shows the impact of the operating states on the downforce exerted on the inside and outside tyres throughout the different cornering conditions. As identified earlier, downforce generation on the inside of the vehicle is slightly promoted in tight corners, whereas more symmetrically distributed aerodynamic loads are generated in large radius corners which approximate symmetric straight-line conditions. Consequently, an increasing variability in the magnitudes of inside and outside downforce coefficients is observed with reducing corner radii.

Throughout the cornering conditions, inside and outside downforce coefficients increase with higher operating settings for the active aerodynamic devices on the respective sides of the vehicle. However, the inside downforce coefficients tend to reduce with higher outside operating settings for a constant inside operating setting and vice versa. This response is to be expected since the aerodynamic load bias to a single side of the vehicle reduces when the opposite operating setting is increased, generating more laterally balanced aerodynamic loads.

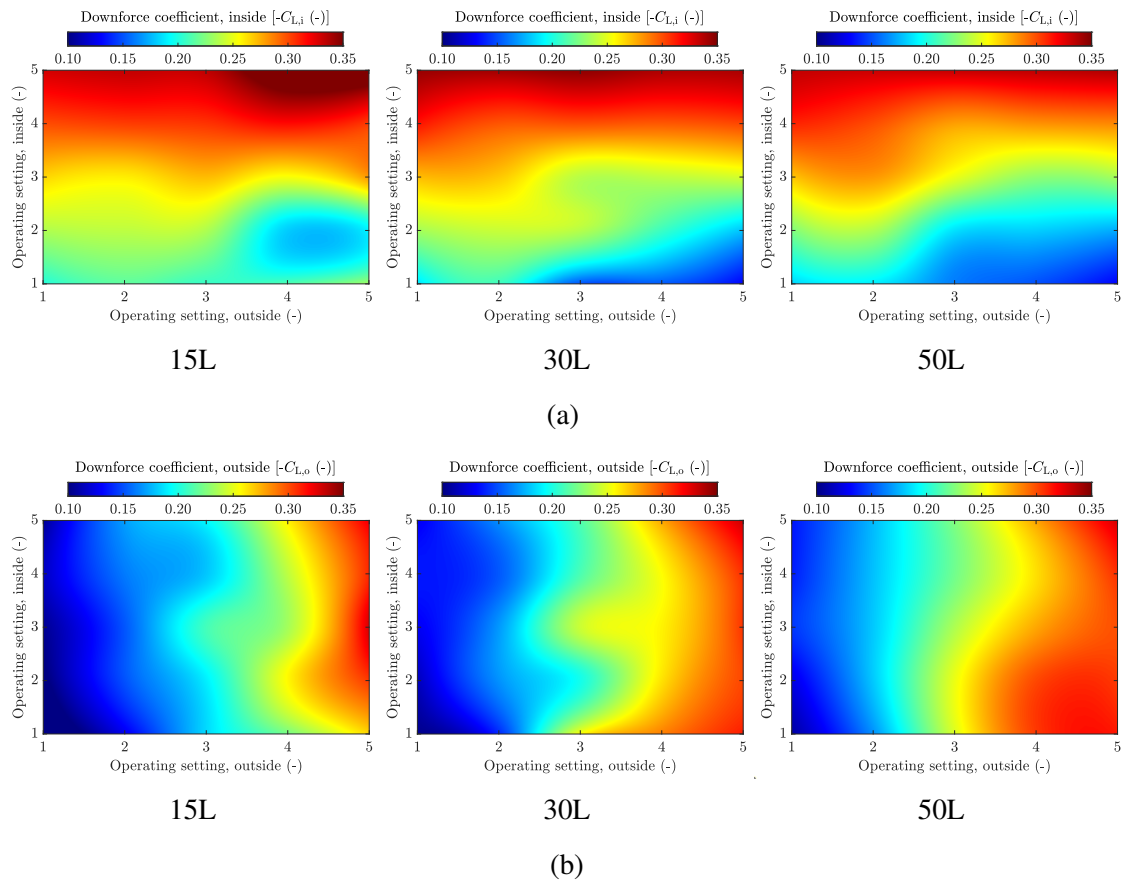


Figure 6.25: Downforce coefficients on the (a) inside and (b) outside tyres for the tilt rear wing high-performance vehicle concept in various cornering conditions, as a function of the inside and outside operating settings of the active aerodynamic systems.

The aerodynamic load distribution induces an aerodynamic roll moment about the vehicle roll axis, as presented in Figure 6.26, which provides insights into the vehicle's AALB capability. As mentioned before, positive coefficients refer to an aerodynamic roll moment toward the inside tyres and negative coefficients to an aerodynamic roll moment toward the outside tyres in this analysis. In straight-line conditions, a desired AALB concept with symmetric settings would not generate an aerodynamic roll moment, whereas inside and outside bias operating states would generate positive and negative aerodynamic roll moments, respectively.

Aerodynamic roll moment coefficients in the 50L radius corner are nearly identical to the desired response in straight-line conditions described above. The contour line of zero aerodynamic roll is, however, marginally below the diagonal with symmetric operating states, since the slight asymmetric conditions in this large corner radius still favour an inside bias aerodynamic load distribution. As previously identified, the bias toward inside downforce generation increases with reducing corner radii, also evidenced from the increasing positive aerodynamic roll moment coefficients across the operating states in tighter corners.

The impact of AALB on the maximum cornering velocity of the tilt rear wing high-performance vehicle concept is provided in Figure 6.27. Evidently, higher cornering velocities are generally observed for inside bias compared to outside bias operating states for each cornering condition. This emphasises the respective performance-enhancing and deteriorating effects of counteracting or reinforcing lateral load transfer during cornering.

As previously noted, the actual magnitudes of aerodynamic forces and moments increase quadratically with the vehicle's velocity as opposed to their respective non-dimensional coefficients discussed above. Hence, subsequent performance analyses will demonstrate larger absolute effects of AALB on cornering performance in high-speed corners compared to low-speed corners.

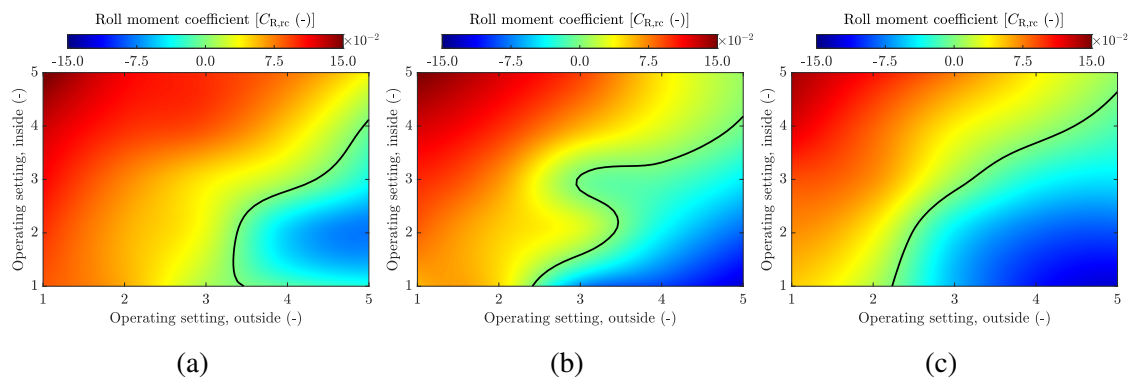


Figure 6.26: Aerodynamic roll moment coefficients about the vehicle's roll axis for the tilt rear wing high-performance vehicle concept in the (a) 15L, (b) 30L, and (c) 50L radius corners, as a function of the inside and outside operating settings of the active aerodynamic systems. Black contour lines indicate a zero aerodynamic roll moment.

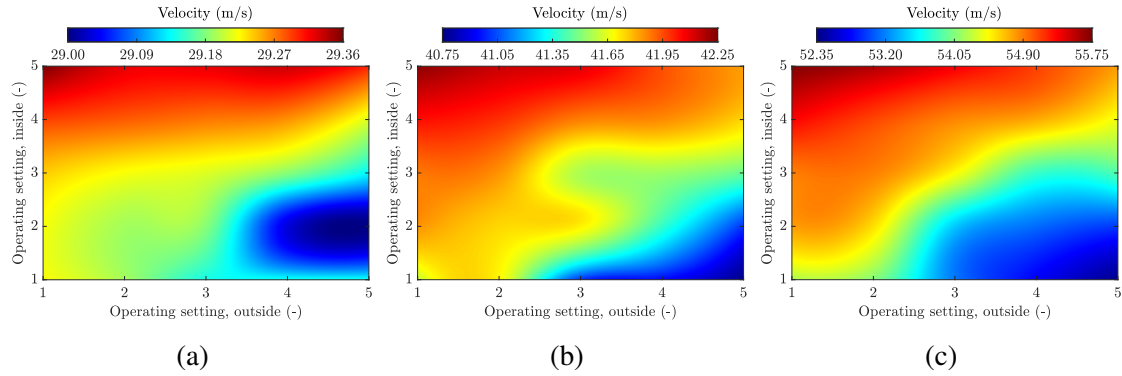


Figure 6.27: Maximum velocity for the tilt rear wing high-performance vehicle concept in the (a) 15L, (b) 30L, and (c) 50L radius corners, as a function of the inside and outside operating settings of the active aerodynamic systems.

In the 50L radius corner, the maximum inside bias operating state (1,5) provides a peak velocity of 55.73 m/s, which is approximately 6.4% higher than the minimum velocity of 52.38 m/s at operating state (5,1) and 1.7% higher than the velocity of 54.79 m/s at the maximum symmetric operating state (5,5). In terms of lateral acceleration, this corresponds to an increase of approximately 13.2% over operating state (5,1) and 3.5% over (5,5). The maximum inside bias state generates approximately 28% less downforce and 18% less drag, enabling it to reach peak velocity while requiring around 18% less power to overcome aerodynamic resistance compared to the maximum symmetric state.

The maximum inside bias state (1,5) achieves a peak velocity of 42.23 m/s in the 30L radius corner, representing a 3.6% increase over the minimum velocity of 40.76 m/s at operating state (5,1) and 1.0% over the velocity of 41.82 m/s at the maximum symmetric state (5,5). Regarding lateral acceleration, this corresponds to a 7.3% increase over state (5,1) and 2.0% over (5,5). Peak velocity at the maximum inside bias state is achieved with approximately 29% less downforce and 19% less drag, reducing the power required to overcome aerodynamic drag by around 18% compared to the maximum symmetric state.

The peak velocity of 29.36 m/s in the 15L radius corner is achieved at the maximum inside bias operating state (1,5), providing a 1.3% improvement over the minimum velocity of 29.00 m/s at operating state (4,8,2) and a 0.2% increase over the velocity of 29.30 m/s at the maximum symmetric operating state (5,5). In terms of lateral acceleration, this corresponds to a 2.5% improvement over operating state (4,8,2) and 0.4% over (5,5). The maximum inside bias state generates approximately 38% less downforce and 15% less drag, allowing it to reach peak velocity while requiring about 15% less power to overcome aerodynamic resistance compared to the maximum symmetric state.

The cornering performance variations arise from the differences in aerodynamic load distributions and their influence on tyre grip. Figure 6.28 presents the lateral tyre forces for the maximum downforce symmetric (5,5) and optimal inside bias (1,5) operating states in the 50L radius corner. In this scenario, the inside bias state reduces lateral load transfer by approximately 2.5% relative to the symmetric state, despite achieving a higher lateral acceleration. Combined with the prioritised inside downforce, this lowers the vertical load disparity between the inside and outside tyres by about 8.5%.

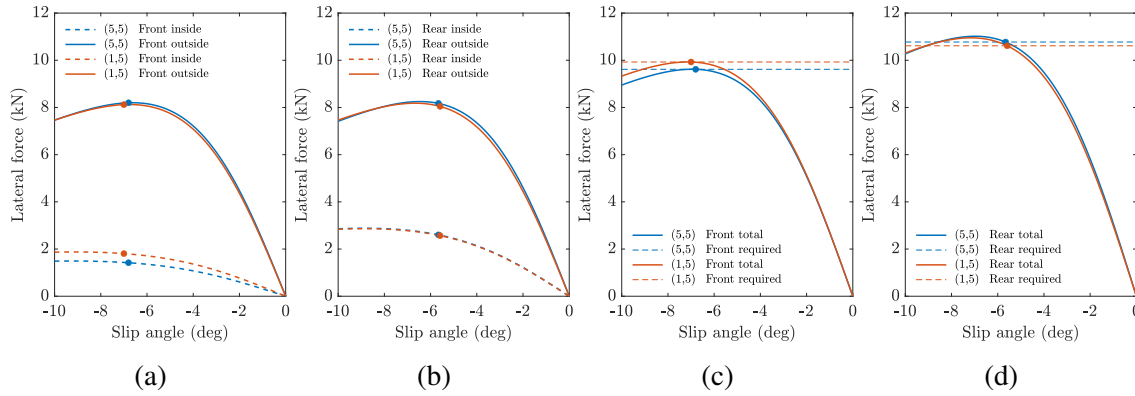


Figure 6.28: Comparison of lateral tyre forces from the tilt rear wing high-performance vehicle concept in its maximum symmetric (5,5) and optimal inside bias (1,5) operating states in the 50L radius corner, showing forces for (a) individual front tyres, (b) individual rear tyres, (c) combined front tyres, and (d) combined rear tyres.

The inside tyres, which experience vertical loads of around 800-1600 N, operate in the steep, high-sensitivity region of the vertical load curve, while the outside tyres, loaded at about 7200-8000 N, operate near saturation, as previously outlined in Section 6.2.5.5. Consequently, the prioritised inside downforce and reduced load transfer from the aerodynamic roll moment cause a net lateral force gain on the grip-limited front axle, as the front inside tyre gains grip more rapidly than the front outside tyre loses with the vertical load redistribution.

Although the inside bias state generates less total rear axle downforce than the maximum downforce symmetric state, the rear inside tyre maintains nearly identical vertical load and grip due to the asymmetric aerodynamic loading, while the rear outside tyre experiences only a minor reduction, resulting in a slight decrease in total rear axle lateral force. Yet, the inside bias state achieves a higher cornering velocity with this slightly reduced rear axle grip because of the tilt rear wing, which not only generates downforce but also a horizontal force component that lessens the lateral force demand on the rear tyres. As a result, the aerodynamic balance can be biased toward enhancing lateral tyre force generation on the grip-limited front axle.

6.3.2.4 Split Rear Wing vs Tilt Rear Wing

The split rear wing and tilt rear wing high-performance vehicle concepts both aim to enhance cornering performance by controlling vertical tyre forces through AALB. However, the underlying mechanisms to achieve AALB and the subsequent efficacy in improving cornering performance differ between these concepts.

The split rear wing concept primarily influences the aerodynamic load distribution by the amount of aerodynamic force generated vertically on either side of the vehicle. In contrast, the tilt rear wing concept is able to generate substantial aerodynamic forces in both vertical and horizontal directions, resulting in a range of aerodynamic roll moment coefficients that is double that of the split rear wing concept.

Both concepts are capable of influencing aerodynamic loads to favour downforce on the vehicle's inside tyres, resulting in positive aerodynamic roll moments. However, the tilt rear wing concept provides a more desirable response to the control strategy than the split rear wing concept. It demonstrates a more consistent transition of positive to negative aerodynamic roll moment coefficients at respective inside and outside bias operating states across the cornering conditions. This consistency is less evident in the split rear wing concept, due to increased flow interactions between the split elements.

The tilt rear wing concept outperforms the split rear wing concept in cornering performance, achieving higher velocities due to its superior AALB capability. In the 50L radius corner, it reaches approximately a 1.2% higher maximum velocity, translating to a 2.4% increase in peak lateral acceleration, while requiring nearly 10% less downforce and 2% less drag. Similarly, in the 30L radius corner, its maximum velocity is about 0.5% higher, reflecting an approximate 1% gain in peak lateral acceleration, achieved with around 12% less downforce and 2% less drag.

In the 15L radius corner, both concepts reach similar maximum velocities. However, the split rear wing needs to operate at the maximum downforce state to reach its peak velocity in such low-speed corners due to its limited AALB capability. In contrast, the tilt rear wing, with its enhanced AALB capability, reaches its peak velocity at the maximum inside bias operating state. This results in a comparable cornering velocity, but with approximately 39% less downforce and 14% less drag.

6.4 Conclusion

This study investigated the potential of active aerodynamic load balancing (AALB) to enhance cornering performance of high-performance vehicles. The AALB capabilities of two vehicle concepts, featuring an active split front wing and either a split or tilt rear wing, were analysed to assess the effectiveness of these active aerodynamic systems.

Cornering performance was evaluated using a quasi-steady static coupling of aerodynamics and dynamics, incorporating CFD-obtained aerodynamic data, lateral load transfer, and non-linear tyre behaviour to determine maximum cornering velocities. Aerodynamic data for the high-performance vehicle concepts were obtained under representative operating conditions, established through an initial performance assessment of a baseline high-performance vehicle using wind tunnel data.

Sufficient levels of AALB were achieved by both the split rear wing and tilt rear wing concepts to noticeably influence cornering performance. The findings supported the initial hypothesis of this study, demonstrating that inside bias operating states, intended to promote downforce on the inside tyres and counteract effects of lateral load transfer, improve cornering performance. In contrast, outside bias operating states, which favour downforce on the outside tyres and reinforce effects of lateral load transfer, were shown to deteriorate cornering performance.

Performance improvements were most substantial in high-speed corners, due to the quadratic increase of aerodynamic forces with the vehicle's velocity. Optimal inside bias operating states of the split rear wing and tilt rear wing concepts reached approximately 0.5% and 2% higher velocities in the highest-speed corner than the maximum symmetric operating states, respectively. Notably, through effectively distributing the aerodynamic loads, the optimal inside bias operating states achieved peak velocities using less downforce and drag compared to the maximum symmetric operating states. Consequently, the propulsion power to overcome drag was also reduced by about 15%-20% in each cornering condition.

Comparing both high-performance vehicle concepts, the tilt rear wing outperformed the split rear wing in terms of its ability to influence the aerodynamic load distribution and achieving peak cornering performance. It was able to generate twice as large aerodynamic roll moment coefficients, and exhibited a more consistent response to the control strategy with a more direct relation between the bias of an operating state and the aerodynamic roll moment magnitude.

The superior AALB capability was attributed to the tilt rear wing's ability to generate both vertical and horizontal aerodynamic force components to influence the aerodynamic balance. Conversely, the split rear wing concept was only able to influence the aerodynamic load distribution by varying the magnitude of aerodynamic forces in the vertical direction on either side of the vehicle. As a result, the tilt rear wing concept achieved the highest maximum cornering velocities, while using less propulsion power to overcome drag in each of the considered cornering conditions.

Overall, this study demonstrated the significant potential of utilising AALB through 2D active aerodynamic systems for enhancing cornering performance. The findings offer crucial insights into the underlying principles of AALB and the fundamental design mechanisms beneficial for effective control over aerodynamic loads. These results establish the foundation for future work on advanced active aerodynamic solutions for enhancing the performance and efficiency of next-generation high-performance automotive vehicles. Explorations considering other dynamic conditions or alternative control strategies could lead to new insights and potentially even greater performance gains, guiding the development of advanced and efficient vehicles.

Data Availability Statement

Experimental data supporting this study are openly available from Cranfield Online Research Data at: <https://dspace.lib.cranfield.ac.uk/handle/1826/22438>. (43)

References

1. Seward D. Race car design. Lavenham, UK: Palgrave; 2014.
2. Dixon JC. Tires, suspension and handling, second edition. 2nd edn. Warrendale, USA: SAE International; 1996.

3. Keogh J. The aerodynamic effects of the cornering flow conditions. PhD Thesis. University of New South Wales; 2016. Available from: <https://doi.org/10.26190/unsworks/19191>.
4. Keogh J, Barber T, Diasinos S, Doig G. The aerodynamic effects on a cornering Ahmed body. *Journal of Wind Engineering and Industrial Aerodynamics*. 2016;154:34-46. Available from: <https://doi.org/10.1016/j.jweia.2016.04.002>.
5. Nakashima T, Mutsuda H, Kanehira T, Tsubokura M. Fluid-dynamic force measurement of Ahmed model in steady-state cornering. *Energies*. 2020;13(24):6592. Available from: <https://doi.org/10.3390/en13246592>.
6. Heft A, Indinger T, Adams N. Introduction of a new realistic generic car model for aerodynamic investigations. SAE Technical Paper 2012-01-0168. 2012. Available from: <https://doi.org/10.4271/2012-01-0168>.
7. Josefsson E, Hagvall R, Urquhart M, Sebben S. Numerical analysis of aerodynamic impact on passenger vehicles during cornering. SAE Technical Paper 2018-37-0014. 2018. Available from: <https://doi.org/10.4271/2018-37-0014>.
8. Nakashima T, Tsubokura M, Okada Y, Nouzawa T, Kono R, Doi Y. Aerodynamic characteristics of a road vehicle in steady-state cornering. *Proceedings of the ASME/JSME/KSME 2015 Joint Fluids Engineering Conference*. Volume 1A: Symposia, Part 2. Seoul, South Korea: The American Society of Mechanical Engineers; 2015. Available from: <https://doi.org/10.1115/AJKFluids2015-17553>.
9. Kono R, Nakashima T, Tsubokura M, Okada Y, Nouzawa T. Aerodynamics simulation of a sedan-type road vehicle in cornering motion with roll angle. *Proceedings of the 34th AIAA Applied Aerodynamics Conference*. AIAA 2016-3726. Washington, USA: AIAA; 2016. Available from: <https://doi.org/10.2514/6.2016-3726>.
10. Albukrek C, Doddegowda P, Ivaldi A, Amodeo J, Bardoscia E. Unsteady flow analysis of a Formula type open wheel race car in cornering. SAE Technical Paper 2006-01-3661. 2006. Available from: <https://doi.org/10.4271/2006-01-3661>.
11. Nara K, Tsubokura M, Ikeda J. A numerical analysis of unsteady aerodynamics of Formula car during dynamic cornering motion. *Proceedings of the 32nd AIAA Applied Aerodynamics Conference*. AIAA 2014-3138. Atlanta, USA: AIAA; 2014. Available from: <https://doi.org/10.2514/6.2014-3138>.
12. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Experimental and numerical investigation of the aerodynamic characteristics of high-performance vehicle configurations under yaw conditions. *Physics of Fluids*. 2024;36(4):045112. Available from: <https://doi.org/10.1063/5.0196979>.
13. Rijns S, Teschner TR, Blackburn K, Brighton J. Effects of cornering conditions on the aerodynamic characteristics of a high-performance vehicle and its rear wing. *Physics of Fluids*. 2024;36(4):045119. Available from: <https://doi.org/10.1063/5.0204204>.

14. Forbes DC, Page GJ, Passmore MA, Gaylard AP. A fully coupled, 6 degree-of-freedom, aerodynamic and vehicle handling crosswind simulation using the DriveAer model. *SAE International Journal of Passenger Cars - Mechanical Systems*. 2016;9(2):710-22. Available from: <https://doi.org/10.4271/2016-01-1601>.
15. Tunay T, O'Reilly CJ, Drugge L. The significance of roll on the dynamics of ground vehicles subjected to crosswind gusts by two-way coupled simulation of aero- and vehicle dynamics. *Advances in Dynamics of Vehicles on Roads and Tracks. Lecture Notes in Mechanical Engineering*. IAVSD 2019. Gothenburg, Sweden: Springer, Cham; 2020. p. 1388-97. Available from: https://doi.org/10.1007/978-3-030-38077-9_160.
16. Tunay T. Two-way coupled aerodynamics and vehicle dynamics simulations of a heavy ground vehicle subjected to crosswind of various frequencies. *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*. 2023;237(6):1406-22. Available from: <https://doi.org/10.1177/09544070221088070>.
17. Forbes DC. Coupling road vehicle aerodynamics and dynamics in simulation. PhD Thesis. Loughborough University; 2017. Available from: <https://hdl.handle.net/2134/25565>.
18. Brandt A, Jacobson B, Sebben S. High speed driving stability of road vehicles under crosswinds: an aerodynamic and vehicle dynamic parametric sensitivity analysis. *Vehicle System Dynamics*. 2022;60(7):2334-57. Available from: <https://doi.org/10.1080/00423114.2021.1903516>.
19. Nakashima T, Tsubokura M, Matsuda S, Doi Y. Coupled analysis of unsteady aerodynamics and 6dof motion of a heavy duty truck in strong wind gusts. *Proceedings of the 29th AIAA Applied Aerodynamics Conference*. AIAA 2011-3672. Honolulu, Hawaii: AIAA; 2011. Available from: <https://doi.org/10.2514/6.2011-3672>.
20. Winkler N, Drugge L, Trigell AS, Efraimsson G. Coupling aerodynamics to vehicle dynamics in transient crosswinds including a driver model. *Computers & Fluids*. 2016;138:26-34. Available from: <https://doi.org/10.1016/j.compfluid.2016.08.006>.
21. Carbonne L, Winkler N, Efraimsson G. Use of full coupling of aerodynamics and vehicle dynamics for numerical simulation of the crosswind stability of ground vehicles. *SAE International Journal of Commercial Vehicles*. 2016;9(2):359-70. Available from: <https://doi.org/10.4271/2016-01-8148>.
22. McKay NJ, Gopalarathnam A. The effects of wing aerodynamics on race vehicle performance. *SAE Technical Paper 2002-01-3294*. 2002. Available from: <https://doi.org/10.4271/2002-01-3294>.
23. Hammad M, Qureshi K, He Y. Safety and lateral dynamics improvement of a race car using active rear wing control. *SAE Technical Paper 2019-01-0643*. 2019. Available from: <https://doi.org/10.4271/2019-01-0643>.

24. Ahangarnejad AH, Melzi S. Numerical analysis of the influence of an actively controlled spoiler on the handling of a sports car. *Journal of Vibration and Control*. 2018;24(22):5437-48. Available from: <https://doi.org/10.1177/1077546318754683>.
25. Diba F, Barari A, Esmailzadeh E. Handling and safety enhancement of race cars using active aerodynamic systems. *Vehicle System Dynamics*. 2014;52(9):1171-90. Available from: <https://doi.org/10.1080/00423114.2014.930158>.
26. Lin MT, Papadopoulos P. Race car aerodynamics simulation and transmitted tire reaction forces. *International Journal of Automotive Engineering*. 2022;13(4):169-76. Available from: https://doi.org/10.20485/jsaeijae.13.4_169.
27. Cai J, Kapoor S, Sikder T, He Y. Effects of active aerodynamic wings on handling performance of high-speed vehicles. *SAE Technical Paper 2017-01-1592*. 2017. Available from: <https://doi.org/10.4271/2017-01-1592>.
28. Kurec K, Remer M, Broniszewski J, Bibik P, Tudruj S, Piechna J. Advanced modeling and simulation of vehicle active aerodynamic safety. *Journal of Advanced Transportation*. 2019;(7308590). Available from: <https://doi.org/10.1155/2019/7308590>.
29. Broniszewski J, Piechna J. A fully coupled analysis of unsteady aerodynamics impact on vehicle dynamics during braking. *Engineering Applications of Computational Fluid Mechanics*. 2019;13(1):623-41. Available from: <https://doi.org/10.1080/19942060.2019.1616326>.
30. Broniszewski J, Piechna JR. Fluid-structure interaction analysis of a competitive car during brake-in-turn manoeuvre. *Energies*. 2022;15(8):2917. Available from: <https://doi.org/10.3390/en15082917>.
31. Kurec K, Remer M, Mayer T, Tudruj S, Piechna J. Flow control for a car-mounted rear wing. *International Journal of Mechanical Sciences*. 2019;152:384-99. Available from: <https://doi.org/10.1016/j.ijmecsci.2018.12.034>.
32. Piechna JR, Kurec K, Broniszewski J, Remer M, Piechna A, Kamieniecki K, et al. Influence of the car movable aerodynamic elements on fast road car cornering. *Energies*. 2022;15(3):689. Available from: <https://doi.org/10.3390/en15030689>.
33. Zavala D, Vicente W, Esquivel R, Salinas-Vazquez M. Numerical simulation of the effect of a centripetal wing spoiler on the flow aerodynamics of a sports car. *SAE Technical Paper 2023-01-5032*. 2023. Available from: <https://doi.org/10.4271/2023-01-5032>.
34. Szudarek M, Piechna J. CFD analysis of the influence of the front wing setup on a time attack sports car's aerodynamics. *Energies*. 2021;14(23):7907. Available from: <https://doi.org/10.3390/en14237907>.
35. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model. *SAE Technical Paper 2018-01-0725*. 2018. Available from: <https://doi.org/10.4271/2018-01-0725>.

36. Rijns S, Teschner TR, Blackburn K, Brighton J. Integrated numerical and experimental workflow for high performance vehicle aerodynamics. SAE Technical Paper 2024-01-5016. 2024. Available from: <https://doi.org/10.4271/2024-01-5016>.
37. Lanfrit M. Best practice guidelines for handling automotive external aerodynamics with Fluent. University of Southampton; 2005. Available from: https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf. (Accessed: 01 May 2022).
38. Saunders JW, Mansour RB. On-road and wind tunnel turbulence and its measurement using a four-hole dynamic probe ahead of several cars. SAE Technical Paper 2000-01-0350. 2000. Available from: <https://doi.org/10.4271/2000-01-0350>.
39. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal. 1994;32(8):1598-605. Available from: <https://doi.org/10.2514/3.12149>.
40. Velenis E, Frazzoli E, Tsiotras P. Steady-state cornering equilibria and stabilisation for a vehicle during extreme operating conditions. International Journal of Vehicle Autonomous Systems. 2010;8(2-4):217-41. Available from: <https://doi.org/10.1504/IJVAS.2010.035797>.
41. Pacejka HB. Tire and vehicle dynamics. 3rd edn. Oxford, UK: Butterworth-Heinemann; 2012. Available from: <https://doi.org/10.1016/B978-0-08-097016-5.00001-2>.
42. Avon Tyres. Technical data resources; 2024. Available from: <https://www.avontyres.com/en-gb/tyre-care/motorsport-technical-data/technical-data-resources/>. (Accessed: 12 April 2024).
43. Rijns S, Teschner TR, Blackburn K, Brighton J. Dataset DrivAer hp-F: force measurements at various rear wing angles of attack. Cranfield Online Research Data (CORD); 2024. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/22438>.

Chapter 7

Lap Time Performance Analysis of Active Aerodynamics

This chapter was originally published as:

Rijns S, Teschner TR, Blackburn K, Siampis E, Brighton J. Optimising vehicle performance with advanced active aerodynamic systems. *Vehicle System Dynamics*. 2025; 1-29. Available from: <https://doi.org/10.1080/00423114.2025.2505619>.

Abstract

This study investigates the performance potential of active aerodynamic systems on high-performance vehicles. Static and active aerodynamic configurations, including asymmetrically actuated systems, are evaluated to identify performance gains and the mechanisms driving these improvements. Vehicle performance is optimised using a minimum lap time simulation framework, which utilises a transient vehicle dynamics model and Computational Fluid Dynamics (CFD) derived aerodynamic data. Results indicate that configurations with greater aerodynamic adaptability enhance acceleration, braking, cornering, and straight-line performance, yielding notable lap time reductions compared to a static aerodynamic configuration. The asymmetrically controlled aerodynamic configuration achieves the highest lap time reduction of approximately 0.92 s (0.76%) due to its ability to modulate downforce both longitudinally and laterally. Optimal control strategies show that aerodynamic elements are actuated to balance vertical tyre load shifts resulting from load transfer, prioritising downforce on underloaded tyres in demanding scenarios like braking, cornering, and acceleration. Additionally, optimal design parameters for the brake, torque, and roll stiffness distributions shifted rearward as configurations provide greater control of aerodynamic loads on the rear axle. Overall, this research highlights the performance advantages of active aerodynamic systems and offers insights into the mechanisms underlying these enhancements, establishing a foundation for further innovations in the field.

Keywords: Active Aerodynamics; Performance Optimisation; Minimum Lap Time; Optimal Control; Vehicle Dynamics; CFD

7.1 Introduction

The pursuit of greater efficiency and performance continues to drive advancements in the automotive industry, especially within motorsport and high-performance vehicle sectors. Recently, active aerodynamic systems, which can dynamically adapt to a vehicle's operating condition, have gained significant attention, highlighted by the upcoming introduction of active aerodynamics in Formula One in 2026 (1). Such innovations not only have the potential to provide a competitive edge on the track, but also pave the way for future trends in automotive engineering.

This study explores the performance potential of active aerodynamic systems on high-performance vehicles. It evaluates static and active aerodynamic setups, including a configuration with asymmetric aerodynamic control capabilities, to determine performance enhancements and the underlying mechanisms driving these gains. The vehicle performance of each aerodynamic configuration is optimised using a comprehensive simulation framework, based on a detailed vehicle dynamics model and representative aerodynamic data, which minimises lap time by optimising the vehicle's trajectory, control inputs, and parameter combinations.

The performance analyses of various aerodynamic configurations require coupling of the vehicle's aerodynamics and dynamics. Among commonly used methods, quasi-steady static coupling, which utilises steady-state Computational Fluid Dynamics (CFD) data, is generally considered the most cost-effective for studies on passenger-sized vehicles (2, 3, 4, 5). This is also reflected by research on the influence of aerodynamic systems on vehicle handling, which has predominantly used quasi-steady static coupling.

Such studies have shown that aerodynamic devices positioned near a vehicle's front and rear axles can influence understeer and oversteer behaviour, thereby enhancing stability during dynamic manoeuvres (6, 7). When combined with an effective control strategy that balanced aerodynamic loads according to mechanical grip, these systems reduced lap times by approximately 3% (8). Additionally, studies on rear wings functioning as air brakes demonstrated improvements in stability during braking events and reductions in braking distance by over 6%, particularly at higher speeds (9, 10, 11).

Research on asymmetric active aerodynamic systems has also shown promising results. Studies on split wings, which can vary the angle of attack between the left- and right-side elements, demonstrated lateral downforce variations of 25%-60%, depending on the wing design (12, 13). By prioritising downforce on the inside tyres during cornering, these systems enhanced lateral stability during evasive manoeuvres, improved cornering velocity, and reduced lap times when paired with an effective control strategy (14, 15, 16, 17). These improvements became more pronounced at higher speeds and were further enhanced by an effective longitudinal downforce distribution.

Previous work on asymmetric active aerodynamic systems has highlighted the potential of tilt wings, which can laterally redistribute downforce and generate a centripetal force (18). Road tests showed that vehicles with tilt rear wings achieved a 13% higher cornering velocity compared to those without aerodynamic devices (19).

Additionally, a comparative study found that split and tilt wings increased cornering velocity by 0.5% and 2%, respectively, compared to conventional symmetric high down-force setups (20). The tilt wing outperformed the split rear wing due to its greater ability to influence vertical tyre loads, generate a centripetal force, and provide a more consistent response to control strategies.

The studies reviewed above have mainly relied on steady-state limit handling analyses in specific driving scenarios to investigate the impact of aerodynamic systems on vehicle performance. Other investigations have employed minimum lap time simulations, formulated as an Optimal Control Problem (OCP), aimed at minimising lap time by optimising control inputs while adhering to track constraints and vehicle performance limits (21, 22). Such simulations can provide a comprehensive view of vehicle performance, offering detailed insights into optimal driver inputs, racing lines, control of active systems, and vehicle design parameters.

Minimum lap time simulations with varying complexities in trajectory planning and dynamics modelling have effectively been used in prior studies to evaluate vehicle performance (23, 24, 25, 26, 27). Findings on conventional aerodynamics indicated that lap times varied by nearly 2% depending on the aerodynamic balance position, with a hypothetical variable aerodynamic balance providing an approximate 0.3% improvement over an optimised static balance (28, 29). Studies on static and active aerodynamic systems, combined with various propulsion configurations, demonstrated lap time improvements of about 0.7%-1% and showed how the aerodynamic systems were actuated to achieve these enhancements (30, 31).

Overall, the literature has highlighted the potential of active aerodynamic systems to enhance vehicle performance and stability, particularly in braking and cornering. However, existing research has largely relied on steady-state limit handling analyses, minimum lap time simulations with fixed-trajectory planning or quasi-steady-state dynamics, and simplified aerodynamic models. This leaves a significant gap in understanding the performance potential and underlying mechanisms driving the enhancements of more advanced active aerodynamic systems on high-performance vehicles, especially through a comprehensive free-trajectory and transient dynamics minimum lap time simulation approach.

This study addresses this gap by analysing static and active aerodynamic configurations, including a setup with asymmetric control capabilities designed to influence aerodynamic balance both longitudinally and laterally. The performance of each aerodynamic configuration is assessed over a lap around the Circuit de Barcelona-Catalunya, which encompasses a wide range of driving scenarios. Vehicle performance is optimised through a lap time simulation framework that leverages a detailed vehicle dynamics model and CFD-derived aerodynamic data, aiming to minimise lap time by simultaneously optimising the vehicle's trajectory, control inputs, and design parameters. Through comprehensive investigations, this study provides detailed insights into the performance gains provided by the active aerodynamic systems, their optimal control strategies to realise these enhancements, and shifts in optimised design parameters in response to these systems.

7.2 Methodology

7.2.1 Vehicle Geometry

This study utilises a high-performance vehicle concept, based on the *DrivAer hp-F* (32), a high-performance variant of the DrivAer Fastback automotive reference model (33). This high-performance vehicle concept, as introduced in Chapter 6 (20), has forebody strakes, a smooth underbody, a 10° multichannel diffuser, and is equipped with an active split front wing and tilt rear wing. The left and right front wing elements can vary their angle of attack between 0° and 10° . The rear wing can adjust its angle of attack between 0° and 30° , and its tilt angle by $\pm 12^\circ$. The high-performance vehicle concept is illustrated in Figure 7.1 and its main dimensions are listed in Table 7.2.

Four aerodynamic configurations, each based on this vehicle geometry, are examined to assess the impact of varying levels of aerodynamic adaptability on vehicle performance. The first configuration is classified as ‘static’, featuring a front and rear wing with fixed angular settings, which serves as a baseline for comparison. The second configuration maintains a fixed front wing angle while introducing active control of the rear wing angle, whereas the third configuration enables active control over both the front and rear wing angles. Through these active control capabilities, the second and third configurations can influence aerodynamic balance longitudinally.

Finally, the fourth configuration allows independent active control of the left and right front wing angles, as well as active control of the rear wing’s angle of attack and tilt angle. This asymmetric actuation functionality enables modulation of the aerodynamic balance both longitudinally and laterally, which is therefore classified as a ‘2D active aerodynamic system’.

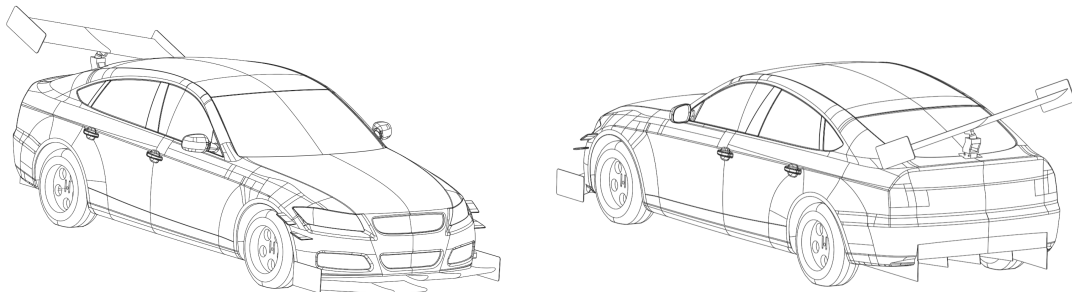


Figure 7.1: Illustration of the high-performance vehicle concept featuring an active split front wing and active tilt rear wing. The aerodynamic systems are displayed in an asymmetric operating setting to illustrate the design.

7.2.2 Computational Fluid Dynamics Simulations

The vehicle's aerodynamic performance data is obtained through CFD simulations, which adopt the authors' well-established methodology used in Chapters 5 and 6 (34, 20). This methodology is founded on a comprehensive grid convergence study, turbulence model evaluation, and extensive validation with wind tunnel data and high-fidelity Delayed Detached Eddy Simulations (DDES) using a wall-resolved mesh with a $y^+ < 1$, as presented in Chapter 2 (35).

The methodology has demonstrated its effectiveness across multiple aerodynamic studies on high-performance vehicles in diverse operating conditions (20, 34, 36, 37), offering a good balance between computational efficiency and solution fidelity. This study presents the essential details of this methodology with respect to the specification of the computational domain, meshing strategy, and solving method, with further details available in the referenced works.

7.2.2.1 Domain and Boundary Conditions

The vehicle is simulated within a rectangular computational domain, illustrated in Figure 7.2. It features a velocity-inlet 3 car-lengths (L) upstream and a pressure-outlet $5L$ downstream of the vehicle, following modelling guidelines for automotive aerodynamics (38). Side and top walls, positioned $1.5L$ from the vehicle, are assigned symmetry conditions to prevent boundary layer growth. Moving ground and wheel rotation velocities, listed along with other simulation settings in Table 7.1, match the freestream velocity of 29 m/s, equivalent to a Reynolds number of 8.88×10^6 . Turbulence conditions follow on-road measurements (39), ensuring representative operating conditions.

7.2.2.2 Mesh

The overset mesh approach used in Chapters 5 and 6 (34, 20) is deployed to construct the computational domain with individual unstructured poly-hexcore meshes in ANSYS Fluent Meshing. This method interpolates data in overlapping mesh regions, enabling targeted transformations of aerodynamic device component meshes without needing to re-mesh the entire domain.

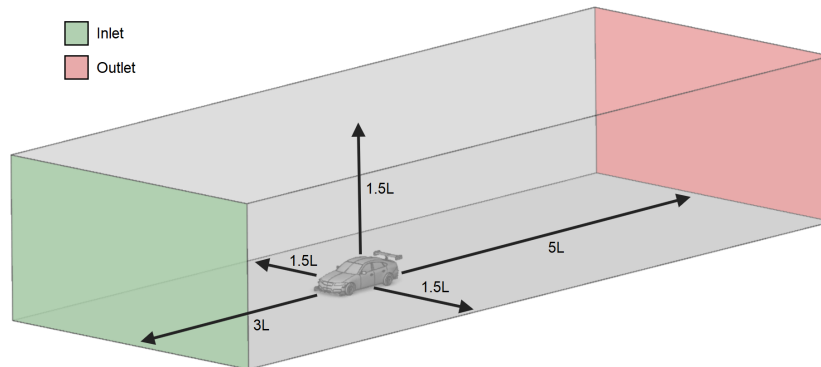


Figure 7.2: Illustration of the computational domain.

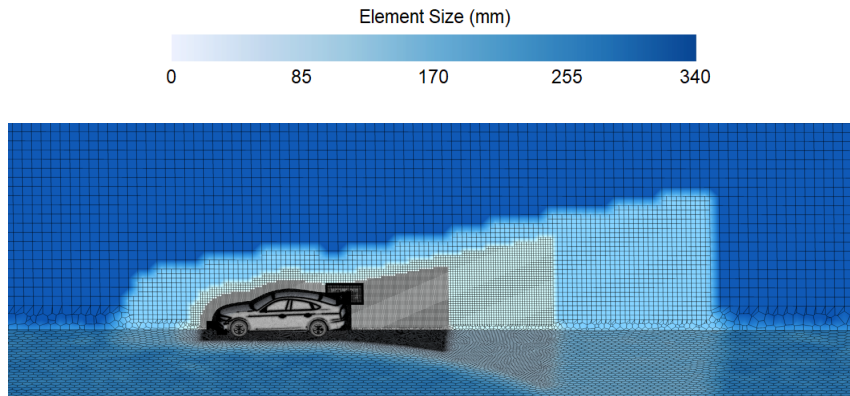
Table 7.1: Overview of the CFD simulation settings.

Setting	Value	Unit
Inlet & Outlet		
Velocity-inlet	29	m/s
Turbulent intensity	4	%
Turbulent length scale	2.5	m
Pressure-outlet	0	Pa
Moving walls		
Ground plane, <i>translational</i>	29	m/s
Wheels, <i>rotational</i>	91.3	rad/s
Air properties		
Density	1.204	kg/m ³
Dynamic viscosity	1.813e-5	kg/(m·s)
Temperature	293	K

Mesh settings include a base and vehicle surface element size of approximately 7.25% and 0.45% of the vehicle's length, respectively. To enhance resolution on aerodynamic devices and small geometric features, surface elements are locally refined by over 80%. A medium near-wall treatment with a y^+ range of 120-150 and 4-6 inflation layers is applied, adhering to ANSYS Fluent's (40) recommended range for the selected turbulence model. Additionally, refinement zones are included to improve resolution in the vehicle's near-field and wake regions. The final mesh, shown in Figure 7.3, consists of approximately 23.9 million cells, with over 99% achieving an orthogonal quality above 0.7.

7.2.2.3 Solve

Reynolds-Averaged Navier-Stokes (RANS) simulations are conducted using the $k-\omega$ SST turbulence model (41) in ANSYS Fluent, solving the continuity, momentum, and transport equations for turbulent kinetic energy and specific dissipation rate. Pressure-velocity coupling uses the coupled scheme, with gradients computed via the least-squares cell-based method. Pressure is discretised using a second-order scheme, while momentum and turbulence transport equations are solved with a second-order upwind scheme.

Figure 7.3: Element sizes near the vehicle, shown using a slice at $y = 0$ (symmetry).

Convergence is evaluated by monitoring the moving means of lift, drag, and side force coefficients, which typically stabilised within 200 iterations. Residuals are monitored to ensure numerical stability, with continuity, turbulent kinetic energy, and specific dissipation rate residuals stabilising below 1×10^{-2} , and x-, y-, and z-velocity residuals below 1×10^{-6} within the first 200 iterations. Each simulation therefore runs for 750 iterations, averaging data over the final 450 iterations. Simulations are performed on two AMD EPYC 7543 processor nodes, utilising 128 cores, with an average runtime of 5 hours per simulation.

7.2.2.4 Aerodynamic Systems Interdependencies

Prior to executing the full simulation set for all wing angle combinations, an initial interdependency analysis assesses whether the front wing angle α_{FW} (deg), rear wing angle α_{RW} (deg), and rear wing tilt angle α_{TW} (deg) independently influence aerodynamic performance. Simulations at minimum, median, and maximum angles are performed to create a combination space. Interdependencies are quantified using the mean absolute deviation (MAD) of combination sweeps relative to single-parameter sweeps with other parameters set to zero.

The aerodynamic effects of each parameter are shown in Figure 7.4, with shaded bars indicating the maximum MAD. MAD values for lift, drag, and side forces remain below 1.5%, 0.5%, and 10% of their respective maximum force coefficients, though their absolute magnitudes are of similar scale. Treating the aerodynamic inputs as independent variables introduces an average absolute force coefficient uncertainty of 5×10^{-3} , less than 1% of the vehicle's maximum force coefficient, while reducing the required simulations by over 97% from 1331 to 33 when sampled at finer one-tenth intervals of each parameter's maximum angle. This approach streamlines the computational process while preserving key aerodynamic effects, aligning with the study's objectives.

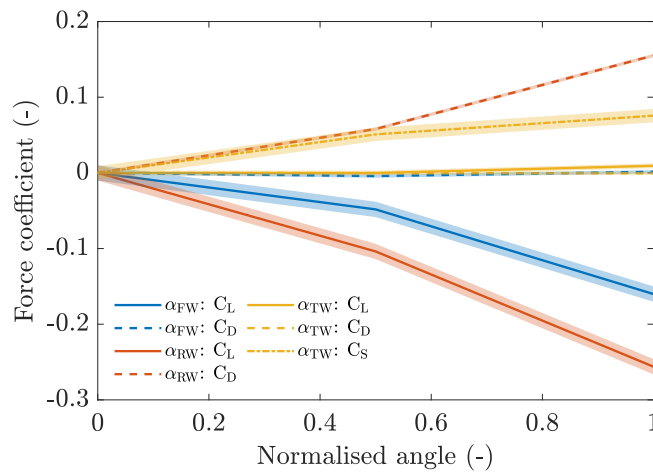


Figure 7.4: Influence of front wing angle, rear wing angle, and rear wing tilt on the vehicle's lift, drag, and side force coefficients. Shaded bars represent the mean absolute deviation (MAD), indicating the degree of parameter interdependence.

7.2.3 Vehicle Model

This study employs a seven degree-of-freedom planar vehicle model, illustrated in Figure 7.5, to simulate the impact of active aerodynamic systems on vehicle performance, building on established mathematical formulations (30, 42). The model incorporates both longitudinal and lateral load transfer, along with non-linear tyre characteristics, which are essential for capturing the strong relation between aerodynamic loads and tyre force generation. Table 7.2 provides an overview of the fixed vehicle parameters used in the model, which are based on the vehicle's geometry and representative specifications for sports and racing vehicles from Dixon (43).

Trajectory tracking is performed in a curvilinear coordinate system to align with the optimisation framework that will be discussed in Section 7.2.4. The vehicle's position is expressed in terms of its travelled distance along the track's centre line s (m), the distance perpendicular to the track's centre line n (m), and the angle relative to the track's centre line tangent ϵ (rad), as expressed by Eqs. (7.1)-(7.3). Here, t (s) denotes time, κ (m^{-1}) the curvature of the track's centre line, v_x (m/s) the vehicle's longitudinal velocity, v_y (m/s) the vehicle's lateral velocity, and r (rad/s) the vehicle's yaw rate.

$$\frac{d}{dt}s = \frac{v_x \cos(\epsilon) - v_y \sin(\epsilon)}{1 - n\kappa} \quad (7.1)$$

$$\frac{d}{dt}n = v_x \sin(\epsilon) + v_y \cos(\epsilon) \quad (7.2)$$

$$\frac{d}{dt}\epsilon = r - \kappa \frac{d}{dt}s \quad (7.3)$$

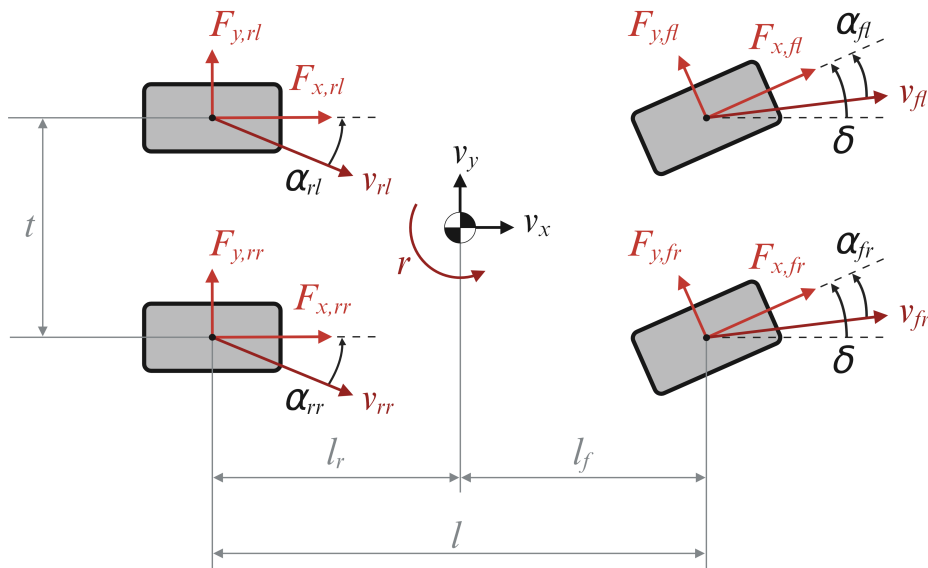


Figure 7.5: Schematic illustration of the vehicle model.

Table 7.2: Overview of the vehicle parameters.

Parameter	Value	Unit	Description
L	4.613	m	Length
W	1.792	m	Width
H	1.397	m	Height
A_{ref}	2.160	m ²	Frontal area
h_0	0.130	m	Ride height
m	1565	kg	Total mass
m_s	1375	kg	Sprung mass
$m_{u,f}$	90	kg	Front unsprung mass
$m_{u,r}$	100	kg	Rear unsprung mass
I_z	1450	kg m ²	Yaw moment of inertia
l	2.786	m	Wheelbase
l_f	1.393	m	Distance between front axle and COG
l_r	1.393	m	Distance between rear axle and COG
t_w	1.520	m	Track width
h_{COG}	0.500	m	Height of COG
$h_{u,f}$	0.297	m	Height of front unsprung mass
$h_{u,r}$	0.297	m	Height of rear unsprung mass
h_f	0.070	m	Height of front roll centre
h_r	0.110	m	Height of rear roll centre
h_{RW}	1.280	m	Height of rear wing
d	0.410	m	Distance between COG and roll axis
R_w	0.314	m	Wheel radius
I_w	0.9	kg m ²	Wheel rotational inertia
k_ϕ	1.146×10^5	Nm/rad	Total roll stiffness
$P_{e,\text{max}}$	456	kW	Maximum electric motor power
$T_{e,\text{max}}$	602	Nm	Maximum electric motor torque
$\Omega_{e,\text{max}}$	17750	RPM	Maximum electric motor speed
γ_g	7.252	-	Final gear ratio

Section 7.2.4 will motivate the use of travelled distance as the independent variable in the optimisation simulations, replacing time to reduce computational complexity and enhance efficiency. Following the methodology by Casanova (21), time-dependent equations are transformed into distance-dependent formulations by introducing the scaling factor S_f , which represents the inverse (dt/ds) of Eq. (7.1). This transformation, applied in Eqs. (7.4) and (7.5), ensures that the equations remain consistent within the distance-based optimisation framework.

$$\frac{d}{ds}n = \left(v_x \sin(\epsilon) + v_y \cos(\epsilon) \right) S_f \quad (7.4)$$

$$\frac{d}{ds}\epsilon = S_f r - \kappa \quad (7.5)$$

The vehicle's equations of motion, expressed as a function of travelled distance, are described by Eqs. (7.6)-(7.8). Throughout these equations, subscripts are used to denote specific components, where k represents the longitudinal (x), lateral (y), or vertical (z) direction, i specifies the front (f) or rear (r) axle location, and j indicates the left (l) or right (r) wheel position.

In these equations, $F_{k,ij}$ (N) denotes the tyre forces, δ (rad) the steering angle, m (kg) the vehicle's total mass, F_R (N) the rolling resistance force, and I_z (kg m²) the yaw moment of inertia. The parameter l_i (m) defines the distance from each axle to the centre of gravity (COG) and t_w (m) the vehicle's track width. Additionally, F_D (N) represents the total aerodynamic drag force opposing the vehicle's motion, while F_S (N) and $F_{S,i}$ (N) correspond to the total aerodynamic side force and its components acting on the front and rear axles, with their positive direction matching that of the lateral tyre forces.

$$\begin{aligned} \frac{d}{ds}v_x = & \left[(F_{x,fl} + F_{x,fr}) \cos(\delta) - (F_{y,fl} + F_{y,fr}) \sin(\delta) \right. \\ & \left. + F_{x,rl} + F_{x,rr} + m v_y r - F_D - F_R \right] \frac{S_f}{m} \end{aligned} \quad (7.6)$$

$$\begin{aligned} \frac{d}{ds}v_y = & \left[(F_{y,fl} + F_{y,fr}) \cos(\delta) + (F_{x,fl} + F_{x,fr}) \sin(\delta) \right. \\ & \left. + F_{y,rl} + F_{y,rr} - m v_x r + F_S \right] \frac{S_f}{m} \end{aligned} \quad (7.7)$$

$$\begin{aligned} \frac{d}{ds}r = & \left[\left((F_{x,fl} + F_{x,fr}) \sin(\delta) + (F_{y,fl} + F_{y,fr}) \cos(\delta) + F_{S,f} \right) l_f \right. \\ & - \left(F_{y,rl} + F_{y,rr} + F_{S,r} \right) l_r + \left(F_{x,rr} - F_{x,rl} \right) \frac{t_w}{2} \\ & \left. + \left((F_{x,fr} - F_{x,fl}) \cos(\delta) + (F_{y,fl} - F_{y,fr}) \sin(\delta) \right) \frac{t_w}{2} \right] \frac{S_f}{I_z} \end{aligned} \quad (7.8)$$

Wheel dynamics are expressed by Eq. (7.9), where Ω_{ij} (rad/s) represents the wheel's angular velocity, T_{ij} (Nm) the applied wheel torque, R_w (m) the wheel radius, and I_w (kg m²) the wheel's rotational inertia.

$$\frac{d}{ds}\Omega_{ij} = \left(T_{ij} - F_{x,ij} R_w \right) \frac{S_f}{I_w} \quad (7.9)$$

7.2.3.1 Tyres

The longitudinal and lateral tyre forces are dependent on the vertical forces acting on the tyres. These vertical loads are described by Eq. (7.10), where the first term represents the static weight distribution, with g (m/s²) denoting the gravitational acceleration, $m_{u,i}$ (kg) the unsprung masses, m_s (kg) the sprung mass, and l (m) the wheelbase length.

Additionally, $F_{L,ij}$ (N) indicates the aerodynamic lift force acting on each individual wheel, and $\Gamma_{x,ij}$ (N) and $\Gamma_{y,ij}$ (N) denote the components of longitudinal and lateral load transfer experienced by each individual wheel.

$$F_{z,ij} = \frac{1}{2}g \left(m_{u,i} + m_s \frac{l_i}{l} \right) - F_{L,ij} \pm \Gamma_{x,ij} \pm \Gamma_{y,ij} \quad (7.10)$$

The longitudinal and lateral tyre forces are modelled using the Pacejka '96 model (44), as expressed by Eqs. (7.11) and (7.12). The Pacejka coefficients, derived from 250/570R13 Avon British F3 tyre data at 0° camber (45), are detailed in Table 7.3.

$$F_{x,ij} = D_{x,ij} \sin \left[C_x \tan^{-1} \left\{ B_x s_{ij} - E_x \left(B_x s_{ij} - \tan^{-1}(B_x s_{ij}) \right) \right\} \right] \quad (7.11)$$

$$F_{y,ij} = D_{y,ij} \sin \left[C_y \tan^{-1} \left\{ B_y \alpha_{ij} - E_y \left(B_y \alpha_{ij} - \tan^{-1}(B_y \alpha_{ij}) \right) \right\} \right] \quad (7.12)$$

The peak value $D_{k,ij}$ (-) is dependent on the tyre's friction coefficient μ_{ij} (-), which are both load dependent as described by Eqs. (7.13) and (7.14). Here, μ_0 (-) denotes the tyre's coefficient of friction at the nominal vertical wheel load F_{z0} (N), and P_{D2} (-) represents the Pacejka coefficient for variation of friction with load.

$$D_{k,ij} = \mu_{ij} F_{z,ij} \quad (7.13)$$

$$\mu_{k,ij} = \mu_0 + P_{D2} \frac{F_{z,ij} - F_{z0}}{F_{z0}} \quad (7.14)$$

The longitudinal and lateral forces produced by each tyre are dependent on the slip ratio s_{ij} (-) and slip angle α_{ij} (rad), described by Eqs. (7.15)-(7.17). Here, v_{ij} (m/s) denotes the effective velocity of the tyre and is expressed by Eq. (7.18).

$$\alpha_{fj} = \delta - \tan^{-1} \left(\frac{r l_f + v_y}{v_x \pm r \frac{t_w}{2}} \right) \quad (7.15)$$

$$\alpha_{rj} = \tan^{-1} \left(\frac{r l_r - v_y}{v_x \pm r \frac{t_w}{2}} \right) \quad (7.16)$$

$$s_{ij} = \frac{R_w \Omega_{ij} - v_{ij}}{v_{ij}} \quad (7.17)$$

$$v_{ij} = \cos(\alpha_{ij}) \sqrt{\left(v_y \pm r l_i \right)^2 + \left(v_x \pm r \frac{t_w}{2} \right)^2} \quad (7.18)$$

Finally, the rolling resistance is modelled using a constant road friction coefficient C_f (-) and the vehicle's total vertical load, as expressed by Eq. (7.19), incorporating aerodynamic load effects on rolling resistance.

$$F_R = C_f (F_{z,fl} + F_{z,fr} + F_{z,rl} + F_{z,rr}) \quad (7.19)$$

Table 7.3: Overview of the tyre model coefficients.

Coefficient	Value	Description
F_{z0}	4905	Nominal Load (N)
μ_0	1.41	Friction coefficient
P_{D2}	-0.6	Variation of friction with load
B_x	6	Stiffness factor, longitudinal
C_x	2.3	Shape factor, longitudinal
E_x	0.9	Curvature factor, longitudinal
B_y	6	Stiffness factor, lateral
C_y	2.5	Shape factor, lateral
E_y	0.5	Curvature factor, lateral

7.2.3.2 Load Transfer

Longitudinal and lateral load transfer occur due to inertial forces which resist the traction forces produced by the tyres. These inertial forces act at the centre of gravity height h_{COG} (m), creating an overturning moment that causes load transfer. The total longitudinal load transfer Γ_x (N) is described by Eq. (7.20), where the nominal drag force F_D^0 (N), which represents the vehicle's drag without any aerodynamic device actuation, is also assumed to act at the centre of gravity height. Section 7.3.1 will discuss that additional drag from the aerodynamic systems F_D^{aero} (N) is primarily produced by the rear wing and therefore assumed to act at the rear wing height h_{RW} (m). The longitudinal load transfer experienced by each wheel is modelled as half of the total transferred load.

$$\Gamma_x = \left[(F_{x,fl} + F_{x,fr}) \cos(\delta) - (F_{y,fl} + F_{y,fr}) \sin(\delta) + F_{x,rl} + F_{x,rr} + F_D^0 \right] \frac{h_{COG}}{l} + F_D^{aero} \frac{h_{RW}}{l} \quad (7.20)$$

The total lateral load transfer Γ_y (N) is expressed by Eq. (7.21). Similar to drag, aerodynamic side force F_S (N) is predominantly generated by the rear wing and is therefore assumed to act at the rear wing height.

$$\Gamma_y = \left[(F_{x,fl} + F_{x,fr}) \sin(\delta) + (F_{y,fl} + F_{y,fr}) \cos(\delta) + F_{y,rl} + F_{y,rr} \right] \frac{h_{COG}}{t_w} + F_S \frac{h_{RW}}{t_w} \quad (7.21)$$

The distribution of lateral load transfer across the front and rear axles is expressed by Eqs. (7.22)-(7.24), where $C_{\Gamma y,i}$ (-) denotes the axle-specific fraction of total lateral load transfer. The equations consider contributions from the unsprung masses, the sprung mass via the suspension links, and the roll couple of the sprung mass through the springs. It depends on vehicle parameters such as unsprung mass heights $h_{u,i}$ (m), roll centre heights h_i (m), and the distance between the COG and the roll axis d (m).

While most of these parameters have limited modifiability due to inherent design constraints, the roll stiffness distribution γ_{k_ϕ} (-), which defines the fraction of total roll stiffness k_ϕ (Nm/rad) at the rear axle, offers some tunability.

$$\Gamma_{y,ij} = \frac{C_{\Gamma y,i}}{C_{\Gamma y,f} + C_{\Gamma y,r}} \Gamma_y \quad (7.22)$$

$$C_{\Gamma y,f} = \frac{h_{u,f} m_{u,f}}{t_w} + \frac{l_r h_f m_s}{l t_w} + \frac{(1 - \gamma_{k_\phi}) k_\phi m_s (\frac{d}{k_\phi - m_s g d})}{t_w} \quad (7.23)$$

$$C_{\Gamma y,r} = \frac{h_{u,r} m_{u,r}}{t_w} + \frac{l_f h_r m_s}{l t_w} + \frac{\gamma_{k_\phi} k_\phi m_s (\frac{d}{k_\phi - m_s g d})}{t_w} \quad (7.24)$$

7.2.3.3 Aerodynamics

The aerodynamic performance of the vehicle equipped with active aerodynamic systems is obtained from CFD simulations, as described in Section 7.2.2, and incorporated into the optimisation framework using polynomial functions. These functions describe the influence of each aerodynamic control input on both the magnitude and distribution of aerodynamic loads acting on the vehicle. Specifically, the effects on key aerodynamic force components, including lift at the front $C_{L,\text{front}}$ (-) and rear wheels $C_{L,\text{rear}}$ (-), lift at the left $C_{L,\text{left}}$ (-) and right wheels $C_{L,\text{right}}$ (-), side force at the front $C_{S,\text{front}}$ (-) and rear axles $C_{S,\text{rear}}$ (-), and total drag C_D (-), are expressed using polynomial formulations.

Eq. (7.25) provides a general formulation for the effect of an aerodynamic element on a given force component coefficient $\Delta C_{u,v}^w$ (-), as a function of its operating angle α_w (deg). The aerodynamic elements are indexed by $w \in \{\text{FW-L, FW-R, RW, TW}\}$, corresponding to the left front wing angle, right front wing angle, rear wing angle, and rear wing tilt. The aerodynamic lift, drag, and side forces are represented by $u \in \{L, D, S\}$, while $v \in \{\text{front, rear, left, right}\}$ specifies the force distribution location. The polynomial coefficients $P_{u,v}^w(\ell)$ (-) define the aerodynamic response, ℓ (-) indexes the polynomial terms, and $N_{u,v}^w$ (-) denotes the polynomial order. The polynomial coefficients for each aerodynamic element's influence on the respective force components are detailed in Table 7.4.

$$\Delta C_{u,v}^w = \sum_{\ell=1}^{N_{u,v}^w} P_{u,v}^w(\ell) \alpha_w^\ell \quad (7.25)$$

The combined aerodynamic force component coefficients $C_{u,v}$ (-) are successively obtained by summing the individual effects from all active aerodynamic elements with the vehicle's nominal coefficients $C_{u,v}^0$ (-), which represent the aerodynamic characteristics without any system actuation, as described in Eq. (7.26).

$$C_{u,v} = C_{u,v}^0 + \Delta C_{u,v}^{\text{FW-L}} + \Delta C_{u,v}^{\text{FW-R}} + \Delta C_{u,v}^{\text{RW}} + \Delta C_{u,v}^{\text{TW}} \quad (7.26)$$

Table 7.4: Overview of the aerodynamic relationship coefficients.

Coefficient	$C_{L,front}$	$C_{L,rear}$	$C_{L,left}$	$C_{L,right}$	C_D	$C_{S,front}$	$C_{S,rear}$
$C_{u,v}^0$	-0.188	-0.103	-0.146	-0.146	0.346	0	0
$P_{u,v}^{FW-L}(1)$	-9.75e-3	1.58e-3	-8.17e-3	0	0	0	0
$P_{u,v}^{FW-R}(1)$	-9.75e-3	1.58e-3	0	-8.17e-3	0	0	0
$P_{u,v}^{RW}(1)$	3.79e-3	-2.18e-2	-9.01e-3	-9.01e-3	2.90e-3	0	0
$P_{u,v}^{RW}(2)$	-1.94e-4	1.11e-3	4.32e-4	4.32e-4	7.87e-5	0	0
$P_{u,v}^{RW}(3)$	4.31e-6	1.56e-5	9.95e-6	9.95e-6	0	0	0
$P_{u,v}^{RW}(4)$	0	-2.75e-6	-1.37e-6	-1.37e-6	0	0	0
$P_{u,v}^{RW}(5)$	0	4.92e-8	2.46e-8	2.46e-8	0	0	0
$P_{u,v}^{TW}(1)$	0	-7.10e-5	4.30e-3	-4.30e-3	0	0	7.19e-3
$P_{u,v}^{TW}(2)$	0	0	3.55e-5	3.55e-5	0	0	0

The total aerodynamic force coefficients are subsequently determined by summing the relevant component contributions, as exemplified for lift in Eq. (7.27).

$$C_L = C_{L,front} + C_{L,rear} \quad (7.27)$$

Given the critical role of aerodynamic loads in tyre force generation, lift coefficients are formulated for each wheel to enable direct integration with the vertical tyre load formulation in the vehicle dynamics model. The distribution is determined based on the side-specific lift contributions previously established. Eq. (7.28) describes this process for the front left wheel, with analogous expressions for the front right, rear left, and rear right wheels.

$$C_{L,FL} = C_{L,front} \frac{C_{L,left}}{C_L} \quad (7.28)$$

Finally, the aerodynamic forces $F_{u,v}$ (N) acting on the vehicle are computed using Eq. (7.29), where ρ (kg/m³) is the air density, A_{ref} (m²) the vehicle's reference frontal area, and v_x (m/s) the longitudinal velocity.

$$F_{u,v} = \frac{1}{2} \rho C_{u,v} A_{ref} v_x^2 \quad (7.29)$$

7.2.3.4 Powertrain

The powertrain is modelled as an electric motor unit coupled with a single-speed transmission and a central differential, enabling torque distribution between the front and rear axles for four-wheel drive capability. Performance metrics are derived from an existing motorsport electric propulsion system (46), with a maximum continuous power output of $P_{e,\max} = 456$ kW, a maximum continuous torque output of $T_{e,\max} = 602$ Nm, and a maximum motor speed of $\Omega_{e,\max} = 17750$ RPM.

The electric motor torque T_e (Nm) is used to determine the individual wheel torques, as expressed by Eqs. (7.30) and (7.31). Here, γ_T (-) denotes the fixed torque distribution, defining the fraction of total torque directed to the rear axle, and γ_g (-) represents the final gear ratio. Additionally, T_b (Nm) represents the total brake torque applied across all four wheels and γ_b (-) the fixed brake balance, defining the fraction of total braking torque directed to the front wheels.

$$T_{fj} = \frac{1}{2} \left((1 - \gamma_T) T_e \gamma_g + T_b \gamma_b \right) \quad (7.30)$$

$$T_{rj} = \frac{1}{2} \left(\gamma_T T_e \gamma_g + T_b (1 - \gamma_b) \right) \quad (7.31)$$

7.2.4 Optimisation Framework

This study utilises an optimisation framework developed by Jiménez et al. (30), which aims to minimise lap time by optimising the vehicle's trajectory, control inputs, and selected design parameters while adhering to track constraints and vehicle performance limitations. This Minimum Lap Time Problem (MLTP) is formulated as an Optimal Control Problem (OCP) and transformed into a Non-Linear Program (NLP) using direct orthogonal collocation. A brief overview is provided here, with a more comprehensive explanation available in the work by Jiménez et al (30).

7.2.4.1 Optimal Control Problem

The OCP is formulated to optimise the vehicle states $\mathbf{x}$, control inputs $\mathbf{u}$, auxiliary variables $\mathbf{y}$ and selected vehicle design parameters $\mathbf{p}$ to minimise total lap time. The distance travelled along the track is used as the independent variable, a method successfully applied in prior studies (21, 28, 42), which maintains an explicit connection to the track position and reduces computational complexity by lowering the problem's dimensionality.

The objective function, expressed by Eq. (7.32), is constrained by Eqs. (7.33)-(7.35), with the optimisation variables specified in Eq. (7.36). The start and finish positions are denoted by s_0 and s_f . The vector function $\mathbf{f}$ describes the vehicle's dynamics through Eqs. (7.4)-(7.9), which are enforced as constraints to ensure accurate simulation of the vehicle's performance. The vector function $\mathbf{h}$ defines constraints on the optimisation variables through bounds and additional constraints defined by Eqs. (7.37)-(7.42). Initial and final states are described by $\mathbf{b}$, used to define the vehicle's starting and finishing conditions.

Minimise

$$J = \int_{s_0}^{s_f} S_f(\mathbf{x}(s), \mathbf{u}(s), \mathbf{y}(s), \mathbf{p}) ds \quad (7.32)$$

Subject to

$$\frac{d}{ds} \mathbf{x} = \mathbf{f}(\mathbf{x}(s), \mathbf{u}(s), \mathbf{y}(s), \mathbf{p}) \quad (7.33)$$

$$\mathbf{h}(\mathbf{x}(s), \mathbf{u}(s), \mathbf{y}(s), \mathbf{p}) \leq 0 \quad (7.34)$$

$$\mathbf{b}(\mathbf{x}(s_0), \mathbf{x}(s_f)) = 0 \quad (7.35)$$

With

$$\mathbf{x} = \begin{bmatrix} v_x \\ v_y \\ r \\ n \\ \varepsilon \\ \Omega_{ij} \end{bmatrix}, \mathbf{u} = \begin{bmatrix} T_e \\ T_b \\ \delta \\ \alpha_{FW(-j)} \\ \alpha_{RW} \\ \alpha_{TW} \end{bmatrix}, \mathbf{y} = \begin{bmatrix} \Gamma_x \\ \Gamma_y \end{bmatrix}, \text{ and } \mathbf{p} = \begin{bmatrix} \gamma_b \\ \gamma_T \\ \gamma_{\kappa\phi} \end{bmatrix} \quad (7.36)$$

Limitations on the maximum electric motor power and angular velocity are incorporated through Eqs. (7.37) and (7.38), while Eq. (7.39) prevents simultaneous application of drive and brake torque control inputs, avoiding unrealistic driving scenarios. Tyre grip is constrained by a friction circle model with a uniform friction coefficient in both longitudinal and lateral directions, as expressed by Eq. (7.40), ensuring tyre forces remain within the available grip. Finally, longitudinal and lateral load transfer are incorporated as auxiliary variables through equality constraints given by Eqs. (7.41) and (7.42).

$$\Omega_e T_e - P_{e,max} \leq 0 \quad (7.37)$$

$$\Omega_e - \Omega_{e,max} \leq 0 \quad (7.38)$$

$$T_e T_b = 0 \quad (7.39)$$

$$\frac{F_{x,ij}^2 + F_{y,ij}^2}{F_{z,ij}^2} - \mu_{ij}^2 \leq 0 \quad (7.40)$$

$$\Gamma_x - \left[\left((F_{x,fl} + F_{x,fr}) \cos(\delta) - (F_{y,fl} + F_{y,fr}) \sin(\delta) + F_{x,rl} + F_{x,rr} + F_D^0 \right) \frac{h_{COG}}{l} + F_D^{aero} \frac{h_{RW}}{l} \right] = 0 \quad (7.41)$$

$$\Gamma_y - \left[\left((F_{x,fl} + F_{x,fr}) \sin(\delta) + (F_{y,fl} + F_{y,fr}) \cos(\delta) + F_{y,rl} + F_{y,rr} \right) \frac{h_{COG}}{t_w} + F_S \frac{h_{RW}}{t_w} \right] = 0 \quad (7.42)$$

7.2.4.2 Non-Linear Programming

The continuous-time OCP is discretised and transcribed into a NLP problem using direct orthogonal collocation to enable numerical optimisation. This involves dividing the track into N segments, each of length ds_k (m). Within each segment s_k , the states are approximated using Legendre polynomials of degree d , where collocation points $s_{k,j}$, with $j = \{1, \dots, d\}$, are selected at the roots of these polynomials. Collocation constraints are enforced at each collocation point, matching the polynomial derivatives of the approximated states to equations governing the vehicle's dynamics. This ensures that the approximated vehicle trajectory evolves according to its modelled dynamic behaviour.

The objective function is evaluated at the collocation points within each track segment, defined by the vector function $\mathbf{S}_{f,k} = [S_f(s_{k,1}), \dots, S_f(s_{k,d})]$. The contribution of each track segment is weighted by $\mathbf{B}$, ensuring that the integral of the objective function over the entire track is accurately approximated. These weighted contributions are summed across all track segments to provide the NLP formulation of the objective function, as expressed by the first term in Eq. (7.43).

$$J = \sum_{k=1}^N \mathbf{B} \mathbf{S}_{f,k} ds_k + \underbrace{\sum (r_m \Delta_s^m U_k)^2 + \sum (r_m \Delta_s^m Y_k)^2}_{\text{regularisation}} \quad (7.43)$$

To ensure realistic and feasible solutions, additional constraints and regularisation terms are introduced. Physical limitations of control inputs are enforced through rate limits, preventing unrealistic steering inputs and reflecting system response times. Regularisation terms are incorporated in the objective function to promote more practical solutions by penalising the sum of squares of specific discrete control inputs U_k , auxiliary variables Y_k , and their m -th derivatives at each track segment Δ_s^m . The regularisation weight factors r_m allow adjustment of each term's influence in the optimisation process.

7.2.4.3 Setup

The minimum lap time simulations are performed on the Circuit de Barcelona-Catalunya, a track known for its combination of high-speed straights and technical corners, making it ideal for comprehensive vehicle performance evaluations. As the track serves solely as a representative case encompassing diverse driving scenarios for the comparative assessment of different aerodynamic configurations, track-specific features such as elevation changes are not included.

The NLP problem is formulated using the automatic differentiation framework CasADi and solved in MATLAB using the IPOPT algorithm with an optimality tolerance of 1×10^{-6} and constraint violation tolerances of 1×10^{-4} . Based on grid convergence data presented in Figure 7.6, a discretisation step size of 10 m is selected, achieving a relative lap time error of approximately 0.01% compared to the smallest step size tested, while reducing computational time by over 23%. Following the work of Jiménez et al. (30), 3rd order Legendre polynomials are used to approximate the states within each track segment.

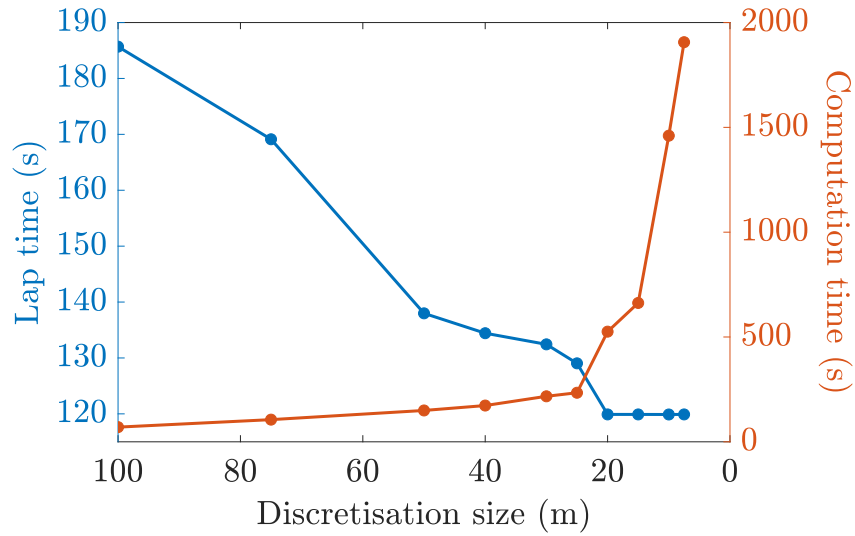


Figure 7.6: Lap time and computation time as a function of the discretisation step size.

Control input rate limits and regularisation weight factors used to ensure practical solutions are detailed in Table 7.5. The rate limits reflect system and driver response times, with aerodynamic systems specifically constrained based on reported response times of approximately 0.5 s in existing high-performance active aerodynamic systems. Furthermore, the regularisation factors are carefully tuned to suppress high-frequency oscillations in the control inputs, ensuring smoother actuation. This prevents excessive corrections and allows clearer trends in the control strategies, supporting the consistency and interpretability of the findings in this study.

Table 7.5: Overview of the rate limits and regularisation weight factors for control inputs and auxiliary variables used for the performance optimisation simulations.

Parameter	Symbol	Rate limits		Regularisation		
		Lower	Upper	r_0	r_1	r_2
Drive torque	T_e	-2e4 Nm/s	2e4 Nm/s	0	0	0
Brake torque	T_b	-2e4 Nm/s	2e4 Nm/s	0	0	0
Steering angle	δ	-0.1 rad/s	0.1 rad/s	0	10	15
Front wing angle	α_{FW}	-20 deg/s	20 deg/s	0	0	0.3
Rear wing angle	α_{RW}	-60 deg/s	60 deg/s	0	0	0.9
Rear wing tilt	α_{TW}	-24 deg/s	24 deg/s	0.005	0	0.4
Longitudinal load transfer	Γ_x	-	-	0	0	1

7.3 Results and Discussion

The analyses begin with an assessment of the influence of each aerodynamic element on the vehicle's aerodynamic performance. Following this, the general performance of each configuration is analysed using lap time results, and areas of performance gains are identified using more detailed performance data. The subsequent section provides an in-depth analysis of the optimal control strategies for each aerodynamic system, offering insights into how these elements are controlled to realise the observed performance improvements. Finally, changes in optimised key vehicle design parameters are assessed to explore how these parameters shift in response to different aerodynamic configurations.

7.3.1 Aerodynamic Performance

The simulated aerodynamic data, which represent the effects of each aerodynamic element, along with the associated fitted functions used for implementation into the optimisation framework, are presented in Figure 7.7. The left and right front wing elements individually enhance front axle downforce by a maximum of approximately $\Delta C_L \approx -0.10$ across their angle of attack range, and affect the downforce on their respective sides of the vehicle by over $\Delta C_L \approx -0.08$. Each element also creates a minor reduction in rear axle downforce, up to approximately $\Delta C_L \approx 0.02$. Furthermore, the front wing elements have negligible effects on drag and downforce on their respective opposite side of the vehicle.

The rear wing exerts the most significant influence on aerodynamic performance. Across its angle of attack range, it increases rear axle downforce by an approximate maximum of $\Delta C_L \approx -0.31$ but also reduces front axle downforce by over $\Delta C_L \approx 0.05$. The rear wing's symmetric configuration results in equal downforce distribution across the left and right sides of the vehicle. Notably, the rear wing exhibits two downforce gain regions, where initial enhancements plateau at the stall angle between 10° - 15° , then rise further due to upstream pressurisation at higher angles, aligning with prior experimental findings (34). Additionally, the rear wing increases drag by nearly $\Delta C_D \approx 0.16$ across its angle of attack range, with drag exhibiting a slight quadratic increase.

The rear wing tilt angle provides an additional layer of control by redistributing downforce laterally. Elevating one side of the wing increases downforce on that side by nearly $\Delta C_L \approx -0.05$, while decreasing it by almost $\Delta C_L \approx 0.06$ on the opposite side, resulting in a slight overall downforce reduction. The tilted rear wing also generates a side force primarily acting at the rear of the vehicle, directed toward the elevated side, and peaking at nearly $\Delta C_S \approx \pm 0.09$. Additionally, the tilt angle has minimal impact on front axle downforce and overall drag.

Building on the analysis of the individual effects of each aerodynamic element, the focus now shifts to assessing the vehicle's overall aerodynamic performance. The combined aerodynamic impact of symmetric front wing and rear wing angle of attack actuation, as presented in Figure 7.8, provides insight into the baseline aerodynamic characteristics of the vehicle. For improved visibility, the downforce coefficient is used in this figure instead of the lift coefficient.

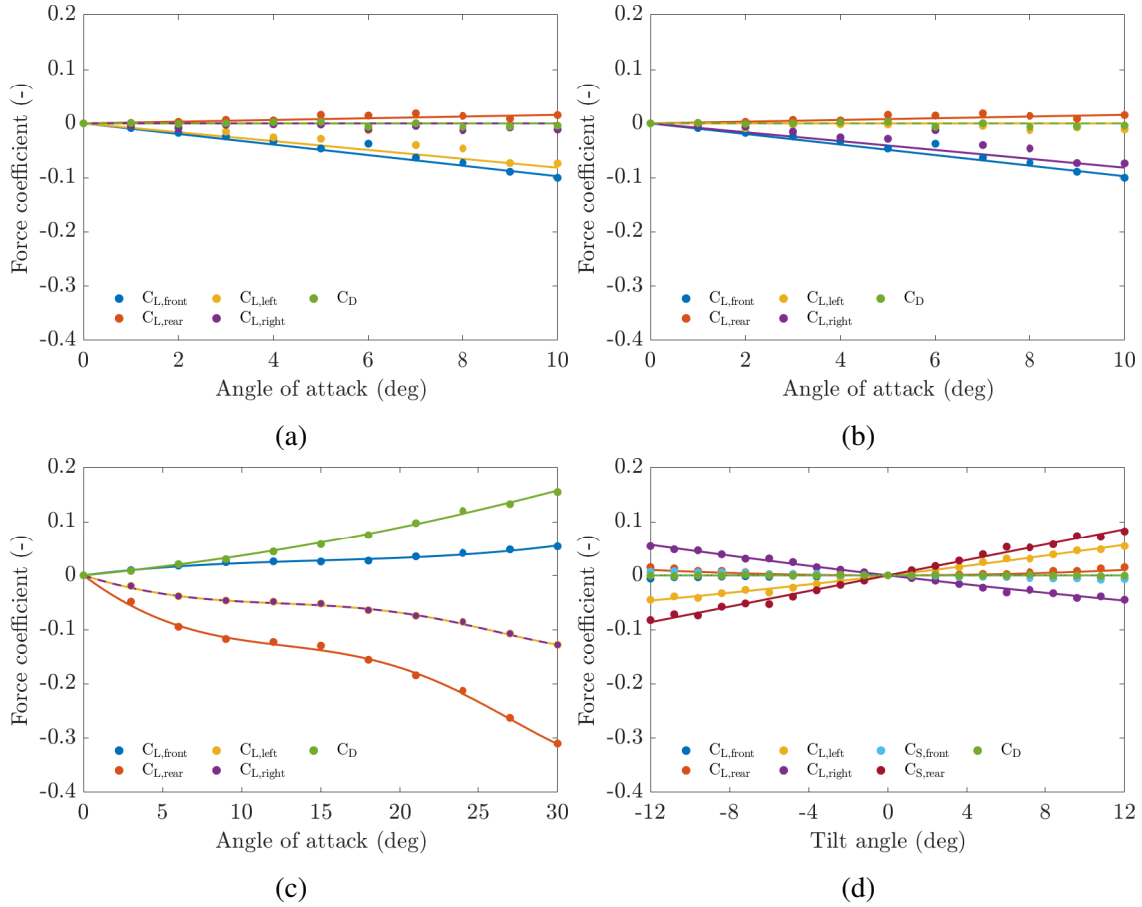


Figure 7.7: Simulated aerodynamic data and corresponding fitted functions illustrating the influence of (a) left front wing element angle of attack, (b) right front wing element angle of attack, (c) rear wing angle of attack, and (d) rear wing tilt angle on the vehicle's aerodynamic performance.

At minimum front and rear wing angles, the vehicle produces a baseline lift coefficient of $C_L \approx -0.29$ and a drag coefficient of $C_D \approx 0.35$. As the angles increase, the rear wing's previously observed two-stage downforce enhancement regions become apparent, while the front wing contributes more linearly to the total downforce. Across their operating ranges, the rear wing increases the base downforce level by $\Delta C_L \approx -0.26$ (+89%) and the front wing by $\Delta C_L \approx -0.16$ (+56%). Together, the wings enhance downforce by a total of $\Delta C_L \approx -0.42$ (+145%) to reach the maximum lift coefficient of $C_L \approx -0.71$.

The downforce distribution, representing the fraction of downforce on the rear axle, starts at approximately 0.35 at the minimum wing angles, indicating a forward bias. Maximising the rear wing angle while minimising the front wing angle shifts the distribution rearward to about 0.76. Conversely, minimising the rear wing angle and maximising the front wing angle shifts the distribution forward to approximately 0.16. With both wing angles maximised, the distribution balances around 0.54, showing a moderate rearward bias as the rear wing generates more downforce at high angles of attack. Additionally, drag is primarily influenced by the rear wing, which increases the baseline drag level by $\Delta C_D \approx 0.16$ (+46%) to a maximum of $C_D \approx 0.50$ across its angle of attack range.

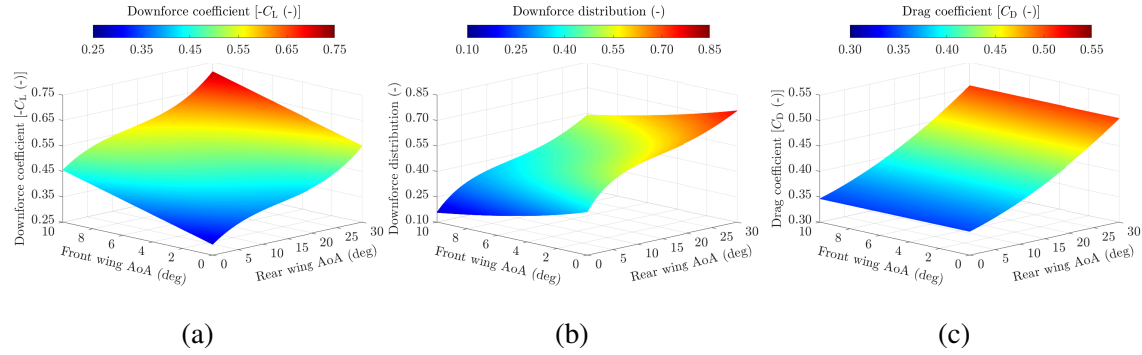


Figure 7.8: Influence of symmetric front and rear wing actuation on the vehicle's (a) downforce coefficient, (b) downforce distribution expressed as the rear axle load fraction, and (c) drag coefficient.

7.3.2 Vehicle Performance

The analysis of lap times reveals clear performance distinctions between the four vehicle configurations, as detailed in Table 7.6. Configuration 1, with an optimised static setup using a maximum front wing angle of 10° and a moderately low rear wing angle of around 8.12° , records the longest lap time of 120.196 s, which serves as the baseline for comparison. Configuration 2, which operates with an optimised fixed front wing angle of 10° and active rear wing control, achieves a lap time of 119.502 s, resulting in an improvement of 0.694 s (0.58% faster). Configuration 3, with active control over both the front and rear wings, records a lap time of 119.452 s, showing an improvement of 0.744 s (0.62% faster) over configuration 1. Finally, configuration 4, which has active asymmetric control of both front and rear wings, achieves the fastest lap time of 119.281 s, reducing lap time by 0.915 s (0.76% faster) compared to configuration 1.

The optimal racing line for configuration 1, shown in Figure 7.9, illustrates the range of driving scenarios encountered on a lap around the Circuit de Barcelona-Catalunya. The adaptability of active aerodynamic configurations to these scenarios can offer notable performance gains. To facilitate subsequent analyses, the circuit's corners are classified as low-, medium-, or high-speed based on average velocity, using the 33rd and 66th percentiles as thresholds. Corners 5, 10, 13, 14, and 15 are classified as low-speed, with an average cornering velocity of 20.62 m/s. Corners 1, 2, 4, 7, 11, and 12 are categorised as medium-speed, averaging 28.39 m/s, while corners 3, 6, 8, 9, and 16 are identified as high-speed, with an average velocity of 40.50 m/s.

Table 7.6: Lap time results and aerodynamic settings for all vehicle configurations.

Configuration	Front wing (deg)	Rear wing (deg)	Sector 1 time (s)	Sector 2 time (s)	Sector 3 time (s)	Lap time (s)	Δ Lap time (s)	Δ Lap time (%)
C1	10.000	8.121	32.953	45.085	42.158	120.196	-	-
C2	10.000	Active	32.721	44.795	41.986	119.502	0.694	0.58
C3	Active	Active	32.701	44.772	41.979	119.452	0.744	0.62
C4	Active asymmetric	Active asymmetric	32.649	44.703	41.928	119.281	0.915	0.76

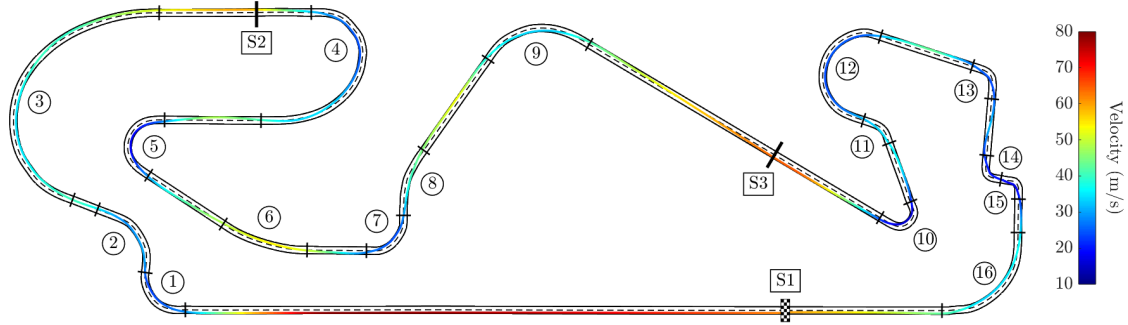


Figure 7.9: Optimal racing line with velocity contours for configuration 1 on the Circuit de Barcelona-Catalunya, featuring labelled sectors and corners with corner start and end point markings.

Insights into the overall performance differences between the vehicle configurations are first drawn from the analysis of velocity, steering angle, and torque data. Generally, the active aerodynamic configurations offer velocity improvements on both the straights and corners, as shown in Figures 7.10a and 7.10b. Increased corner exit velocities in longer medium-to-high-speed bends such as corners 3, 4, 9, and 16 are particularly noticeable from the high delta velocity in the latter half of these corners. Additionally, later braking in heavy-braking zones is observable, particularly evident by delta velocity peaks just before corners 1, 4, 9, and 10.

The steering angle inputs, presented in Figure 7.10c, show only minor deviations between the configurations, suggesting small variations in their optimal racing lines. However, noticeably less steering input is observed in corners such as 3, 4, 5, 9, 12, and 16. This suggests that the active aerodynamic configurations offer improved grip, leading to higher steering responsiveness.

The drive and brake torque inputs, shown in Figure 7.10d, further reinforce the findings above. In long bends such as corners 3, 4, 9, and 16, the active aerodynamic configurations enable earlier application of drive torque, resulting in faster corner exits. In addition, these configurations allow more average brake torque to be applied, enabling later braking in heavy-braking zones before corners 1, 4, 9, and 10, as previously identified.

To further illustrate the performance enhancements provided by the active aerodynamic configurations, Figure 7.11 presents GG diagrams that map the relationship between longitudinal and lateral acceleration. In these diagrams, positive longitudinal acceleration represents forward acceleration, while negative values correspond to braking. Similarly, positive and negative lateral accelerations denote left- and right-hand cornering, respectively. The borders of each GG diagram are highlighted to indicate the performance limits of each configuration, emphasising the operating boundaries within which the vehicle performs. Across all configurations, the diagrams show a concentration of operating points along these boundaries, demonstrating that each configuration consistently operates near its maximum potential.

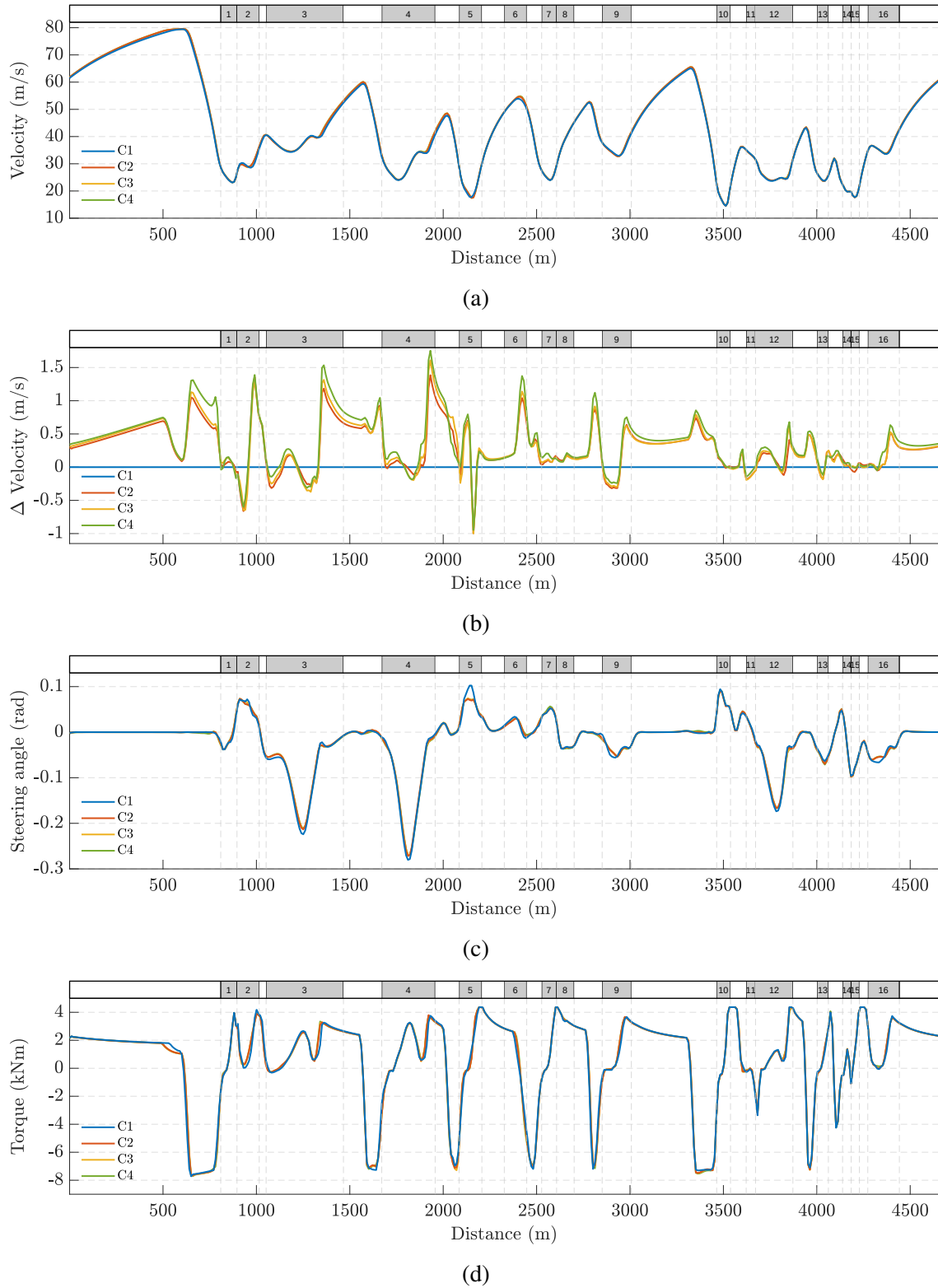


Figure 7.10: (a) Velocity, (b) velocity variations, (c) steering angle input, and (d) total combined wheel torque as a function of distance for all vehicle configurations.

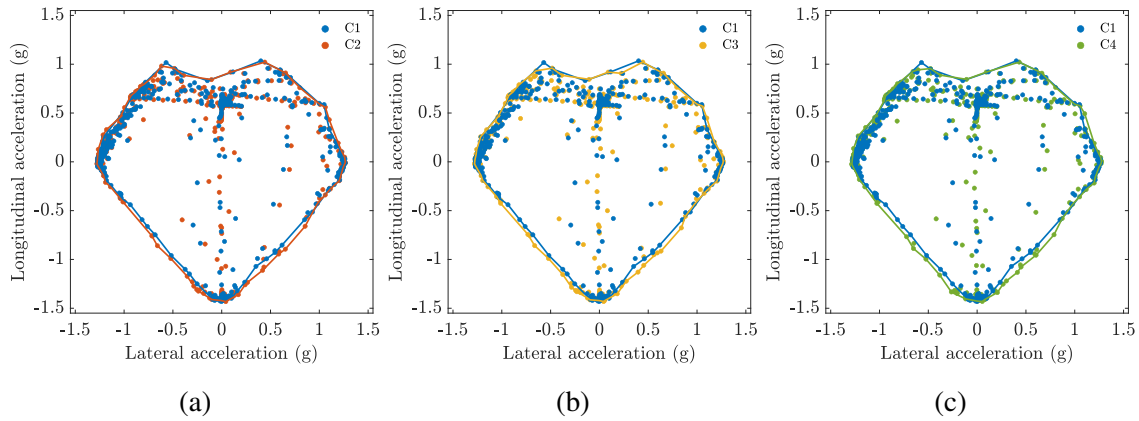


Figure 7.11: GG-diagrams for (a) configuration 2, (b) configuration 3, and (c) configuration 4 in comparison to configuration 1, with highlighted borders indicating the performance limits of each configuration.

The GG diagrams further confirm configuration 4 as the most capable setup, extending the performance envelope with the highest limits in both longitudinal and lateral acceleration. Relative to configuration 1, configurations 2 and 3 exhibit approximately a 2.5% increase in GG area, while configuration 4 achieves an increase of over 3.5%. In terms of pure lateral acceleration, configurations 2 and 3 show a modest improvement of less than 0.5%, whereas configuration 4 reaches nearly 1% higher lateral acceleration.

Configuration 4 also demonstrates the greatest improvements in combined deceleration and lateral acceleration in the lower quadrants of the GG diagrams, representing trail braking scenarios. Here, configurations 2 and 3 achieve an average 4.2% increase in combined acceleration, while configuration 4 sees a further improvement to 5.2% relative to configuration 1. In contrast, combined lateral and forward acceleration in the upper quadrants of the GG diagrams, which corresponds to acceleration out of corners, shows a more modest increase of less than 0.5% for configurations 2 and 3, and nearly 3% for configuration 4. These results highlight the greater overall performance gains of configuration 4, particularly in scenarios involving combined acceleration.

7.3.3 Optimal Control of Aerodynamic Systems

With the key performance gains established, the analysis proceeds to examine the optimal control inputs for the active aerodynamic systems driving these improvements, as illustrated in Figure 7.12. In addition, longitudinal and lateral load transfer data are presented to highlight their influence on the control strategies.

For configuration 2, the rear wing angle is typically maximised in low-speed corners to enhance downforce and improve cornering performance, as depicted in Figure 7.12a. In medium-speed corners, the angle varies between medium and high, while in high-speed corners, it ranges from low to medium. This strategy is attributed to the quadratic relationship between aerodynamic forces and velocity, allowing lower angles of attack to generate sufficient downforce at high speeds while also mitigating excessive drag generation.

Generally, the rear wing control shows notable similarities to an inverse relationship with longitudinal load transfer, increasing the angle under forward load transfer during braking and decreasing the angle under rearward load transfer during acceleration. High angles of attack during braking not only aid deceleration by increasing aerodynamic drag, but also by enhancing rear axle load, allowing the rear tyres to contribute more effectively to braking. Conversely, lowering the angle during sustained acceleration reduces aerodynamic drag, and avoids excessive rear axle loading and front axle unloading. Additionally, on straights, the rear wing is set to its minimum angle to reduce aerodynamic drag and rolling resistance, maximising straight-line speed.

The rear wing control for configuration 3, demonstrated in Figure 7.12a, largely mirrors that of configuration 2, with minor adjustments to account for the active front wing's influence on the aerodynamic balance. The front wing angle is typically maximised under high lateral load transfer during cornering to increase downforce on the front axle, thereby enhancing tyre grip and cornering performance. In heavy-braking zones, when forward load transfer peaks, the front wing angle is reduced to improve load balance, ensuring both the front and rear tyres contribute effectively to deceleration.

Adjustments to the front wing are less frequent compared to the rear wing, due to the rear wing's greater trade-off between downforce and drag. Front wing angle adjustments generally occur only when the rear wing is at its maximum angle, and vice versa, reflecting a coordinated approach in which the wings manage axle-specific downforce requirements in response to load shifts during acceleration, braking, and cornering. On straights, both front and rear wing angles are reduced to minimise drag and rolling resistance, maximising straight-line speed.

Configuration 4's control strategy, shown in Figure 7.12b, optimises the independent left and right front wing elements, along with the adjustable rear wing angle of attack and tilt angle, providing the highest adaptability to diverse driving scenarios. During trail braking, as the vehicle initiates cornering, peak forward load transfer starts to diminish while lateral load transfer increases with steering angle application. In this phase, the inside front wing element angle is maximised to enhance grip on the less loaded front inside tyre, while the outside front wing angle is reduced to prevent further unloading of the rear axle. The rear wing angle of attack remains at its maximum to enhance rear axle load and increase aerodynamic drag, aiding deceleration. Concurrently, the rear wing maximally tilts away from the corner centre, providing a lateral force at the rear axle and a favourable downforce distribution for the inside tyres. This strategy consistently emerges at the end of braking zones, when approaching corner entry.

As the vehicle transitions from trail braking to pure cornering, where lateral load transfer predominates, both front wing elements are set to their maximum angle to enhance front axle grip. The rear wing angle of attack typically varies between low and medium levels depending on cornering speed to maintain aerodynamic balance, while the rear wing tilt remains maximised away from the corner. The tilt angle control closely mirrors the lateral load transfer patterns, generating a side force that supports lateral forces at the rear axle and enhances downforce on the inside tyres, which become underloaded with increasing lateral load transfer. This coordinated control strategy is particularly evident in longer medium-to-high-speed corners, such as corners 3, 4, 9, 11, 12, and 16.

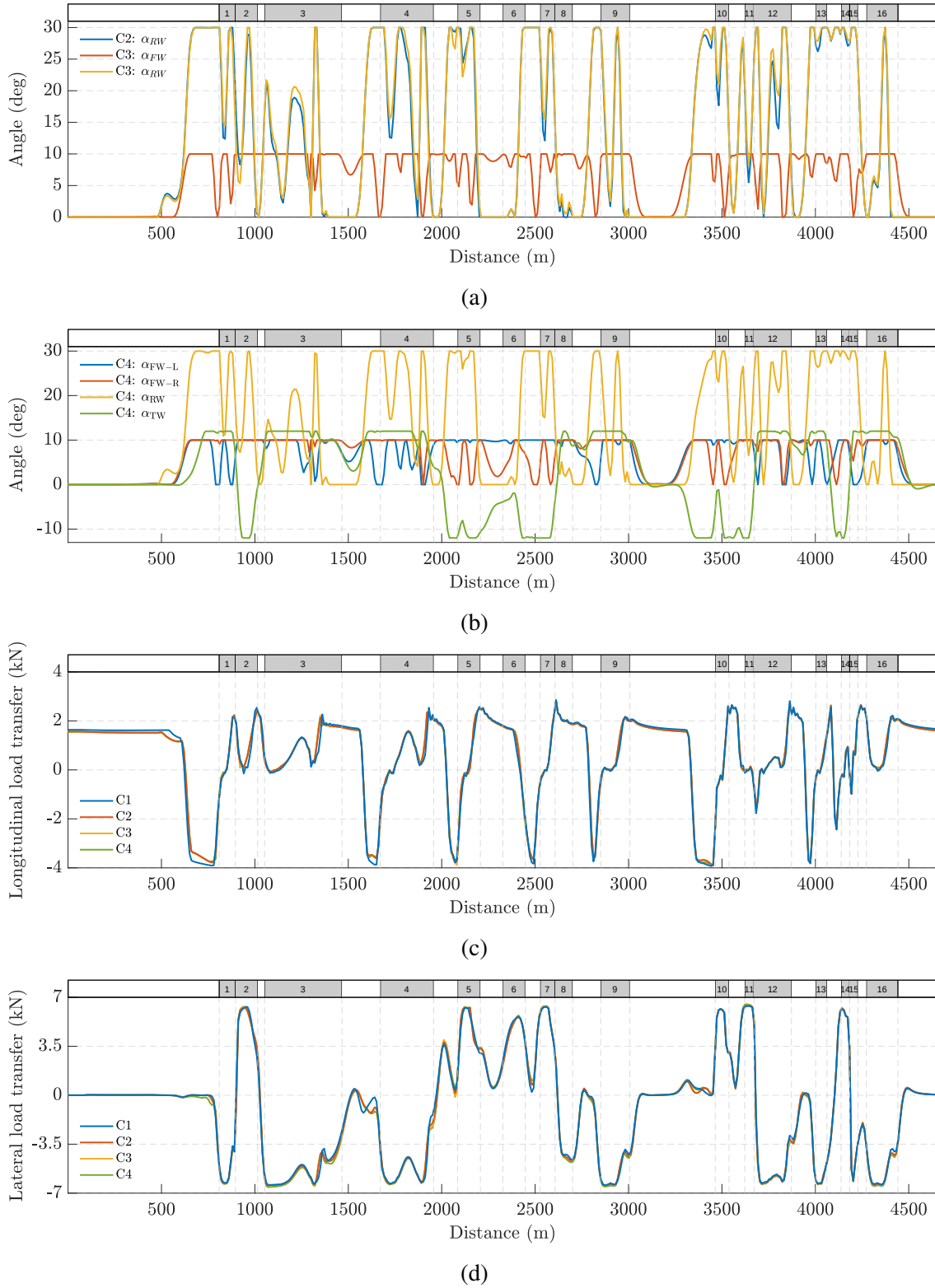


Figure 7.12: Optimal control inputs for the aerodynamic systems of (a) configurations 2 and 3, (b) configuration 4, along with (c) longitudinal load transfer and (d) lateral load transfer as functions of distance for all vehicle configurations.

During the initial acceleration phase when exiting a corner, lateral load transfer starts to reduce as steering input lessens, with a minor rearward load transfer occurring as initial drive torque is applied. In this phase, the inside front wing element is maximised to enhance downforce on the underloaded inside front tyre, while the outside front wing element is minimised to prevent rear axle unloading. Simultaneously, the rear wing angle of attack and tilt angle are maximised to increase downforce on the rear axle and underloaded inside tyres. This configuration effectively distributes downforce across the axles to maximise the four-wheel drive system, with the rear axle bearing most load to accommodate the majority of initial drive torque, while sufficient load remains on the front wheels for effective utilisation of the torque directed to the front wheels.

This configuration remains only briefly, as rearward load transfer intensifies and lateral load transfer decreases further during corner exit. In this phase, both front wing elements are set to their maximum angle to increase downforce on the underloaded front axle. Simultaneously, the rear wing angle of attack is minimised to prevent rear axle overloading and mitigate front axle unloading, thereby balancing the rearward load transfer. The rear wing tilt remains maximised while lateral load transfer persists, until the vehicle exits the corner fully and enters a straight. At this point, all angles are set to zero, minimising drag, rolling resistance, and side forces, to maximise straight-line speed.

7.3.4 Vehicle Design Parameters

The optimal vehicle design parameters for brake, torque, and roll stiffness distributions, listed in Table 7.7, vary noticeably with different aerodynamic configurations. The optimal brake balance shifts progressively rearward with increasing aerodynamic control, as the active aerodynamic configurations offer improved grip at the rear tyres during braking.

Configuration 1, with a high front wing angle and moderately low rear wing angle, generates limited rear axle downforce, resulting in the most front-biased brake balance. Configuration 2's active rear wing control enhances rear axle downforce during braking, resulting in a nearly 1.4% rearward shift of the optimal brake balance. In addition to the rear wing's high downforce, configuration 3 reduces its front wing angle near peak deceleration to mitigate rear axle unloading, contributing to a rearward brake balance shift of over 1.6% compared to configuration 1. Configuration 4 exhibits the highest rearward brake balance shift of over 1.7%, due to effective balancing of downforce during trail braking, further improving the utilisation of the rear tyres during braking.

Table 7.7: Optimal vehicle design parameters of brake balance, torque distribution, and roll stiffness distribution for all vehicle configurations.

Configuration	Brake Balance	Torque distribution	Roll stiffness distribution
	γ_b (-)	γ_T (-)	γ_{k_ϕ} (-)
C1	0.677	0.727	0.462
C2	0.667	0.739	0.470
C3	0.666	0.741	0.472
C4	0.665	0.740	0.475

The optimal torque distribution also shifts rearward for the active aerodynamic configurations, varying between approximately 1.6% to 1.9%. Maximising the rear wing and reducing the front wing angle during initial torque application increases downforce and thereby traction at the rear tyres compared to configuration 1. This enhanced grip allows more torque to be applied to the rear wheels, shifting the optimal torque distribution rearward.

The optimal roll stiffness distribution shifts rearward for active aerodynamic configurations as well. Configurations 2 and 3 enhance rear axle downforce during cornering compared to configuration 1, allowing it to accommodate greater lateral load transfer, resulting in a 1.7% to 2.1% rearward shift in the optimal roll stiffness distribution. Configuration 4 shows the most pronounced shift of approximately 2.7% relative to configuration 1, due to its asymmetric aerodynamic control. Its aerodynamic elements can increase downforce on the underloaded inside tyres and generate an aerodynamic side force near the rear axle during cornering, allowing configuration 4 to accommodate the highest lateral load transfer at the rear axle.

7.4 Conclusion

This study examined the influence of active aerodynamic systems on high-performance vehicles by analysing the effects of different configurations on lap time, control inputs, and optimal design parameters. It compared static and active setups, including a configuration with 2D active aerodynamic systems capable of influencing aerodynamic balance longitudinally and laterally. The analysis addressed both the magnitude of performance gains and the underlying mechanisms through which these gains were achieved.

Vehicle performance was optimised and evaluated for a lap at the Circuit de Barcelona-Catalunya using a minimum lap time simulation framework. The simulations employed a transient vehicle dynamics model that incorporated longitudinal and lateral load transfer, non-linear tyre characteristics, and aerodynamic forces obtained through CFD simulations.

The results demonstrated that active aerodynamic configurations significantly reduced lap times compared to a static setup, with key performance improvements in braking capability, cornering speed, acceleration out of corners, and straight-line velocity. The fully active asymmetric configuration provided the highest lap time reduction of approximately 0.915 s (0.76%), as its ability to modulate downforce laterally improved tyre grip utilisation, enabling the vehicle to sustain higher longitudinal and lateral accelerations.

Optimal control strategies for each configuration revealed that performance gains were achieved by actuating aerodynamic elements to balance shifts in vertical tyre loads induced by load transfer. Configurations with control over either the rear wing or both wings showed a response that correlated inversely with longitudinal load transfer, adjusting wing angles to support the underloaded axle. The configuration with fully asymmetric control capabilities was able to further maximise downforce on the underloaded inside tyres during high lateral load transfer and prioritise downforce on the most underloaded tyres in scenarios with combined load transfers.

Additionally, optimal design parameters for brake, torque and roll stiffness distributions shifted progressively rearward for the active aerodynamic configurations. These shifts reflected the enhanced control over aerodynamic loads on the rear axle, enabling more effective utilisation of the rear tyres during braking, initial acceleration, and cornering.

Overall, this research demonstrated the substantial potential of active aerodynamic systems, particularly with 2D active configurations, to enhance vehicle performance. The findings provided valuable insights into the extent and areas of performance improvements, the aerodynamic control strategies that drive these gains, and the resulting shifts in key vehicle design parameters. The study presented an effective simulation framework for evaluating aerodynamic control strategies and informing vehicle setup decisions, establishing a foundation for future research on active aerodynamic technologies. Future work exploring control strategy implementations, integration with other active vehicle systems, and real-world applications could lead to new insights and potentially even greater performance gains, guiding the development of next-generation high-performance and automotive vehicles.

Data Availability Statement

The minimum lap time simulation framework and the corresponding generated performance data supporting this study are openly available from Cranfield Online Research Data at: <https://dspace.lib.cranfield.ac.uk/handle/1826/23604>. (47)

References

1. Fédération Internationale de l'Automobile. FIA 2026 F1 regulations: section c: technical regulations, issue 9; 2024. Available from: https://www.fia.com/sites/default/files/fia_2026_f1_regulations_-_section_c_technical_-_iss09_-_2024-10-17.pdf. (Accessed: 01 November 2024).
2. Forbes DC. Coupling road vehicle aerodynamics and dynamics in simulation. PhD Thesis. Loughborough University; 2017. Available from: <https://hdl.handle.net/2134/25565>.
3. Winkler N, Drugge L, Trigell AS, Efraimsson G. Coupling aerodynamics to vehicle dynamics in transient crosswinds including a driver model. *Computers & Fluids*. 2016;138:26-34. Available from: <https://doi.org/10.1016/j.compfluid.2016.08.006>.
4. Nakashima T, Tsubokura M, Vázquez M, Owen H, Doi Y. Coupled analysis of unsteady aerodynamics and vehicle motion of a road vehicle in windy conditions. *Computers & Fluids*. 2013;80:1-9. Available from: <https://doi.org/10.1016/j.compfluid.2012.09.028>.
5. Carbonne L, Winkler N, Efraimsson G. Use of full coupling of aerodynamics and vehicle dynamics for numerical simulation of the crosswind stability of ground vehicles. *SAE International Journal of Commercial Vehicles*. 2016;9(2):359-70. Available from: <https://doi.org/10.4271/2016-01-8148>.

6. Ahangarnejad AH, Melzi S. Numerical analysis of the influence of an actively controlled spoiler on the handling of a sports car. *Journal of Vibration and Control*. 2018;24(22):5437-48. Available from: <https://doi.org/10.1177/1077546318754683>.
7. Mashrouteh S, Khajepour A, Kasaiezadeh A, Esmailzadeh E, Chen SK, Litkouhi B. Multi-actuation controller for performance vehicles: optimal torque allocation and active aerodynamic. *IEEE Transactions on Vehicular Technology*. 2022;71(3):2721-33. Available from: <https://doi.org/10.1109/TVT.2022.3144133>.
8. Ferraris A, De Cupis D, de Carvalho Pinheiro H, Messana A, Sisca L, Giancarlo Airale A, et al. Integrated design and control of active aerodynamic features for high performance electric vehicles. *SAE Technical Paper 2020-36-0079*. 2021. Available from: <https://doi.org/10.4271/2020-36-0079>.
9. Broniszewski J, Piechna J. A fully coupled analysis of unsteady aerodynamics impact on vehicle dynamics during braking. *Engineering Applications of Computational Fluid Mechanics*. 2019;13(1):623-41. Available from: <https://doi.org/10.1080/19942060.2019.1616326>.
10. Broniszewski J, Piechna JR. Fluid-structure interaction analysis of a competitive car during brake-in-turn manoeuvre. *Energies*. 2022;15(8):2917. Available from: <https://doi.org/10.3390/en15082917>.
11. Kurec K, Remer M, Broniszewski J, Bibik P, Tudruj S, Piechna J. Advanced modeling and simulation of vehicle active aerodynamic safety. *Journal of Advanced Transportation*. 2019;(7308590). Available from: <https://doi.org/10.1155/2019/7308590>.
12. Ayyagari DT, He Y. Aerodynamic analysis of an active rear split spoiler for improving lateral stability of high-speed vehicles. *International Journal of Vehicle Systems Modelling and Testing*. 2017;12(3-4):217-39. Available from: <https://doi.org/10.1504/IJVSMT.2017.089978>.
13. Sikder T, Kapoor S, He Y. Optimizing dynamic performance of high-speed road vehicles using aerodynamic aids. *Proceedings of the ASME 2016 International Mechanical Engineering Congress and Exposition*. vol. Volume 7: Fluids Engineering. Phoenix, USA: The American Society of Mechanical Engineers; 2016. Available from: <https://doi.org/10.1115/IMECE2016-65414>.
14. Hammad M, Qureshi K, He Y. Safety and lateral dynamics improvement of a race car using active rear wing control. *SAE Technical Paper 2019-01-0643*. 2019. Available from: <https://doi.org/10.4271/2019-01-0643>.
15. van Aaken JPT. Influence of asymmetric aerodynamics on lateral acceleration on a Formula Student car. MSc Thesis. Eindhoven University of Technology; 2023. Available from: https://pure.tue.nl/ws/portalfiles/portal/312301921/BEP_report_JobvanAaken.pdf.
16. Cai J, Kapoor S, Sikder T, He Y. Effects of active aerodynamic wings on handling performance of high-speed vehicles. *SAE Technical Paper 2017-01-1592*. 2017. Available from: <https://doi.org/10.4271/2017-01-1592>.

17. Diba F, Barari A, Esmailzadeh E. Handling and safety enhancement of race cars using active aerodynamic systems. *Vehicle System Dynamics*. 2014;52(9):1171-90. Available from: <https://doi.org/10.1080/00423114.2014.930158>.
18. Zavala D, Vicente W, Esquivel R, Salinas-Vazquez M. Numerical simulation of the effect of a centripetal wing spoiler on the flow aerodynamics of a sports car. *SAE Technical Paper 2023-01-5032*. 2023. Available from: <https://doi.org/10.4271/2023-01-5032>.
19. Piechna JR, Kurec K, Broniszewski J, Remer M, Piechna A, Kamieniecki K, et al. Influence of the car movable aerodynamic elements on fast road car cornering. *Energies*. 2022;15(3):689. Available from: <https://doi.org/10.3390/en15030689>.
20. Rijns S, Teschner TR, Blackburn K, Brighton J. Performance analyses of active aerodynamic load balancing designs on high-performance vehicles in cornering conditions. *Physics of Fluids*. 2024;36(8):085199. Available from: <https://doi.org/10.1063/5.0222780>.
21. Casanova D. On minimum time vehicle manoeuvring: the theoretical optimal lap. PhD Thesis. Cranfield University; 2000. Available from: <http://etheses.dur.ac.uk/2671/>.
22. Massaro M, Limebeer DJN. Minimum-lap-time optimisation and simulation. *Vehicle System Dynamics*. 2021;59(7):1069-113. Available from: <https://doi.org/10.1080/00423114.2021.1910718>.
23. Veneri M, Massaro M. A free-trajectory quasi-steady-state optimal-control method for minimum lap-time of race vehicles. *Vehicle System Dynamics*. 2020;58(6):933-54. Available from: <https://doi.org/10.1080/00423114.2019.1608364>.
24. Lovato S, Massaro M. Three-dimensional fixed-trajectory approaches to the minimum-lap time of road vehicles. *Vehicle System Dynamics*. 2022;60(11):3650-67. Available from: <https://doi.org/10.1080/00423114.2021.1969024>.
25. Lovato S, Massaro M. A three-dimensional free-trajectory quasi-steady-state optimal-control method for minimum-lap-time of race vehicles. *Vehicle System Dynamics*. 2022;60(5):1512-30. Available from: <https://doi.org/10.1080/00423114.2021.1878242>.
26. Limebeer DJN, Bastin M, Warren E, Fensham HG. Optimal control of a NASCAR – specification race car. *Vehicle System Dynamics*. 2023;61(5):1210-35. Available from: <https://doi.org/10.1080/00423114.2022.2067573>.
27. Christ F, Wischniewski A, Heilmeier A, Lohmann B. Time-optimal trajectory planning for a race car considering variable tyre-road friction coefficients. *Vehicle System Dynamics*. 2021;59(4):588-612. Available from: <https://doi.org/10.1080/00423114.2019.1704804>.
28. Perantoni G, Limebeer DJN. Optimal control for a Formula One car with variable parameters. *Vehicle System Dynamics*. 2014;52(5):653-78. Available from: <https://doi.org/10.1080/00423114.2014.889315>.

29. Imani Masouleh M, Limebeer DJN. Optimizing the aero-suspension interactions in a Formula One car. *IEEE Transactions on Control Systems Technology*. 2016;24(3):912-27. Available from: <https://doi.org/10.1109/TCST.2015.2475396>.
30. Jiménez Elbal A, Zarzuelo Conde A, Siampis E. Simultaneous optimisation of vehicle design and control for improving vehicle performance and energy efficiency using an open source minimum lap time simulation framework. *World Electric Vehicle Journal*. 2024;15(8):366. Available from: <https://doi.org/10.3390/wevj15080366>.
31. de Buck P, Martins JRRA. Minimum lap time trajectory optimisation of performance vehicles with four-wheel drive and active aerodynamic control. *Vehicle System Dynamics*. 2023;61(8):2103-19. Available from: <https://doi.org/10.1080/00423114.2022.2101930>.
32. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model. *SAE Technical Paper 2018-01-0725*. 2018. Available from: <https://doi.org/10.4271/2018-01-0725>.
33. Heft A, Indinger T, Adams N. Introduction of a new realistic generic car model for aerodynamic investigations. *SAE Technical Paper 2012-01-0168*. 2012. Available from: <https://doi.org/10.4271/2012-01-0168>.
34. Rijns S, Teschner TR, Blackburn K, Brighton J. Effects of cornering conditions on the aerodynamic characteristics of a high-performance vehicle and its rear wing. *Physics of Fluids*. 2024;36(4):045119. Available from: <https://doi.org/10.1063/5.0204204>.
35. Rijns S, Teschner TR, Blackburn K, Brighton J. Integrated numerical and experimental workflow for high performance vehicle aerodynamics. *SAE Technical Paper 2024-01-5016*. 2024. Available from: <https://doi.org/10.4271/2024-01-5016>.
36. Rijns S, Teschner TR, Blackburn K, Brighton J. Aerodynamic analysis of the ride height dependency of a high-performance vehicle equipped with a multichannel diffuser in ground effect. *SAE Technical Paper 2023-01-5064*. 2023. Available from: <https://doi.org/10.4271/2023-01-5064>.
37. Rijns S, Teschner TR, Blackburn K, Proenca AR, Brighton J. Experimental and numerical investigation of the aerodynamic characteristics of high-performance vehicle configurations under yaw conditions. *Physics of Fluids*. 2024;36(4):045112. Available from: <https://doi.org/10.1063/5.0196979>.
38. Lanfrit M. Best practice guidelines for handling automotive external aerodynamics with Fluent. University of Southampton; 2005. Available from: https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf. (Accessed: 01 May 2022).
39. Saunders JW, Mansour RB. On-road and wind tunnel turbulence and its measurement using a four-hole dynamic probe ahead of several cars. *SAE Technical Paper 2000-01-0350*. 2000. Available from: <https://doi.org/10.4271/2000-01-0350>.

40. Modeling turbulent flows - introductory Fluent training. Ansys Inc; 2006. Available from: https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence_Notes_Fluent-v6.3.06.pdf. (Accessed: 01 May 2022).
41. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*. 1994;32(8):1598-605. Available from: <https://doi.org/10.2514/3.12149>.
42. Tremlett AJ, Massaro M, Purdy DJ, Velenis E, Assadian F, Moore AP, et al. Optimal control of motorsport differentials. *Vehicle System Dynamics*. 2015;53(12):1772-94. Available from: <https://doi.org/10.1080/00423114.2015.1093150>.
43. Dixon JC. Tires, suspension and handling, second edition. 2nd edn. Warrendale, USA: SAE International; 1996.
44. Pacejka HB. Tire and vehicle dynamics. 3rd edn. Oxford, UK: Butterworth-Heinemann; 2012. Available from: <https://doi.org/10.1016/B978-0-08-097016-5.00001-2>.
45. Avon Tyres. Technical data resources; 2024. Available from: <https://www.avontyres.com/en-gb/tyre-care/motorsport-technical-data/technical-data-resources/>. (Accessed: 12 April 2024).
46. Helix. SPM242-176 electric motor unit specification sheet; 2024. Available from: <https://www.ehelix.com/products/spm242-electric-motor/>. (Accessed: 01 September 2024).
47. Rijns S, Siampis E, Teschner TR, Blackburn K, Brighton J. Minimum lap time simulation framework: active aerodynamics and torque distribution. *Cranfield Online Research Data (CORD)*; 2025. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/23604>.

Chapter 8

Lap Time Performance Analysis of Active Aerodynamics Combined With Active Torque Distribution

This chapter was originally published as:

Rijns S, Teschner TR, Siampis E, Blackburn K, Brighton J. Performance optimisation of high-performance vehicles with active aerodynamics and active torque distribution [in review]. Advanced Vehicle Engineering Centre. Cranfield University. 2025.

Abstract

This study investigates the performance potential of active aerodynamic systems combined with active torque distribution (ATD) on high-performance vehicles. Minimum lap time simulations are employed to simultaneously optimise the vehicle's trajectory, control inputs, design parameters, and tyre characteristics for various active aerodynamic configurations combined with all-wheel-drive (AWD) and ATD powertrains, evaluated on the Circuit de Barcelona-Catalunya. Results show that ATD setups reduce lap times by up to 4.8% compared to AWD setups, with further reductions of up to 5.4% when combined with active aerodynamics. AWD configurations favour moderately low downforce setups to balance drag, while ATD configurations benefit from higher downforce to maximise total tyre grip. Active aerodynamic systems with asymmetric actuation further enhance performance by prioritising downforce on underloaded tyres, improving combined grip on each axle. Additionally, the findings reveal that AWD configurations perform best with tyres featuring lower nominal vertical loads and higher cornering stiffness to maximise grip for less loaded tyres, while ATD setups excel with tyres featuring slightly higher nominal loads and lower cornering stiffness, achieving more balanced grip across all tyres. Overall, this study highlights the synergistic performance potential of active aerodynamics with ATD and provides detailed insights into the mechanisms driving their performance gains, guiding next-generation vehicle development.

Keywords: Active Aerodynamics; Active Torque Distribution; Performance Optimisation; Minimum Lap Time; Optimal Control; Vehicle Dynamics

8.1 Introduction

Active systems on vehicles, which dynamically adapt to operating conditions, have gained significant attention for their potential to improve vehicle handling, efficiency, and overall performance. Among these innovations, active aerodynamic systems, which influence aerodynamic forces, and active torque distribution (ATD) systems, which allocate torque to individual wheels, have emerged as particularly promising solutions. Combined, these innovations could potentially offer significant performance enhancements for automotive and motorsport applications.

Studies on active aerodynamic systems have demonstrated their capability to enhance vehicle stability during dynamic manoeuvres, reduce braking distances by over 6%, and improve lap times by nearly 3% compared to static aerodynamic configurations (1, 2, 3, 4, 5, 6). Asymmetric aerodynamic designs, such as split and tilt wings, have been shown to vary aerodynamic loads longitudinally and laterally, improving lateral stability and acceleration during dynamic events by prioritising downforce on underloaded tyres (7, 8, 9, 10, 11, 12, 13, 14). Notably, tilt wing designs have been reported to outperform split wings, achieving a 1.2% increase in cornering velocity due to their greater influence on aerodynamic load modulation and consistent response to control strategies (15).

ATD has been extensively studied for its potential to enhance vehicle performance. Research has demonstrated its ability to optimise tyre grip utilisation, enabling higher longitudinal and lateral accelerations (16, 17, 18, 19, 20). ATD has been shown to improve handling performance by mitigating oversteer and understeer tendencies through the generation of corrective yaw moments, providing more responsive vehicle behaviour during dynamic manoeuvres (21). Previous work has also reported that integrating active aerodynamics and ATD improved stability during harsh manoeuvres compared to using ATD alone (2). Additionally, minimum lap time simulations demonstrated that ATD reduced lap times by 4%, with a further 1% improvement achieved through the addition of an active rear wing (22).

Minimum lap time simulations, with varying levels of trajectory planning and vehicle dynamics modelling, have been widely used to optimise and evaluate vehicle performance by minimising lap time while adhering to track constraints and dynamic limitations (23, 24, 25, 26, 27, 28, 29). Such studies have demonstrated lap time improvements of 1%-2% with conventional aerodynamics across different powertrain configurations (30, 31, 32, 22). Additionally, improvements of approximately 1% have been reported using 2D active aerodynamic systems capable of modulating aerodynamic balance longitudinally and laterally, outperforming other conventional active aerodynamic configurations (33).

Although prior research has established the individual benefits of active aerodynamics and ATD, few studies have investigated their combined implementation. The limited research available typically employed simplified simulation techniques, which did not fully capture the dynamic interactions between these systems, leaving their synergistic potential largely underexplored. To address this gap, this study evaluates the combined effects of active aerodynamics and ATD on vehicle performance. It also investigates how these systems influence optimal vehicle setups and analyses the tyre characteristics most effective for further enhancing performance.

This study employs minimum lap time simulations using a free-trajectory approach, transient vehicle dynamics, and Computational Fluid Dynamics (CFD) derived aerodynamic data. The simulation framework simultaneously optimises the vehicle’s trajectory, control inputs, and selected design parameters. The performance of various active aerodynamic configurations, including asymmetrically actuated systems, combined with all-wheel-drive (AWD) and ATD powertrains, is evaluated on the Circuit de Barcelona-Catalunya. Through comprehensive analyses, this study quantifies the individual and combined performance gains delivered by each system, identifies the underlying mechanisms driving these improvements, and evaluates the influence of vehicle design parameters and tyre characteristics on maximising performance.

8.2 Methodology

8.2.1 Vehicle Geometry

This study evaluates the performance of various active aerodynamic configurations integrated with AWD and ATD powertrains. It employs a high-performance vehicle concept, introduced in Chapter 6 (15), which is based on the *DrivAer hp-F* (34), a high-performance variant of the DrivAer Fastback model (35). The vehicle features forebody strakes, a flat underbody, and a 10° multichannel diffuser. It is equipped with an active split front wing and tilt rear wing. The front wing elements can individually adjust their angle of attack from 0° to 10° , while the rear wing’s angle of attack can be varied between 0° and 30° , with a tilt angle adjustment range of $\pm 12^\circ$. The high-performance vehicle concept is depicted in Figure 8.1 and its main dimensions are detailed in Table 8.2.

Four aerodynamic configurations of this vehicle are analysed to evaluate the influence of varying levels of aerodynamic adaptability on vehicle performance. The first configuration (C1) features a fixed front and rear wing angle, serving as a baseline for comparison. The second configuration (C2) retains the fixed front wing but incorporates active rear wing control, whereas the third configuration (C3) allows for active control of both the front and rear wings. The fourth configuration (C4) introduces active asymmetrical control, with independent actuation of the left and right front wing elements, in addition to active control of the rear wing’s angle of attack and tilt angle. This asymmetric actuation functionality allows influence on the aerodynamic balance both longitudinally and laterally, which is therefore classified as a ‘2D active aerodynamic system’. Each aerodynamic configuration is evaluated with an AWD and ATD powertrain, as detailed in Table 8.1.

Table 8.1: Overview of the vehicle configurations.

Feature	C1	C2	C3	C4	C1-ATD	C2-ATD	C3-ATD	C4-ATD
Front wing	Static	Static	Active	Active asymmetric	Static	Static	Active	Active asymmetric
Rear wing	Static	Active	Active	Active asymmetric	Static	Active	Active	Active asymmetric
Powertrain	AWD	AWD	AWD	AWD	ATD	ATD	ATD	ATD

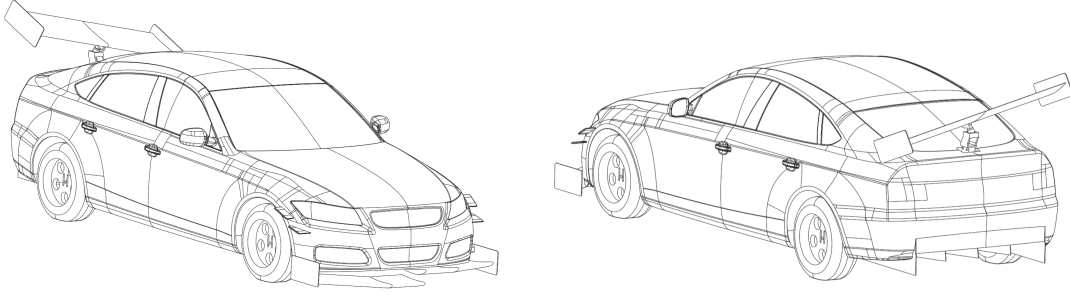


Figure 8.1: Illustration of the high-performance vehicle concept featuring an active split front wing and active tilt rear wing. The aerodynamic systems are displayed in an asymmetric operating setting to illustrate the design.

8.2.2 Vehicle Model

This study employs the seven degree-of-freedom planar vehicle model introduced in Chapter 7 (33), illustrated in Figure 8.2. The simulation framework is extended to optimise ATD control inputs in addition to those of the active aerodynamic systems. The model captures longitudinal, lateral, and yaw rate motions, together with individual wheel rotations. It also accounts for longitudinal and lateral load transfers, non-linear tyre behaviour, and aerodynamic forces. Table 8.2 lists the vehicle parameters, which are based on the geometry and representative specifications of high-performance vehicles (36).

To align with the optimisation framework, vehicle trajectory tracking is described in a curvilinear coordinate system, which uses the travelled distance s (m), lateral displacement n (m), and angle ϵ (rad), with respect to track's centreline. Since the optimisation framework is distance-based, as will be detailed in Section 8.2.3, time-dependent equations are transformed into functions of travelled distance. This transformation follows the methodology by Casanova (23), introducing the scaling factor S_f , which represents the inverse (dt/ds) of Eq. (8.1), as applied in Eqs. (8.2) and (8.3). In the equations below, t (s) denotes time, κ (m^{-1}) the track's curvature, r (rad/s) the vehicle's yaw rate, and v_x (m/s) and v_y (m/s) the vehicle's longitudinal and lateral velocities, respectively.

$$\frac{d}{dt}s = \frac{v_x \cos(\epsilon) - v_y \sin(\epsilon)}{1 - n\kappa} \quad (8.1)$$

$$\frac{d}{ds}n = \left(v_x \sin(\epsilon) + v_y \cos(\epsilon) \right) S_f \quad (8.2)$$

$$\frac{d}{ds}\epsilon = S_f r - \kappa \quad (8.3)$$

The equations of motion are described by Eqs. (8.4)-(8.6). Subscripts are used to specify components, where k indicates the longitudinal (x), lateral (y), or vertical (z) direction, i represents the front (f) or rear (r) axle location, and j specifies the left (l) or right (r) wheel position. $F_{k,ij}$ (N) represents the tyres forces, F_D (N) the aerodynamic drag, F_R (N) the rolling resistance, and $F_{S,i}$ (N) the aerodynamic side forces.

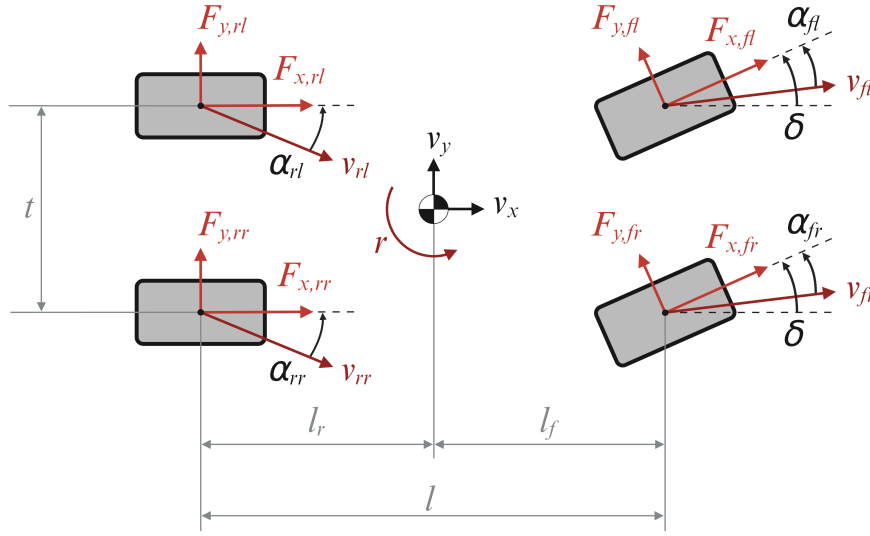


Figure 8.2: Schematic illustration of the vehicle model.

The steering angle is denoted by δ (rad), the vehicle's total mass by m (kg), the yaw moment of inertia by I_z (kg m²), the axle distances from the centre of gravity (COG) by l_i (m), and the track width by t_w (m). Additionally, wheel dynamics are modelled by Eq. (8.7), where Ω_{ij} (rad/s) indicates the wheel angular velocity, T_{ij} (Nm) the applied wheel torque, R_w (m) the wheel radius, and I_w (kg m²) the wheel rotational inertia.

$$\begin{aligned} \frac{d}{ds}v_x = & \left[(F_{x,fl} + F_{x,fr}) \cos(\delta) - (F_{y,fl} + F_{y,fr}) \sin(\delta) \right. \\ & \left. + F_{x,rl} + F_{x,rr} + m v_y r - F_D - F_R \right] \frac{S_f}{m} \end{aligned} \quad (8.4)$$

$$\begin{aligned} \frac{d}{ds}v_y = & \left[(F_{y,fl} + F_{y,fr}) \cos(\delta) + (F_{x,fl} + F_{x,fr}) \sin(\delta) \right. \\ & \left. + F_{y,rl} + F_{y,rr} - m v_x r + F_S \right] \frac{S_f}{m} \end{aligned} \quad (8.5)$$

$$\begin{aligned} \frac{d}{ds}r = & \left[\left((F_{x,fl} + F_{x,fr}) \sin(\delta) + (F_{y,fl} + F_{y,fr}) \cos(\delta) + F_{S,f} \right) l_f \right. \\ & - \left(F_{y,rl} + F_{y,rr} + F_{S,r} \right) l_r + \left(F_{x,rr} - F_{x,rl} \right) \frac{t_w}{2} \\ & \left. + \left((F_{x,fr} - F_{x,fl}) \cos(\delta) + (F_{y,fl} - F_{y,fr}) \sin(\delta) \right) \frac{t_w}{2} \right] \frac{S_f}{I_z} \end{aligned} \quad (8.6)$$

$$\frac{d}{ds}\Omega_{ij} = \left(T_{ij} - F_{x,ij} R_w \right) \frac{S_f}{I_w} \quad (8.7)$$

Table 8.2: Overview of the vehicle parameters.

Parameter	Value	Unit	Description
L	4.613	m	Length
W	1.792	m	Width
H	1.397	m	Height
A_{ref}	2.160	m ²	Frontal area
h_0	0.130	m	Ride height
m	1565	kg	Total mass
m_s	1375	kg	Sprung mass
$m_{u,f}$	90	kg	Front unsprung mass
$m_{u,r}$	100	kg	Rear unsprung mass
I_z	1450	kg m ²	Yaw moment of inertia
l	2.786	m	Wheelbase
l_f	1.393	m	Distance between front axle and COG
l_r	1.393	m	Distance between rear axle and COG
t_w	1.520	m	Track width
h_{COG}	0.500	m	Height of COG
$h_{u,f}$	0.297	m	Height of front unsprung mass
$h_{u,r}$	0.297	m	Height of rear unsprung mass
h_f	0.070	m	Height of front roll centre
h_r	0.110	m	Height of rear roll centre
h_{RW}	1.280	m	Height of rear wing
d	0.410	m	Distance between COG and roll axis
R_w	0.314	m	Wheel radius
I_w	0.9	kg m ²	Wheel rotational inertia
k_ϕ	1.146×10^5	Nm/rad	Total roll stiffness
$P_{e,\text{max}}$	456	kW	Maximum electric motor power
$T_{e,\text{max}}$	602	Nm	Maximum electric motor torque
$\Omega_{e,\text{max}}$	17750	RPM	Maximum electric motor speed
γ_g	7.252	-	Final gear ratio

8.2.2.1 Tyres

Tyre forces depend on vertical loads, calculated using Eq. (8.8). The first term describes the static weight distribution, influenced by the unsprung masses $m_{u,i}$ (kg), the sprung mass m_s (kg), and the wheelbase length l (m). The remaining terms represent the gravitational acceleration g (m/s²), aerodynamic lift forces $F_{L,ij}$ (N), and longitudinal $\Gamma_{x,ij}$ (N) and lateral $\Gamma_{y,ij}$ (N) load transfer components.

$$F_{z,ij} = \frac{1}{2}g \left(m_{u,i} + m_s \frac{l_i}{l} \right) - F_{L,ij} \pm \Gamma_{x,ij} \pm \Gamma_{y,ij} \quad (8.8)$$

Total longitudinal Γ_x (N) and lateral Γ_y (N) load transfer are described by Eqs. (8.9) and (8.10), where h_{COG} (m) signifies the COG height, F_D^0 (N) the nominal drag force acting at the COG height, and F_D^{aero} (N) the additional drag from aerodynamic elements acting at the rear wing height h_{RW} (m).

$$\Gamma_x = \left[(F_{x,fl} + F_{x,fr}) \cos(\delta) - (F_{y,fl} + F_{y,fr}) \sin(\delta) + F_{x,rl} + F_{x,rr} + F_D^0 \right] \frac{h_{COG}}{l} + F_D^{aero} \frac{h_{RW}}{l} \quad (8.9)$$

$$\Gamma_y = \left[(F_{x,fl} + F_{x,fr}) \sin(\delta) + (F_{y,fl} + F_{y,fr}) \cos(\delta) + F_{y,rl} + F_{y,rr} \right] \frac{h_{COG}}{t_w} + F_S \frac{h_{RW}}{t_w} \quad (8.10)$$

Longitudinal load transfer is equally distributed between the left and right wheels. The lateral load transfer distribution accounts for contributions from the unsprung masses, the sprung mass through suspension links, and the roll couple of the sprung mass via the springs, as described by Eqs. (8.11)-(8.13). These equations incorporate the unsprung mass height $h_{u,i}$ (m), the roll centre height h_i (m), and the distance between the COG and the roll axis d (m) to calculate the axle-specific fraction of total lateral load transfer $C_{\Gamma y,i}$ (-). Although most of these parameters are constrained by design limitations, the roll stiffness distribution γ_{k_ϕ} (-), which represents the proportion of total roll stiffness k_ϕ (Nm/rad) at the rear axle, can be adjusted to some extent.

$$\Gamma_{y,ij} = \frac{C_{\Gamma y,i}}{C_{\Gamma y,f} + C_{\Gamma y,r}} \Gamma_y \quad (8.11)$$

$$C_{\Gamma y,f} = \frac{h_{u,f} m_{u,f}}{t_w} + \frac{l_r h_f m_s}{l t_w} + \frac{(1 - \gamma_{k_\phi}) k_\phi m_s (\frac{d}{k_\phi - m_s g d})}{t_w} \quad (8.12)$$

$$C_{\Gamma y,r} = \frac{h_{u,r} m_{u,r}}{t_w} + \frac{l_f h_r m_s}{l t_w} + \frac{\gamma_{k_\phi} k_\phi m_s (\frac{d}{k_\phi - m_s g d})}{t_w} \quad (8.13)$$

Longitudinal and lateral tyre forces are modelled using the Pacejka '96 model (37), as expressed by Eqs. (8.14) and (8.15). Tyre model coefficients are derived from Avon British F3 tyres (38), listed in Table 8.3. The peak value $D_{k,ij}$ (-) and friction coefficient μ_{ij} (-) are load-dependent, as expressed in Eqs. (8.16) and (8.17), where the nominal vertical tyre load F_{z0} (N), friction coefficient at the nominal load μ_0 (-), and Pacejka's coefficient for friction variation with load P_{D2} (-) are incorporated. The optimisation parameter γ_{z0} (-) is introduced to shift the nominal tyre load without affecting the peak force magnitude.

$$F_{x,ij} = D_{x,ij} \sin \left[C_x \tan^{-1} \left\{ B_x s_{ij} - E_x \left(B_x s_{ij} - \tan^{-1}(B_x s_{ij}) \right) \right\} \right] \quad (8.14)$$

$$F_{y,ij} = D_{y,ij} \sin \left[C_y \tan^{-1} \left\{ B_y \alpha_{ij} - E_y \left(B_y \alpha_{ij} - \tan^{-1}(B_y \alpha_{ij}) \right) \right\} \right] \quad (8.15)$$

Table 8.3: Overview of the tyre model coefficients.

Coefficient	Value	Description
F_{z0}	4905	Nominal Load (N)
μ_0	1.41	Friction coefficient
P_{D2}	-0.6	Variation of friction with load
B_x	6	Stiffness factor, longitudinal
C_x	2.3	Shape factor, longitudinal
E_x	0.9	Curvature factor, longitudinal
B_y	6	Stiffness factor, lateral
C_y	2.5	Shape factor, lateral
E_y	0.5	Curvature factor, lateral

$$D_{ij} = \mu_{ij} \frac{F_{z,ij}}{\gamma_{z0}} \quad (8.16)$$

$$\mu_{ij} = \mu_0 + P_{D2} \frac{F_{z,ij} - F_{z0} \gamma_{z0}}{F_{z0} \gamma_{z0}} \quad (8.17)$$

Tyre slip ratios s_{ij} (-) and slip angles α_{ij} (rad) are described by Eqs. (8.18)-(8.20), with the effective tyre velocity v_{ij} (m/s) calculated by Eq. (8.21). Rolling resistance is modelled using a constant road friction coefficient C_f (-), as expressed by Eq. (8.22).

$$\alpha_{fj} = \delta - \tan^{-1} \left(\frac{r l_f + v_y}{v_x \pm r \frac{t_w}{2}} \right) \quad (8.18)$$

$$\alpha_{rj} = \tan^{-1} \left(\frac{r l_r - v_y}{v_x \pm r \frac{t_w}{2}} \right) \quad (8.19)$$

$$s_{ij} = \frac{R_w \Omega_{ij} - v_{ij}}{v_{ij}} \quad (8.20)$$

$$v_{ij} = \cos(\alpha_{ij}) \sqrt{\left(v_y \pm r l_i \right)^2 + \left(v_x \pm r \frac{t_w}{2} \right)^2} \quad (8.21)$$

$$F_R = C_f (F_{z,fl} + F_{z,fr} + F_{z,rl} + F_{z,rr}) \quad (8.22)$$

8.2.2.2 Aerodynamics

Aerodynamic data of the high-performance vehicle and its aerodynamic devices are obtained from CFD simulations conducted in Chapter 7 (33). The deployed methodology was established on a grid convergence study, turbulence model evaluation, and validation with experimental data and high-fidelity Delayed Detached Eddy Simulation (DDES) results from Chapter 2 (39). This study focuses on the implementation of the aerodynamic data into the vehicle model, with a comprehensive description of the CFD simulation methodology available in Section 7.2.2 (33).

The aerodynamic performance data is incorporated using polynomial fitted functions of the simulated results. The functions describe the influence of the left front wing element angle of attack $\alpha_{\text{FW-L}}$ (deg), right front wing element angle of attack $\alpha_{\text{FW-R}}$ (deg), rear wing angle of attack α_{RW} (deg), and rear wing tilt angle α_{TW} (deg), on both the magnitude and distribution of aerodynamic forces acting on the vehicle. Specifically, they define variations in key aerodynamic force components, including lift at the front $C_{L,\text{front}}$ (-) and rear wheels $C_{L,\text{rear}}$ (-), lift at the left $C_{L,\text{left}}$ (-) and right wheels $C_{L,\text{right}}$ (-), side force at the front $C_{S,\text{front}}$ (-) and rear axles $C_{S,\text{rear}}$ (-), and total drag C_D (-).

The influence of each element on a specific aerodynamic force component coefficient $\Delta C_{u,v}^w$ (-) is expressed as a function of its operating angle α_w (deg) using polynomial formulations, as described by Eq. (8.23). Here, $w \in \{\text{FW-L}, \text{FW-R}, \text{RW}, \text{TW}\}$ represents the aerodynamic elements, including the left and right front wing angles, rear wing angle, and rear wing tilt, respectively. The aerodynamic lift, drag, and side forces are indexed by $u \in \{L, D, S\}$, while force distribution locations are specified by $v \in \{\text{front}, \text{rear}, \text{left}, \text{right}\}$. The polynomial coefficients, listed in Table 8.4, are denoted by $P_{u,v}^w(\ell)$, with ℓ (-) indexing the polynomial terms and $N_{u,v}^w$ indicating the polynomial order.

$$\Delta C_{u,v}^w = \sum_{\ell=1}^{N_{u,v}^w} P_{u,v}^w(\ell) \alpha_w^\ell \quad (8.23)$$

The combined aerodynamic force coefficients $C_{u,v}$ (-) are obtained by adding the individual effects of all aerodynamic elements with the nominal aerodynamic coefficients $C_{u,v}^0$ (-), which represent the vehicle's baseline aerodynamic performance without system actuation, as formulated in Eq. (8.24). The total aerodynamic coefficients are then computed by summing the relevant contributions, with lift exemplified in Eq. (8.25).

$$C_{u,v} = C_{u,v}^0 + \Delta C_{u,v}^{\text{FW-L}} + \Delta C_{u,v}^{\text{FW-R}} + \Delta C_{u,v}^{\text{RW}} + \Delta C_{u,v}^{\text{TW}} \quad (8.24)$$

$$C_L = C_{L,\text{front}} + C_{L,\text{rear}} \quad (8.25)$$

Given the significance of aerodynamic loads in tyre force generation, lift coefficients are formulated for each wheel individually to enable direct integration with the vertical load calculations in the vehicle dynamics model. This distribution follows the previously established side-specific lift contributions. Eq. (8.26) provides the formulation for the front left wheel, with analogous expressions applied to the remaining wheels.

$$C_{L,\text{FL}} = C_{L,\text{front}} \frac{C_{L,\text{left}}}{C_L} \quad (8.26)$$

Finally, the aerodynamic forces $F_{u,v}$ (N) acting on the vehicle are calculated by Eq. (8.27), where ρ (kg/m³) represents the air density, A_{ref} (m²) the vehicle's reference frontal area, and v_x (m/s) the longitudinal velocity.

$$F_{u,v} = \frac{1}{2} \rho C_{u,v} A_{\text{ref}} v_x^2 \quad (8.27)$$

Table 8.4: Overview of the aerodynamic relationship coefficients.

Coefficient	$C_{L,front}$	$C_{L,rear}$	$C_{L,left}$	$C_{L,right}$	C_D	$C_{S,front}$	$C_{S,rear}$
$C_{u,v}^0$	-0.188	-0.103	-0.146	-0.146	0.346	0	0
$P_{u,v}^{FW-L}(1)$	-9.75e-3	1.58e-3	-8.17e-3	0	0	0	0
$P_{u,v}^{FW-R}(1)$	-9.75e-3	1.58e-3	0	-8.17e-3	0	0	0
$P_{u,v}^{RW}(1)$	3.79e-3	-2.18e-2	-9.01e-3	-9.01e-3	2.90e-3	0	0
$P_{u,v}^{RW}(2)$	-1.94e-4	1.11e-3	4.32e-4	4.32e-4	7.87e-5	0	0
$P_{u,v}^{RW}(3)$	4.31e-6	1.56e-5	9.95e-6	9.95e-6	0	0	0
$P_{u,v}^{RW}(4)$	0	-2.75e-6	-1.37e-6	-1.37e-6	0	0	0
$P_{u,v}^{RW}(5)$	0	4.92e-8	2.46e-8	2.46e-8	0	0	0
$P_{u,v}^{TW}(1)$	0	-7.10e-5	4.30e-3	-4.30e-3	0	0	7.19e-3
$P_{u,v}^{TW}(2)$	0	0	3.55e-5	3.55e-5	0	0	0

8.2.2.3 Powertrain

The aerodynamic configurations considered in this study are evaluated with AWD and ATD powertrains, utilising a high-performance electric motor unit paired with a single-speed transmission, capable of delivering a maximum continuous power output of $P_{e,max} = 456$ kW, a maximum continuous torque output of $T_{e,max} = 602$ Nm, and a maximum motor speed of $\Omega_{e,max} = 17750$ RPM (40).

Even though high-performance electric vehicles typically employ multiple motor units for advanced torque distribution, such as one per axle or wheel, both powertrain configurations are modelled with a single electric motor for simplicity. This serves as an elementary use case to explore key performance trends under varying torque allocation capabilities, ensuring consistent total power and torque output. This simplification enables a focused analysis of the interaction between torque allocation and active aerodynamic control, with implementation details considered beyond the scope of this study.

Torque allocation for the AWD configurations is governed by a fixed torque distribution ratio γ_T (-), which determines the fraction of total electric motor torque T_e (Nm) directed to the rear axle. Similarly, total braking torque T_b (Nm) is distributed between the axles using a fixed brake balance γ_b (-), which specifies the fraction allocated to the front axle. Torque at the front T_{fj} (Nm) and rear wheels T_{rj} (Nm) is then expressed by Eqs. (8.28) and (8.29), where γ_g (-) denotes the final gear ratio.

$$T_{fj} = \frac{1}{2} \left((1 - \gamma_T) T_e \gamma_g + T_b \gamma_b \right) \quad (8.28)$$

$$T_{rj} = \frac{1}{2} \left(\gamma_T T_e \gamma_g + T_b (1 - \gamma_b) \right) \quad (8.29)$$

ATD configurations can dynamically allocate drive and brake torque to each individual wheel using wheel-specific torque ratios γ_{ij} (-), where individual wheel torques T_{ij} (Nm) are described by Eq. (8.30).

$$T_{ij} = \gamma_{ij} T_e + \gamma_{ij} T_b \quad (8.30)$$

8.2.3 Optimisation Framework

The optimisation framework employed in this study builds upon the methodology developed by Jiménez et al. (32) and subsequently extended in Chapter 7 (33). The present work further adapts the framework to enable the integrated optimisation of active aerodynamic systems and active torque distribution. A brief overview is provided below, while readers are referred to the referenced works for more detailed descriptions.

The Minimum Lap Time Problem (MLTP) is formulated as an Optimal Control Problem (OCP) and transcribed into a Non-Linear Programming (NLP) problem using direct orthogonal collocation. Travelled distance is used as the independent variable, reducing computational complexity by eliminating time as a variable. The OCP is transcribed into a NLP by discretising the track into N segments of length ds_k (m).

The OCP is formulated to optimise the vehicle states X_k , control inputs U_k , auxiliary variables Y_k , and selected vehicle design parameters p to minimise total lap time. Vehicle states within each segment are approximated using Legendre polynomials of degree d . Collocation points $s_{k,j}$ with $j = \{1, \dots, d\}$ are selected at the polynomial roots. Collocation constraints enforce equality between the polynomial derivatives and governing dynamic equations, ensuring the vehicle's trajectory evolves according to its dynamics model.

The contributions of collocation points within each track segment, denoted by $\mathbf{S}_{f,k}$, are weighted by vector $\mathbf{B}$ and summed to approximate lap time in the NLP objective function, as defined in Eqs. (8.31) and (8.32). To promote physically plausible solutions, regularisation terms are introduced that penalise the sum of squares of control inputs, auxiliary variables, and their m -th derivatives at each segment Δ_s^m . The weight factors r_m are used to govern their influence on the objective function.

Minimise

$$J = \sum_{k=1}^N \mathbf{B} \mathbf{S}_{f,k} ds_k + \underbrace{\sum (r_m \Delta_s^m U_k)^2 + \sum (r_m \Delta_s^m Y_k)^2}_{\text{regularisation}} \quad (8.31)$$

With

$$X = \begin{bmatrix} v_x \\ v_y \\ r \\ n \\ \varepsilon \\ \Omega_{ij} \end{bmatrix}, U = \begin{bmatrix} T_e \\ T_b \\ \delta \\ \alpha_{\text{FW}(\cdot j)} \\ \alpha_{\text{RW}} \\ \alpha_{\text{TW}} \\ \gamma_j \end{bmatrix}, Y = \begin{bmatrix} \Gamma_x \\ \Gamma_y \end{bmatrix}, \text{ and } p = \begin{bmatrix} \gamma_b \\ \gamma_T \\ \gamma_{\phi} \\ \gamma_{z0} \end{bmatrix} \quad (8.32)$$

The optimisation is subject to bounds on the optimisation variables and additional constraints described in Eqs. (8.33)-(8.39). Powertrain capabilities are imposed through maximum electric motor power and angular velocity constraints, as defined in Eqs. (8.33) and (8.34). To avoid unrealistic scenarios, simultaneous application of drive and brake torque is prevented by Eq. (8.35). Tyre forces are limited by a friction circle model with a uniform friction coefficient, ensuring combined longitudinal and lateral forces remain within the available grip, as expressed in Eq. (8.36).

A constraint on the individual wheel torque ratios for the ATD powertrain is enforced through Eq. (8.37), preventing the total allocated torque from exceeding the electric motor's available output. Additionally, longitudinal and lateral load transfer are introduced as auxiliary decision variables, constrained by Eqs. (8.38) and (8.39), to avoid an algebraic loop arising from the interdependence between load transfer, tyre forces, and vehicle accelerations.

$$\Omega_e T_e - P_{e,max} \leq 0 \quad (8.33)$$

$$\Omega_e - \Omega_{e,max} \leq 0 \quad (8.34)$$

$$T_e T_b = 0 \quad (8.35)$$

$$\frac{F_{x,ij}^2 + F_{y,ij}^2}{F_{z,ij}^2} - \mu_{ij}^2 \leq 0 \quad (8.36)$$

$$1 - (\gamma_{FL} + \gamma_{FR} + \gamma_{RL} + \gamma_{RR}) = 0 \quad (8.37)$$

$$\Gamma_x - \left[\left((F_{x,fl} + F_{x,fr}) \cos(\delta) - (F_{y,fl} + F_{y,fr}) \sin(\delta) + F_{x,rl} + F_{x,rr} + F_D^0 \right) \frac{h_{COG}}{l} + F_D^{aero} \frac{h_{RW}}{l} \right] = 0 \quad (8.38)$$

$$\Gamma_y - \left[\left((F_{x,fl} + F_{x,fr}) \sin(\delta) + (F_{y,fl} + F_{y,fr}) \cos(\delta) + F_{y,rl} + F_{y,rr} \right) \frac{h_{COG}}{t_w} + F_S \frac{h_{RW}}{t_w} \right] = 0 \quad (8.39)$$

Table 8.5: Overview of the rate limits and regularisation weight factors for control inputs and auxiliary variables used for the performance optimisation simulations.

Parameter	Symbol	Rate limits		Regularisation		
		Lower	Upper	r_0	r_1	r_2
Drive torque	T_e	-2e4 Nm/s	2e4 Nm/s	0	0	0
Brake torque	T_b	-2e4 Nm/s	2e4 Nm/s	0	0	0
Active torque distribution	γ_{ij}	-2e4 Nm/s	2e4 Nm/s	0	0	0
Steering angle	δ	-0.1 rad/s	0.1 rad/s	0	10	15
Front wing angle	α_{FW}	-20 deg/s	20 deg/s	0	0	0.3
Rear wing angle	α_{RW}	-60 deg/s	60 deg/s	0	0	0.9
Rear wing tilt	α_{TW}	-24 deg/s	24 deg/s	0.005	0	0.4
Longitudinal load transfer	Γ_x	-	-	0	0	1

The lap time simulations are conducted on the Circuit de Barcelona-Catalunya, featuring both high-speed straights and technical corners. Elevation changes are omitted, as the track serves purely as a representative case for comparative performance evaluation. The NLP problem is implemented in MATLAB using the automatic differentiation framework CasADi and solved with the IPOPT solver, configured with an optimality tolerance of 1×10^{-6} and constraint violation tolerances of 1×10^{-4} . The vehicle states are approximated using third-order Legendre polynomials and the track is discretised in 10 m segments, balancing computational efficiency and accuracy (32, 33).

Rate limits and regularisation weight factors, listed in Table 8.5, are calibrated to reflect the physical response times of both the driver and vehicle systems. In particular, the aerodynamic actuators are constrained to response times of approximately 0.5 s, consistent with those reported in existing active aerodynamic applications. Regularisation factors are further tuned to suppress high-frequency oscillations in control inputs, promoting smoother actuation and preventing excessive corrections, thereby enhancing the robustness and interpretability of the control strategy results in this study.

8.3 Results and Discussion

This section first evaluates the influence of individual aerodynamic elements on overall aerodynamic performance. Subsequently, it examines lap times, optimal setups, key performance metrics, and control inputs across the vehicle configurations. Finally, optimal tyre characteristics are analysed to identify desired configuration-specific properties.

8.3.1 Aerodynamic Performance

The aerodynamic contributions of individual components are presented in Figure 8.3. The front wing elements increase front axle downforce by up to $\Delta C_L \approx -0.10$, primarily on their respective side of the vehicle, while causing minor rear axle downforce reductions of up to $\Delta C_L \approx 0.02$ and minimal drag penalties, reflecting their high aerodynamic efficiency.

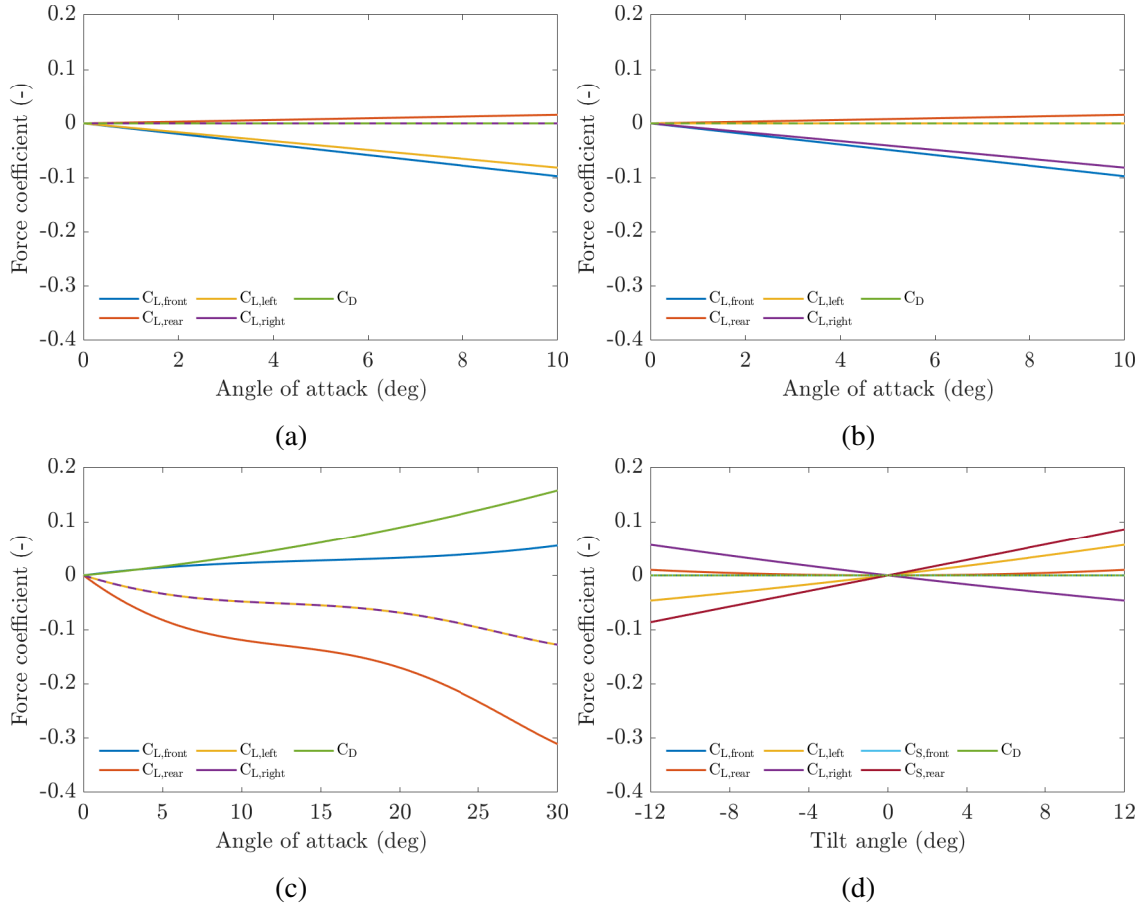


Figure 8.3: Influence of the (a) left front wing element angle of attack, (b) right front wing element angle of attack, (c) rear wing angle of attack, and (d) rear wing tilt angle on the vehicle's aerodynamic performance.

The rear wing has the greatest aerodynamic impact, increasing rear axle downforce by up to $\Delta C_L \approx -0.31$ while slightly reducing front axle downforce by up to $\Delta C_L \approx 0.05$. Its performance shows an initial rise in downforce, a stall plateau between 10° – 15° , and additional gains at higher angles due to upstream pressurisation effects. Drag increases quadratically, peaking at $\Delta C_D \approx 0.16$. Tilting the rear wing redistributes downforce laterally, increasing it by up to $\Delta C_L \approx -0.05$ on the elevated side while reducing it by $\Delta C_L \approx 0.06$ on the opposite side. It also generates a side force of up to $\Delta C_S \approx \pm 0.09$, with negligible effects on drag or front axle downforce.

Altogether, the vehicle achieves a lift coefficient of $C_L \approx -0.29$, a drag coefficient of $C_D \approx 0.35$, and a rear axle downforce distribution of 0.35 at minimised wing angles. Maximising the front wing increases downforce by $\Delta C_L \approx -0.16$ (+56%) and shifts the balance forward to 0.16. Maximising the rear wing enhances downforce by $\Delta C_L \approx -0.26$ (+89%) and shifts the balance rearward to 0.76. At combined maximised wing angles, total downforce increases by $\Delta C_L \approx -0.42$ (+145%) to $C_L \approx -0.71$, while drag rises by $\Delta C_D \approx 0.16$ (+46%) to $C_D \approx 0.50$, with a downforce distribution of 0.54.

8.3.2 Lap Times

Lap time results are summarised in Table 8.6 to quantify performance gains delivered by active aerodynamics and ATD. Aerodynamic configurations C1 to C4 with AWD powertrains are compared against their ATD-enhanced counterparts C1-ATD to C4-ATD. Configuration C1, featuring optimised static front and rear wing settings, serves as the baseline with a lap time of 120.196 s. The front wing operates at its maximum angle due to its high efficiency, while the rear wing is set at 8.12° to balance downforce and drag. Increasing aerodynamic adaptability progressively reduces lap times. Configuration C2, with a static front wing and active rear wing, improves lap time by 0.694 s (0.58%), while configuration C3, employing active control of both wings, achieves a 0.744 s (0.62%) reduction. Configuration C4, with asymmetric aerodynamic control, provides the greatest improvement among the AWD configurations, reducing lap time by 0.915 s (0.76%).

Incorporating ATD yields significant gains, with configuration C1-ATD reducing lap time by 5.638 s (4.80%) to 114.558 s over C1. Notably, it benefits from a significantly higher downforce setup, with a rear wing angle of 28.46° . Active aerodynamics further improve lap times, with configurations C2-ATD, C3-ATD, and C4-ATD achieving reductions of 0.486 s (0.42%), 0.494 s (0.43%), and 0.665 s (0.58%) over C1-ATD, respectively. While the relative gains from active aerodynamics reduce by 20%-30% when paired with ATD, their integration remains valuable for improving performance.

8.3.3 Vehicle Design Parameters

Optimised design parameters in Table 8.6, including brake torque, drive torque, and roll stiffness distributions, illustrate the influence of active aerodynamics and ATD on vehicle setup. For AWD configurations, increasing aerodynamic adaptability shifts these parameters progressively rearward, driven by greater control over rear axle downforce. Enhanced rear grip enables the axle to receive more torque during braking and initial acceleration, while also supporting greater lateral load transfer. Owing to its superior asymmetric aerodynamic control, configuration C4 exhibits the largest rearward shifts of 1.7% in brake balance, 1.7% in torque distribution, and 2.7% in roll stiffness distribution.

Table 8.6: Lap time and optimal design parameter results for all vehicle configurations.

Configuration	Front wing (deg)	Rear wing (deg)	Brake balance γ_b (-)	Torque distribution γ_T (-)	Roll stiffness distribution γ_{k_ϕ} (-)	Lap time (s)	Δ Lap time (s)	Δ Lap time (%)
C1	10.000	8.12	0.677	0.727	0.462	120.196	-	-
C2	10.000	Active	0.667	0.739	0.470	119.502	0.694	0.58
C3	Active	Active	0.666	0.741	0.472	119.452	0.744	0.62
C4	Active asymmetric	Active asymmetric	0.665	0.740	0.475	119.281	0.915	0.76
C1-ATD	10.000	28.46	ATD	ATD	0.448	114.558	5.638	4.80
C2-ATD	10.000	Active	ATD	ATD	0.446	114.072	6.124	5.23
C3-ATD	Active	Active	ATD	ATD	0.452	114.064	6.131	5.24
C4-ATD	Active asymmetric	Active asymmetric	ATD	ATD	0.457	113.893	6.302	5.39

Leveraging the ability to better utilise available tyre grip, the ATD configurations have more forward roll stiffness distributions compared to their AWD counterparts, with configuration C1-ATD exhibiting a 3.2% forward shift compared to C1. However, similar to the AWD configurations, the roll stiffness distribution shifts rearward with increasing aerodynamic adaptability relative to configuration C1-ATD, with C3-ATD and C4-ATD demonstrating shifts of around 1.0% and 2.0%, respectively.

Throughout the remainder of this study, configurations C2 and C2-ATD are excluded from further analysis due to minimal performance differences compared to C3 and C3-ATD. This is attributed to the front wing's high aerodynamic efficiency, limiting the benefits of front wing angle reductions for this specific vehicle design. Configurations C3 and C3-ATD are retained to provide insights into the front wing's performance and control.

8.3.4 Vehicle Performance

Racing lines for configurations C1 and C1-ATD, shown in Figure 8.4, highlight noticeable differences in long bends such as corners 3 and 4, attributed to variations in performance limits and handling responsiveness. Understeer characteristics in Figure 8.5a reveal that configuration C1-ATD exhibits an approximately 30% lower understeer gradient and less severe steering angle gain in the non-linear region, indicating enhanced steering responsiveness and handling predictability. Figures 8.5b and 8.5c demonstrate that for both AWD and ATD powertrains, active aerodynamic configurations exhibit negligible changes in understeer characteristics relative to static aerodynamic setups but show slight increases in maximum lateral acceleration, which will be elaborated upon later.

Velocity variations in Figure 8.6 illustrate the performance improvements achieved by the active aerodynamic configurations. The magnitude and location of these velocity gains depend on the characteristics of their respective static aerodynamic setups. Configurations C3 and C4 primarily enhance downforce compared to the low-downforce, low-drag setup of C1. As a result, significant velocity gains are observed in grip-limited scenarios, such as heavy braking zones (e.g., before corners 1, 4, 9, and 10) and during acceleration out of long bends (e.g., corners 3, 4, 9, and 16).

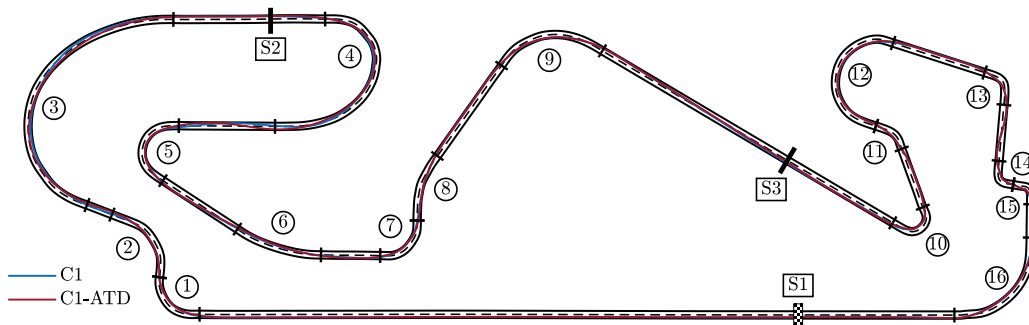


Figure 8.4: Optimal racing lines for configurations C1 and C1-ATD on the Circuit de Barcelona-Catalunya, featuring labelled sectors and corners, including corner start and end point markings.

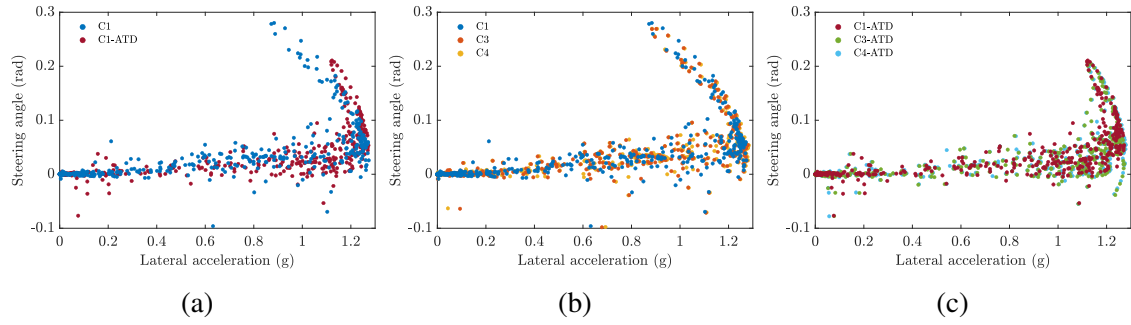


Figure 8.5: Steering angle versus lateral acceleration for (a) configurations C1 and C1-ATD, (b) configurations C1, C3 and C4, and (c) for configurations C1-ATD, C3-ATD and C4-ATD.

Conversely, configurations C3-ATD and C4-ATD primarily improve performance by reducing drag relative to the high-downforce, high-drag setup of C1-ATD. As a result, they achieve substantial velocity gains during power-limited phases, such as along the straights following turns 9 and 16. Although less pronounced, both configurations also achieve modest velocity improvements through most medium- to high-speed corners, attributed to slight increases in downforce, and for C4-ATD specifically, to a more effective distribution of downforce.

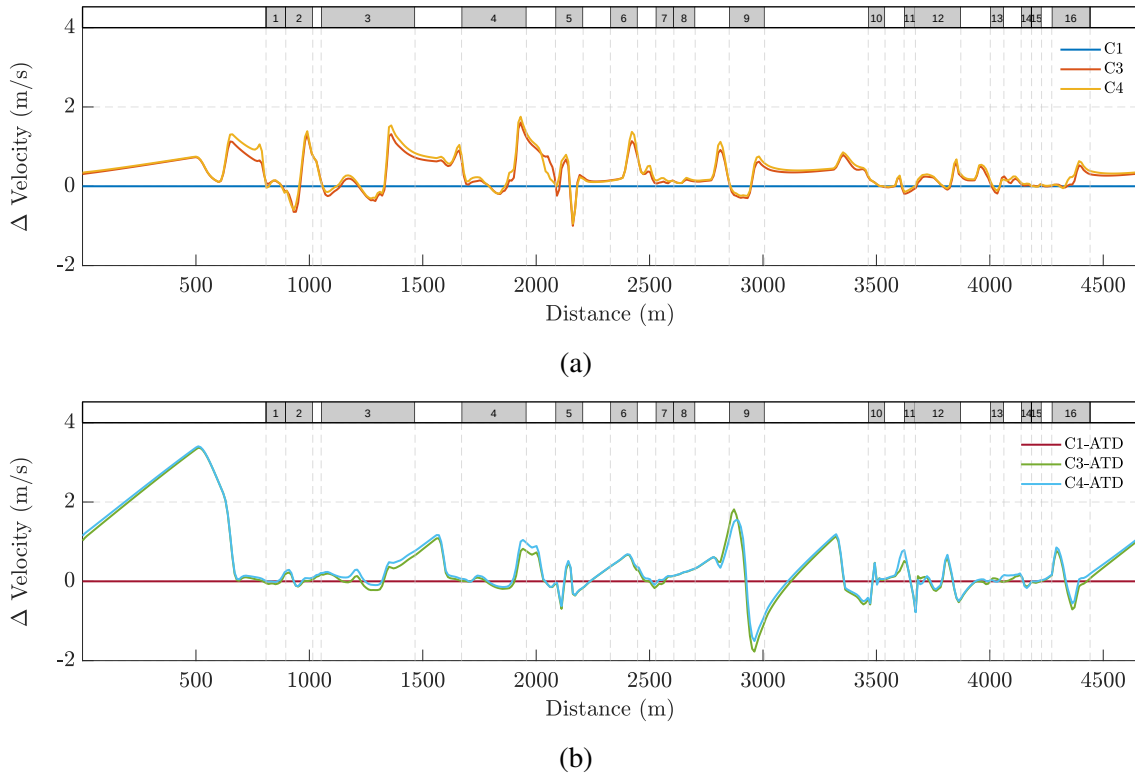


Figure 8.6: Delta velocity as a function of distance for (a) configurations C3 and C4 relative to C1, and (b) configurations C3-ATD and C4-ATD relative to C1-ATD.

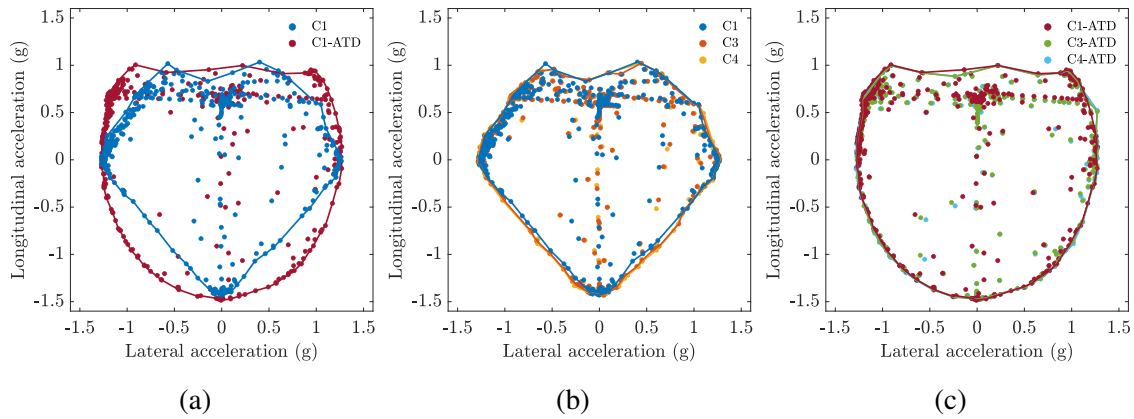


Figure 8.7: GG-diagrams, presenting longitudinal versus lateral acceleration, for (a) configurations C1 and C1-ATD, (b) configurations C1, C3 and C4, and (c) configurations C1-ATD, C3-ATD and C4-ATD, with highlighted borders indicating the performance limits of each configuration.

GG diagrams presented in Figure 8.7 demonstrate the expanded performance envelope for each configuration. Configuration C1-ATD achieves a 30% larger GG area than C1, with 4% higher pure lateral acceleration, 20% higher combined forward and lateral acceleration, and 30% higher combined deceleration and lateral acceleration.

Among AWD configurations, C3 expands the GG area by over 2% compared to C1, with improvements of 0.5% in pure lateral acceleration, 1% in combined forward and lateral acceleration, and over 4% in combined deceleration and lateral acceleration. The asymmetric aerodynamic control of C4 expands the GG area by nearly 4%, achieving 1% higher pure lateral acceleration, 3% higher combined forward and lateral acceleration, and over 5% higher combined deceleration and lateral acceleration. For ATD configurations, the high-downforce setup of C1-ATD limits further GG area expansion by C3-ATD, while C4-ATD achieves a modest increase of less than 0.5%, with minor gains of 0.5%-1% in pure lateral and combined accelerations through asymmetric aerodynamic loading.

8.3.5 Control Inputs

The underlying mechanisms driving the performance gains are investigated through analyses of steering, torque, and aerodynamic control inputs. Steering traces in Figure 8.8a show distinct differences between AWD and ATD configurations. ATD setups exhibit greater responsiveness, requiring up to 30% less steering input throughout multiple corners (e.g., 2, 3, 4, 5, and 12). While not directly evident from the previously discussed understeer characteristics, active aerodynamic configurations further reduce steering input compared to their static counterparts, particularly in medium- to high-speed corners (e.g., 3, 4, 6, 8, and 16), enhancing handling responsiveness.

Torque inputs presented in Figure 8.8b reveal notable differences between AWD and ATD configurations, where C1-ATD demonstrates later and more intense braking in heavy-braking zones (e.g., before corners 1, 4, and 10) and maintains more consistent drive torque in long turns (e.g., 3 and 4) compared to C1.

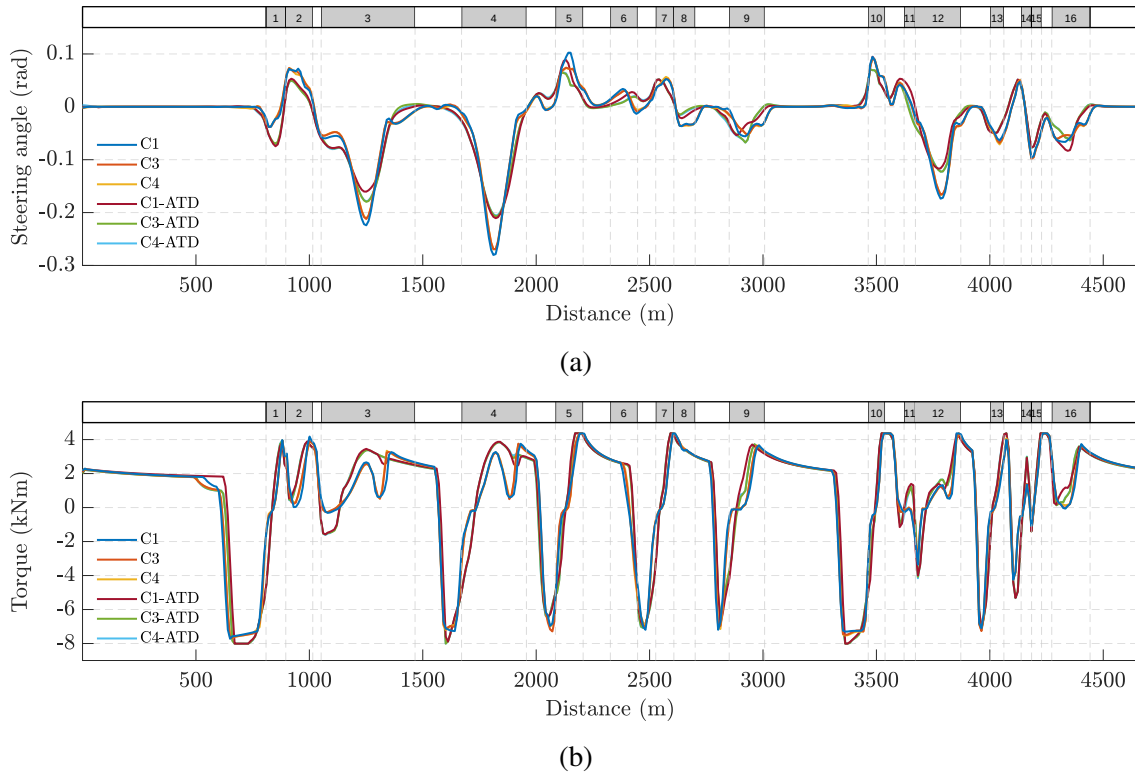


Figure 8.8: (a) Steering angle and (b) combined total wheel torque as a function of distance for all vehicle configurations.

For AWD configurations, C3 and C4 enable earlier torque application and greater brake torque due to enhanced rear axle grip. In contrast, configurations C3-ATD and C4-ATD show diminishing returns in these areas due to the already high-downforce setup of C1-ATD, but still achieve slightly higher peak brake torque and more consistent torque during acceleration.

Figure 8.9 illustrates individual wheel torque allocation for the ATD configurations, with C4-ATD serving as a representative example. The torque allocation patterns are largely consistent across ATD setups, with minor variations arising from aerodynamic influences. Key trends are observed across different driving scenarios and detailed below.

During straight-line and trail braking (e.g., before corners 1, 4, 9, and 10), brake torque is initially evenly distributed across the wheels on each axle, with approximately 64% allocated to the heavily loaded front axle at peak forward load transfer. As lateral load transfer increases during trail braking, brake torque allocation shifts to the heavily loaded outside tyres, while the underloaded inside tyres receive less torque. Brake torque across all tyres eventually diminishes with increasing steering angle to accommodate larger lateral forces.

In rapid successive manoeuvres (e.g., corners 5–7), substantial combined longitudinal and lateral load transfer occurs immediately. The most heavily loaded front outside tyre therefore directly receives up to 35% of the total brake torque. Approximately 25% is distributed to the front inside and rear outside tyres each, while the least-loaded rear inside tyre receives just over 10%.

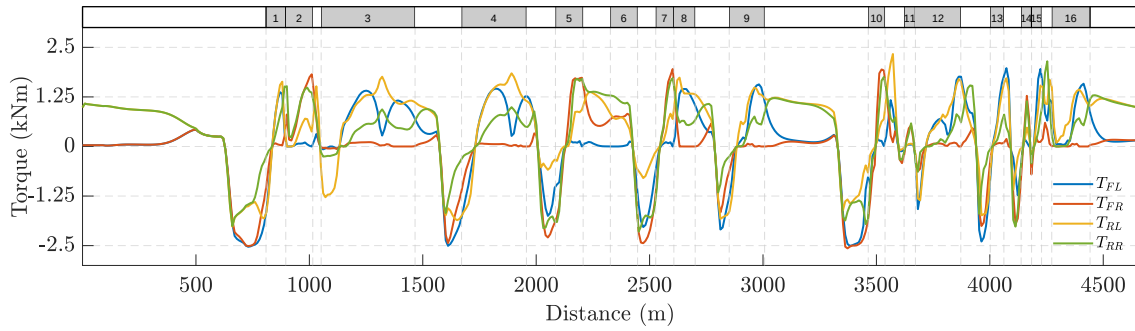


Figure 8.9: Individual wheel torque as a function of distance for configuration C4-ATD.

During corner exits, particularly in medium- to high-speed corners in the first two sectors, drive torque shifts dynamically as load transfers rearward. Initially, the heavily loaded outside tyres each receive nearly 40% of the total drive torque, while the rear inside tyre is allocated around 20% and the least-loaded front inside tyre less than 2%.

As the vehicle exits the corner and rearward load transfer increases, torque on the front outside tyre decreases while torque on the rear inside tyre increases. By the time straight-line acceleration is achieved, the torque distribution becomes predominantly rearward due to the increased loading on the rear axle.

Figure 8.10 compares the control strategies for the active aerodynamic systems of the AWD and ATD configurations. The rear wing control for configurations C3 and C3-ATD, shown in Figure 8.10a, is closely tied to load transfer during torque application. During initial acceleration, the rear wing angle is maximised to increase rear axle downforce. As peak torque and rearward load transfer are reached, the rear wing angle reduces to restore balance and minimise drag during the subsequent power-limited phase. During braking, the rear wing angle is maximised to enhance deceleration by increasing downforce on the underloaded rear axle and generating drag.

Key differences between the configurations include sensitivity and frequency of rear wing adjustments. Configuration C3 varies rear wing angles significantly between high- and low-speed corners to balance aerodynamic loads across the grip-limited tyres on each axle, whereas configuration C3-ATD maintains consistently high rear wing angles throughout corners to maximise downforce, benefiting from the ability to allocate torque to individual wheels. Similarly, C3 exhibits more frequent front wing adjustments, especially during braking. Configuration C3-ATD primarily operates at maximum front wing angles except during power-limited phases to reduce resistive forces. Both configurations rapidly reduce front wing angles during initial acceleration to maximise rear axle grip and torque allocation to the more heavily loaded rear wheels.

Figures 8.10b and 8.10c compare aerodynamic control strategies for configurations C4 and C4-ATD, mirroring trends observed for C3 and C3-ATD. Configuration C4 frequently adjusts the rear wing's angle of attack during cornering to balance aerodynamic loads, while C4-ATD maintains high rear wing angles to maximise tyre grip with minimal rebalancing.

The rear wing's tilt angle control for configurations C4 and C4-ATD is nearly identical. During cornering, the rear wing tilts away from the corner centre to prioritise downforce on the underloaded inside tyres. When inclined, the rear wing also generates a centripetal force that primarily acts at the rear of the vehicle due to its mounting position, thereby improving cornering performance.

The front wing control for configuration C4 features more frequent asymmetric adjustments, particularly during braking and cornering, to optimise aerodynamic load distribution. In braking zones, the outside front wing element is reduced to prevent overloading of the front outside tyre and further unloading of the rear axle.

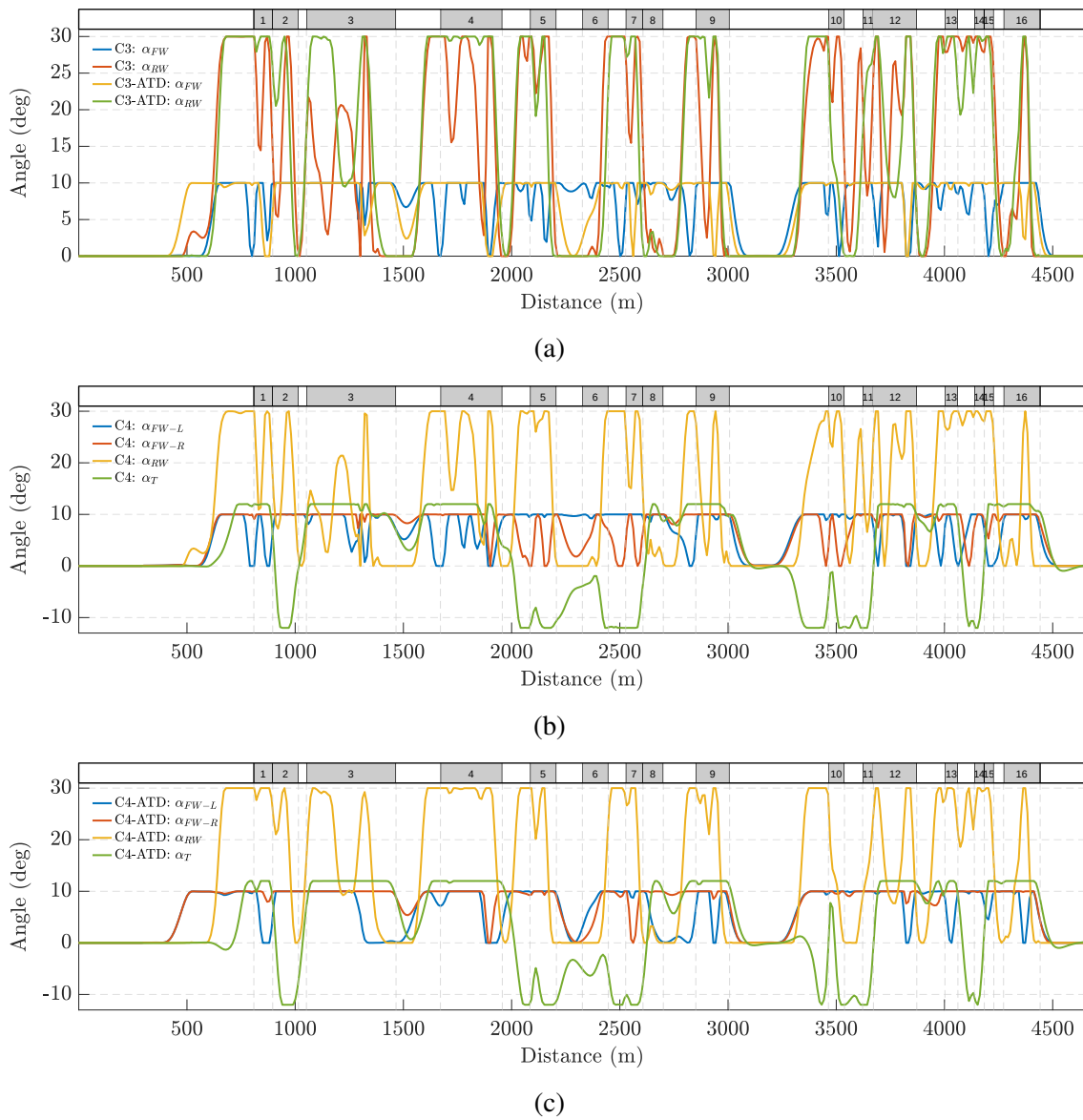


Figure 8.10: Control inputs of the active aerodynamic systems as a function of distance for configurations (a) C3 and C3-ATD, (b) C4, and (c) C4-ATD.

In contrast, C4-ATD exhibits more restrained adjustments, with minor reductions in the outside front wing angle before specific corners such as 4 and 9. During corner exits, both configurations reduce the outside front wing element to prioritise downforce on the rear axle and underloaded inside tyres.

8.3.6 Tyre Characteristics

The influence of tyre characteristics on vehicle performance is explored by optimising the nominal tyre load F_{z0} (N) for each configuration without altering the peak longitudinal and lateral forces. As illustrated in Figure 8.11, lower nominal loads are modelled to increase cornering stiffness, reduce the vertical load at which peak forces are generated, and steepen force reduction in the saturated region. These tyre characteristics, though theoretical in nature, enable an exploratory analysis of tyre properties most effective in maximising the performance gains delivered by active aerodynamics and ATD.

The original tyre was selected to reach peak force near the maximum vertical load encountered during cornering, as recommended by Seward (41). As shown in Table 8.7, optimised tyres with lower nominal loads reduce lap times by over 2% for AWD and by about 1.5% for ATD configurations. The optimal nominal loads significantly differ between powertrains, with AWD setups benefiting from $0.68F_{z0}$ and ATD from $0.77F_{z0}$.

Variations in optimised nominal tyre loads across aerodynamic setups are within 0.6% for both powertrain configurations, indicating reduced sensitivity to aerodynamic setup. However, the influence of active aerodynamics on lap time gains differs notably between the AWD and ATD setups. With optimised tyres, the lap time improvements of configurations C3 and C4 over the C1 are approximately 8% lower than with standard tyres. In contrast, for the ATD configurations, the lap time gains of C3-ATD and C4-ATD over C1-ATD increase by over 15% when using optimised tyres compared to the standard tyres.

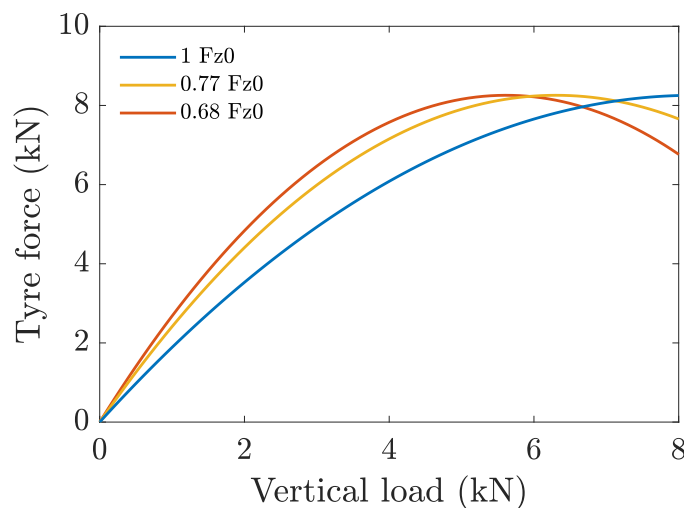


Figure 8.11: Tyre force, representing longitudinal and lateral components, as a function of vertical tyre load for the original nominal load F_{z0} , and optimised loads of $0.68 F_{z0}$ (AWD) and $0.77 F_{z0}$ (ATD).

Table 8.7: Lap time results for each vehicle configuration with original and optimised tyres. Improvements are expressed relative to the respective static aerodynamic configuration of the same powertrain and tyre setup.

Configuration	Original tyre			Optimised tyre			Nominal tyre load
	Lap time (s)	Δ Lap time (s)	Δ Lap time (%)	Lap time (s)	Δ Lap time (s)	Δ Lap time (%)	
C1	120.196	-	-	117.447	-	-	$0.684 F_{z0}$
C3	119.452	0.744	0.62	116.780	0.667	0.57	$0.684 F_{z0}$
C4	119.281	0.915	0.76	116.620	0.827	0.70	$0.683 F_{z0}$
C1-ATD	114.558	-	-	112.934	-	-	$0.770 F_{z0}$
C3-ATD	114.064	0.494	0.43	112.307	0.627	0.56	$0.767 F_{z0}$
C4-ATD	113.893	0.665	0.58	112.177	0.757	0.67	$0.765 F_{z0}$

The GG diagrams in Figure 8.12 show that the performance gains achieved with optimised tyres stem primarily from increased combined longitudinal and lateral accelerations, particularly during trail braking and corner exits. The GG areas expand by over 15% for AWD and 7% for ATD configurations, aligning with the greater absolute lap time improvements observed for AWD setups.

Figure 8.13 shows tyre forces during high-demand trail braking and corner exit scenarios, illustrating how tyre characteristics affect performance across configurations relative to the original tyre. For AWD setups, torque application to an axle is limited by the tyre with the least grip on that axle. In contrast, ATD configurations allocate torque to each individual wheel, fully utilising the available grip of each tyre. The original tyre (blue) demonstrates a trade-off in tyre selection, with its peak force occurring near the maximum vertical load experienced. While this favours the most heavily loaded tyre, it reduces grip for less loaded tyres, limiting combined force generation. This trade-off is particularly impactful for AWD systems due to their axle-based torque allocation.

Optimised tyres for AWD configurations (orange) feature lower nominal vertical loads, peaking near the load experienced by the second most heavily loaded tyre. These characteristics enhance grip on the underloaded inside tyres through increased cornering stiffness, allowing greater torque allocation to each axle. While this compromises lateral force on the most heavily loaded tyre, which now operates in the saturated region, overall tyre force utilisation is improved, yielding notable performance gains.

For ATD configurations, optimised tyres (yellow) exhibit slightly higher nominal vertical loads than their AWD counterparts, peaking between the loads experienced by the two most heavily loaded outside tyres. Since ATD systems reduce the dependency on enhancing grip for the most underloaded tyres, these optimised characteristics provide a more balanced distribution of grip across all tyres. The tyres avoid excessive force reduction on heavily loaded tyres while maintaining improved grip for less loaded tyres. This balance enhances combined tyre force utilisation, delivering the highest overall performance.

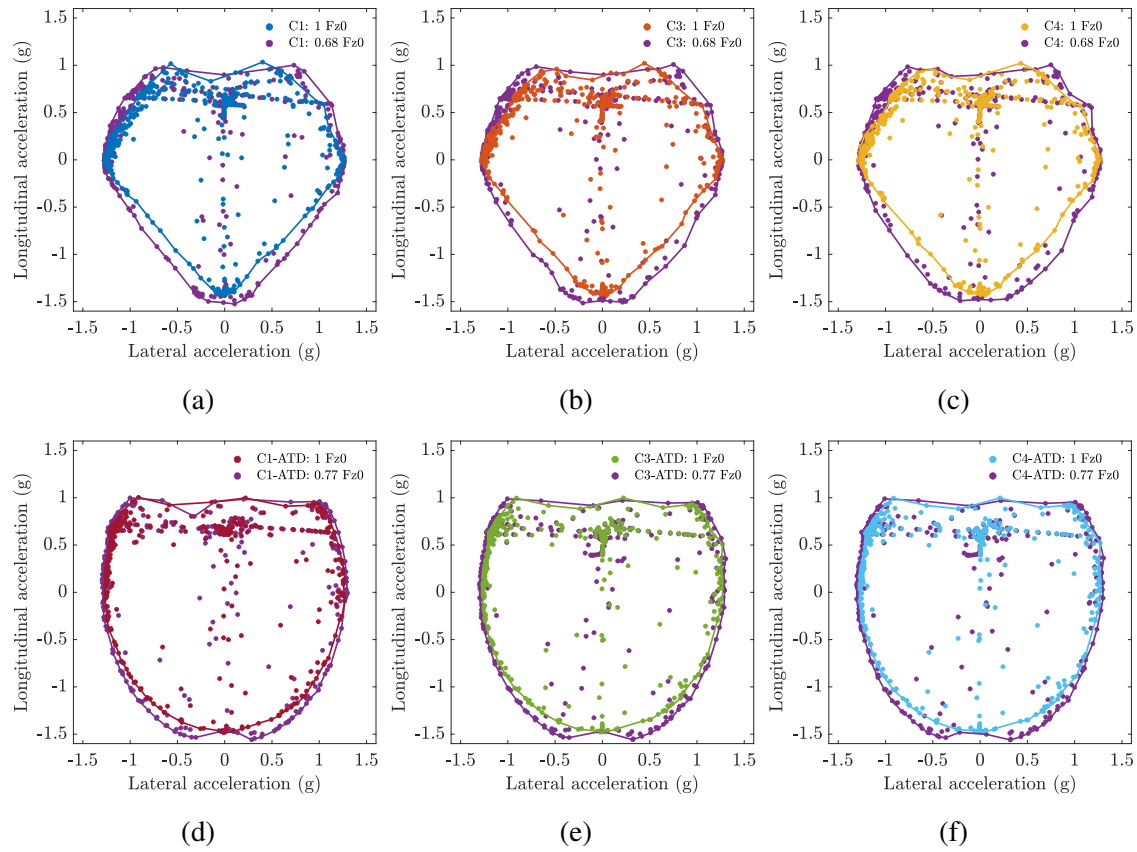


Figure 8.12: GG-diagrams, presenting longitudinal versus lateral acceleration, for configurations (a) C1, (b) C3, (c) C4, (d) C1-ATD, (e) C3-ATD, and (f) C4-ATD with the original nominal tyre load F_{z0} and their respective optimised nominal tyre load.

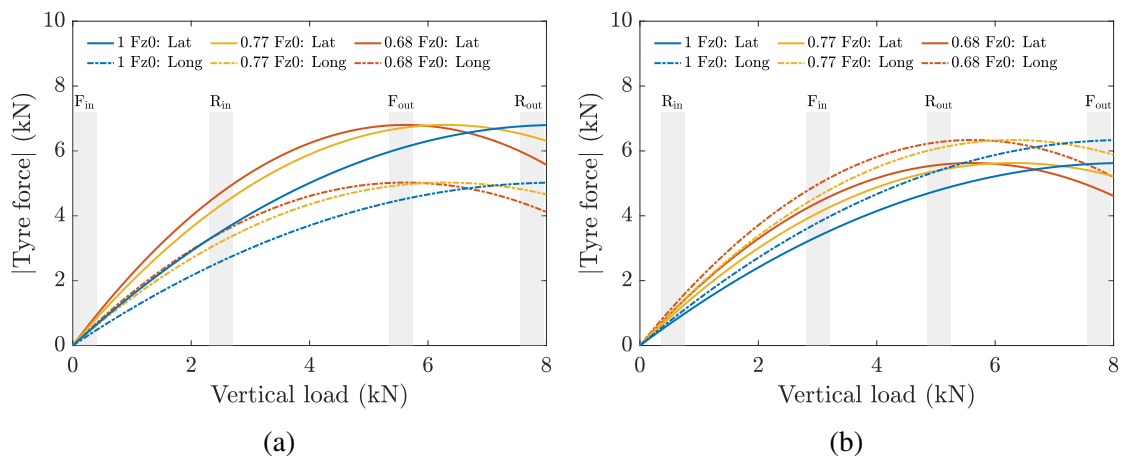


Figure 8.13: Absolute tyre forces during (a) corner exit acceleration at a slip angle of 4° and a slip ratio of 0.05, and during (b) trail braking at a slip angle of 3° and a slip ratio of -0.07. Grey shaded bars indicate typical vertical tyre loads for the front inside/outside F_{in}/F_{out} (N) and rear inside/outside R_{in}/R_{out} (N) tyres for each scenario.

8.4 Conclusion

This study explored the performance potential of integrating active aerodynamic systems with ATD through comprehensive minimum lap time simulations. By optimising the vehicle's trajectory, control inputs, and selected design parameters, the analysis evaluated various active aerodynamic configurations paired with AWD and ATD powertrains. Performance was assessed on the Circuit de Barcelona-Catalunya, providing valuable insights into performance gains, control strategies, vehicle setups, and tyre characteristics.

The results showed that ATD systems achieved lap time reductions of approximately 4.8% compared to AWD configurations, with additional improvements of up to 5.4% when combined with active aerodynamic systems. The AWD configuration favoured a lower downforce static aerodynamic setup to balance drag, while the ATD configuration benefited from a higher downforce setup to maximise overall tyre grip.

Active aerodynamics further enhanced performance relative to static setups, reducing lap times by up to 0.76% for AWD and 0.58% for ATD powertrains. In both cases, the configuration with 2D active aerodynamic systems capable of asymmetric actuation delivered the greatest gains. Increasing aerodynamic adaptability shifted the optimal brake balance, torque distribution, and roll stiffness distribution rearward due to the improved control over rear axle downforce. ATD configurations benefited from more forward roll stiffness distributions, leveraging their ability to utilise front axle grip more effectively.

ATD configurations demonstrated superior performance by dynamically allocating torque to the heavily loaded tyres, utilising tyre grip more effectively compared to AWD configurations. This capability resulted in significantly higher combined accelerations during braking and cornering scenarios. The performance gains from active aerodynamics were influenced by the characteristics of the respective static setups. For AWD configurations, braking and cornering performance improved primarily through increased downforce, while for ATD configurations, straight-line acceleration and top speed benefited from reduced drag. Asymmetric aerodynamic control further enhanced performance for both powertrains by generating lateral aerodynamic forces and prioritising downforce on underloaded tyres, increasing overall tyre grip.

Control inputs for both symmetric and asymmetric active aerodynamic systems showed more frequent adjustments for AWD configurations compared to ATD configurations, as AWD configurations required continuous balancing of loads across grip-limited tyres on each axle. In contrast, ATD configurations maintained more consistent high-downforce settings, leveraging dynamic torque allocation to optimise tyre grip utilisation.

Tyre characteristics were also crucial in maximising performance. AWD configurations benefited from tyres with lower nominal vertical loads and higher cornering stiffness, prioritising grip on the most underloaded tyres to enable greater torque application to each axle, while compromising lateral grip on the heavily loaded tyres. In contrast, torque application for ATD configurations is not limited by the least available tyre grip on an axle, which therefore performed better with tyres featuring slightly higher nominal vertical loads and lower cornering stiffness. These more balanced properties minimised grip losses on heavily loaded tyres while maintaining adequate grip for less loaded tyres.

Overall, this research demonstrated the substantial performance benefits of combining active aerodynamic systems with active torque distribution. The findings quantified the extent of performance improvements and uncovered the underlying mechanisms driving these gains. Additionally, the study introduced an effective optimisation simulation framework capable of evaluating vehicle performance and guiding decisions on vehicle setup and tyre selection. This framework offers significant potential for broader applications in the development and optimisation of other active vehicle systems. Collectively, this work provides a strong foundation for advancing active aerodynamic systems for next-generation automotive and motorsport vehicles.

Data Availability Statement

The minimum lap time simulation framework and the corresponding generated performance data supporting this study are openly available from Cranfield Online Research Data at: <https://dspace.lib.cranfield.ac.uk/handle/1826/23604>. (42)

References

1. Ahangarnejad AH, Melzi S. Numerical analysis of the influence of an actively controlled spoiler on the handling of a sports car. *Journal of Vibration and Control*. 2018;24(22):5437-48. Available from: <https://doi.org/10.1177/1077546318754683>.
2. Mashrouteh S, Khajepour A, Kasaiezadeh A, Esmailzadeh E, Chen SK, Litkouhi B. Multi-actuation controller for performance vehicles: optimal torque allocation and active aerodynamic. *IEEE Transactions on Vehicular Technology*. 2022;71(3):2721-33. Available from: <https://doi.org/10.1109/TVT.2022.3144133>.
3. Broniszewski J, Piechna J. A fully coupled analysis of unsteady aerodynamics impact on vehicle dynamics during braking. *Engineering Applications of Computational Fluid Mechanics*. 2019;13(1):623-41. Available from: <https://doi.org/10.1080/19942060.2019.1616326>.
4. Broniszewski J, Piechna JR. Fluid-structure interaction analysis of a competitive car during brake-in-turn manoeuvre. *Energies*. 2022;15(8):2917. Available from: <https://doi.org/10.3390/en15082917>.
5. Kurec K, Remer M, Broniszewski J, Bibik P, Tudruj S, Piechna J. Advanced modeling and simulation of vehicle active aerodynamic safety. *Journal of Advanced Transportation*. 2019;(7308590). Available from: <https://doi.org/10.1155/2019/7308590>.
6. Ferraris A, De Cupis D, de Carvalho Pinheiro H, Messana A, Sisca L, Giancarlo Airale A, et al. Integrated design and control of active aerodynamic features for high performance electric vehicles. *SAE Technical Paper 2020-36-0079*. 2021. Available from: <https://doi.org/10.4271/2020-36-0079>.

7. Ayyagari DT, He Y. Aerodynamic analysis of an active rear split spoiler for improving lateral stability of high-speed vehicles. *International Journal of Vehicle Systems Modelling and Testing*. 2017;12(3-4):217-39. Available from: <https://doi.org/10.1504/IJVSMT.2017.089978>.
8. Sikder T, Kapoor S, He Y. Optimizing dynamic performance of high-speed road vehicles using aerodynamic aids. *Proceedings of the ASME 2016 International Mechanical Engineering Congress and Exposition*. vol. Volume 7: Fluids Engineering. Phoenix, USA: The American Society of Mechanical Engineers; 2016. Available from: <https://doi.org/10.1115/IMECE2016-65414>.
9. Zavala D, Vicente W, Esquivel R, Salinas-Vazquez M. Numerical simulation of the effect of a centripetal wing spoiler on the flow aerodynamics of a sports car. *SAE Technical Paper 2023-01-5032*. 2023. Available from: <https://doi.org/10.4271/2023-01-5032>.
10. Hammad M, Qureshi K, He Y. Safety and lateral dynamics improvement of a race car using active rear wing control. *SAE Technical Paper 2019-01-0643*. 2019. Available from: <https://doi.org/10.4271/2019-01-0643>.
11. van Aaken JPT. Influence of asymmetric aerodynamics on lateral acceleration on a Formula Student car. MSc Thesis. Eindhoven University of Technology; 2023. Available from: https://pure.tue.nl/ws/portalfiles/portal/312301921/BEP_report_JobvanAaken.pdf.
12. Cai J, Kapoor S, Sikder T, He Y. Effects of active aerodynamic wings on handling performance of high-speed vehicles. *SAE Technical Paper 2017-01-1592*. 2017. Available from: <https://doi.org/10.4271/2017-01-1592>.
13. Diba F, Barari A, Esmailzadeh E. Handling and safety enhancement of race cars using active aerodynamic systems. *Vehicle System Dynamics*. 2014;52(9):1171-90. Available from: <https://doi.org/10.1080/00423114.2014.930158>.
14. Piechna JR, Kurec K, Broniszewski J, Remer M, Piechna A, Kamieniecki K, et al. Influence of the car movable aerodynamic elements on fast road car cornering. *Energies*. 2022;15(3):689. Available from: <https://doi.org/10.3390/en15030689>.
15. Rijns S, Teschner TR, Blackburn K, Brighton J. Performance analyses of active aerodynamic load balancing designs on high-performance vehicles in cornering conditions. *Physics of Fluids*. 2024;36(8):085199. Available from: <https://doi.org/10.1063/5.0222780>.
16. Smith EN, Velenis E, Tavernini D, Cao D. Effect of handling characteristics on minimum time cornering with torque vectoring. *Vehicle System Dynamics*. 2018;56(2):221-48. Available from: <https://doi.org/10.1080/00423114.2017.1371771>.
17. Antunes J, Antunes A, Outeiro P, Carneira C, Oliveira P. Testing of a torque vectoring controller for a Formula Student prototype. *Robotics and Autonomous Systems*. 2019;113:56-62. Available from: <https://doi.org/10.1016/j.robot.2018.12.010>.

18. Chatzikomis C, Sorniotti A, Gruber P, Bastin M, Shah RM, Orlov Y. Torque-vectoring control for an autonomous and driverless electric racing vehicle with multiple motors. *SAE International Journal of Vehicle Dynamics, Stability, and NVH*. 2017;1(2):338-51. Available from: <https://doi.org/10.4271/2017-01-1597>.
19. Ghosh J, Tonoli A, Amati N. A torque vectoring strategy for improving the performance of a rear wheel drive electric vehicle. *Proceedings of the 2015 IEEE Vehicle Power and Propulsion Conference (VPPC)*. Montreal, Canada: IEEE; 2015. p. 1-6. Available from: <https://doi.org/10.1109/VPPC.2015.7352887>.
20. Broere S, Van Kampen J, Salazar M. Minimum-lap-time control strategies for all-wheel drive electric race cars via convex optimization. *Proceedings of the 2022 European Control Conference (ECC)*. London, United Kingdom: IEEE; 2022. p. 1204-11. Available from: <https://doi.org/10.23919/ECC55457.2022.9838115>.
21. Wang J, Gao S, Qiang Y, Xu M, Guan C, Zhou Z. Structural topology and dynamic response analysis of an electric torque vectoring drive-axle for electric vehicles. *Automotive Innovation*. 2022;5(2):164-79. Available from: <https://doi.org/10.1007/s42154-022-00178-0>.
22. de Buck P, Martins JRRA. Minimum lap time trajectory optimisation of performance vehicles with four-wheel drive and active aerodynamic control. *Vehicle System Dynamics*. 2023;61(8):2103-19. Available from: <https://doi.org/10.1080/00423114.2022.2101930>.
23. Casanova D. On minimum time vehicle manoeuvring: the theoretical optimal lap. PhD Thesis. Cranfield University; 2000. Available from: <http://etheses.dur.ac.uk/2671/>.
24. Massaro M, Limebeer DJN. Minimum-lap-time optimisation and simulation. *Vehicle System Dynamics*. 2021;59(7):1069-113. Available from: <https://doi.org/10.1080/00423114.2021.1910718>.
25. Veneri M, Massaro M. A free-trajectory quasi-steady-state optimal-control method for minimum lap-time of race vehicles. *Vehicle System Dynamics*. 2020;58(6):933-54. Available from: <https://doi.org/10.1080/00423114.2019.1608364>.
26. Lovato S, Massaro M. Three-dimensional fixed-trajectory approaches to the minimum-lap time of road vehicles. *Vehicle System Dynamics*. 2022;60(11):3650-67. Available from: <https://doi.org/10.1080/00423114.2021.1969024>.
27. Lovato S, Massaro M. A three-dimensional free-trajectory quasi-steady-state optimal-control method for minimum-lap-time of race vehicles. *Vehicle System Dynamics*. 2022;60(5):1512-30. Available from: <https://doi.org/10.1080/00423114.2021.1878242>.
28. Limebeer DJN, Bastin M, Warren E, Fensham HG. Optimal control of a NASCAR – specification race car. *Vehicle System Dynamics*. 2023;61(5):1210-35. Available from: <https://doi.org/10.1080/00423114.2022.2067573>.

29. Christ F, Wischniewski A, Heilmeier A, Lohmann B. Time-optimal trajectory planning for a race car considering variable tyre-road friction coefficients. *Vehicle System Dynamics*. 2021;59(4):588-612. Available from: <https://doi.org/10.1080/00423114.2019.1704804>.
30. Perantoni G, Limebeer DJN. Optimal control for a Formula One car with variable parameters. *Vehicle System Dynamics*. 2014;52(5):653-78. Available from: <https://doi.org/10.1080/00423114.2014.889315>.
31. Imani Masouleh M, Limebeer DJN. Optimizing the aero-suspension interactions in a Formula One car. *IEEE Transactions on Control Systems Technology*. 2016;24(3):912-27. Available from: <https://doi.org/10.1109/TCST.2015.2475396>.
32. Jiménez Elbal A, Zarzuelo Conde A, Siampis E. Simultaneous optimisation of vehicle design and control for improving vehicle performance and energy efficiency using an open source minimum lap time simulation framework. *World Electric Vehicle Journal*. 2024;15(8):366. Available from: <https://doi.org/10.3390/wevj15080366>.
33. Rijns S, Teschner TR, Blackburn K, Siampis E, Brighton J. Optimising vehicle performance with advanced active aerodynamic systems. *Vehicle System Dynamics*. 2025:1-29. Available from: <https://doi.org/10.1080/00423114.2025.2505619>.
34. Soares RF, Knowles A, Olives SG, Garry K, Holt JC. On the aerodynamics of an enclosed-wheel racing car: an assessment and proposal of add-on devices for a fourth, high-performance configuration of the DrivAer model. *SAE Technical Paper 2018-01-0725*. 2018. Available from: <https://doi.org/10.4271/2018-01-0725>.
35. Heft A, Indinger T, Adams N. Introduction of a new realistic generic car model for aerodynamic investigations. *SAE Technical Paper 2012-01-0168*. 2012. Available from: <https://doi.org/10.4271/2012-01-0168>.
36. Dixon JC. *Tires, suspension and handling*, second edition. 2nd edn. Warrendale, USA: SAE International; 1996.
37. Pacejka HB. *Tire and vehicle dynamics*. 3rd edn. Oxford, UK: Butterworth-Heinemann; 2012. Available from: <https://doi.org/10.1016/B978-0-08-097016-5.00001-2>.
38. Avon Tyres. Technical data resources; 2024. Available from: <https://www.avontyres.com/en-gb/tyre-care/motorsport-technical-data/technical-data-resources/>. (Accessed: 12 April 2024).
39. Rijns S, Teschner TR, Blackburn K, Brighton J. Integrated numerical and experimental workflow for high performance vehicle aerodynamics. *SAE Technical Paper 2024-01-5016*. 2024. Available from: <https://doi.org/10.4271/2024-01-5016>.
40. Helix. SPM242-176 electric motor unit specification sheet; 2024. Available from: <https://www.ehelix.com/products/spm242-electric-motor/>. (Accessed: 01 September 2024).

41. Seward D. Race car design. Lavenham, UK: Palgrave; 2014.
42. Rijns S, Siampis E, Teschner TR, Blackburn K, Brighton J. Minimum lap time simulation framework: active aerodynamics and torque distribution. Cranfield Online Research Data (CORD); 2025. Available from: <https://dspace.lib.cranfield.ac.uk/handle/1826/23604>.

Chapter 9

General Discussion

This chapter provides a comprehensive reflection on the research, synthesising the scope, methodologies, and key findings of each research phase defined in Section 1.3. By comparing studies and integrating results, it identifies overarching trends, interdependencies, and complementary insights while addressing any discrepancies. Subsequently, the chapter discusses the research contributions and their real-world implications. Finally, it outlines opportunities for future research, providing direction for further advancements in high-performance vehicle aerodynamics and active aerodynamic systems.

9.1 Synthesis of Research Phases

9.1.1 Phase 1: Development of Methodological Foundation

The first research phase aimed to establish a rigorous methodological foundation suited for the assessment of the aerodynamic complexities of high-performance vehicles. Using the *DrivAer hp-F* vehicle model, it focused on developing a cost-effective Computational Fluid Dynamics (CFD) workflow that balances predictive accuracy and computational efficiency, ensuring its applicability for the extensive analyses of operating conditions and active aerodynamic systems in later phases. Additionally, it sought to establish a structured wind tunnel correlation process for consistent data interpretation and validation to support the integration of experimental and numerical analyses throughout the research.

9.1.1.1 Approach

To achieve these objectives, a meshing strategy that balances resolution and computational expense was developed through a grid independence study in Chapter 2. To ensure predictive accuracy while maintaining practical computation times, multiple Reynolds-Averaged Navier–Stokes (RANS) turbulence models were evaluated against wind tunnel measurements and high-fidelity Delayed Detached Eddy Simulation (DDES) results across a range of ride heights. Additionally, a numerical approach replicating wind tunnel conditions was deployed to provide deeper insights into wind tunnel-induced effects and improve the interpretation of discrepancies between CFD and experimental data.

9.1.1.2 Key Findings

A key outcome of this research phase was the identification of the $k-\omega$ SST model as the most consistently accurate among the considered RANS approaches. It demonstrated high predictive accuracy, reproducing experimental downforce and drag trends with predictions within 2.5% of high-fidelity DDES results, while operating approximately 75 times faster using only half the computational resources. Beyond force trends, it generally exhibited the highest agreement with DDES results in capturing flow field characteristics, reinforcing its suitability for computationally efficient aerodynamic analyses.

However, Chapter 3 demonstrated the model's limitations at critically low ride heights, where highly unsteady flow behaviour dominated aerodynamic performance. In these conditions, the $k-\omega$ SST model failed to capture vortex instability and transitional flow separation observed in DDES results, leading to significant deviations from experimental force trends. In contrast, DDES simulations accurately reproduced experimental trends across the full ride height range, showing that downforce initially increased with decreasing ride height due to underbody flow acceleration and diffuser vortex strengthening, before abruptly declining as the vortices destabilised and severe flow separation occurred.

Insights from the numerical correlation process revealed that wind tunnel blockage induced a dynamic pressure rise, which, for the considered case, increased downforce and drag by approximately 10% and 15%, respectively, compared to open-road conditions. While conventional blockage correction methods provided reasonable estimates of this pressure rise, they did not account for its unequal influence on downforce and drag. Additionally, interference from the overhead measurement system reduced downforce and drag by nearly 5% and 10%, respectively, further underscoring the importance of considering facility-induced effects.

9.1.1.3 Comparative Remarks

The numerical correlation process developed in this phase was later applied in Chapters 4 and 5 to systematically correlate CFD simulations with wind tunnel experiments, supporting data interpretation and validation practices. The current section seeks to discuss the predictive accuracy of the CFD methodology along with sources of discrepancies between CFD and experiments across the chapters, reinforcing confidence in the established methodology, clarifying its limitations, and supporting correlation processes.

Validation results from Chapter 2, Chapter 3, and Chapter 5 demonstrated consistent predictive trends for the $k-\omega$ SST model under moving ground conditions. In relatively steady flow scenarios, the model accurately reproduced experimental force trends with a near-constant 10% downforce overprediction, coupled with a slight drag underprediction across the validation cases. However, as aerodynamic performance became increasingly dominated by highly unsteady flow behaviour, such as at critically low ride heights in Chapter 3 or near the rear wing's stall angle in Chapter 5, the model transitioned to slight downforce underprediction. This shift was likely due to its tendency to predict abrupt flow transitions, failing to capture transitional flow separation observed in the DDES simulations, as discussed in Chapter 3.

In contrast, validation against wind tunnel measurements without a moving ground in Chapter 4 revealed significantly larger downforce overpredictions, nearly five times greater than those observed in moving ground conditions. This discrepancy was primarily attributed to differences in the ground boundary layer, which was affected by wind tunnel floor irregularities and imperfect boundary layer suction. Furthermore, while the $k-\omega$ SST model reliably captured dominant wake structures, it demonstrated stronger agreement with slant surface pressure distributions than with highly unsteady wake pressure data, reinforcing previously observed trends in moving ground validations.

9.1.1.4 Summary

This research phase successfully achieved Objective 1, establishing a cost-effective CFD workflow and a structured wind tunnel correlation process to support subsequent aerodynamic analyses. The $k-\omega$ SST model was identified as a computationally efficient RANS approach that demonstrated strong predictive accuracy, with validation results across multiple chapters reinforcing its reliability and consistency for general aerodynamic analyses and comparative studies. However, in conditions where aerodynamic performance was dominated by highly unsteady flow behaviour, such as critically low ride heights, higher-fidelity simulations like DDES were required to capture complex unsteady phenomena, including vortex instability and severe flow separation.

The numerical wind tunnel correlation methodology proved effective in identifying and quantifying wind tunnel-induced effects, underscoring the importance of replicating experimental conditions in CFD to enhance correlation and improve data interpretation. Validation results across later chapters further reinforced its effectiveness, enabling setup-specific investigations into sources of discrepancies and strengthening confidence in combined experimental and numerical analyses. Ultimately, this phase established a robust methodological foundation that underpinned the aerodynamic investigations in subsequent research phases, ensuring a consistent and practical framework for evaluating aerodynamic performance and active aerodynamic systems in real-world operating conditions.

9.1.2 Phase 2: Aerodynamic Analysis and Active Aerodynamic Design Evaluation

Building on the methodological framework established in the first phase, this research phase focused on analysing the aerodynamic behaviour of the *DrivAer hp-F* model in real-world operating conditions, specifically yaw and cornering. The objective was to investigate how these conditions influence flow field characteristics and impact overall aerodynamic performance. Additionally, this phase focused on evaluating the effectiveness of 2D active aerodynamic systems in modulating the aerodynamic balance both longitudinally and laterally to enhance cornering performance. Where possible, wind tunnel experiments were conducted alongside CFD simulations, ensuring a comprehensive and validated understanding of aerodynamic behaviour under dynamic operating conditions.

9.1.2.1 Approach

The aerodynamic effects of yaw were examined in Chapter 4 through a series of wind tunnel experiments and CFD simulations on the baseline, spoiler-equipped, and rear wing-equipped *DrivAer hp-F* configurations. The experiments measured aerodynamic forces, surface pressures, and wake pressures across a range of yaw angles. CFD simulations following the methodology established in the first phase were conducted to replicate the wind tunnel conditions, improving the interpretation of experimental data. Additionally, the methodology was extended with a mesh interface technique to eliminate the need for re-meshing at each yaw angle, enhancing computational efficiency.

The influence of cornering on aerodynamic performance was investigated in Chapter 5 by analysing the *DrivAer hp-F* with a rear wing across various cornering scenarios and comparing the results to straight-line conditions. The methodology developed in the first research phase was refined with modified inlet and outlet positions, a moving reference frame, and an overset mesh technique, enabling the efficient simulation of multiple driving conditions with different vehicle operating states. To complement the simulations, moving ground wind tunnel experiments were conducted to measure aerodynamic forces at varying rear wing angles. CFD simulations replicating these conditions were performed to facilitate validation and support data interpretation.

The aerodynamic behaviour and performance impact of 2D active aerodynamic systems during cornering were assessed in Chapter 6. Two high-performance vehicle concepts featuring split wing and tilt wing designs, capable of modulating aerodynamic balance both longitudinally and laterally, were introduced. A control strategy was implemented, linking front and rear wing settings on each side of the vehicle to maintain a target aerodynamic balance range. Aerodynamic data was obtained through CFD simulations using the methodology established in Chapter 5 and integrated with a steady-state vehicle dynamics model to evaluate the impact on cornering performance.

9.1.2.2 Key Findings

The yaw analysis in Chapter 4 demonstrated that across a 20° yaw range, lift on the baseline *DrivAer hp-F* more than doubled, while drag increased by over 25%. Adding a spoiler or rear wing enhanced downforce but also increased yaw sensitivity by approximately 25% and 30%, respectively. Side force increased nearly linearly with yaw angle, with the spoiler and rear wing configurations generating about 5% and 10% higher peak side forces. Flow field analyses identified several mechanisms contributing to these aerodynamic changes, including enhanced wheel wake interference reducing underbody downforce, flow acceleration over windward upper surfaces amplifying lift, and altered flow interactions with aerodynamic devices diminishing their effectiveness. Additionally, wake characteristics became highly asymmetric, dominated by an inboard-rotating vortex that intensified with yaw angle, increasing pressure drag.

The cornering analysis in Chapter 5 showed downforce and drag penalties of up to 20% and 35%, respectively, with severity dependent on corner radius. These effects were primarily driven by increased underbody wheel wake interference reducing downforce, flow acceleration over upper surfaces amplifying lift, and reductions in base pressure increasing drag. Lateral pressure gradients induced a centripetal force that became more pronounced in tighter corners. Additionally, significant wake asymmetries developed, with a spanwise recirculation vortex forming on the inboard side and interacting with a streamwise wake structure on the outboard side. The rear wing exhibited strong sensitivity to cornering, with notable changes in vortical flow and pressure distribution, diminishing its aerodynamic performance.

The evaluation of the aerodynamic load modulation capabilities of the two 2D active aerodynamic systems introduced in Chapter 6 identified the tilt wing concept as the most effective configuration. It generated nearly twice the aerodynamic roll moment of the split wing concept and provided a more direct response to the control strategy. As a result, the tilt wing concept improved maximum cornering velocity by 2% over a symmetric maximum-downforce setup, whereas the split wing concept achieved a 0.5% gain. Beyond improving cornering performance, both systems enhanced aerodynamic efficiency by optimising downforce distribution, reducing drag-related propulsion power demands by 15%–20% compared to the symmetric maximum-downforce configuration in the analysed cornering conditions.

9.1.2.3 Comparative Remarks

This research phase provided key insights into how yaw and cornering conditions influence flow field characteristics and aerodynamic performance. To better understand their relationship, this section compares findings from both studies, identifying areas of agreement and variation.

Experimental findings are first considered, where the yaw study in Chapter 4 measured forces on both the vehicle and stationary wheels using an underfloor balance without a moving ground, whereas the straight-line experiments in Chapter 5 employed a moving ground with rotating wheels, measuring only vehicle body forces using an overhead balance. The stationary ground setup recorded approximately half the downforce measured with the moving ground, while drag remained comparable. While this discrepancy is partly due to the measurement configurations, it underscores the significant role of replicating ground effect in high-performance vehicle aerodynamics.

To assess the relative influence of yaw and cornering on aerodynamic performance, conditions with equivalent flow impact angles are compared. For a 5° yaw case and a $5.75L$ radius corner, yaw increased lift by $\Delta C_L \approx 0.10$, whereas lift remained largely unchanged during cornering relative to the corresponding straight-line scenarios. Conversely, the drag penalty in yaw was 60% lower than in cornering, while the side force increase was more than twice as large. A similar trend was observed at 10° yaw and the equivalent $2.9L$ radius corner, with yaw inducing more than four times the lift increase seen in cornering, accompanied by a 60% smaller drag penalty and a threefold greater side force increment. These results demonstrate that yaw conditions amplify lift and side forces more strongly, whereas cornering imposes comparatively higher drag penalties.

These force trends correspond to differences in the vehicle's surface pressure distribution between yaw and cornering conditions. During cornering, the relative flow angle varies along the vehicle's longitudinal axis, decreasing near the centre before reversing toward the rear, whereas in yaw it remains constant. As a result, the pressure distribution near the nose in cornering resembles that of a positive yaw angle, while the pressure near the rear aligns with that of an opposite yaw angle. The uniform flow angle in yaw promotes stronger flow acceleration over the upper surfaces and intensifies wheel wake interference along the underbody, contributing to the larger lift increments observed in these conditions. Additionally, the orientation of the resultant aerodynamic forces relative to the vehicle's trajectory contributes to smaller drag penalties but larger side force increases in yaw conditions.

While yaw and cornering produce distinct aerodynamic effects, similarities emerge at the front and rear of the vehicle, where the local flow angle aligns more closely between the two conditions. Given the challenges of replicating full cornering scenarios in wind tunnel experiments, measurements in yaw conditions could provide valuable insights into the sensitivity of front- and rear-mounted aerodynamic devices to varying flow conditions, informing their behaviour during cornering. Additionally, the significant downforce variation between stationary and moving ground experiments highlights the critical role of ground effect, reinforcing the necessity of a moving ground setup when assessing full aerodynamic performance.

9.1.2.4 Summary

This research phase addressed Objective 2 through Chapter 4, which analysed the aerodynamic effects of yaw on three high-performance vehicle configurations using combined experimental and numerical investigations. Chapter 5 achieved Objective 3 by examining the influence of various cornering conditions on the aerodynamic characteristics of a high-performance vehicle and its rear wing relative to straight-line conditions through CFD simulations and wind tunnel measurements. Finally, Chapter 6 addressed Objective 4 by evaluating 2D active aerodynamic concepts in various cornering conditions, demonstrating their ability to modulate aerodynamic balance and enhance cornering performance.

Overall, this research phase provided a comprehensive characterisation of aerodynamic performance in yaw and cornering conditions. It also presented an evaluation of 2D active aerodynamic systems in enhancing cornering performance, with the tilt wing concept proving most effective. The findings of this research phase formed a solid foundation for the subsequent phase, which shifts toward performance-oriented analyses of active aerodynamic systems in representative on-track driving scenarios.

9.1.3 Phase 3: Performance Optimisation and Analysis of Active Aerodynamic Systems

Building upon the preceding analysis of 2D active aerodynamic systems during cornering, the final research phase focused on evaluating the influence of active aerodynamic systems on overall vehicle performance in on-track driving scenarios. Additionally, it explored a complementary research direction, identified in collaboration with the Vehicle Dynamics and Control group at Cranfield University, aimed at examining the performance of active aerodynamic systems when integrated with advanced powertrain architectures capable of active torque distribution (ATD). This research phase sought to establish a quantitative understanding of the performance impact of various active aerodynamic configurations, including 2D active aerodynamic systems, while identifying underlying mechanisms and control strategies driving their performance enhancements.

9.1.3.1 Approach

The influence of active aerodynamic systems on vehicle performance was investigated in Chapter 7 using a minimum lap time simulation framework that simultaneously optimised the vehicle's trajectory, control inputs, and selected design parameters. This framework was extended to enable the optimisation of vehicle performance with varying levels of active aerodynamic control, incorporating aerodynamic data derived from CFD simulations following the methodology established in the second research phase. The analysis considered a range of aerodynamic configurations, including static, active, and 2D active systems, evaluated over a lap of the Circuit de Barcelona-Catalunya.

The performance potential of active aerodynamics when integrated with ATD was examined in Chapter 8 through lap time simulations on the Circuit de Barcelona-Catalunya. This study further extended the minimum lap time simulation framework to support performance evaluation across a range of aerodynamic and torque distribution configurations. In addition to optimising selected vehicle design parameters, the framework was enhanced to include the optimisation of the tyres' nominal vertical load. This extension facilitated an investigation into the influence of vehicle setup and tyre characteristics on maximising the performance gains from active aerodynamics and ATD.

9.1.3.2 Key Findings

The results from Chapter 7 demonstrated that active aerodynamic configurations significantly enhanced vehicle performance compared to static aerodynamic setups, with improvements in braking, cornering, acceleration, and straight-line speed. The 2D active aerodynamic configuration achieved the greatest lap time reduction of approximately 0.92 s (0.76%) by modulating downforce laterally between the inboard and outboard tyres, improving grip and enabling higher longitudinal and lateral accelerations.

Optimal control strategies revealed that performance gains were driven by the active adjustment of aerodynamic elements to counteract vertical tyre load shifts induced by load transfer. Configurations featuring symmetric control of either the rear wing alone or both the front and rear wings adjusted wing angles in a manner generally opposing longitudinal load transfer, redistributing downforce to support the underloaded axle.

The 2D active aerodynamic systems further optimised load distribution by prioritising downforce on the most underloaded tyres during scenarios with combined longitudinal and lateral load transfers. Additionally, increasing aerodynamic adaptability led to progressively rearward shifts in the optimal brake balance, torque distribution, and roll stiffness distribution. These trends reflected the improved aerodynamic control over the rear axle, enabling more effective tyre utilisation during braking and acceleration while accommodating greater lateral load transfer during cornering.

The results presented in Chapter 8 demonstrated a substantial performance difference between conventional all-wheel-drive (AWD) and more advanced ATD powertrain configurations. When combined with a static aerodynamic setup, the ATD configuration achieved an approximately 5.64 s (4.80%) faster lap time, attributed to its capacity to dynamically allocate drive and brake torque to individual wheels, thereby optimising tyre force utilisation. In contrast, AWD configurations are limited to axle-based torque distribution, which is constrained by the most grip-limited tyre on each axle. The improved traction utilisation offered by ATD enabled higher combined longitudinal and lateral accelerations, particularly during trail braking and corner exits, and resulted in enhanced steering response and greater handling predictability.

The integration of active aerodynamics further enhanced the performance of ATD configurations, reducing lap times by up to 0.67 s (0.58%), with the 2D active aerodynamic system yielding the greatest gains. Control strategies revealed that conventional active aerodynamic setups generally adjusted wing angles inversely with longitudinal load transfer, distributing downforce to the most underloaded axle. Meanwhile, the 2D active aerodynamic configuration provided a more refined response by prioritising downforce on the most underloaded tyres in conditions involving both longitudinal and lateral load transfer.

Increasing aerodynamic adaptability resulted in a progressively rearward shift in the optimal roll stiffness distribution, driven by greater aerodynamic influence on the rear axle and a corresponding increase in its capacity to accommodate lateral load transfer. Additionally, optimisation of nominal tyre loads underscored the critical role of tyre characteristics, improving the lap time of the ATD configuration with static aerodynamics by 1.62 s (1.40%). Tyre characteristics also influenced the effectiveness of active aerodynamic systems on the ATD configurations. The 2D active aerodynamic setup achieved a lap time reduction of 0.76 s (0.67%) when paired with optimised tyres, representing an approximately 15% greater improvement than that achieved with standard tyres.

9.1.3.3 Comparative Remarks

To provide deeper insight into the performance potential of active aerodynamics across different powertrain configurations, this section discusses their achieved lap time improvements, control strategies, and optimal vehicle setups when integrated with AWD and ATD powertrains. Starting with optimised static aerodynamic setups, the AWD configuration employed a moderately low-downforce setup to balance drag for straight-line performance. In contrast, the ATD configuration adopted a high-downforce setup to maximise tyre grip during dynamic manoeuvres.

This distinction arises from the ATD configuration's capability to distribute torque between individual wheels, enabling more effective utilisation of the additional tyre grip provided by the high-downforce setup. The associated increase in drag compromises straight-line performance but is offset by enhanced traction during cornering. Consistent with this behaviour, the ATD configuration adopted a slightly more front-biased roll stiffness distribution than the AWD setup, reflecting its improved capacity to exploit front axle grip.

Torque distribution capabilities also influenced the optimal aerodynamic control strategies. Both the symmetric and 2D active systems exhibited less frequent adjustments and generally higher downforce settings in ATD configurations compared to AWD. In AWD setups, more continuous adjustments were required to balance aerodynamic loads across the most grip-limited tyres on each axle, which constrain the axle-based torque allocation.

In terms of lap time gains, the symmetric and 2D active aerodynamic setups improved lap times for the AWD configuration by approximately 0.25 s more than for the ATD configuration, corresponding to 40% and 30% greater relative gains, respectively. These differences partially stem from the characteristics of their respective baseline static aerodynamic setups, which influence whether performance gains arise primarily from increased downforce during dynamic manoeuvres or from improved aerodynamic efficiency on straights.

Tyre characteristics optimisation based on the nominal vertical load revealed a significant influence on the effectiveness of active aerodynamics. AWD configurations performed best with tyres featuring lower nominal vertical loads and higher cornering stiffness, which prioritised grip on underloaded tyres, enabling greater torque application to each axle, albeit at the expense of reduced lateral grip on the most heavily loaded tyres. In contrast, ATD configurations, unconstrained by axle-based grip limitations, benefited from tyres with slightly higher nominal loads and lower cornering stiffness. These more balanced characteristics reduced grip losses on heavily loaded tyres while maintaining sufficient grip on less loaded tyres. With optimised tyres, the lap time gains from the active aerodynamic systems became more comparable across both powertrains, underscoring the critical role of tyre characteristics in maximising the benefits of active aerodynamics.

9.1.3.4 Summary

This research phase successfully addressed Objective 5 through the investigations presented in Chapter 7, which evaluated the performance potential of active aerodynamic systems and the underlying mechanisms driving their gains. The analysis employed a minimum lap time simulation framework, incorporating CFD-derived aerodynamic data and detailed vehicle modelling, to assess system performance over a lap of the Circuit de Barcelona-Catalunya. Objective 6 was fulfilled in Chapter 8, which extended the study to consider active aerodynamic systems integrated with active torque distribution. This investigation identified key performance trends across different powertrain configurations, emphasising both their individual and combined contributions, and demonstrated the importance of tyre characteristics in maximising overall vehicle performance.

Overall, this phase demonstrated the effectiveness of active aerodynamic systems in enhancing vehicle performance, offering key insights into lap time improvements, optimal control strategies, and the influence of vehicle and tyre setup on system effectiveness. Beyond advancing the understanding of active aerodynamics, including 2D systems capable of modulating aerodynamic balance both longitudinally and laterally, the integrated methodology established a comprehensive simulation framework for optimising and evaluating the impact of active aerodynamics on vehicle performance. The findings, together with the published simulation framework, provide a strong foundation for future developments in active aerodynamics and support continued innovation in high-performance vehicle engineering.

9.2 Research Contributions and Implications

This section outlines the key research contributions of this thesis and their broader implications. It presents methodological advancements, findings from experimental and numerical aerodynamic analyses, and insights gained through simulation-based performance evaluations.

A key contribution of this research was the development of a cost-effective numerical workflow tailored for high-performance vehicle aerodynamics. Through systematic meshing strategy development, turbulence model evaluation, and wind tunnel correlation methodologies, this workflow enabled computationally efficient simulations with high predictive accuracy. Successfully applied across varying flow conditions, it demonstrated strong consistency, increasing confidence in its predictive capabilities. Additionally, the structured correlation approach improved validation processes and experimental data interpretation, providing a framework for accurate and practical aerodynamic analyses in both research and industry.

The numerical analysis of ride height dependency provided new insights into the complex flow phenomena governing aerodynamic performance in ground effect. The study demonstrated the crucial role of vortex structures in sustaining diffuser performance and identified critical low ride heights where vortex breakdown caused abrupt aerodynamic performance losses. These findings advance the understanding of diffuser aerodynamics, offering valuable insights for future research and the improvement of underbody aerodynamic design in high-performance vehicles.

The experimental and numerical investigation of yaw conditions advanced the understanding of yaw-induced aerodynamic effects on high-performance vehicles. The study identified key flow field alterations and their impact on aerodynamic performance, offering insights to support vehicle design improvements for enhanced performance in yaw conditions. Furthermore, the published experimental datasets, including wind tunnel measurements of forces, surface pressures, and wake pressures in yaw conditions, expanded the limited experimental database on vehicle aerodynamics. These datasets serve as valuable validation resources for researchers and practitioners, with current applications in industry reinforcing their real-world impact.

The analysis of cornering conditions provided new insights into the aerodynamic behaviour of high-performance vehicles. It identified key corner-induced effects on flow field characteristics, including rear wing interactions and their impact on overall aerodynamic performance. These findings support the development of improved vehicle designs for enhanced aerodynamic performance during cornering. The study also introduced an effective methodology for evaluating a wide range of vehicle setup configurations across various operating conditions, integrating overset meshing with a moving reference frame, delivering a streamlined workflow for future analyses. The accompanying published experimental dataset, featuring force measurements at varying rear wing angles under moving ground conditions, expanded the limited database on vehicle aerodynamics, supporting researchers and industry practitioners in refining aerodynamic modelling approaches.

The design and evaluation of two 2D active aerodynamic configurations, implemented as split and tilt wing concepts, provided key insights into their ability to influence aerodynamic balance both longitudinally and laterally. By comparing their aerodynamic capabilities and impact on cornering performance, the study identified the most effective concept and demonstrated the advantages of modulating aerodynamic balance laterally. These findings advance the understanding of 2D active aerodynamic systems and their influence on vehicle performance, supporting future research and development in both conventional and 2D active aerodynamic systems.

The investigation into the performance potential of active aerodynamic systems, including the 2D tilt wing concept, provided key insights into their effectiveness across a range of on-track driving scenarios. Using a comprehensive minimum lap time simulation framework, the study quantified the performance gains of various active aerodynamic configurations, identified optimal control strategies, and evaluated the influence of vehicle setup on maximising system effectiveness. These findings advance the understanding of the role of active aerodynamics, including 2D systems, in enhancing vehicle performance, providing a strong foundation for further academic research and industry developments.

The extended research into active aerodynamics integrated with active torque distribution (ATD) yielded new insights into their individual and synergistic contributions to vehicle performance. Analysis of their interaction revealed the mechanisms underpinning performance gains, shifts in optimal vehicle setup, and the critical influence of tyre characteristics in maximising their effectiveness. Collectively, these findings advance the understanding of the performance of active aerodynamics integrated across different powertrain architectures and provide support for future research and development toward their application in next-generation high-performance vehicles.

The extended functionalities integrated into the minimum lap time simulation framework enable performance optimisation across various active aerodynamic and torque distribution configurations, including the simultaneous optimisation of design parameters and tyre characteristics. The published framework demonstrated its effectiveness in supporting comprehensive performance analyses throughout this research and serves as a valuable tool for future studies on active aerodynamics and related vehicle systems. Additionally, it could support informed decision-making in development processes and pre-event setup planning within industry applications.

Overall, this research made fundamental contributions to the field of high-performance vehicle aerodynamics by advancing methodological approaches, expanding the experimental database, deepening the understanding of aerodynamic effects in dynamic operating conditions, and demonstrating the performance potential of active aerodynamic systems. These methodological, empirical, theoretical, and technical contributions have significant implications for both academia and industry, supporting future research and development on vehicle aerodynamics and active aerodynamic systems. A summary of the key contributions is provided below:

1. Development of a cost-effective CFD workflow for high-performance vehicle aerodynamics, integrating a refined meshing strategy, turbulence model evaluation, and wind tunnel correlation methodology. This framework provides a structured reference for practical aerodynamic simulations, supporting both research and industry applications.
2. Identification of critical ride height dependencies in high-performance vehicle aerodynamics through numerical analysis, revealing vortex-driven diffuser performance mechanisms and the onset of vortex breakdown with its detrimental effects at critically low ride heights. These findings advance knowledge of underbody aerodynamics and inform their design for enhanced performance and stability.
3. Characterisation of yaw-induced aerodynamic effects through combined experimental and numerical investigations, identifying key flow field alterations and their impact on aerodynamic performance. These findings deepen the understanding of vehicle aerodynamics in yaw conditions and provide insights for improving aerodynamic design in dynamic operating conditions.
4. Analysis of the aerodynamics of a high-performance vehicle with a rear wing in straight-line and cornering conditions using wind tunnel experiments and CFD simulations, revealing corner-induced flow field changes and their impact on aerodynamic performance. This study refines cornering simulation methodologies and advances the understanding of vehicle aerodynamics during cornering, informing vehicle and rear wing design for improved performance in dynamic driving conditions.
5. Design and evaluation of two 2D active aerodynamic systems, developed as split and tilt wing concepts, using CFD simulations and steady-state cornering performance analysis, demonstrating their ability to modulate aerodynamic balance both longitudinally and laterally, along with their impact on cornering performance. These findings advance the understanding of 2D active aerodynamic systems and support research aimed at enhancing vehicle performance through active aerodynamics.
6. Optimisation and analysis of vehicle performance with active aerodynamic systems, including 2D configurations, using a minimum lap time simulation framework, identifying performance gains, optimal control strategies, and vehicle setup considerations. These findings advance knowledge on the performance potential of active aerodynamics and provide a foundation for further research and development on active aerodynamics for automotive and high-performance vehicles.

7. Optimisation and evaluation of active aerodynamics combined with active torque distribution using minimum lap time simulations, identifying individual and combined performance gains, key performance-enhancing mechanisms, and optimal vehicle setup and tyre characteristics. The findings advance the understanding of active aerodynamics integrated with different powertrain architectures and support future research into their application in automotive and high-performance vehicles.
8. Expansion of the limited experimental database on vehicle aerodynamics through publicly available datasets comprising wind tunnel measurements of forces, surface pressures, and wake pressures in yaw conditions, as well as force measurements at varying rear wing angles under moving ground conditions. These datasets serve as essential validation resources for aerodynamic research and development, with their active use in industry highlighting their practical relevance.
9. Extension of an existing minimum lap time simulation framework, enhancing its capabilities to optimise vehicle performance across active aerodynamic and torque distribution configurations, alongside parametric optimisation of vehicle setup and tyre characteristics. This publicly available extension serves as a valuable resource for future research on active aerodynamics and related vehicle systems, as well as for supporting decision-making in vehicle development and pre-event setup planning.

9.3 Future Work

While this research has successfully achieved all defined objectives, several avenues for further investigation can be identified. These potential areas of future work build upon the methodologies and findings established in this research and could provide deeper insights into vehicle aerodynamics and active aerodynamic systems.

One suggested direction for further research is the experimental investigation of yaw effects using a moving ground. The yaw experiments in this study were conducted with a stationary ground due to the limitations of the available facility. However, some wind tunnels are equipped with rotating moving ground capabilities, allowing additional refinements in the assessment of yaw-induced changes to underbody flow characteristics. Implementing such an experimental setup could offer further insights into the effects of yaw on ground-effect aerodynamics and facilitate a direct comparison with moving-ground cornering simulations, further clarifying the relationship between these conditions.

Another avenue for future research is the inclusion of vehicle body motions, such as roll, pitch, and heave, in lap time and aerodynamic simulations. These dynamic effects influence underbody aerodynamics, and incorporating them could facilitate the refinement of performance predictions. Coupling aerodynamics and vehicle dynamics can be approached at varying levels of complexity, ranging from interpolation between aerodynamic datasets at predefined body positions to fully dynamic coupling of CFD and lap time simulations. While the latter could potentially yield more detailed insights into the interaction between active aerodynamics and vehicle dynamics, it would impose significantly higher computational costs.

Such investigations could focus on the role of active aerodynamics in stabilising the aerodynamic platform by counteracting body motion effects to some extent. Conventional active aerodynamics are expected to mitigate pitch, while 2D active aerodynamic systems could influence both pitch and roll, further enhancing their potential benefits over static aerodynamic configurations.

Further research could investigate a wider range of conventional and 2D active aerodynamic systems tailored to specific vehicle applications. In high-performance vehicles, active diffuser systems or alternative wing configurations could offer additional methods for optimising aerodynamic balance. Active aerodynamics could also be explored in vehicles with a greater reliance on aerodynamics, such as hypercars or Formula race cars, which operate at higher velocities and may experience even greater performance benefits from conventional and 2D active aerodynamic systems. Additionally, research could explore active aerodynamics in automotive applications to balance aerodynamic efficiency with other performance factors. For example, in electric vehicles, where cooling demands are increasing, future studies could investigate how active aerodynamic systems can manage thermal requirements while minimising drag, ultimately improving efficiency and driving range.

The implementation and control of active aerodynamic systems presents another opportunity for future research. This study evaluated optimal control strategies and identified key related quantities, highlighting relationships that could guide real-time implementation. Future work could investigate the feasibility of real-time controllers, assessing how effectively control architectures operate relative to the identified optimal aerodynamic control cases. Additionally, transient aerodynamic effects associated with dynamically adjusting aerodynamic elements could be explored. Analysing flow stabilisation times and hysteresis effects could provide valuable insights into the dynamic response of active aerodynamic systems, informing real-time controller development to improve aerodynamic performance.

Chapter 10

Conclusion

This research investigated the performance potential of active aerodynamic systems for high-performance vehicles across dynamic driving scenarios. To support this, it developed a cost-effective Computational Fluid Dynamics (CFD) methodology for aerodynamic analysis, and examined the influence of real-world operating conditions, specifically yaw and cornering, on aerodynamic performance. Building on this foundation, several active aerodynamic concepts were designed, including 2D systems capable of modulating aerodynamic balance both longitudinally and laterally, and their impact on vehicle performance was comprehensively analysed.

A key achievement of this research was the development and validation of a cost-effective CFD workflow, alongside a structured wind tunnel correlation process. Together, these enabled reliable aerodynamic predictions and improved the interpretation of experimental data. Additional core contributions included combined experimental and numerical aerodynamic analyses conducted under varying ride heights, yaw conditions, and cornering scenarios. These investigations provided insights into ground-effect aerodynamics and flow field alterations induced by dynamic operating conditions, highlighting their influence on aerodynamic performance.

A central component of the research was the performance analysis of conventional and 2D active aerodynamic systems, evaluated in combination with all-wheel-drive and active torque distribution powertrain architectures using minimum lap time simulations. These analyses demonstrated the performance benefits of active aerodynamic systems, revealed the underlying performance-enhancing mechanisms, and identified optimal control strategies. Furthermore, the results highlighted the influence of vehicle setup and tyre characteristics on maximising system effectiveness. Ultimately, the analyses showed that 2D active aerodynamic systems consistently outperformed conventional configurations. By prioritising downforce on underloaded tyres while generating an aerodynamic roll moment opposing lateral load transfer, these systems exploited the tyres' non-linear vertical load sensitivity, achieving grip levels unattainable with symmetric aerodynamic loading.

This research provides significant contributions to both academia and industry. The methodological advancements offer guidance for improving simulation accuracy and experimental correlation. Insights from the aerodynamic analyses deepen the understanding of vehicle aerodynamics in dynamic conditions and the associated published wind tunnel datasets expand the limited publicly available experimental database on vehicle aerodynamics. The demonstrated performance potential of active aerodynamic systems, particularly the 2D configurations, emphasises their relevance for future vehicle designs, while the published minimum lap time simulation framework provides a valuable tool for ongoing research and development in vehicle engineering.

Overall, this research offers a holistic exploration of active aerodynamic systems for high-performance vehicles, bridging methodological development, aerodynamic characterisation, and vehicle performance optimisation. Through its methodological, empirical, theoretical, and technical contributions, it provides a solid foundation for future advances in active aerodynamic systems and their integration into next-generation automotive and motorsport applications.