INSIGHT INTO THE AERODYNAMICS OF RACE AND IDEALIZED ROAD VEHICLES USING SCALE-RESOLVED AND SCALE-AVERAGED CFD SIMULATIONS

by

Adit Misar

A dissertation submitted to the faculty of The University of North Carolina at Charlotte in partial fulfillment of the requirements for the degree of Doctor of Philosophy in Mechanical Engineering

Charlotte

2023

Approved by:

Dr. Mesbah Uddin

Dr. Chen Fu

Dr. Srinivas Pulugurtha

Dr. Praveen Ramaprabhu

©2023 Adit Misar ALL RIGHTS RESERVED

ABSTRACT

ADIT MISAR. Insight into the aerodynamics of race and idealized road vehicles using scale-resolved and scale-averaged cfd simulations. (Under the direction of DR. MESBAH UDDIN)

Aerodynamics has long been perceived as the single most important aspect amongst all factors that contribute to the on-track performance of a racecar. As such, in all forms of motorsports, race teams dedicate a significant portion of their budget and efforts to aerodynamic development. As track testing of racecars is cost prohibitive and is mostly controlled by the sports sanctioning bodies, wind-tunnel testing and Computational Fluid Dynamics (CFD) are the commonly used tools in racecar aero development. However, in an effort to ensure level playing fields, race sanctioning bodies introduced limits on how much wind tunnel time or CFD resources each team can utilize in its aerodynamic development. For CFD, the implication of these caps means that the solution turn around time, accuracy and reliability must be improved to overcome the challenges caused by the imposed resource-utilization-restrictions. In order to achieve the goal of finding a fast, yet reliable CFD methodology, this project presents the development of a Reynolds-Averaged Navier Stokes (RANS) CFD framework for NASCAR Cup stock-racecars using a Scale Averaged (SAS) approach based on the SST $k - \omega$ turbulence model. The methodology development process involves a thorough understanding of the effects of solver parameters, closure coefficients, and boundary conditions on the prediction veracity. The prediction accuracy is validated against test data obtained from a closed-return, open-jet rolling-road wind tunnel for a range of racecar on-track operating conditions, such as the changes in ride-heights and yaw. Results using the CFD framework presented in this dissertation achieved a correlation of 98% with the wind-tunnel lift and drag measurements data over a range of operating conditions. However, existing literature suggests that the scale and time-resolved (or SRS) Detached Eddy Simulation (DES) approach produces a better overall flow field predictions for simplified road vehicles, such as the Ahmed body. The aerodynamic characteristics of racecars are starkly different even from the passenger vehicles, let alone the simplified vehicles. As a study comparing the effectiveness of SAS and SRS approaches in flow-filed predictions is not available in existing literature, this work also investigates the aerodynamics of a stock-racecar using Improved Delayed DES (IDDES). This study finds that the IDDES resolved a range of finer vortical structures that are almost entirely missed by the RANS approach. To better understand the roles of these vortices on the aero characteristics of the race car, spectral analyses of the aerodynamic forces and moments are carried out. The distribution of Power Spectral Density (PSD) is found to be largely independent of the operating conditions. This implies that the dominant modes stemmed from the racecar geometry with a ramification that an understanding on the contribution to the dominant energy modes by different race-car geometry components would be very beneficial for performance improvement. To identify the dominant modes, a more advanced and informative modal decomposition tool is required. The Dynamic Mode Decomposition (DMD), which was seen in existing literature to be a very effective tool for low Reynolds number flows around canonical geometries, is considered to be an ideal candidate. However, due to the nonavailability of the volume of flow-field data required to train a DMD algorithm for the flows past NASCAR race-cars, the process is sought to develop using a simplified road vehicle, the Ahmed body. When the DMD algorithm from the existing literature was applied to the high Reynolds number, separation-dominated flow past an Ahmed body, the DMD reconstruction of the flow field suffered nonphysical dampening of the medium-to-high frequency modes. To circumvent this, a modified DMD algorithm is proposed in this work which involves introduction of a mode filtration process. The proposed DMD algorithm is found to be very effective in both flow-field reconstruction and predictions of the future state.

DEDICATION

This work is dedicated to my parents, Medha Misar and Sunil Misar, and my brother Anuj Misar, for providing me with constant support, motivation and encouragement.

Their continual inspiration to deliver my best at every task and selfless love provided throughout my life builds the foundation that makes this dissertation possible.

Thank You

ACKNOWLEDGEMENTS

I would like to thank my academic advisor, Dr. Mesbah Uddin, who provided me with the opportunity to put together my passions of aerodynamics and engineering. A career in aerodynamics has been my dream since I was first introduced to aerodynamics back in high school. I am thankful for all the learning opportunities that he has presented to me that includes research on various projects, as well as teaching and mentoring opportunities. Mostly, I thank him for his patience and insights over this long and challenging journey to complete a doctoral dissertation. I am grateful to have him as a mentor.

I would like to thank my industry mentors, Ted Pandaleon, Josh Wilson, Dr. Nathon Tison, and Dr. Vamshi Korivi for all their insights and support.

I would like to thank my data sponsor for providing the CAD model and all wind tunnel experimental data of a Gen-6 NASCAR Cup Racecar. These are proprietary data obtained through a Non-Disclosure Agreement (NDA) and thus to protect the confidential data, all force and moment coefficients presented in this dissertation are normalized by an arbitrary reference area.

I also thankfully acknowledge partial funding support from the Office of Naval Research (ONR grant# N00014-19-1-2245) through which I was supported as a Research Assistant to work on the development of CFD based application oriented Virtual Engineering tools.

I would like to thank my dissertation committee members I have had the honor to work with, Dr. Chen Fu, Dr. Srinivas Pulugurtha, and Dr. Praveen Ramaprabhu, for their feedback, support and commitment to high-level education.

I would like to thank UNC Charlotte University Research Computing and MOSAIC Computing divisions for providing dependable and adequate computational resources for my research.

I would also like to thank my peers in our research group, Clay Robinson, Chunhui

Zhang, Sudhan Rajesakar, Patrick Bounds, Spencer Owen, Brett Peters, Ayushi Jain, Hamed Ahani, and Vincent Lee, to name a few, for their companionship, support, and correcting my English grammatical errors.

I also thank my family for their continued support and motivation, extending across the physical distance of over 13450 km (8358 miles) from each other almost every day. I am eager to host you when you come here to visit me.

I would like to extend my humble gratitude to everyone who has been part of my journey since I arrived in this country. There are too many to name here but some are, Amol Sathe, Anay Joshi, Anish Venkatraman, Anurag Doshi, Arjun Yeravdekar, Jaydeep Kshirsagar, Manish Patil, Nagarjun Chandrashekar, and Pratik Kulkarni. Our shared experiences and memories have helped sustain the push over the finish line and make me excited for life beyond graduate school.

Beyond academics, I would like to acknowledge the facilities and activity groups at UNC Charlotte that helped adopt healthier lifestyles. The friendly staff at the Student Health Center, Counseling at Psychological Services (CAPS), Belk Gym, and University Recreation Center (UREC) provided all necessary support. I resumed my activities of swimming and table tennis. I also picked up several new activities of racquetball, tennis, boxing, badminton, and resistance training. Walking and jogging on the Mallard Creek Greenway also provided rejuvenation. These helped to significantly improve my overall fitness. I would like to thank my friends Arjun Yeravdekar, Jaydeep Kshirsagar, Manish Patil, Hamed Ahani, and Vincent Lee for their company and motivation throughout these activities.

Finally, I would like to thank my extended family, both in India and overseas, for their support and encouragement. Many in the United States have hosted me for holidays and I look forward to visiting everyone. I am comforted in the knowledge that my network of people are always available to help me navigate the various adventures of life.

TABLE OF CONTENTS

LIST OF TABLE	ES	xii
LIST OF FIGUR	LES	xiii
LIST OF ABBRH	EVIATIONS	xxi
LIST OF SYMBO	OLS	xxiii
CHAPTER 1: IN	TRODUCTION	1
1.1. Comput	ational Fluid Dynamics (CFD)	2
1.1.1.	Governing Equations	3
1.1.2.	Direct Numerical Simulation (DNS)	5
1.1.3.	Improved Delayed Detached Eddy Simulation (ID- DES)	6
1.1.4.	Reynolds-Averaged Navier Stokes (RANS)	7
1.1.5.	CFD Tools	9
1.2. Backgro	ound	10
1.3. Motivat	ion	13
1.4. Objectiv	ves	15
1.5. Disserta	tion Outline	17
CHAPTER 2: (A AND BOUN DICTIONS (ARTICLE 1) EFFECTS OF SOLVER PARAMETERS IDARY CONDITIONS ON RANS CFD FLOW PRE- OVER A GEN-6 NASCAR RACECAR	18
2.1. Introduc	ction	18
2.2. Comput	ational Method & Simulation Setup	22
2.2.1.	Turbulence Model	24
2.2.2.	Geometry and Mesh	25

			ix
	2.2.3.	Physics Setup	28
	2.2.4.	Wall Treatment	29
	2.2.5.	Solver and Convergence	30
2.3.	Results a	and Discussion	30
	2.3.1.	Boundary Condition Effect	31
	2.3.2.	Validation	31
	2.3.3.	Tunnel Size	32
	2.3.4.	Realizability	33
	2.3.5.	Grid Independence Study	34
	2.3.6.	Compressibility	35
	2.3.7.	Under-Relaxation Factors (URF's)	38
	2.3.8.	Delta Accumulated C_D nd C_L lots	39
2.4.	Summar	y/Conclusions	42
CHAPT AVI CUI	ER 3: ERAGED P SERIES	(ARTICLE 2) SCALE-RESOLVED AND TIME- SIMULATIONS OF THE FLOW OVER A NASCAR S RACECAR	52
3.1.	Introduc	tion	52
3.2.	Governir	ng Equations	58
	3.2.1.	The Improved Delayed Detached Eddy Simulation Mode	60
3.3.	Geometr	y and Mesh	62
3.4.	Physics \$	Setup	67
3.5.	RESULI	TS AND DISCUSSION	68
3.6.	Conclusi	ons	87

CHAPTER 4: (A AVERAGED ELLING AI FIELD OVE	ARTICLE 3) ON THE EFFECTIVENESS OF SCALE- O AND SCALE-RESOLVED TURBULENCE MOD- PPROACHES IN PREDICTING THE PRESSURE OR A NASCAR RACECAR	90
4.1. Introduc	ction	90
4.2. Methode	ology	96
4.2.1.	Governing Equations	96
4.2.2.	Geometry	102
4.2.3.	Computational Domain and Boundary Conditions	102
4.2.4.	Initialization	104
4.2.5.	Discretization	104
4.2.6.	Physics Setup	104
4.2.7.	Stopping Criteria and Data Averaging	105
4.2.8.	Computational Resources	105
4.3. Results	and Discussion	106
4.3.1.	Coefficient Plots	106
4.3.2.	Accumulated forces	109
4.3.3.	Pressure Probe Plots	115
4.4. Conclus	ions	133
CHAPTER 5: (FLOW ARO DYNAMIC I	ARTICLE 4) INSIGHT INTO THE TURBULENT UND AN IDEALIZED ROAD VEHICLE USING THE MODE DECOMPOSITION APPROACH	136
5.1. Introduc	ction	136
5.2. DMD E	quations	140

х

			xi
5.3.	Methodo	blogy	143
	5.3.1.	Solver Settings	143
	5.3.2.	Geometry, Domain, and Boundary Conditions	145
	5.3.3.	Discretization Scheme	147
	5.3.4.	DMD Workflow	147
	5.3.5.	Data Collection Strategy	148
5.4.	Results		149
	5.4.1.	CFD Validation	149
	5.4.2.	Application of DMD to a Canonical Flow Case	149
	5.4.3.	Ahmed Body Simulations	151
	5.4.4.	Future State Predictions using DMD	167
	5.4.5.	Computational Resources	169
5.5.	Conclusi	on	170
CHAPT	ER 6: CC	ONCLUSIONS	173
REFERI	ENCES		176

LIST OF TABLES

TABLE 2.1: Configurations of the racecar considered in this study.	31
TABLE 2.2: VWT sizes.	37
TABLE 3.1: Racecar ride-height and yaw configurations used in this study	69
TABLE 4.1: Configurations of the racecar that are considered in this study.	95
TABLE 5.1: Mean of all aerodynamic coefficients and RMS of their fluc- tuations as obtained from CFD simulation and DMD reconstruction.	168
TABLE 5.2: Mean of all aerodynamic coefficients and RMS of their fluc- tuations as obtained from CFD simulation and a future prediction by a DMD based ROM.	169
TABLE 5.3: Computational resources required by DMD and CFD	170

LIST OF FIGURES

FIGURE 1.1: Energy spectrum for a turbulent flow, where κ is the wavenumber (inverse of length scale) and $E(\kappa)$ is the turbulent kinetic energy.	5
FIGURE 1.2: Generations of the NASCAR Cup racecar (image source: https://www.autoweek.com/racing/nascar/a36107106/nascar-generations/	11
FIGURE 1.3: A journalists representation of the qualify- ing results of the 2022 Singapore F1 GP (image source: https://twitter.com/autosport/status/1576251439579598848	14
FIGURE 2.1: Gen-6 NASCAR with unique surfaces highlighted in orange.	21
FIGURE 2.2: Layout of Windshear Wind Tunnel; Image source: Windshear website.	22
FIGURE 2.3: The two ride height configurations used in this study.	22
FIGURE 2.4: The two yaw conditions, 0° and -3° considered in this study.	23
FIGURE 2.5: Computational domain.	27
FIGURE 2.6: Final mesh: Near car, @Y=Om center plane.	28
FIGURE 2.7: y + Distribution over the surface of the vehicle.	29
FIGURE 2.8: Histogram plot of wall $y+$ distribution.	29
FIGURE 2.9: Streamwise velocity profiles on $Z/SG = 1$ plane (a, top) with right wall as velocity inlet, Left wall as pressure outlet (b, bottom) Side walls as zero-gradient boundaries.	32
FIGURE 2.10: Cross-stream velocity profiles on $Z/SG =1$ plane (a, top) with right wall as velocity inlet, left wall as pressure outlet (b, bottom) Side walls as zero-gradient boundaries.	33
FIGURE 2.11: Headline C_D for validation study.	34
FIGURE 2.12: Headline C_L for validation study.	35

FIGURE 2.13: %Front for validation study.	36
FIGURE 2.14: Normalized streamwise velocity @ Z=SG plane.	36
FIGURE 2.15: Headline C_D for Configuration I for different VWT sizes.	37
FIGURE 2.16: Headline C_L for Configuration I for different VWT sizes.	38
FIGURE 2.17: Headline C_D for Configuration II for different realizability coefficient values.	39
FIGURE 2.18: Headline C_L for Configuration II for different realizability coefficient values.	40
FIGURE 2.19: Headline C_{LF} for Configuration II for different realizability coefficient values.	41
FIGURE 2.20: CD for different grid sizes.	42
FIGURE 2.21: CL for Configuration I at different grid sizes.	43
FIGURE 2.22: CL for Configuration II at different grid sizes.	44
FIGURE 2.23: CL for Configuration III at different grid sizes.	45
FIGURE 2.24: Headline C_D for all three Configurations for incompressible and compressible solvers.	46
FIGURE 2.25: Headline C_L for all three configurations for incompressible and compressible solvers.	47
FIGURE 2.26: Slower convergence before ramping up the URFs for Con- figuration II.	47
FIGURE 2.27: Faster convergence after ramping up the URFs for Config- uration II	48
FIGURE 2.28: Headline C_D for all three configurations with default and ramped URF's.	48
FIGURE 2.29: Headline C_L for all three Configurations with default and ramped URF's.	49

xiv

FIGURE 2.30: Delta accumulated C_D plot for Configuration II with different realizability coefficients.	49
FIGURE 2.31: Delta accumulated C_L plot for Configuration II with different realizability coefficients.	50
FIGURE 2.32: Delta accumulated C_L plot for Configuration I between compressible and incompressible solvers.	50
FIGURE 2.33: Delta accumulated C_L plot for Configuration III between compressible and incompressible solvers.	51
FIGURE 3.1: Gen-6 NASCAR with unique surfaces highlighted in orange	56
FIGURE 3.2: Layout of the Windshear Wind Tunnel. Image Source: Windshear website (https://www.windshearinc.com/; accessed on 18-Nov-2022)	57
FIGURE 3.4: Two yaw conditions, 0° and -3° , as considered in this study	58
FIGURE 3.3: Two ride height configurations used in this study; note that the splitter gap is expressed using a symbol 'SG' in subsequent analysis. (a) top: low SG, (b) bottom: high SG	58
FIGURE 3.5: Schematics of the computational domain	63
FIGURE 3.6: Mesh for the RANS case $@ = 0$ center plane	64
FIGURE 3.7: A histogram of wall $y+$ distribution over the surface of the vehicle obtained from the RANS simulation at zero-yaw	65
FIGURE 3.8: Wall y^+ Distribution Over the Surface of the Vehicle	65
FIGURE 3.9: Mean Turbulent Viscosity Ratio (TVR) $@y = 0$ plane	66
FIGURE 3.10: Mean Turbulent Viscosity Ratio (TVR) $@z = SG$ plane	67
FIGURE 3.11: Comparison of drag force predictions against wind tunnel measurements for all three configurations	69
FIGURE 3.12: Comparison of lift force predictions against wind tunnel measurements for all three configurations	70

XV

FIGURE 3.13: Comparison of %Front-Downforce predictions against wind tunnel measurements	70
FIGURE 3.14: Comparison of Front-lift coefficient predictions against wind tunnel measurements for all three configurations	71
FIGURE 3.15: Comparison of Rear-lift coefficient predictions against wind tunnel measurements for all three configurations	72
FIGURE 3.16: Delta accumulated force coefficients for the IDDES cases relative to the respective RANS cases for all three configurations; Top: Drag, Middle: Lift, Bottom: Side-Force.	73
FIGURE 3.17: Accumulated CD for configuration I; Solid line: IDDES, dotted line: <i>RANS</i> .	74
FIGURE 3.18: Predicted surface pressure distribution on the upper surface as obtained from the IDDES solver relative to the predictions of RANS solver.	75
FIGURE 3.19: Predicted surface pressure distribution as obtained from the IDDES solver relative to the predictions of RANS solve (bottom view).	75
FIGURE 3.20: Mean streamwise velocity normalized by the reference velocity $@ z = SG$ plane. Top: RANS; Middle: IDDES; bottom: Delta between IDDES and RANS.	77
FIGURE 3.21: Underbody skin friction coefficient (C_f) with Line Integral Convolutions (LIC) of the wall shear stress along <i>x</i> -direction; Top: RANS, bottom: IDDES.	78
FIGURE 3.22: Zoomed-in view (around the front splitter region) of un- derbody skin friction coefficient (C_f) with Line Integral Convolutions (LIC) of the wall shear stress along x-direction; Top: RANS, bottom: IDDES.	79
FIGURE 3.23: Upper body mean skin Friction coefficient (C_f) with Line Integral Convolutions (LIC) of the magnitude of wall shear stress; Top: RANS, bottom: IDDES	80
FIGURE 3.24: Mean Turbulent Kinetic Energy (TKE) @ y=0 Plane	81
FIGURE 3.25: Mean Turbulent Kinetic Energy (TKE)@z=SG plane	82

xvi

	xvii
FIGURE 3.26: Mean Specific Dissipation Rate (SDR) @ z=SG Plane	83
FIGURE 3.27: Streamlines in the near wake region; Top: RANS, Bottom: IDDES.	84
FIGURE 3.28: Normalized vorticity distribution on the $y = 0$ plane.	85
FIGURE 3.29: Normalized vorticity distribution on the $z = SG$ plane.	85
FIGURE 3.30: Power Spectral Density (PSD) of the force coefficients for all three configurations. Top: drag; Middle: down-force (negative lift); Bottom: Sideforce	89
FIGURE 4.1: % Δ of CD and CL between CFD and WT	107
FIGURE 4.2: % Δ of CLF and CLR between CFD and WT	108
FIGURE 4.3: % Δ of %_Front and L/D between CFD and WT	109
FIGURE 4.4: Plot of accumulated force coefficients from each configura- tion from RAS-C and DES-C solvers. Top: (a) Accumulated CD, middle: (b) Accumulated CL, and bottom: (c) Accumulated CS	111
FIGURE 4.5: Plot of differences in accumulated force coefficients from DES-C solver w.r.t RAS-C solver from each configuration. Top: (a) Delta accumulated CD, middle: (b) Delta accumulated CL, and bottom: (c) Delta accumulated CS	114
FIGURE 4.6: Plot of C_p distribution at pressure probes on the splitter	116
FIGURE 4.7: Plot of C_p distribution at pressure probes on the splitter extension panel	117
FIGURE 4.8: Plot of C_p distribution at pressure probes on the floor	118
FIGURE 4.9: Plot of C_p distribution at pressure probes on the LHS side skirts	119
FIGURE 4.10: Plot of C_p distribution at pressure probes on the RHS side skirts	120
FIGURE 4.11: Plot of C_p distribution at pressure probes on the fuel cell and rear crash structures	121

FIGURE 4.12: Plot of ${\cal C}_p$ distribution at pressure probes on the rear windshield	122
FIGURE 4.13: Plot of C_p distribution at pressure probes on the decklid	123
FIGURE 4.14: Plot of C_p distribution at pressure probes on the spoiler (a) top row, (b) middle row, and (c) bottom row	125
FIGURE 4.15: Plot of C_p distribution at pressure probes on the front fascia	127
FIGURE 4.16: Plot of C_p distribution at pressure probes on the hood	128
FIGURE 4.17: Plot of Mach number and mass flow rate distribution at the splitter and front face geometry (a) top: RAS-C, (b) bottom: DES-C	129
FIGURE 4.18: Plot of C_p distribution at pressure probes on the P1 (engine filter,) P2 (roof front), P3 (cabin filter), and P4 (rear fascia)	130
FIGURE 4.19: Plot of C_p distribution at pressure probes on the upper body centerline	131
FIGURE 4.20: Plot of C_p distribution at pressure probes on the vehicle's LHS	132
FIGURE 4.21: Plot of C_p distribution at pressure probes on the vehicle's RHS	133
FIGURE 5.1: Validation of the CFD simulation approach and methodol- ogy	149
FIGURE 5.2: Instantaneous Normalized streamwise velocity for flow past a 2D cylinder: (a) CFD prediction, (b) DMD re-construction and (c) the difference between (b) and (a).	150
FIGURE 5.3: Mean streamwise velocity for flow past a 2D cylinder: (a) CFD prediction, (b) DMD re-construction and (c) the difference be- tween (b) and (a).	152
FIGURE 5.4: RMS of the streamwise velocity fluctuations for flow past a 2D cylinder: (a) CFD prediction, (b) DMD reconstruction and (c) the difference between (b) and (a).	153

xviii

FIGURE 5.5: Mean of surface C_p as obtained using data sampled at 4 kHz: (a) from DMD; (b) from CFD; (c) difference between (a) and (b); (d) same as (c) but bottom-right isometric view	154
FIGURE 5.6: RMS of surface C_p fluctuations obtained using data sampled at 4 kHz: (a) from DMD; (b) from CFD; (c) difference between (a) and (b); (d) same as (c) but bottom-right isometric view	155
FIGURE 5.7: Forces and moments obtained from CFD calculations and DMD reconstructions, sampled at 4kHz; (a) drag, (b) lift, (c) side- force, (d) pitching moment, (e) rolling moment, and (f) yawing mo- ment	156
FIGURE 5.8: PSD of forces and moments obtained from CFD calculations and DMD reconstructions, sampled at 4kHz; (a) drag, (b) lift, (c) sideforce, (d) pitching moment, (e) rolling moment, and (f) yawing moment	157
FIGURE 5.9: Mean of surface C_p as obtained using data sampled at 10 kHz. (a) Mean of DMD, (b) Mean of CFD, (c and d) difference between mean of DMD and mean of CFD, where (d) bottom-right isometric view	159
FIGURE 5.10: RMS of the fluctuating component of surface C_p as obtained using data sampled at 10 kHz: (a) DMD; (b) CFD; (c) difference between (a) and (b); (d) same as (c), but bottom-right isometric view	160
FIGURE 5.11: Forces and Moments of CFD vs DMD, sampled at 10 kHz;coefficients of (a) drag, (b) lift, (c) sideforce, (d) pitching moment,(e) rolling moment, and (f) yawing moment	161
FIGURE 5.12: PSD of Forces and Moments of CFD vs DMD, sampled at 10 kHz; coefficients of (a) drag, (b) lift, (c) sideforce, (d) pitching moment, (e) rolling moment, and (f) yawing moment	162
FIGURE 5.13: Mean surface C_p distribution, with 10kHz sampling and custom filtering: (a) DMD; (b) CFD (c and d) difference between (a) and (b)	164
FIGURE 5.14: RMS of fluctuating component of surface C_p with 10kHz sampling and custom filtering: (a) DMD reconstruction, (b) CFD simulation, (c & d) the difference between (b) and (a).	165

xix

FIGURE 5.15: Forces and moments of CFD simulation verses DMD re- construction, sampled at 10kHz and obtained using custom filters	166
FIGURE 5.16: PSD of Forces and moments of CFD simulation verses DMD reconstruction, sampled at 10kHz and obtained using custom filters	167
FIGURE 5.17: Differences between future predictions of DMD and CFD data: (a) delta of force coefficients, (b) delta of moment coefficients; delta implies the difference between the DMD predictions and CFD values	169
FIGURE 5.18: PSD of future predictions of DMD relative to known CFD data; coefficients of (a) drag, (b) lift, (c) sideforce, (d) pitching moment, (e) rolling moment, and (f) yawing moment	170

 $\mathbf{X}\mathbf{X}$

LIST OF ABBREVIATIONS

- CFD Computational Fluid Dynamics
- CRFM Condenser, Refrigerator, and Fan Module
- DDES Delayed Detached Eddy Simulation
- DES Detached Eddy Simulation
- DMD Dynamic Mode Decomposition
- DNS Direct Numerical Simulation
- GIS Grid Induced Separation
- GV Ground Vehicle
- GVSC Ground Vehicles Systems Center
- IDDES Improved Delayed Detached Eddy Simulation
- LES Large Eddy Simulation
- NVH Noise, Vibration, and Harshness
- OEM Original Equipment Manufacturer
- PSD Power Spectral Density
- RANS Reynolds-Averaged Navier-Stokes
- SAS Scale Averaged Simulation
- SDR Specific Dissipation Rate
- SG Splitter Gap
- SGS Sub Grid Scale

SRANS Steady Reynolds-Averaged Navier-Stokes

- SRS Scale Resolved Simulation
- SST Shear Stress Transport
- SVD Singular Value Decomposition
- TD Time Dynamics
- TKE Turbulent Kinetic Energy
- TVR Turbulent Viscosity Ratio
- TVS Tangential Velocity Specification
- VWT Virtual Wind Tunnel
- WT Wind Tunnel

LIST OF SYMBOLS

$C_{(.)}$	Coefficient of pressure, force and moment
$\mathrm{d}t$	time step
F	X Force acting on the vehicle
f,g	generic functions
H, L and W	Height, Length, and the Width the vehicle geometry,
	respectively
i, and j	indexing variables
K	Turbulence Kinetic Energy per unit mass
Re	Reynolds number
X	Collected data matrix
x, y, z, and t	Stream-wise, lateral and vertical directions,
	and time respectively
U	Mean velocity
y^+	Non-dimensional wall distance
Subscripts:	
x, y, and z	Components in $x, y, $, and z , respectively
p, D, L, and S	Pressure, drag, lift, and side-force, respectively
PM, YM, and RM	Pitch, yaw and roll moments, respectively

CHAPTER 1: INTRODUCTION

Willy Rampf, former Technical Director of BMW-Sauber Formula 1 (F1) Race team, said that, "If you look at all the components that affect the performance of a Formula One car, aerodynamics represent by far the single most important factor" [1]. James Allison, currently the Chief Technical Officer of the Mercedes AMG F1 team, said that, "Aerodynamics is the start, middle and the end of whether a car is quick. Generally the car with the best aerodynamics wins the championship." [2]. These assessments highlight the vital role aerodynamics play as a performance differentiator between the race teams in competitive motorsports. Race teams thus spend significant resources towards the racecars' aerodynamic development and testing.

The three avenues exist for conducting aerodynamic development: (a) road tests, (b) Wind Tunnel (WT) tests, and (c) numerical simulation using Computational Fluid Dynamics (CFD). Each approach has its own pros and cons and race teams deploy a mix of all three tools for aerodynamic development while balancing costs, availability and dependability.

Road testing is seen as the ultimate litmus test of the racecars' performance. As said by Toto Wolff, the Team Principal of Mercedes AMG F1 team, "The stopwatch never lies. We get our first understanding of the competitive truth on Saturday in qualifying." [3]. Road tests provide a direct and instant measure of the racecars' performance. There are however certain challenges that are faced with road tests. Uncontrollable parameters such as weather, driver error, and noise induced by the test environment would cause significant unpredictability and non-repeatability of these tests. Additionally, only a finite amount of sensors can be carried by the onboard data acquisition system (DAQ). And finally, track time, and operation costs the test vehicle can be expensive.

Engineers thus resort to the use of simulated environments, such as Wind Tunnels and CFD, that are designed to replicate open-air conditions encountered by the racecars on track. Both the methods, of Wind Tunnels and CFD, are significantly more affordable to road tests and serve the added benefit of being available in the early design stages when an actual vehicle can simply be unavailable for road testing. Of these two approaches, Wind tunnel experiments are seen as a more realistic due to their use of a tangible vehicle model and its interaction with controlled air flow. The use of Wind Tunnels is restricted by their known deficiencies such as the need to correct for blockage ratios, need to simulate moving ground and operation costs.

Complementary to wind tunnel tests, CFD simulations can provide a significantly more detailed description of the flow field around the vehicle using non-intrusive measurements in a virtual environment, making it a cost-effective companion tool in the aerodynamic analysis. With proper discretization, use of appropriate boundary conditions, and physics models, CFD simulations can now predict the flow field with accuracy comparable to wind tunnel tests [4, 5, 6, 7]. CFD analyses are sometimes used to explain and rationalize observations from the wind-tunnel tests [8, 9, 10, 11]. CFD is thus a reliable and indispensable tool for racecar aerodynamic development.

1.1 Computational Fluid Dynamics (CFD)

The CFD methodology and governing equations are described in sections 2.2.1, 3.2, and 4.2.1. However, for benefit of the reader these are reproduced here.

In recent times, the rapid development of computational resources and enhanced capabilities of numerical simulation methods have enabled CFD to be a practical first-approximation tool. Additionally, professional race sanctioning bodies, like the Federation Internationale de l'Automobile (FIA) or The National Association for Stock Car Auto Racing (NASCAR), have restrictions on the number of wind-tunnel hours a team can spend in their racecar development. As such, the racing industry requires the development of reliable CFD methods with faster turnaround times.

1.1.1 Governing Equations

The fundamental basis of CFD are the Navier-Stokes (N-S) equations that are the governing equations for fluid flow. These equations are a mathematical representation of the principles of Conservation of Mass (also called the Continuity Equation), Conservation of Momentum, and Conservation of Energy. As vehicle aerodynamics are generally studied under isothermal flow conditions, with this assumption being valid for this dissertation also, the energy equation can be excluded. For a Newtonian flow these are given by equations 1.1, and 1.2 respectively, using Einstein notation where repeating index variables (i) or (j) imply summation over all possible values, e.g. (i = 1, 2, 3).

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \tag{1.1}$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_i} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$$
(1.2)

where, t represents time and the variables u_i , p, ρ , and τ_{ij} represent the timedependent values of the velocity in x_i direction, pressure, fluid density, and fluid viscous stress tensor, respectively. The viscous stress tensor, τ_{ij} , is defined as:

$$\tau_{ij} = 2\mu s_{ij} \tag{1.3}$$

where μ is the fluid kinematic viscosity and s_{ij} represents instantaneous rate of strain tensor defined as:

$$s_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{1.4}$$

The N-S equations completely and entirely describe the turbulent flow field from

the largest to the smallest scales of motion. The challenge faced by CFD comes from turbulence. Turbulent flows contain a mix of different coherent flow structures that span a range of length and time scales [12, 13, 14]. The largest structures are called integral scales, represented by l, and the smallest scales are called the Kolmogorov scales, represented by η . From dimensional analysis it can be shown that the largest and smallest scales for length, time, and velocity are related to the Reynolds number of the flow by the relationships shown in Equations 1.5, 1.6, and 1.7.

$$\frac{\eta}{l} \approx Re^{\frac{-3}{4}} \tag{1.5}$$

$$\frac{\tau}{t} \approx R e^{\frac{-1}{2}} \tag{1.6}$$

$$\frac{\upsilon}{V} \approx Re^{\frac{-1}{4}} \tag{1.7}$$

where the Reynolds number is a non-dimensional ratio of viscous to inertial forces given by Eq. 1.8

$$Re = \frac{\rho VD}{\mu} \tag{1.8}$$

The computational cost increases with the need to resolve the range of length scales. As an example, for a racecar application such as the one under consideration in this dissertation, the Kolmogorov length scale can be the order of $\eta = 0.01$ mm where l is taken as the vehicle length. Thus, turbulence models are broadly classified based upon the scales resolved and modeled by each approach. The four (4) broad approaches are, in the order of computational cost, Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), Detached Eddy Simulation (DES) and Reynolds-Averaged Navier-Stokes (RANS) simulation. Figure 1.1 shows the energy spectrum of a turbulent flow and the ranges of length scales, represented by their wavenumber (κ) , resolved by the turbulence models.



Figure 1.1: Energy spectrum for a turbulent flow, where κ is the wavenumber (inverse of length scale) and $E(\kappa)$ is the turbulent kinetic energy.

1.1.2 Direct Numerical Simulation (DNS)

In DNS, the entire range of spatio-temporal scales are resolved. Such a numerical solution requires that CFD resolve the spatial and temporal computational domain to the Kolmogorv scales. The computational resources required for such a simulation can be shown to scale with $\text{Re}^{11/4}$, and is impractical for an engineering application at high Reynolds number. For example, the flow field studied in this dissertation has a Reynolds number of 2×10^7 and would require about 110 exabytes of memory. Therefore we need to model the turbulent flow.

1.1.3 Improved Delayed Detached Eddy Simulation (IDDES)

LES simulations work by directly resolving the largest scales of motion and using a spatial low-pass filter on the smaller scales of motion. It is believed that LES is more accurate than RANS as fewer of the turbulent scales are modeled, however, the implementation of the LES approach is computationally expensive for automotive flows, and thus not discussed further here. A more practical hybrid RANS/LES approach of DES was proposed by Spalart et al. [15, 16, 17]. DES approaches use a switching function to use LES in the regions far from the wall and RANS in the boundary layer regions. The switch between the LES solver and RANS solver is achieved via the computation of two local parameters, a local turbulent length scale, l_T , and a local grid size, ℓ_{LES} .

$$\ell_T \equiv \frac{\sqrt{k}}{\omega} \tag{1.9}$$

$$\ell_{\rm LES} \equiv C_{\rm DES} \ \Delta_{\rm DES} \tag{1.10}$$

A limitation of this hybrid approach is that when the numerical value of ℓ_T and ℓ_{LES} reduces below a critical value, then the LES solver may be erroneously applied inside a boundary layer region. The effect of this local grid size can then be observed as a nonphysical separation being predicted and is thus known as Grid Induced Separation (GIS). GIS is thus a negative consequence of the switching function and is mitigated by modifying the switching function to include a delay based on the wall normal distance and local eddy viscosity [15]. This new approach with the modification to the switching function is called the Delayed DES or DDES. Another version of DES makes a further modification to the switching function between LES and RANS regions with the aim of providing further shielding to the boundary layer regions in high Reynolds number flows [17, 18]. This second modification is called the Improved DDES or

IDDES model which has been used for this dissertation. The IDDES model includes a Sub-Grid Scale (SGS) dependence on the wall-distance that further prevents LES modeling where the wall-distance is much smaller than the boundary-layer thickness.

$$\widetilde{\omega} = \frac{\sqrt{k}}{\ell_{\text{Hybrid}} f_{\beta^* \beta^*}} \tag{1.11}$$

where f_{β^*} is the free-shear modification factor, β^* is an SST $k - \omega$ model constant, and the parameter ℓ_{Hybrid} is defined as:

$$l_{\text{Hybrid}} = \tilde{f}_d \left(1 + f_e\right) \ell_{\text{RANS}} + \left(1 - \tilde{f}_d\right) C_{\text{DES}} \Delta_{\text{IDDES}}$$
(1.12)

1.1.4 Reynolds-Averaged Navier Stokes (RANS)

(Much of this section is reproduced from sections 4.2.1.1) The Reynolds Averaged Navier-Stokes (RANS) approach is a commonly used method for solving an engineering problem using CFD. In this approach Reynolds decomposition is used to decompose the instantaneous velocity and pressure fields into mean and fluctuating components, mathematically expressed in the form $a_i = A_i + a'_i$, and followed by ensemble-averaging the original N-S equations. Thus, as an example, in this convention, u_i , U_i , and u'_i represent the time-dependent instantaneous, time averaged, and time-dependent fluctuating parts of the velocity component in *i*-direction respectively. The RANS equations are then expressed by equations 1.13 and 1.14. Here, we describe the turbulent flow statistically in terms of the mean velocity field $U_i(\mathbf{x}, t)$ and mean rate of strain $S_{ij}(\mathbf{x}, t)$ instead of the instantaneous velocity field $u_i(\mathbf{x}, t)$ and instantaneously rate of strain field $s_{ij}(\mathbf{x}, t)$, respectively. These equations are commonly referred to as the Unsteady Reynolds Averaged Navier-Stokes (URANS) equations.

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1.13}$$

$$\frac{\partial U_i}{\partial t} + \frac{\partial U_j U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left(2\mu S_{ij} - \rho \overline{u'_i u'_j} \right)$$
(1.14)

where the terms, $-\rho \overline{u'_i u'_j}$, which is a symmetric tensor, are known as the Reynolds stresses. These are six additional terms that are challenge introduced into the system of equations as a consequence of the emerges from the Reynolds averaging process. This presents the classical closure problem in fluid mechanics in the fact that the six new independent terms now give us a total of 10 variables to determine using 4 equations. This is often resolved using the turbulent-viscosity hypothesis by Boussinesq in 1877 (see equation 1.15). As per Boussinesq's hypothesis, a relationship is needed between the turbulent stresses and the mean rate of strain, similar to the viscous stress relationship as shown in equation 1.3. However, in this case the constant of proportionality is a fictitious flow variable, called the *turbulent eddy viscosity*, ν_t , shown in equation 1.15.

$$\overline{u'_i u'_j} = \frac{2}{3} k \delta_{ij} - \nu_t \left(\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i} \right)$$
(1.15)

where k is turbulence kinetic energy per unit mass, $k \equiv (1/2) \ \overline{u'_i u'_i}$, and δ_{ij} is Kronecker delta. The determination of this flow variable ν_t is the central element of turbulence modeling approach. All the various eddy viscosity based turbulence models found in literature differ primarily in the way they estimate ν_t . All of the modern turbulence modelling approaches solve additional transport equation(s) to determine ν_t ; this type of modelling approaches are classified on the basis of the number of transports equations involved, and what transport variables are used in the modelled equations. For example, a one-equation turbulence model will involve the solution of one additional transport equations, and a two-equation $k-\omega$ modelling approach will involve transports of turbulence kinetic energy (k), and specific rate of turbulence kinetic energy dissipation (ω) . This dissertation uses the SST Menter $k - \omega$ (SST) [19, 20] turbulence model in all four articles. An interested reader is referred to Zhang et al. [21] and the original articles of Menter and coworkers [22, 20, 19] for all relevant details.

1.1.5 CFD Tools

A CFD pipeline typically consists of three (3) stages: (a) pre-processing, (b) processing (i.e. running the iterative solver), and (c) post-processing. The initial preprocessing step included cleaning the CAD model to obtain a valid "water-tight" surface for CFD processing. This was achieved by using a commercial pre-processing software ANSA version 20.1.2, by Beta CAE Systems. The CAD was manually cleaned with care taken to ensure that all intricate geometry details were retained. The cleaned CAD surface was then surface meshed, resulting in 13 million triangles, to retain all the geometric features when exporting to Star-CCM+. The surface mesh was exported to Star-CCM+ via NASTRAN format where further pre-processing was performed. This included the surface re-meshing, volume meshing, and solver setup. Then simulation was run using UNC Charlotte HPC and post-processing operations were conducted using a combination of in-built tools of Star-CCM+ and custom scripts written in MATLAB.

All the CFD simulations presented in this dissertation were performed using a commercial finite volume based CFD code of Star-CCM+ version 2020.2.1. The commercial FVM based CFD codes are capable of handling the complex CAD geometries encountered in automotive and motorsport applications. Using Star-CCM+, the conservation laws were solved iteratively in their integral form using the Eulerian approach. The continuity and momentum equations were solved using a segregated solver with the pressure and velocity coupling attained by a predictor-corrector method of the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) solver. An interested reader is directed to the Star-CCM+ version 15.04 user manual for further details. Given that the authors have previously experienced a significant variation in CFD predictions because of domain decomposition [23], care was taken to maintain the same parallelization scheme throughout this study. All simulations were run on UNC Charlotte High Performance Computing clusters using 144 processors across 3 nodes having 48 processors each.

1.2 Background

NASCAR is the largest race sanctioning and operating body for auto racing in North America, and the NASCAR Cup Series is the top-tier automotive competition series within NASCAR. This dissertation is designed to gain further insights into the aerodynamics of the NASCAR Cup racecar and idealized road vehicles using scaleresolved and scale-averaged CFD simulations. The NASCAR Cup series is currently on the 7^{th} generation of the racecar evolution. The 7 generations of NASCAR Cup racecar are shown in Figure 1.2 below.

The Generation 6 car, also called Gen-6, is the style that was used between 2013 – 2021. The vehicle model used in this dissertation is a detailed full-scale NASCAR Gen-6 Cup car model. The model features most of the production parts including the engine, drive-train, chassis, cabin interior, suspension, wheels, radiator, and gear cooler. The characteristic aerodynamic features of this car include: a front splitter with underbody splitter extension panel, a rear spoiler, very low groundclearance side-skirts, front by-pass ducts, a camera pod, radio communication and GPS antennas, NACA ducts for cabin and driveline cooling, roof-rails and sharkfin. These features make the aerodynamics of a NASCAR significantly different to generic automotive flows. There are some CFD studies with a NASCAR geometry available in literature and are summarized below. Very early numerical experiments using CFD as a tool focused on understanding the car performance in different conditions and were limited to simplified, and now outdated Gen-4 and Gen-5, NASCAR geometries. Brzustowicz (2002) et al. of Daimler Chrysler presented an experimental and numerical investigation of a Gen-4 car. With moving ground simulation they



Figure 1.2: Generations of the NASCAR Cup racecar (image source: https://www.autoweek.com/racing/nascar/a36107106/nascar-generations/

reported between 20 - 40% discrepancy between CFD and wind tunnel predictions [24]. Duncan & Golsch (General Motors) (2004) presented a CFD investigation using a Gen-4 car from another manufacturer. They studied the surface pressure patterns and turbulent flow features over a range of air velocities. They however do not present any validation data for their numerical simulations. [25]. Singh from General Motors (2008) presented a CFD investigation using a simplified Gen-5 geometry. They studied the effects of moving ground, wing angle and splitter length. This study also does not present any validation data [26]. All of these researchers were focused on the effects of various geometry changes to the aerodynamic characteristics.

While more recent work by Fu et al. [27, 28, 29, 30, 31, 32] used a detailed Gen-6 geometry. Their data is validated with AeroDyn data, a closed-jet, open-return wind tunnel, at a single operating condition. Aerodyn also uses boundary layer suction and tangentially blown jets for moving ground simulation. The work of Fu et al. [27, 28, 29, 30, 31, 32] was focused on effects of the choice of RANS turbulence models, the sensitivity of the RANS model closure coefficients, and upstream turbulence parameters of intensity and length scale on the flow predictions. The most recent work by Jacuzzi et al. [10, 11] also used a detailed Gen-6 geometry and is validated with AeroDyn data. Their work focused on the effects of passively blown ducts on trailing vehicle drag and yaw moment stability.

The primary objective of most of these investigations was to use CFD as a tool to study the racecar performance under various geometric configurations, such as design changes to the various aerodynamic components, or different settings of ground simulations and wind velocities. Only the publications of Fu et al. [27, 28, 29, 30, 31, 32] evaluated the performance of the CFD simulations themselves by focusing on the effects of turbulence modeling. All these studies used a variant of a RANS turbulence model.

Additionally in recent times, aerodynamic research is being conducted on adaptive driving conditions such as vehicle platooning [33, 10]. Other adaptive conditions encountered include changing sidewall proximity, crosswind changes, and ride height changes [8, 28, 11]. It is thus desired to apply a control mechanism to a moving vehicle to obtain desired aerodynamic characteristics [34]. To apply such a control signal to a moving vehicle, we need the ability to predict the future state of aerodynamic forces and moments. A Reduced Order Model (ROM) can be used to make future state predictions of the aerodynamic flow field ([35]). For adaptive systems, the future state predictions can then be coupled with a control input to obtain the desired performance characteristics ([36]). One such ROM, the Dynamic Mode Decomposition (DMD) which is a data-driven linearization tool, has shown the ability to obtain the modal decomposition and flow field reconstruction of low Reynolds number flows [37, 38, 39].

1.3 Motivation

Professional race sanctioning bodies, like the FIA and NASCAR, have recently introduced restrictions on the amount of CPU hours and CFD items a team can spend in their racecar development. In this new era of restricted CFD development, Mattia Binotto (former Team Principal of Scuderia Ferrari F1 team) said, "Normally a week of development is less than 0.1 seconds per lap" [40]. And similarly, Gary Anderson (former Technical Director of both Lotus F1 and Jaguar F1 teams) said, "1% more usable downforce is roughly one tenth of a second (per lap)" [41]. To provide context to these numbers, one can go through the recent race results published on the governing bodies' websites and find that the difference in laptime between the fastest and slowest cars is generally within 4%, which for a reference 90 second lap amounts to 3.6 seconds. This means for a 20 (F1) to 40 (NASCAR) racecar field, the average difference between consecutive racecars is about one to two tenths of a second per lap. This is visualized by a journalists representation of the qualifying results of the 2022 Singapore F1 Grand Prix (GP) shown in Figure 1.3.

As such, the racing industry requires the development of accurate and informationrich CFD methods to make the most of the limited CPU time allotted to them. While the works of Fu et al. [27, 28, 29, 30, 31, 32] focused on turbulence modeling effects, a study of the effects of boundary conditions and solver parameters on the aerodynamic predictions of flow around a NASCAR Cup racecar was absent.

As seen in Figure 1.1, the DES approach requires lesser modelling of the energy spectrum compared to a RANS based approach. Thus, the DES approach is believed to be more accurate, containing a wider range of the resolved turbulent scales relative to the RANS based approach. Recent publications using generic automotive


Figure 1.3: A journalists representation of the qualifying results of the 2022 Singapore F1 GP (image source: https://twitter.com/autosport/status/1576251439579598848

geometries such as the Ahmed body geometry [42], the DrivAer geometry [5] and a passenger car [21] have shown promising results using variants of the DES turbulence modelling approach. A similar investigation using the NASCAR Cup racecar geometry was absent.

Lastly, a race car is tuned to behave a certain way on track when in clean air [11, 43, 44, 45, 46]. When the operating condition changes, such as ride height or crosswind angle, the vehicle undergoes adaptive driving conditions. Our IDDES study on the NASCAR Cup racecar identified spectral analysis of the aerodynamic forces

and moments as a potential tool for racecar aerodynamic development [47]. We thus evolved into the use of a more advanced modal decomposition tool, DMD, for the prediction of the future state of aerodynamic forces and moments given a change in input parameters.

1.4 Objectives

The main goal of this dissertation is to gain insight into the aerodynamics of race and idealized road vehicles using scale-resolved and scale-averaged CFD simulations. This dissertation consists of four technical articles. The first three articles center on a Gen-6 NASCAR Cup racecar using scale-resolved and scale-averaged CFD simulations. Each of these three articles assesses in detail the CFD performance using three cases of the NASCAR Cup racecar that include two ride heights and two yaw angles. Validation data for all three cases is available in the form of wind tunnel data from Windshear, an open-jet closed-return type wind tunnel that has a rolling belt and boundary layer suction for moving ground simulation. Thus, this wind tunnel configuration resembles open-air conditions more closely than the AeroDyn wind tunnel. The fourth article explores the development of a ROM using DMD for the prediction of the future state of aerodynamic forces and moments.

The first article explores the effects of solver parameters and boundary conditions on the RANS CFD flow prediction over a Gen-6 NASCAR racecar. The parameters studied include the Virtual Wind Tunnel (VWT) size for open road simulation, the effect of the Realizability coefficient as used in the SST turbulence modelling calculations, the effect of the compressibility solver, the effect of the Under-Relaxation Factors (URF's) and boundary types for crosswind simulation.

The second article studies the feasibility and prediction veracity of transient SRS approaches like IDDES for the aerodynamic characterization of stock racecar. This article presents an in-depth analysis comparing and contrasting the CFD simulation of the flow field around a Gen-6 NASCAR racecar as obtained using the popular and commonly used SAS RANS approach (c.f. [46]) and a potentially more accurate transient IDDES approach. The primary objectives were to investigate the predictive difference between these two methods and to understand the root causes of these differences.

Significantly different flow field predictions were obtained from SAS and SRS approaches from the first two articles (Misar, A.S., and Uddin, M (2022) [46], Misar et al. (2023) [47]). The third article sheds some light as to which flow features around the vehicle may be contributing the most to those discrepancies, directing future studies to those specific areas likely to produce improved simulation accuracy. This was done by probing the correlation between the static pressure data obtained from surface-mounted probes from the wind tunnel experiments of a racecar to the predictions obtained from the different CFD simulations of the same geometry.

The objective of the fourth article was to study the performance of the DMD algorithm when attempting to predict the future flow field of a moving Ground Vehicle (GV) at a high Reynolds number. In this article, we applied the DMD algorithm to a high Reynolds number, hugely separated bluff body flow. We had to modify the DMD algorithm as the version as available in published literature failed to capture the frequency response of the high Reynolds number flow. Our modified DMD algorithm was validated using reconstructions and future predictions of aerodynamic coefficients of forces and moments. An idealized road vehicle geometry was used for this developmental work as experimental data for pressure and velocity profile of the NASCAR Cup racecar is not available. Also, the analysis of the modal decomposition of a complex geometry is complicated and the multiple simulations needed are prohibitive when using a NASCAR Cup racecar. The extension of the developed ROM to the NASCAR Cup racecar is left for a future study.

1.5 Dissertation Outline

This dissertation is comprised of four articles and organized in the following manner,

- Chapter 2 (Article 1) A. S. Misar and M. Uddin, "Effects of solver parameters and boundary conditions on RANS CFD flow predictions over a Gen-6 NASCAR racecar," Tech. Rep. 2022-01-0372, SAE WCX Technical Paper, United States, 2022. (Published by SAE International [46])
- Chapter 3 (Article 2) A. S. Misar, M. Uddin, T. Pandaleon, and J. Wilson, "Scaleresolved and time-averaged simulations of the flow over a NASCAR cup series racecar," Tech. Rep. 2023-01-0735, SAE Technical Paper, United States, 2023. (Published by SAE International [47])
- Chapter 4 (Article 3) A. S. Misar, P. Davis, and M. Uddin, "On the Effectiveness of Scale-Averaged and Scale-Resolved Turbulence Modelling Approaches in Predicting the Pressure Field over a NASCAR Racecar" (Submitted for publication to *Fluids*, awaiting editorial response)
- Chapter 5 (Article 4) A. S. Misar, N. Tison, V. Korivi, and M. Uddin "On the Application of the DMD Approach to the High Reynolds Number Turbulent Flow Around an Idealized Road Vehicle" (To be submitted for publication to Vehicles)

The summarized conclusions of the dissertation are given in Chapter 6

CHAPTER 2: (ARTICLE 1) EFFECTS OF SOLVER PARAMETERS AND BOUNDARY CONDITIONS ON RANS CFD FLOW PREDICTIONS OVER A GEN-6 NASCAR RACECAR

2.1 Introduction

In the auto-racing, aerodynamics of the racecar is the single most important contributor to the vehicle's on-track performance [1]. Race teams allocate significant resources towards the vehicle's aerodynamic development and testing. Wind tunnel testing and Computational Fluid Dynamics (CFD) are the two main avenues to perform aerodynamic development. Wind tunnel tests are often referenced due to their use of a full-size physical model. CFD is a reliable and indispensable tool for racecar aerodynamic development. Complementary to wind tunnel tests, CFD simulations can provide a significantly more detailed description of the flow field around the vehicle using non-intrusive measurements in a virtual environment, making it a cost-effective companion tool in the aerodynamic analysis [25, 26, 24]. With proper discretization, use of appropriate boundary conditions, and physics models, CFD simulations can now predict the flow field with accuracy comparable to wind tunnel tests [4, 5, 6, 7]. CFD analyses are sometimes used to explain and rationalize observations from the wind-tunnel tests [8, 9, 10, 11]. However, professional race sanctioning bodies, like the Federation Internationale de l'Automobile (FIA) or The National Association for Stock Car Auto Racing (NASCAR), have restrictions on the number of wind-tunnel hours and on the maximum CPU clock time a team can spend in their racecar development. As such, the racing industry requires the development of reliable CFD methods with faster turnaround times.

With advances in available computing power, Large Eddy Simulation (LES) and

Detached Eddy Simulation (DES) have shown greater accuracy in some industrial CFD applications due to their ability to provide better predictions of the flow fields [4, 5, 6, 7]. However, the Reynolds-averaged Navier-Stokes equation (RANS) approach is still widely used as the first approximation tool in the racing industry because of its relatively quick turnaround time [25, 26, 24, 10, 11, 27, 28, 29, 30, 31, 32]. A study using Delayed Detached Eddy Simulation (DDES) took approximately 15 times longer for a passenger car simulation than an equivalent RANS would require [48]. It has been has shown that a suitably designed RANS CFD simulation is a cost-effective approach for developing the external aerodynamics of a full-sized DrivAer model [49]. Thus, the successful implementation of a suitable RANS model can represent an order of magnitude saving of computational efforts.

The accuracy of flow field prediction by RANS simulations is heavily influenced by the chosen turbulence model [50, 51], with this effect holding true for vehicle external aerodynamic predictions [4, 5, 27]. Turbulence modeling literature is plentiful for both internal and external flows involving simple geometries, such as channel flows [52] and flows past bluff bodies [53]. A more thorough review may be found in published books and review articles [54, 13, 55, 56, 57]. In terms of automotive applications, a vast majority of the published validation works for turbulence models are confined to simple automotive models, like Ahmed body or DrivAer model [58, 59]. Based upon prior experience with a NASCAR geometry [27, 28, 29, 30, 31, 32] this paper uses the Shear Stress Transport (SST) k-w turbulence model developed by Menter [22, 20, 19].

Race vehicles are aerodynamically unique as these include many aerodynamic devices specifically designed for high Downforce-to-drag ratio [48]. The Generation 6 NASCAR, a.k.a Gen-6, that is in use since 2013 has many such unique features. As can be seen in Figure 2.1, the characteristic aerodynamic surfaces of this car include a front splitter, a rear spoiler, very low ground-clearance side-skirts, front bypass ducts, a camera pod, communication antennae, NACA ducts for cabin cooling, roof-rails and a shark-fin. The front splitter usually operates at a very low-ground clearance. Where a typical passenger vehicle normally produces a small lift with a lift-to-drag ratio of about 0.3 [49, 59], a racecar must achieve high-speed cornering performance, typically with down-force (or a negative lift) and having a lift to drag ratio of 2.0 or larger [25, 26, 24, 10, 11, 27, 28, 29, 30, 31, 32]. Also, due to dynamic on-track conditions experienced by a racecar on account of high operating speeds, vehicle ride-height and orientation changes, the aerodynamic behaviors significantly change between cornering and straight-line driving conditions. A racecar on corner entry is subject to braking while on corner-exit is subjected to a high longitudinal acceleration. This causes the pitch of the racecar to change in what are called as dive-and-squat angles. The vehicle also experiences a yaw during corner entry, apex, and exit. Thus, the vehicle's aerodynamic characteristics must be analyzed under an envelope of yaw and pitch orientations. Existing literature covers yaw and pitch effects on generalized car shapes, such as the Ahmed body and DrivAer body, both experimentally and numerically [60, 61, 62, 63, 64, 65, 66]. Some limited work is also published for performance cars focusing on specific areas such as wings or using simplified geometries [67, 68, 69]. Very early numerical experiments using CFD as a tool focused on understanding the car performance in different conditions and were limited to very simplified, and now outdated Gen-4 and Gen-5, NASCAR geometries due to the computational limits at the time [25, 26, 24]. While more recent work by Fu et al [27, 28, 29, 30, 31, 32] uses detailed Gen-6 geometry, their data is validated with AeroDyn data, a closed-jet, open-return wind tunnel, at a single operating condition. Also, that wind tunnel used boundary layer suction for ground-plane emulation. The work of Fu et al focusses more on effects of turbulence parameters on the flow predictions [27, 28, 29, 30, 31, 32].



Figure 2.1: Gen-6 NASCAR with unique surfaces highlighted in orange.

In this paper, all CFD simulations are validated against data from Windshear, an open-jet closed-return type wind tunnel with a moving-belt for rolling road simulation that is particularly designed for road-ready motorsport vehicles [70]. Wind tunnel data for 3 operating configurations are available covering high and low ride height as well as zero and -3.0° yaw angles. This can be seen in Figures 2.2, 2.3, and 2.4. This paper explores the effects of solver parameters, Virtual Wind Tunnel (VWT) size for open road simulation, effect of the Realizability coefficient as used in the SST turbulence modelling calculations, effect of the compressible solver and boundary conditions for crosswind simulation on the RANS CFD flow prediction over a Gen-6 NASCAR racecar. Further details of the simulation setup and computational methodology will be addressed in subsequent sections.



Figure 2.2: Layout of Windshear Wind Tunnel; Image source: Windshear website.



Figure 2.3: The two ride height configurations used in this study.

2.2 Computational Method & Simulation Setup

The Navier-Stokes (NS) equations for instantaneous velocity and pressure fields can be decomposed into mean and fluctuating components using RANS equations for an incompressible turbulent flow. The resulting mass and momentum equations,



Figure 2.4: The two yaw conditions, 0° and -3° considered in this study.

together known as the Navier-Stokes equations for continuity and momentum, are represented in Einstein indicial notation by Equations 2.1 and 2.2:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{2.1}$$

$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} \left[-\bar{p} \delta_{ij} + \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j} \right]$$
(2.2)

These equations for the mean quantities are essentially identical to the original instantaneous equations but have the additional term $-\rho u'_i u'_j$ in the momentum transport equation. This term constitutes the Reynolds stress tensor. It has a total of nine components, six of which are independent. These six additional stress terms make the number of unknowns larger than the number of available mathematical equations to solve them. This gives rise to what is known as the turbulence closure problem. To account for this, fluid-dynamists resort to what is traditionally known as turbulence modeling. Two approaches can be used to model the Reynolds stresses in terms of mean flow quantities and to provide closure of the governing equations: (i) eddy viscosity models, and (ii) the Reynolds stress transport models. Eddy viscosity models use the concept of a turbulent viscosity μ_t , and model the turbulent stresses in a way analogous to laminar flows. Most of the turbulence models currently used by CFD engineers in engineering applications are based off the eddy viscosity models, of which, $\mathbf{k}-\varepsilon$ and $\mathbf{k}-\omega$ variants are the most widely used ones.

2.2.1 Turbulence Model

The Shear-Stress Transport (SST) $k - \omega$ The $k - \omega$ model replaces the dissipation rate ε in $k - \varepsilon$ model with a specific dissipation rate ω which is defined as $\omega \approx \frac{\varepsilon}{k}$. Compared to the standard $k - \omega$, the SST $k - \omega$ turbulence model, proposed by Menter and his colleagues [22, 20, 19], has an additional non-conservative cross-diffusion term containing $\Delta k \cdot \Delta \omega$ in the ω transport equation. By using a blending function that includes the cross-diffusion term far from the wall but not near the wall, the SST model effectively blends a $k - \varepsilon$ model in the far-field free stream while keeping the $k - \omega$ boundary layer calculation advantages in the near wall region. The expressions for the eddy viscosity μ_t , and the transport equations are given in Equations 2.3 to 2.10.

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_i} = \hat{P}_k - \beta^* k \omega + \frac{\partial}{\partial x_i} \left[\left(v + \sigma_k v_t \frac{\partial k}{\partial x_i} \right) \right]$$
(2.3)

$$\frac{\partial\omega}{\partial t} + U_j \frac{\partial\omega}{\partial x_i} = \alpha \frac{1}{\mu_t} \hat{P}_k - \beta \omega^2 + \frac{\partial}{\partial x_i} \left[\left(v + \sigma_k v_t \frac{\partial k}{\partial x_i} \right) \right]
+ 2 \left(1 - F_1 \right) \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial\omega}{\partial x_i}$$
(2.4)

$$v_{\rm t} = \frac{a_1 k}{\max\left(a_1 \omega, SF_2\right)} \tag{2.5}$$

$$S = \sqrt{2S_{ij}S_{ij}} \tag{2.6}$$

$$P_{k} = v_{t} \frac{\partial U_{i}}{\partial x_{j}} \left(\frac{\partial U_{i}}{\partial x_{j}} + \frac{\partial U_{j}}{\partial x_{i}} \right) \rightarrow \hat{P}_{k} = \min\left(P_{k}, 10\beta^{*}k\omega\right)$$
(2.7)

$$F_{1} = \tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^{*}\omega y}, \frac{500v}{y^{2}\omega}\right), \frac{4\rho\sigma_{\omega^{2}}k}{CD_{k\omega}y^{2}}\right]\right\}^{4}\right\}$$
(2.8)

$$F_2 = \tanh\left[\left[\max\left(\frac{2\sqrt{k}}{\beta^*\omega y}, \frac{500v}{y^2\omega}\right)\right]^2\right]$$
(2.9)

$$CD_{k\omega} = \max\left(2\rho\sigma_{\omega^2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i}, 10^{-10}\right)$$
(2.10)

Where $\alpha, \beta, \beta^*, \sigma_k, \sigma_\omega$ and $\sigma_{\omega 2}$ are closure coefficients that are computed by using blending functions F_1, F_2 and corresponding constants of $k - \varepsilon$ and $k - \omega$ models via $\alpha = \alpha 1F1 + \alpha 2(1 - F1)$, etc. The *a*1, constant in ν_t is initially set to 0.31 per the STAR-CCM+ version 2020.2.1 user manual. A production limiter is used in the SST model to prevent the build-up of turbulence in stagnation regions.

2.2.2 Geometry and Mesh

The CAD geometry was imported into ANSA v15 and cleaned of all surface tessellation errors. Sufficient care was taken to retain as much of the original surface description as possible. Thus, the geometry consists of the external aerodynamic surface, cabin interior flow, NACA cooling ducts, underhood flow with detailed powertrain and exhaust assemblies, and porous regions for the radiator and gear cooler. The fully detailed and error-free final surface consisting of 13 million triangles was imported into Star-CCM+ for CFD simulation.

The reader is reminded that wind tunnel experiments are also a type of simulation. The open-jet configuration of the WindShear wind tunnel also attempts to simulate openroad conditions [70]. Thus, a large computational domain is required to perform an open-road CFD simulation for vehicle aerodynamics. At the start, the test model was placed in a large VWT, shown in Figure 2.5, with dimensions of $51L \times 50W \times$ 50H, where L, W, and H are the characteristic length, width, and height of the test geometry; the inlet and outlet were 10 L upstream and 40L downstream, respectively. This large wind tunnel was used to avoid the influence of blockage ratio and numerical pressure waves which can occur at boundaries. The dimensions were chosen from prior experience and StarCCM+ version 2020.2.1 best-practices recommendations. After the Tunnel Size Study, the dimensions were upped to $211 \text{ L} \times 200 \text{ W} \times 200 \text{ H}$ for all remaining simulations; with the inlet and outlet placed 50 L upstream and 160 L downstream, respectively. The inlet was given a velocity of 67.056 m/s(150 mph), a turbulence intensity of 1.0%, and a turbulent length scale of 0.01 m. A pressure outlet was placed at the opposite end of the tunnel and the sides of the tunnel were given a zero-gradient condition. For the crosswind simulations this configuration was modified to change one side of the tunnel from a zero-gradient to a velocity inlet and the other side to a pressure outlet. The velocity inlets then had their velocity specified in x and y components to give the flow the proper angle to simulate the desired yaw, or crosswind to be more precise. To correlate to the moving ground simulations, the no-slip ground was given a tangential velocity corresponding to the given inlet velocity. The wheel rotation was modelled using a Tangential Velocity Specification (TVS) with a small vertical wall to simulate the tire-ground contact patch to retain simplicity and cost-effectiveness of the model [29, 71]. The discretization scheme used for porous media modelling includes porous baffles to simulate the front and inner grilles of the radiator ducting. Thus, the radiator consists of 3 regions, the primary cooling duct, the secondary cooling duct, and the radiator core itself. Porous media modelling is tuned against the mass flow rate measurements from the wind tunnel.

To discretize the domain, an unstructured hexahedral cell mesh was created using Star-CCM's "trimmed cell" mesher. This meshing algorithm seeks to create cells which are some multiple of 2^n greater/smaller than the selected base-size. To properly take advantage of this behavior and correctly resolve the near field flow, nine different volume sources were used around the body to control cell refinement, each one being



Figure 2.5: Computational domain.

half the size of the previous. Eight more volume sources were used near the vehicle to capture the flow around special areas of interest such as the spoiler, splitter, gear cooler duct, wheels, cabin interior, and the underhood regions. For example, the narrow but expanding gap between the splitter and the ground boundaries is resolved with 3 different volume controls having anisotropic cells. The vertical dimensions have the finest mesh with $\Delta z = 1.5$, and 3.0, respectively. Figure 2.6 shows the final mesh around the body.

To properly resolve the high gradient boundary layer on the upper skin, 16 prism layer cells with a total thickness of 15.6 mm were placed along the body surface. The prism layer first node height was specified to be 0.01 mm. 18 more surface controls were used to similarly define appropriate prism layer values to the various surfaces around the car. For example, the rolling road boundary had 4 prism layer cells with a total thickness of 2.4 mm and the prism layer first node height was specified to be 0.1



Figure 2.6: Final mesh: Near car, @Y=Om center plane.

mm. The bottom surface of the splitter had 16 prism layer cells with a total thickness of 3.6 mm and the prism layer first node height was specified to be 0.005 mm. This gave a y^+ distribution as shown in Figures 2.7 and 2.8. Using these parameters, the y^+ value for 86% of the body cells were seen to have a value less than 1.11% cells had $1 < y^+ < 2$, these were located at regions of locally accelerated flow such as around the headlight edges and the A-pillar. The majority of cells with $y^+ > 2$ were found at areas of sharp corners where the prism layer mesher collapsed. The final mesh used a base size of 24 mm and consisted of 128 million cells.

2.2.3 Physics Setup

The simulations performed in this study were done using a commercial finite volume solver CFD package, Star-CCM+ version 2020.2.1. Each simulation unless specified was performed using a segregated incompressible solver on an unstructured grid using the SIMPLE solver.



Figure 2.7: y+ Distribution over the surface of the vehicle.



Figure 2.8: Histogram plot of wall y+ distribution.

2.2.4 Wall Treatment

A two-layer all y^+ wall treatment was applied to the simulation to ensure reasonably accurate boundary layer calculation even in some complex locations of the geometry where the y^+ was not sufficiently small.

The $k-\omega$ SST turbulence model was used along with its default closure coefficients. This model has been shown to have good prediction capability for automotive based flows. It has also been shown to respond well to closure coefficient tuning, which may be explored for future investigations.

2.2.5 Solver and Convergence

Given that the authors have previously experienced a significant variation in CFD predictions because of domain decomposition [23], care was taken to maintain the same parallelization scheme throughout this study. All simulations were run on UNC Charlotte High Performance Computing clusters using 144 processors across 3 nodes having 48 processors each. The simulations were set to run for 10,000 iterations, more than enough to ensure convergence. The authors do note that the convergence was monitored using the aerodynamic force coefficients and was seen to occur at roughly 6,000 iterations. Subsequently, in this paper, all the results presented are from an averaging window of 6,000 - 10,000 iterations.

2.3 Results and Discussion

Table 2.1 lists the three configurations of the Gen-6 NASCAR considered in this study. The authors extend thanks to Chip Ganassi Racing (CGR) for providing the Gen-6 NASCAR Cup Racecar CAD Model and all relevant wind-tunnel experimental data. These are CGR's proprietary data obtained through an NDA. The reader must note that for the sake of protecting CGR's confidential data, all force and moment coefficients presented in this paper are normalized by an arbitrary reference area. The results for each CFD simulation have been validated against 14 data-points from the wind tunnel data including all 6 force and moment coefficients, Front and Rear downforce, % Front balance, L/D ratio, Front and Rear side force, radiator, and gear cooler mass flow rates. However, for conciseness, only the relevant and interesting

parameters are shown in subsequent sections. Also, a detailed flow field investigation is a work in progress and will be published in a subsequent paper.

2.3.1 Boundary Condition Effect

First, we considered Case I to study the effect of boundary conditions of the side walls of the VWT on the flow field predictions. Figure 2.9(b) and Figure 2.10(b) show that the zero gradient boundaries on the side walls have an unphysical impact on the flow predictions even though they are far away from the racecar geometry. Figure 2.9(a) and Figure 2.10(a) show expected flow field development far away from the racecar geometry. Thus, henceforth this set of boundary conditions is used exclusively for both crosswind angles.

2.3.2 Validation

All 3 configurations listed in Table 2.1 were simulated using the VWT of dimmensions 51 L \times 50 W \times 50H. The results are compared against wind tunnel data below.

Case	Yaw (deg)	Splitter Gap (SG)
Ι	-3.0	Low
II	-3.0	High
III	0.0	High

Table 2.1: Configurations of the racecar considered in this study.

From Figures 2.11 and 2.12 we can see that the numerical predictions are well correlated to the wind tunnel data for all three configurations of ride height and crosswind angles. We can observe that the % delta between CFD and wind tunnel is very small at < 2% and has within 1% variation. However, in Figure 2.13 we see that CFD significantly overpredicts the % Front in configuration III. This could be due to the limitation of RANS to fully capture the dissipation of the underbody splitter jet seen Figure 2.14. This will be investigated further in a subsequent paper using DES



Figure 2.9: Streamwise velocity profiles on Z/SG = 1 plane (a, top) with right wall as velocity inlet, Left wall as pressure outlet (b, bottom) Side walls as zero-gradient boundaries.

modelling.

We also have surface pressure tap data from the wind tunnel test for all three configurations. However, a detailed surface pressure distribution study is beyond the scope of the present paper and will be published in a subsequent study.

2.3.3 Tunnel Size

Next, we explored 3 larger VWT sizes to validate open air simulation conditions. All 3 configurations were run using larger tunnel sizes explained in Table 2.2 below.

From Figures 2.15 and 2.16 we observe a small dependence on the VWT size that has a variation within 0.5% of the corresponding wind tunnel values. Thus, we chose



Figure 2.10: Cross-stream velocity profiles on Z/SG = 1 plane (a, top) with right wall as velocity inlet, left wall as pressure outlet (b, bottom) Side walls as zero-gradient boundaries.

the largest VWT size to minimize blockage ratio effects. With the discretization scheme using very coarse cells near the VWT boundaries, the increment in cell count is less than 0.5%.

2.3.4 Realizability

Next, we explored the effect of the Realizability coefficient using Configuration II. The realizability coefficient was varied from 0.6 (default) to 0.9 in increments of 0.1

We see in Figure 2.17 that C_D is barely affected by the change in Realizability coefficient. However, Figures 2.18 and 2.19 show that C_L , particularly C_{LF} , is having a very strong dependence on the reliability coefficient. Interestingly the 36 -count



Figure 2.11: Headline C_D for validation study.

increment between Re = 0.6 and Re = 0.9 is identical in C_L and C_{LF} plots. This suggests that the splitter gap flow prediction is highly sensitive to the Realizability coefficient. This will be explored more in subsequent delta plot sections.

2.3.5 Grid Independence Study

The results so far are well correlated to the wind tunnel data. However, we wanted to cross-check the grid dependence of the predictions. Thus, all three configurations were run on 3 mesh sizes using the largest VWT size and realizability coefficient as 0.9. The meshes were labelled as Coarse, Baseline and Fine. The Coarse and Fine meshes were obtained by changing the base size of 24 mm by $\pm 10\%$. Thus, the Coarse mesh consisted of 96 million cells and the Fine mesh consisted of 165 million cells.

We can see in Figure 2.20 that the C_D predictions barely change with the mesh. Figures 2.21, 2.22, and 2.23 show that downforce is more sensitive to the grid size



Figure 2.12: Headline C_L for validation study.

than the drag. The variation in C_L is observed to be < 1% and thus the baseline mesh was deemed a sufficient compromise between reasonable accuracy and computational cost.

2.3.6 Compressibility

Next, we studied the effect of compressibility. This is of interest as due to the high-speed nature of the flow there are local spots such as near the splitter where the local Mach number is greater than 0.3 and thus local compressibility effects are expected. For this incompressible flow solutions are compared against compressible solver with segregated temperature solver for the energy equation.

We can see in Figure 2.24 that C_D prediction improves overall with the compressible solver. However, in Figure 2.25 the downforce prediction improves for Configurations II & III but underpredicts in Configuration I. The general downward (reduction in



Figure 2.13: %Front for validation study.



Figure 2.14: Normalized streamwise velocity @ Z=SG plane.

Table 2.2: VWT sizes.

Last	Tunnel	Tunnel Dimmensions
Suffix	Extents	
А	Baseline	51 L \times 50 W \times 50H(10 L upstream \times
	Tunnel	40 L downstream)
В	2x Tunnel	101 L \times 100 W \times 100H(20 L upstream \times
		80 L downstream)
С	Long Tunnel	111 L \times 50 W \times 50H(10 L upstream \times
		100 L downstream)
D	4x Tunnel	$211 \text{ L} \times 200 \text{ W} \times 200 \text{H}(50 \text{ L} \text{ upstream})$
		160 L downstream)



Figure 2.15: Headline C_D for Configuration I for different VWT sizes.

downforce magnitude) trend remains the same for all three configurations. However, these swings are large and will be investigated further in subsequent sections.



Figure 2.16: Headline C_L for Configuration I for different VWT sizes.

2.3.7 Under-Relaxation Factors (URF's)

Finally, we began exploring the effect of URF's. URFs for pressure, velocity and turbulence were ramped for all configurations. A linear ramping algorithm was used, and all URF's attained a stable value by iteration 1000. In the Configuration II example seen in Figures 2.26 and 2.27 we can see that this change in the URF's has resulted in early convergence from 2000 iterations. This indicates that while the lower values of the URF's are helping maintain numerical stability of the solver during the initial iterations, they may be potential to reduce this relaxation to achieve convergence faster. Star-CCM+ v15.04 User manual does state that "the default pressure and velocity under-relaxation factors are conservative. They lead to converged solution in most cases, including when the grid is poor...Optimal values of under-relaxation parameters are problem dependent.".



Figure 2.17: Headline C_D for Configuration II for different realizability coefficient values.

a high-quality grid and achieve a gridindependent solution, an optimization of the URF's is desirable to achieve faster convergence. There may be potential to reduce computational time by 30%.

Additionally, we can also see from the C_D and C_L trends in Figures 2.28 and 2.29 that ramping up of the URF's is generally improving accuracy. We hypothesize that this may indicate inadequate convergence with the default URF values. The solver may be trapped into a local minimum thus unable to reduce error. These phenomenon will be explored in detain in subsequent publications.

2.3.8 Delta Accumulated C_D nd C_L lots

To identify regions where the force coefficients differences come from, we look at accumulated C_D and C_L plots. In Figures 2.30 and 2.31 the reference simulation is



Figure 2.18: Headline C_L for Configuration II for different realizability coefficient values.

Configuration II with a realizability coefficient of 0.9. This case was chosen as the reference as it has highest accuracy w.r.t the wind tunnel data. We plotted the delta between the simulations to highlight where along the streamwise length of the vehicle we see major differences in force coefficient predictions.

We know that Re = 0.9 case slightly overpredicts CD. In Figure 2.30 we can see that relative to the Re = 0.6 case, this overprediction of C_D stems from the underhood and decklid regions. This indicates that the higher realizability likely overpredicting the separation and recirculation zones as the local flow velocity magnitude reduces.

However, we know that realizability coefficient of 0.9 gives the most accurate prediction of C_L . We can see in Figure 2.31 that as the realizability coefficient is increased, the underprediction of downforce consistently reduces. Also, we see that the magnitude of underprediction increases steadily from front of the car until the decklid



Figure 2.19: Headline C_{LF} for Configuration II for different realizability coefficient values.

region. This suggests that the underhood and underbody flows are most affected by this coefficient change.

Next, in Figures 2.32 and 2.33, we look at the effect of compressibility solver using the incompressible simulations as the reference. We know that the compressible solver predicts C_L more accurately w.r.t the wind tunnel data. The overprediction of downforce by the incompressible solver is seen to come from the entire length of the vehicle. The largest contributors are the front splitter, roof, and rear spoiler regions. This is expected as these regions experience the fastest local velocity flow fields. This shows that the local compressibility effects are important for accurate resolution of the underbody pressure field.



Figure 2.20: CD for different grid sizes.

2.4 Summary/Conclusions

An investigation of effect of boundary conditions and solver parameters on the flow predictions around a Gen-6 NASCAR using steady-state RANS simulations are presented in this paper. Three configurations of the vehicle, including two ride heights and two yaw angles, were considered. Zero-gradient boundaries were compared with inlet and outlet type boundaries for the simulation side walls in crosswind simulation. The effects of realizability coefficient in the SST $k - \omega$ turbulence model and the effects of the compressibility solver were studied. Force and moment coefficients from all three configurations were validated against wind tunnel data. Both CFD and wind tunnel experiments were setup to simulate openroad conditions. Specific conclusions of this study are:

• Sufficient care must be taken during CAD cleanup to retain the detailed surface



Figure 2.21: CL for Configuration I at different grid sizes.

description and subsequently during mesh generation to resolve the surface. This contributes towards achieving very high accuracy w.r.t wind tunnel data. Thus, not only is the trend from the wind tunnel data captured, but also the magnitude of the predicted changes is accurate.

- Even with a large computational domain with negligible blockage ratio (0.04%), the side wall boundary conditions are critically important for crosswind simulation. It is recommended to use inlet and outlet type boundary conditions. This is because even for the small crosswind angles considered, the zero-gradient side boundaries were causing an unphysical effect on the mean flow.
- Also, a much larger size (211 L × 200 W × 200H (50L upstream ×160 L downstream)) of the VWT is recommended for an open-air-simulation. Such a computational domain has an even smaller blockage ratio of 0.0025%. Such a long



Figure 2.22: CL for Configuration II at different grid sizes.

VWT, especially in the downstream direction, is required to ensure the far wake region is resolved and thus subsequently improves the near wake and surface pressure predictions.

- Significant influences of both the realizability coefficient and the compressibility solver were found on the lift coefficient. Both, a smaller value of realizability coefficient and the incompressible solver are seen to significantly overpredict the suction pressure under the racecar, hence overpredicting the overall downforce generated. It is recommended to use a higher value of the realizability coefficient (0.9) and use the compressibility solver to accurately capture the suction pressure in the undercar flow field.
- Additional sources of lift discrepancy are ventilation drag, support struts, and various boundary corrections used by the wind tunnel to report open-road co-



Figure 2.23: CL for Configuration III at different grid sizes.

efficients. An interested reader is referred to the paper by Walter etal. (2012).

A more comprehensive study of the flow field prediction is required to fully understand the impact of these solver changes. However as mentioned earlier, these investigations will be published in a subsequent paper. Future work includes an indepth study of the differences in predicted flow fields, surface pressure distributions and higher resolution DES simulations to capture higher frequency motions and the effects of a transient solver.



Figure 2.24: Headline C_D for all three Configurations for incompressible and compressible solvers.



Figure 2.25: Headline C_L for all three configurations for incompressible and compressible solvers.



Figure 2.26: Slower convergence before ramping up the URFs for Configuration II.



Figure 2.27: Faster convergence after ramping up the URFs for Configuration II



Figure 2.28: Headline C_D for all three configurations with default and ramped URF's.



Figure 2.29: Headline C_L for all three Configurations with default and ramped URF's.



Figure 2.30: Delta accumulated C_D plot for Configuration II with different realizability coefficients.


Figure 2.31: Delta accumulated C_L plot for Configuration II with different realizability coefficients.



Figure 2.32: Delta accumulated C_L plot for Configuration I between compressible and incompressible solvers.



Figure 2.33: Delta accumulated C_L plot for Configuration III between compressible and incompressible solvers.

CHAPTER 3: (ARTICLE 2) SCALE-RESOLVED AND TIME-AVERAGED SIMULATIONS OF THE FLOW OVER A NASCAR CUP SERIES RACECAR

3.1 Introduction

Aerodynamics is considered as the single most important contributor to a racecar's on-track performance [1]. This led professional race teams to allocate a significant portion of their resources to the aerodynamic testing and development of their race vehicles. Race teams are making significant investments in advancing both of the two tools of the trade: Wind Tunnel (WT) testing methodologies optimization and Computational Fluid Dynamics (CFD) process improvement. Recent trends suggest that while WT tests are often a desired approach as these involve testing of the full-size physical model of the actual racecar, CFD has also evolved into a reliable and indispensable supporting tool for racecar aerodynamic development. Complementary to the WT tests, CFD simulations can provide a significantly more detailed description of the flow field around the vehicle using non-intrusive data collection in a virtual environment, making it a cost-effective companion tool in the aerodynamic analysis [25, 26, 24]. With a proper discretization scheme, and applying appropriate boundary conditions and physics models, CFD simulations can now predict the flow field with an accuracy comparable to wind tunnel tests [4, 5, 6, 7]. Nevertheless, CFD analyses are sometimes used to explain and rationalize observations from the wind-tunnel tests [8, 9, 10, 11]. However, professional race sanctioning bodies, like the Federation Internationale de l'Automobile (FIA) or the National Association for Stock Car Auto Racing (NASCAR), have restrictions on the number of wind-tunnel hours. FIA also has a yearly cap on the maximum CPU clock time a team can spend in their racecar development, NASCAR has a similar monthly limit on the number of CFD runs a team may perform. As such, the racing industry requires the development of accurate and reliable CFD methods yet with faster turnaround times. Because of its relatively quicker turnaround time, time- and scale- averaged (SAS) Reynoldsaveraged Navier-Stokes equation (RANS) approaches are widely used as the first tool in the designdecision-making process in motorsports [25, 26, 24, 10, 11, 27, 28, 29, 30, 31, 32].

The flow over an automotive body, be it a passenger or race vehicle, is a complex, inherently transient and turbulent one. In literature, time-resolved numerical solution approaches like the Large Eddy Simulation (LES) and Detached Eddy Simulation (DES) have shown greater accuracy in some industrial CFD applications due to their ability to better capture the dynamic evolution of the transient flow fields [4, 5, 6, 7]; these approaches are commonly referred to as the Scale Resolving Simulations or SRS for short. However, compared to SAS or RANS approaches, computational time requirements for SRS approaches are significantly higher. For example, to run a simulation using a variant of the DES, called the Delayed Detached Eddy Simulation (DDES), takes approximately 15 times longer for a passenger car simulation than an equivalent RANS would have required [48]. Nevertheless, it has been shown that a suitably designed RANS CFD simulation is a cost-effective approach for developing the external aerodynamics of a full-sized DrivAer model [49]. A similar conclusion can be drawn for cases involving stock racecars based on the high-fidelity RANS based CFD simulation of NASCAR Cup racecars previously presented by Uddin and coworkers (see [30, 46]).

It is common knowledge that the accuracy of flow field prediction by RANS simulations is heavily influenced by the chosen turbulence model [50, 51], with this effect holding true for vehicle external aerodynamic predictions [4, 5, 27]. On the other hand, recent literature shows that DES simulations involving various automotive geometries are providing encouraging results and better insight into the flow dynamics. These include better correlation with WT experiments in terms of force and moment prediction, and realistic characterization of flow structures in both the near and far wake regions [6, 5, 42, 72, 73, 21, 33] in terms of their dynamic evolution. Turbulence modeling literature detailing the drawbacks of the RANS approach and highlighting the superiority of the DES approaches in elucidating the vortical structures embedded in the flow is plentiful for both internal and external flows. The studied cases include both simple geometries, such as channel flows [52] and flows past bluff bodies [53], and other complex shapes; a more thorough review may be found in published books and review articles [54, 13, 55, 56, 57]. As a result, the general trend observed in automotive OEM external aero CFD application is a shift towards the transient SRS simulations, even though the racing industry still primarily relies on the SAS type CFD.

However, a vast majority of the published turbulence modeling validation studies pertaining to automotive external aerodynamics are based on either idealized automotive models, like the Ahmed body or standardized passenger vehicle model like the DrivAer model [21, 58, 59], and studies investigating the efficacy of DES type SRS approaches in motorsports is simply non-existent. In this backdrop, and leveraging our prior experience with the flow around NASCAR Cup racecars [27, 28, 29, 30, 31, 32, 46], this paper explores the feasibility and efficacy of the SRS approaches in the external aerodynamics investigations of a stockcar type racecar using a DES approach, similar to the ones used in [22, 20, 19, 74]. The investigations presented in this paper are carried out using a Delayed Detached Eddy Simulation (DDES) with Improved wall-modeling capability; this approach, which is to be discussed later, is commonly referred to as the IDDES approach.

Race vehicles are aerodynamically very different from road vehicles as these include many aerodynamic devices specifically designed for high downforce-to-drag ratio [48]. The Generation 6 NASCAR, (called Gen-6 for short) Cup racecar, which had been in use since 2013, has many such features. As can be seen in Figure 3.1, the characteristic aerodynamic features of this car include: a front splitter with underbody splitter extension panel, a rear spoiler, very low ground-clearance side-skirts, front by-pass ducts, a camera pod, radio communication and GPS antennas, NACA ducts for cabin and driveline cooling, roof-rails and shark-fin. The front splitter usually operates at a very low-ground clearance. Where a typical passenger vehicle normally produces a small lift with a lift-to-drag ratio of about 0.3 [49, 59], a racecar must achieve high-speed cornering performance, typically with down-force (or a negative lift) and having a lift to drag ratio of -2.0 or larger [25, 26, 24, 10, 11, 27, 28, 29, 30, 31, 32]. Also, due to dynamic on-track conditions experienced by a racecar due to high operating speeds and vertical acceleration, vehicle ride-height and orientation changes, the vehicle's aerodynamic behavior significantly changes between cornering and straightline driving conditions. A racecar on corner entry is subject to braking while on corner-exit is subjected to a high longitudinal acceleration. This causes the pitch of the racecar to change in what are called dive-and-squat angles. The vehicle also experiences a yaw during corner entry, apex, and exit. Thus, the vehicle's aerodynamic characteristics must be analyzed under an envelope of yaw and pitch orientations. Existing literature covers yaw and pitch effects on generalized car shapes, such as the Ahmed body and DrivAer body, both experimentally and numerically [60, 61, 62, 63, 64, 65, 66]. Some limited work is also published for performance cars focusing on specific areas such as wings or using simplified geometries [67, 68, 69]. Very early numerical experiments using CFD as a tool focused on understanding the car performance in different conditions and were limited to very simplified, and now outdated Gen-4 and Gen-5, NASCAR geometries. While more recent work by Fu et al. [27, 28, 29, 30, 31, 32] uses detailed Gen-6 geometry, their data is validated with AeroDyn data, a closed-jet, open-return wind tunnel, at a single operating condition. Also, that wind tunnel used boundary layer suction for ground-plane emulation. Nevertheless, the work of Fu et al. focused more on effects of turbulence parameters,

boundary conditions, solver parameters and the choice of turbulence models on the flow predictions [27, 28, 29, 30, 31, 32]. In the absence of authentic studies on the feasibility and prediction veracity of transient SRS approaches like IDDES for the aerodynamic characterization of stock racecar, this paper presents an in-depth analysis comparing and contrasting the CFD simulation of the flow field around a Gen-6 NASCAR racecar as obtained using the popular and commonly used SAS RANS approach (c.f. [46]) and a potentially more accurate transient IDDES approach. The primary objectives are to investigate the predictive difference between these two methods and to understand the root causes of these differences.



Figure 3.1: Gen-6 NASCAR with unique surfaces highlighted in orange

All CFD simulations presented in this paper are validated against data from Windshear, an open-jet closed-return type wind tunnel with a moving-belt for rolling road simulation that is particularly designed for road-ready motorsport vehicles [75]. The layout for the windshear wind tunnel is shown in Figure 3.2. Wind tunnel data for the three operating configurations made available to us include: high and low ride height as well as zero and -3.0° yaw angles. The ride heights in terms of the splitter gap clearance as used in the study are shown in Figure 3.3, while the yaw configurations are shown in Figure 3.4. Further details of the simulation setup and computational methodology will be addressed in subsequent sections.



Figure 3.2: Layout of the Windshear Wind Tunnel. Image Source: Windshear website (https://www.windshearinc.com/; accessed on 18-Nov-2022)

NOTE that in the subsequent discussions, the splitter gap, i.e. the vertical clearance between the ground and the bottom-most part of the splitter, will be expressed using a symbol 'SG'.



Figure 3.4: Two yaw conditions, 0° and -3° , as considered in this study



High Splitter Gap

Figure 3.3: Two ride height configurations used in this study; note that the splitter gap is expressed using a symbol 'SG' in subsequent analysis. (a) top: low SG, (b) bottom: high SG

3.2 Governing Equations

The Navier-Stokes (NS) mass, momentum and energy equations govern the dynamics of fluid flows. In case of a subsonic nonreacting flow, these are a set of four transport equations in terms of instantaneous velocity and pressure gradients. However, a direct numerical solution (DNS) of these equations as applied to racecars is way beyond the capacity of any computer in the world, hence, we resort to finding an approximate or modeled solution to this problem. The next coarse-grained approach is called the Large Eddy Simulation (LES), which resolves the large energy-containing scales of turbulence and models only the small-scale motions on the argument that these small scales are often considered isotropic with an assumption that they can be modeled without much loss of accuracy. And secondly, since the contribution of these small scales to Reynolds stresses is much weaker when compared to the contributions from the larger eddies, the induced error is expected to be minimal. In fine, the LES approach segregates the wide turbulent scales into the resolved larger scales and the modeled sub-grid scales (SGS).

However, LES is still too computationally costly for most of the automotive external aerodynamics flows. A more practical approach is the Detached Eddy Simulation (DES), proposed by Spalart and his coworkers [75, 76]. DES is a hybrid approach that uses: (1) the next coarse-grained approach, the RANS, in the region close to solid boundary that requires very fine mesh for LES-quality resolution, and (2) LES in the wake and far from the wall regions. The RANS approach models all scales of the flow in which the governing equations are obtained by first decomposing the instantaneous velocity and pressure fields into mean and fluctuating components, using what is known as the Reynolds decomposition, and then timeaveraging the resultant equations. The RANS continuity (or mass conservation) and momentum equations are shown in Equations 3.1 and 3.2, respectively, using the Einstein summation notation where a repeating index implies summation over all possible values, and in this case *i* or j = 1, 2, 3.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{3.1}$$

$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} \left[-\bar{p} \delta_{ij} + \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j} \right]$$
(3.2)

In these above equations, the variable u_i represents velocity in x_i direction, u'_i is the fluctuating component of the instantaneous velocity u_i , the overbar denotes a time-averaged quantity (i.e. $\overline{u_i}$ denotes the time-averaged value of u_i), and t, p, ρ , and μ represent time, pressure, fluid density and fluid molecular viscosity, respectively; δ_{ij} is Kronecker delta, $\delta_{ij} = 1$ for (i = j), and $\delta_{ij} = 0$, for $(i \neq j)$.

Note that these equations for the mean quantities are essentially identical to the original instantaneous equations, but have the additional term $-\rho \overline{u'_i u'_j}$ in the momentum transport equations. This term constitutes the Reynolds stress tensor. It has a total of nine components, six of which are independent. These six additional stress terms make the number of unknowns larger than the number of available equations to solve them. This gives rise to what is known as the turbulence closure problem, which is solved using various turbulence modeling approaches. One of these popular approaches involves expressing the Reynolds stresses in terms of the mean flow quantities, such as the eddy viscosity model which uses the concept of a turbulent viscosity μ_t , and model the turbulent stresses or Reynolds stresses in a way analogous to shear stress in laminar flows. Turbulence models normally vary in terms of how the eddy viscosity is defined, how the turbulence quantities are related to the eddy viscosity, and what transport equations and constitutive relations are used in the determination of v_t . For this study we will be using the SST-Mentor $k - \omega$ modeling approach, SST for short hereinafter. Literature on the use of the SST model is very rich, even for the automotive applications, and hence, for brevity, the details of these equations are omitted. An interested reader is referred to the original articles by Menter and his coworkers [22, 20, 19] or an automotive external aerodynamics article by Zhang et al. [21] for all relevant equations.

3.2.1 The Improved Delayed Detached Eddy Simulation Mode

The DES approach was proposed by Spalart as a more practical application of LES [75, 76]. DES is a hybrid approach that combines LES in the regions away-from-the-

wall regions and RANS within the boundary layers. The switching between LES and RANS is done by computing a local turbulent length scales, l_T , and a local grid size, l_{LES} .

$$\ell_T \equiv \frac{\sqrt{k}}{\omega} \tag{3.3}$$

$$\ell_{\rm LES} \equiv C_{\rm DES} \ \Delta_{\rm DES} \tag{3.4}$$

The weakness of DES is that LES may be applied inside the boundary layer when l_T and l_{LES} drop below a critical value. This causes a prediction of unphysical separation due to the local grid size and is thus known as Grid Induced Separation (GIS). This is mitigated by introducing a delay in the switching function based on the wall normal distance and local eddy viscosity [15]. This approach is called the Delayed DES or DDES. In this paper we used the Improved DDES or IDDES model. This version of DES further modifies the switching function between LES and RANS regions and is aimed at application in high Reynolds number flows [17, 18]. The IDDES model incorporates SGS dependence on the wall-distance that allows RANS modeling where the wall-distance is much smaller than the boundary-layer thickness.

$$\widetilde{\omega} = \frac{\sqrt{k}}{\ell_{\text{Hybrid}} f_{\beta^* \beta^*}} \tag{3.5}$$

where f_{β^*} is the free-shear modification factor, β^* is an SST $k - \omega$ model constant, and the parameter l_{hybrid} is defined as:

$$l_{\text{Hybrid}} = \tilde{f}_d \left(1 + f_e\right) \ell_{\text{RANS}} + \left(1 - \tilde{f}_d\right) C_{\text{DES}} \Delta_{\text{IDDES}}$$
(3.6)

3.3 Geometry and Mesh

The racecar geometry used in this study is a fully detailed NASCAR Gen-6 Cup racecar as used during the 2019 race reason. The CAD geometry was imported into ANSA v15 and cleaned for all surface tessellation errors. Sufficient care was taken to retain as much of the original surface description as possible. Thus, the geometry consists of the external aerodynamic surface, cabin interior flow, NACA cooling ducts, underhood flow with detailed powertrain and exhaust assemblies, and porous regions for the radiator and gear cooler. The fully detailed and error-free final surface consisting of 13 million triangles was imported into Star-CCM+ for CFD simulation.

Wind tunnel experiments can also be viewed as a type of simulation. The openjet configuration of the WindShear wind tunnel also attempts to simulate open-road conditions [70]. Thus, a large computational domain is required to perform an openroad CFD simulation for vehicle aerodynamics. As shown in Figure 3.5, the test model was placed in a large Virtual Wind Tunnel (VWT) with dimensions of $211L \times 200W \times$ 200H, where L, W, and H are the length, width, and height of the test geometry; the inlet and outlet were 50L upstream and 160L downstream, respectively. This large wind tunnel was chosen on the basis of a previous study [46] to avoid the influence of blockage ratio and numerical pressure waves which can occur at boundaries. The inlet was given a velocity of 67.056 m/s (150 mph), a turbulence intensity of 1.0%, and a turbulent length scale of 10 mm. A pressure outlet was placed at the opposite end of the tunnel. For the crosswind simulations, one side of the tunnel was set to a velocity inlet and the other side to a pressure outlet. The velocity inlets then had their velocity specified in x and y components to give the flow the proper angle to simulate the desired yaw, or crosswind to be more precise. To emulate a moving ground wind tunnel test scenario, the no-slip floor was given a tangential velocity corresponding to the given inlet velocity. The wheel rotation was modeled using a Tangential Velocity Specification (TVS) with a small vertical wall to simulate the tire-ground contact patch; this was done for simplicity and cost-effectiveness [29, 72, 71].



Figure 3.5: Schematics of the computational domain

Proper discretization and modeling of the underhood airflow is crucial for accurate mass flow and force predictions [77]. Thus, special care was taken to develop a modeling strategy for the condenser, radiator and fan module (CRFM). The discretization scheme used for porous media modeling includes porous baffles to simulate the front and inner grilles of the radiator ducting. Thus, the radiator consists of 3 regions, the primary cooling duct, the secondary cooling duct, and the radiator core itself. These three regions can be seen in Figure 3.6 colored with light green, dark green and blue respectively. Porous media modeling is tuned to achieve mass flow rate that matches almost accurately with the mass-flow measurements from the wind tunnel.

To discretize the domain, an unstructured predominantly hexahedral cell mesh was created using Star-CCM's "trimmed cell" mesher. This meshing algorithm seeks to create cells which are some multiple of 2^n greater/smaller than the selected basesize. To correctly resolve flow around the racecar, nine different volume sources were used around the body to control cell refinement, each one being half the size of the next coarser one. Eight more volume sources were used near the vehicle to capture flows around the critical areas of interest, such as the spoiler, splitter, gear cooler duct, wheels, cabin interior, and the underhood regions. For example, the narrow but expanding gap between the splitter and the ground boundaries is resolved with 3 different volume controls having anisotropic cells. The vertical dimensions have the finest mesh with $\Delta z = 0.75$ mm and 1.5 mm respectively. Figure 3.6 shows the final mesh around the body.



Figure 3.6: Mesh for the RANS case @ = 0 center plane

To properly resolve high gradient boundary layers on the wetted surfaces, 18 custom surface controls were used to define appropriate prism layer settings. The objective was to have a wall $y^+ < 1$ and to ensure a smooth growth up to the core mesh. A histogram for the all y^+ distribution thus obtained are shown in Figures 3.7 and a y^+ scalar plot is shown in Figure 3.8 for the zero yaw RANS case. From Figures 3.7 and 3.8, it can be seen that y^+ value for 95% of the body cells have a value less than 1, while 3% cells had $1 < y^+ < 2$. These higher y^+ value cells are located at regions of locally accelerated flow, such as around the headlight edges and the A-pillar. The majority of cells with $y^+ > 2$ were found in areas of sharp corners where the prism



layer mesher collapsed. The final RANS mesh consisted of 128 million cells

Figure 3.7: A histogram of wall y+ distribution over the surface of the vehicle obtained from the RANS simulation at zero-yaw



Figure 3.8: Wall y^+ Distribution Over the Surface of the Vehicle

There is a notable difference between the mesh requirements for the RANS and IDDES type simulations. As noted earlier, for a good mesh for SST based RANS simulation, it is desirable to have a wall y^+ value less than 1. However, for the IDDES cases, it is important to achieve a desirable value of the Turbulent Viscosity Ratio (TVR), v_{τ}/v . TVR is essentially a measure of the ratio of the Sub-Grid Scale (SGS) dissipation to the overall dissipation. It has been suggested in previous studies with DES that, for a well-correlated prediction of the near wall velocity profile and the length of the recirculation bubble, the associated TVR in regions of shear flow should be maintained at approximately 10 [6, 78, 79, 80, 81]. Thus, the mesh refinement volume sources in the target regions, such as the splitter, spoiler and near wake, were adjusted, through several iterations, to obtain a desirable TVR. Figures 3.9 and 3.10 show the TVR distribution obtained from three simulations: (1) RANS, (2) IDDES using the RANS grid, and (3) IDDES using the refined IDDES grid consisting of 200 million cells. It can be seen from these figures that for the refined IDDES grid, high TVR regions are primarily confined to the near-body RANS region, and thus satisfies the requirements of a good DES simulation as suggested by the previous studies.



Figure 3.9: Mean Turbulent Viscosity Ratio (TVR) @y = 0 plane



Figure 3.10: Mean Turbulent Viscosity Ratio (TVR) @z = SG plane

3.4 Physics Setup

The simulations presented in this paper were carried out using a commercial finite volume solver CFD package, Star-CCM+ version 2020.2.1. All simulations, unless specified otherwise, were performed using a segregated flow, implicit-unsteady, compressible solver on an unstructured grid using the SIMPLE solver. The $k - \omega$ SST based IDDES turbulence model was used along with its default closure coefficients. A two-layer all y^+ wall treatment was applied to the simulation to ensure reasonably accurate boundary layer calculation in some complex locations of the geometry where the y^+ was not sufficiently small.

A 2nd order discretization scheme was used for the diffusion terms and a 2nd order upwind scheme was used for the convection terms of the momentum equations. The time step was normalized by vehicle length (L) and freestream velocity (U_{∞}). The non-dimensionalized time step used was $\Delta t = (L/U_{\infty}) 1.2 \times 10^{-3}$; this corresponds to a nominal CFL (Courant-Friedrichs-Lewy) number of around unity for near-car grids, and even smaller for the far wake regions. This Δt has been reported as a sufficiently small time-step size for automotive IDDES applications [73]; this was also verified through our time-step independence verification study. Six inner iterations were found to be sufficient for all residuals to drop by 3 orders of magnitude within each time-step.

The authors have previously observed a significant variation in CFD predictions of road vehicle external aerodynamic characteristics when simulations are carried out using Message Passing Interface (MPI) as the parallelization tool if sufficient care was not exercised to ensure consistent domain decomposition [23]. Thus, care was taken to maintain the same parallelization schemes and hardware consistency throughout this study. All simulations were run on UNC Charlotte's High-Performance Computing clusters using 144 processors across 3 nodes having 48 processors each. The simulations were set to run for 90 LETOTs, where 1 Large Eddy Turn Over Time (LETOT) = L/U_{∞} . Note that LETOTs are often referred to as the flow through time as well. Our preliminary exercise has shown that convergence of force and moment coefficients start to occur at roughly 50 LETOTs. Based on this all results presented in this paper are from an averaging window of the last 30 LETOTs, that is averaged between 60 – 90 LETOTs

3.5 RESULTS AND DISCUSSION

As mentioned earlier the three configurations of the Gen-6 NASCAR were considered in this study and are tabulated in Table 3.1 below. These proprietary CAD and wind tunnel data were obtained through a Non-disclosure Agreement (NDA), and in order to protect our sponsor's confidential data, all force and moment coefficients presented in this paper are normalized by an arbitrary reference area. The CFD predicted results are validated against 14 data-points from the wind tunnel data, which includes: all 6 force and moment coefficients, Front and Rear downforce, Lift to Drag (L/D) ratio, Front and Rear side-force, Front-balance (in percent), and radiator and gear cooler mass-flow-rates. However, for conciseness, only a selection of the relevant parameters is presented here.

Case ID	Yaw (deg)	Splitter Gap (SG)
Ι	-3.0	Low
II	-3.0	High
III	0.0	High

Table 3.1: Racecar ride-height and yaw configurations used in this study

Drag and lift coefficients and %Front-balance (defined as the ratio of the front downforce to total downforce) corresponding to all three configurations as obtained from the RANS and IDDES simulations are compared against the wind tunnel measurements in Figures 3.11, 3.12 and 3.13, respectively. Note that the quantity "%Frontbalance" is often referred to simply as "%Front" in racing community, and, hence, this paper will use this terminology as well.



Figure 3.11: Comparison of drag force predictions against wind tunnel measurements for all three configurations



Figure 3.12: Comparison of lift force predictions against wind tunnel measurements for all three configurations



Figure 3.13: Comparison of %Front-Downforce predictions against wind tunnel measurements

From Figures 3.11 and 3.12, in comparison to the wind tunnel test data, IDDES overpredicted both drag and lift forces for all three configurations. We can observe that while, for drag, the % delta between RANS CFD and WT is very small at < 3%,

the % delta between IDDES CFD and WT is larger at ~ 10%. This is consistent with the findings of Zhang et. al. [21] for a passenger vehicle. However, in Figure 3.13, we see that IDDES CFD predictions have a better agreement with the wind tunnel data for % Front-downforce. This indicates that the better total down-force prediction of RANS may result from the cancellation of positive and negative errors in the prediction of front and rear downforces. Note that the over-prediction by the IDDES is resulted from a consistent over-prediction of both front and rear downforce. A comparison of front and rear down force predictions by the two approaches against the wind tunnel test data presented in Figures 3.14 and 3.15, respectively supports this conjecture. It is very interesting to observe that IDDES prediction of the front downforce corresponding to configuration I shows a much better correlation with the wind tunnel measurements. As this case involves a lower splitter Gap, it appears that the IDDES approach is superior in capturing the small-gap flow interactions.



Figure 3.14: Comparison of Front-lift coefficient predictions against wind tunnel measurements for all three configurations

These trends are further corroborated in Figure 3.16 where the delta accumulated force coefficients for all 3 cases are shown; note that, in these figures, the accumu-



Figure 3.15: Comparison of Rear-lift coefficient predictions against wind tunnel measurements for all three configurations

lated forces from the IDDES are shown relative to the RANS case. This implies that a positive delta indicates an overprediction by the IDDES approach. Generally, compared to the RANS predictions, the accumulated forces at every streamwise location are overpredicted by the IDDES approach. The exception is the accumulated CD at x/L = -0.2, i.e., around the cowl region. The slight dip in the accumulated C_D for configuration I (see Figure 3.17), which has a longitudinal offset at the firewall region. This and the abnormally large discrepancy between the two simulation approaches for configuration I is currently under investigation. Of particular interests are also the sharp rise and fall in delta accumulated C_L in Figure 3.16 at x/L equal to -0.45and +0.35. These again suggest larger prediction discrepancies for the two methods around the splitter, and decklid regions. Clearly, configuration III (the one with zero crosswind at high splitter-gap) showed the largest prediction discrepancy between the two solvers, and hence was subsequently selected as the case to investigate further.

Figures 3.18 and 3.19 show the pressure distribution on the upper surface as obtained from the IDDES solver relative to the predictions of RANS solver for configu-



Figure 3.16: Delta accumulated force coefficients for the IDDES cases relative to the respective RANS cases for all three configurations; Top: Drag, Middle: Lift, Bottom: Side-Force.



Figure 3.17: Accumulated CD for configuration I; Solid line: IDDES, dotted line: *RANS*.

ration III. This is symbolically expressed as ΔC_P where positive and negative values indicate overprediction and underprediction, respectively, by the IDDES solver relative to the RANS solver. The regions of slightly positive ΔCP over the front fascia, hood, and windshield (in Figure 3.18, marked as A) coupled with the slightly negative ΔCP over the rear fascia (in Figure 3.19, marked as A) explains the higher drag prediction by the IDDES solver. The significantly higher positive ΔC_P on the decklid and spoiler (marked as B) explain the rear downforce overprediction. This could be due to a premature separation of the decklid airflow causing a stronger stagnation region there. Figure 3.19 clearly shows (region C) a significantly positive ΔC_P on the splitter surface as well as a significant negative ΔC_P at the entrance to the front diffuser. This could be due to a separation of the front diffuser airflow at the larger Splitter Gap configurations, indicating a mini recirculation bubble region there. This premature separation will cause an increase in lift, which is evident in the cumulative lift plot in Figure 3.16 which shows a higher lift prediction by the IDDES solver up to x/L approximately equal to -0.45. Additional analyses of these aspects will be presented in a subsequent paper.



Figure 3.18: Predicted surface pressure distribution on the upper surface as obtained from the IDDES solver relative to the predictions of RANS solver.



Figure 3.19: Predicted surface pressure distribution as obtained from the IDDES solver relative to the predictions of RANS solve (bottom view).

For the category of stock racecars in general, an understanding of the flow in the underbody region is critical to the aerodynamic development, in particular for producing the downforce. Of particular importance to the NASCAR aero engineers is the strong underbody jet that originates from the splitter region. However, as can be seen from Zhang et al. [77], the prediction of the underhood flow is very challenging for RANS which gets even more challenging due to the radiator flow modeling process using a porous media approximation. This "porous media modeling" approach makes the flow out of the radiator much smoother compared to the actual one that comprises a superposition of several hundred small 3D irregular jets. In spite of the limitation/uncertainty of our current CFD process in terms of predicting the radiator out flow, the underbody jet predictions obtained from the RANS and IDDES simulations corresponding to Configuration III are shown in Figure 3.20; this figure also contains the prediction delta, i.e. the difference between streamwise velocities as predicted by the two solvers. Again, a positive delta indicates overprediction by the IDDES method. We can clearly see that this jet being diffused in the IDDES case. This starts by what appears to be a flow separation near the front diffuser and continuously larger dissipation in the near wake of this underbody jet.

Figures 3.21 and 3.22 show contours of the mean skin friction coefficient (C_f) plotted using line integral convolutions (LIC) of the magnitude of wall shear stress; Figure 3.22 is a zoomed-in view of Figure 3.21 around the splitter region to focus on the region surrounding the front diffuser as marked by " A " in Figure 3.21. From the LIC, it is clear that IDDES predicts a leading-edge separation, whereas RANS approach shows a smooth unseparated flow. Additionally, IDDES predicted a significantly lower C_f around the bottom of the fuel cell, but for flows past this, IDDES is predicting higher C_f . The leading edge of the front diffuser (A) shows a region of reverse flow (colored blue, indicating negative skin friction) in the IDDES case. This flow separation would cause a loss of momentum. Additionally, upper body



Figure 3.20: Mean streamwise velocity normalized by the reference velocity @ z = SG plane. Top: RANS; Middle: IDDES; bottom: Delta between IDDES and RANS.

skin friction coefficient (C_f) with Line Integral Convolutions (LIC) of the wall shear stress along x-direction, as shown in Figure 3.23, indicates that RANS prediction



Figure 3.21: Underbody skin friction coefficient (C_f) with Line Integral Convolutions (LIC) of the wall shear stress along x-direction; Top: RANS, bottom: IDDES.

In order to probe factors (turbulence quantities) contributing to the predictive differences between these two methods, we intend to look at the predictions by the RANS and IDDES solvers for two turbulence quantities of interest: the mean Turbulent Kinetic Energy (TKE) or k, and mean Specific Dissipation Rate (SDR) or ω . The mean TKE contours shown in Figures 3.24 and 3.25 indicate that the RANS solver is predicting significantly higher TKE over the decklid and in the wake re-



Figure 3.22: Zoomed-in view (around the front splitter region) of underbody skin friction coefficient (C_f) with Line Integral Convolutions (LIC) of the wall shear stress along x-direction; Top: RANS, bottom: IDDES.

gions. This contrasts with the observations of Ashton et. al., (2016) [5] wherein the IDDES solver was seen to predict slightly higher TKE than the RANS solver. This can be attributed to the different CFD setups in both the studies. Ashton et. al., (2016)[5] used a DrivAer model geometry, which is a simplified passenger car model, and solved with an incompressible solver whereas this study uses a fully detailed NASCAR racecar geometry, solved with a compressible solver. Additionally, Ashton



Figure 3.23: Upper body mean skin Friction coefficient (C_f) with Line Integral Convolutions (LIC) of the magnitude of wall shear stress; Top: RANS, bottom: IDDES

et. al., (2016)[5] had a Reynolds number of 7×10^5 whereas this study has a Reynolds number of 2×10^7 . These differences mean that the CFD solvers are attempting to resolve a significantly different flow field. To obtain a thorough comparison of the near wake TKE predictions influenced by the effects of turbulence modelling and Reynolds number will require a future study using a range of Reynolds numbers. In this study, we conjecture that the higher TKE prediction by the RANS solver in the decklid region prevents local flow separation by energizing the flow. This suppression of the local separation bubble is the reason why we saw significantly fewer vortical structures on the decklid. The same argument can be applied to the TKE of flow around the underbody region. Additionally, the IDDES shows lower TKE prediction over the side skirt region. This could be enough to energize and maintain flow attachment in those near wall regions. In figure 3.26 we can see the SDR in the IDDES case is significantly greater around the front diffuser, underbody, and near wake regions. This combination of reduced TKE and increased SDR in the flow predictions is causing the momentum loss and increased local surface pressure seen Figures 3.16 to 3.20.



Figure 3.24: Mean Turbulent Kinetic Energy (TKE) @ y=0 Plane

Clearly, the largest prediction discrepancy between the two solvers is observed



Figure 3.25: Mean Turbulent Kinetic Energy (TKE)@z=SG plane

in the wake region making it necessary to investigate it further. For that matter, Figure 3.27 shows zoomed-in views of the streamline patterns in the wake region as obtained from the RANS and IDDES simulations. It can be seen in Figure 3.27 that RANS detects a total of three recirculation regions (marked by the red arrow and labeled as A): one near the decklid and spoiler interface, and the other two in the near wake. On the contrary, the IDDES predicts additional recirculation regions. In particular, the IDDES is picking up secondary recirculation bubbles on the decklid,



Figure 3.26: Mean Specific Dissipation Rate (SDR) @ z=SG Plane

and four additional recirculation regions further down the wake. Furthermore, a small recirculation region near the ground is also visible. All these additional recirculation regions are encircled in Figure 3.27.

One of the most effective ways of investigating the wake dynamics is interpreting this region dominated by vortical structure in terms of vorticity distribution. Vorticity scalar field normalized by $U_{\infty} \times L$ where U_{∞} and L are the freestream velocity and vehicle length, respectively, are shown in Figures 3.28 and 3.29, on the y = 0 and z =



Figure 3.27: Streamlines in the near wake region; Top: RANS, Bottom: IDDES.

SG planes, respectively. Clearly, the IDDES simulation is able to elucidate the finer scales of motion whereas these are completely smudged out in the RANS case. We have seen in the work of Fu et. al. (2016) [82], that upstream turbulence significantly alters the force and moment predictions. This implies that if one is interested in investigating racecar drafting or racing in proximity to other vehicles, then one must be cognizant that RANS based simulations are significantly lacking in predicting the finer scales of motion. It is our conjecture that this could adversely affect the force and moment predictions of the drafting vehicle. However, additional investigations

aimed at gaining insight into the evolution and dynamics of these structure are left out for a future publication.



Figure 3.28: Normalized vorticity distribution on the y = 0 plane.



Figure 3.29: Normalized vorticity distribution on the z = SG plane.
Finally, taking the advantage of the transient nature of the IDDES computations, we intend to investigate the Power Spectral Density (PSD) of the force coefficients as shown in Figure 3.30. We anticipate that this spectral decomposition will illustrate the dominant modes of the flow in terms of frequency. We conjecture that, through a Strouhal number analogy, the contributors to these dominant modes can be determined. With f as the frequency, and D as a length scale, the Strouhal number is defined as $St = \frac{f*D}{V}$. Our first observation from Figure 3.30 is that the PSD distribution is largely independent of the configurations. The PSD distribution of the drag coefficient has two major dominant modes, at 80 and 110 Hz, and several minor dominant modes in the range 200 - 300 Hz. A drag reduction can be achieved through aero modifications if one knows which component of the vehicle body is contributing to these modes. Having said this, we would like to warn the readers that simplified tools like PSD analysis may be good starting point but may not be a sophisticated and informative enough tool to be used in effective modal identification, and that a more advanced tool, like the Dynamic Mode Decomposition (DMD), may need to be used (see [35]). However, although the authors are currently investigating this, the DMD analyses of flow around the NASCAR body shapes is not only very complex, but also requires huge computational resources. For example, the storage requirements for 1 case would be more than 5 Terabytes.

Reverting to Figure 3.30, it can be seen that a significantly larger number of dominant modes, in the range 40 - 200 Hz, are contributing to the down-force production, and there exist at least one mode around 55 Hz that negatively impacts the downforce. In terms of the sideforce, all of the cases show a single dominant mode at 110 Hz, except for CASE-I (low ground clearance case) which shows an additional low frequency dominant mode at 40 Hz. By comparing the PSD plots for lift and side-force, we can intuitively argue that this mode is somewhat related to the splitter gap. However, we have no explanation at this point how a smaller splitter gap can contribute to the side force production.

Despite cautions stated earlier on the limitations of the PSD based modal analysis, data presented in Figure 3.30 brings out a few interesting points. As it was stated earlier, there exists a dominant mode at 110 Hz for all three force components. Now applying Strouhal analogy, and assuming that Strouhal number asymptotes to a value of around 0.22 for large Reynolds number flows, and using the 80% of the free-stream velocity as a characteristic velocity (this is essentially the convection velocity of the energy containing eddies for a boundary layer flow, see [83]), we can calculate that the associated length-scale of the mode at 110 Hz is 0.1013 meters (or 4 inches), which is equal to the spoiler height of the racecar. This example simply illustrates the power of the PSD based tools, and the authors are currently working on this. Note that in addition to the PSD of the force components, the authors are also probing additional quantities like front- and rear-downforce, force balance and three components of the force-moment. The final word of caution is that although averaging over the last of 30 LETOT is sufficient for force and moment coefficient, this length of run-time is not enough for the PSD analyses of modes associated with large scales which are on the order of the whole racecar characteristic length-scales, and we need to calculate the PSD over at least 300 LETOTS. This will eventually quadruple the simulation run time and will significantly increase the computational cost.

3.6 Conclusions

In this paper we have investigated the flow prediction around a fullscale, fully detailed, Gen-6 NASCAR Cup series racecar using RANS and IDDES turbulence modeling approaches. The force and moment coefficients were validated against wind tunnel data from an open-jet, closed-return wind tunnel with a rotating belt and boundary layer suction for moving ground simulation. It was found that the drag and lift predictions obtained from the IDDES are within 10% of the wind tunnel predictions, which is worse than the prediction from a RANS model. However, the

better result from the RANS method may be merely a coincidence, and probably is consequence of cancellation of the positive and negative errors in predicting the front and rear lift forces. This is evident from the inferior percent frontdownforce prediction by the RANS method. The discrepancies in force predictions by these two methods are explained using the predicted flow fields. Clearly, the superiority of the IDDES approach is its capability to depict a more realistic picture of the vortical structures embedded in the flow, in particular in the wake region. This phenomenon is very important for race teams that intend to optimize aero characteristics of the car based off the on-track behavior, especially when the racecar is to be optimized as the trailing car in the pack.

Having said this, the over prediction by the IDDES approach requires additional investigation. Considering the overprediction of drag and lift by IDDES, the big question that remains unanswered is whether the IDDES is unintentionally shielding a real separation in its prediction by wrongly assuming a true separation as a grid-induced separation (GIS), and the switching function is improperly triggered on. Probably, the best way to verify this conjecture is through comparing the CFD predicted pressure fields against the experimental measurements, which is a topic of future investigation by the authors.

Finally, spectral analysis of the forces showed some potentials of being used as a tool in racecar aerodynamic optimization as it can discern modes which either positively and negatively impacts the force coefficient(s) more dominant than the others, and then identifying the body-component associated with that mode. However, this work is still at its infancy and requires additional tuning before it becomes an applicable tool.



Figure 3.30: Power Spectral Density (PSD) of the force coefficients for all three configurations. Top: drag; Middle: down-force (negative lift); Bottom: Sideforce

CHAPTER 4: (ARTICLE 3) ON THE EFFECTIVENESS OF SCALE-AVERAGED AND SCALE-RESOLVED TURBULENCE MODELLING APPROACHES IN PREDICTING THE PRESSURE FIELD OVER A NASCAR RACECAR

4.1 Introduction

Aerodynamics is a vital contributor to a racecar's performance, thus race teams invest significant resources into the aerodynamic development of their competition vehicles. The three standard procedures for conducting aerodynamic development are road tests, Wind Tunnel (WT) tests, and numerical simulation using Computational Fluid Dynamics (CFD). Advances in computing power and numerical simulation methodologies have enabled CFD to be used as a reliable first-approximation tool to obtain the flow fields around a racecar and to predict the aerodynamic forces acting on it. The preference for CFD over traditional aerodynamic testing methods is due to the advantages it offers such as cost-effectiveness, fast turn-around times, a high degree of control over the test environment, and the ability to provide a significantly more detailed flow field description using non-intrusive measurements. In order to correlate the validity of these methodologies, WT tests are typically the preferred reference models for aerodynamic development. A CFD simulation framework demonstrates flow field predictions that are very well correlated to WT tests when implemented with appropriate discretization schemes, boundary conditions, physics models, and data averaging strategies. However, with the aim of limiting costs and encouraging closer competition, race-sanctioning bodies have introduced restrictions on both the maximum number of wind-tunnel hours and the maximum CPU time a team can spend on their racecar development. For example, the Federation Internationale de l'Automobile (FIA) caps the annual wind tunnel and CPU hours for each team, while the National Association for Stock Car Auto Racing (NASCAR), has an annual cap on wind tunnel hours and a monthly limit on the number of CFD runs for each manufacturer. As such, the racing industry requires accurate, reliable, and time-efficient CFD methods. [47, 46, 28, 84, 30, 27, 31, 32]

CFD methods may be classified into two broad classes, steady-state Scale-Averaged Simulations (SAS) and time-resolved Scale-Resolved Simulations (SRS). SAS approaches such as Reynolds Averaged Navier Stokes (RANS) simulations have been the preferred CFD methodology in the racing industry due to their relative simplicity and quick turnaround times. Meanwhile, SRS approaches involving Large Eddy Simulation (LES) such as hybrid RANS/LES or variants of Detached Eddy Simulation (DES) have gained popularity within the automotive industry due to their capacity to capture the dynamic behavior of the flow field, thus giving higher confidence in the aerodynamic coefficient predictions. However, SRS simulations of full-sized automotive geometries require an order of magnitude more computational resources than equivalent SAS simulations. Due to this computational cost penalty and the aerodynamic testing restrictions posed by the race governing bodies, SRS approaches are prohibitive for the competitive racing industry. As SAS approaches have shown suitable prediction accuracy at much reduced computational cost, they remain a favored tool for the racing industry. [47, 71, 5, 21] It is therefore critically important which turbulence model is selected, and that proper solver parameters are chosen [28, 46, 85].

Examining the underlying physics, both SAS and SRS approaches contain a RANS model. It has been seen in the literature that the predictions by RANS simulations are highly dependent on the turbulence modeling approach chosen [50, 51, 4, 27]. In this regard, there exists turbulence modeling literature for canonical flows, such as channel flows and bluff body wakes, as well as automotive flows using generic geometries such as the Ahmed body and the DrivAer model [52, 53, 58, 5, 42]. Substantial review

papers are available in published literature and the interested reader is directed to these references for further details [54, 13, 55, 56, 57]. Based upon the prior experience of the authors with a NASCAR geometry, all the CFD cases presented in this paper use the Shear Stress Transport (SST) $k - \omega$ turbulence model developed by Menter. [47, 46, 23, 71, 8]

Previously the authors published a CFD framework (Misar, A.S., and Uddin, M (2022) [46]), using an SAS approach with Menter's $k-\omega$ turbulence model [22, 20, 19], for predicting the aerodynamic behaviour of a race car. The aerodynamic drag and lift coefficients predicted by this framework were within 2% of WT values. While these predictions were well-correlated, the front downforce was overpredicted and rear downforce was underpredicted. This resulted in significant error of front-to-rear downforce balance, defined forward as %_Front. Automotive CFD literature suggests that SRS approaches are seen to produce more accurate and detailed flow field predictions, thus the authors, Misar et al. (2023) [47], developed another a framework for an SRS approach using the IDDES model by Shur et al. [16], using the previous SAS framework as a baseline. It was found that the IDDES framework overpredicted both lift and drag relative to RANS and WT values but had a much better correlation to WT prediction of %_Front balance. The IDDES flow field also resolved many more vortical structures resulting in significant differences in the macroscopic flow field, particularly in the underbody and decklid regions.

Zhang et al. (2019) [21] used automotive geometry to study the effect of various RANS and DES variants on the aerodynamic force predictions of a hatchback-style passenger car, utilizing four (4) variants of RANS models and two (2) variants of the DES model. The largest discrepancy observed by Zhang in aerodynamic coefficients was between the realizable $k - \epsilon$ (RKE) RANS model and the Detached DES (DDES) model. Similar to Misar (2022, 2023), drag predictions from the RKE-based SAS were well-correlated to WT values, but all SRS approaches were overpredicting drag. Only

drag data from a wind tunnel was available to Zhang for validation. Ashton et al. (2016) [5] studied a DrivAer geometry in both estateback and fastback configuration, and Guilmineau et al. (2018) [42] studied an Ahmed body geometry at 25° and 35° slant angles. Both Ashton and Guilmineau had similar observations, with the lift and drag coefficients of the SRS approach being overpredicted. To better appreciate where these flow prediction differences between SAS, SRS, and WT data occur, it is first necessary to understand the particular geometric features of the racecar geometry studied in this paper and the wind tunnel configuration from where the validation data was collected.

Race vehicles have many aerodynamic devices that are specifically designed for a high downforce-to-drag ratio, resulting in aerodynamic characteristics distinctly different from the passenger vehicles they represent. The Generation 6 NASCAR Cup racecar (called Gen-6 for short), which had been in use since 2013, has many such aerodynamic features. These include: a rear spoiler, roof-rails, a shark-fin, a front splitter with an underbody splitter extension panel, very low ground-clearance side-skirts, front by-pass ducts, NACA ducts for cabin and driveline cooling, a camera pod, and radio communication and GPS antennas. The front splitter is maintained at a very low ground clearance. Where a typical passenger vehicle normally produces a small lift with a lift-to-drag ratio of about 0.3, a racecar must achieve high-speed cornering performance, typically with down-force (or a negative lift) and having a lift-to-drag ratio of -2.0 or larger [49, 59, 47, 46, 28, 84, 30, 27, 31, 32]. Race cars experience dynamic on-track conditions having significantly different aerodynamic behaviors between cornering and straight-line driving conditions due to the effects of high operating speeds, vertical acceleration, vehicle ride-height and orientation changes. A racecar on corner entry is subject to braking, while on corner exit it is subjected to a high longitudinal acceleration. This causes the pitch of the racecar to change in what are called dive-and-squat angles. The race vehicle experiences a yaw during corner entry, apex, and exit. Therefore, the vehicle's aerodynamic characteristics must be analyzed under an envelope of yaw and pitch orientations. Existing literature covers yaw and pitch effects on generalized car shapes, such as the Ahmed body and DrivAer body, both experimentally and numerically [60, 61, 62, 63, 64, 65, 66]. Some limited work is also published for performance cars focusing on specific areas such as wings or using simplified geometries [67, 68, 69]. Very early numerical experiments using CFD as a tool focused on understanding the car performance in different conditions and were limited to very simplified, and now outdated, Gen-4 and Gen-5 NASCAR geometries [25, 26, 24]. Nevertheless, the work of Fu et al. focused more on effects of turbulence parameters, boundary conditions, solver parameters and the choice of turbulence models on the flow predictions [28, 84, 30, 27, 31, 32].

The reader must note that a wind tunnel is an experiment by itself and is therefore susceptible to its own various sources of error. Each wind tunnel is unique and the operators have to apply corrections in order to report open-air results. The work by Fu et al. [28, 84, 30, 27, 31, 32] and Jacuzzi [86] uses detailed Gen-6 geometries with their data validated using data collected at the AeroDyn wind tunnel. This full-scale tunnel is a closed-jet, open-return design using boundary layer suction for groundplane simulation. A closed-jet wind tunnel, such as AeroDyn, has a high blockage ratio, requiring the application of blockage correction ratio factors [32, 87]. All CFD simulations presented in this paper are validated against data from Windshear, an open-jet closed-return type wind tunnel with a moving-belt for rolling road simulation, a design that is particularly well-suited for road-ready motorsport vehicles. All aerodynamic forces and moments are non-dimensionalized using reference geometric and wind velocity measurements. These non-dimensionalized forces and moments are then presented as coefficients to 3 decimal places. Each 1/1000th place is commonly referred to as a "count". The Computer-Aided Design (CAD) and WT data for the Gen-6 NASCAR geometry presented in this study was obtained from a sponsor through a Non-Disclosure Agreement (NDA). Due to NDA constraints, all aerodynamic coefficients presented in this paper are non-dimensionalized using arbitrary reference values.

The challenge CFD methodologies present to racecar simulations, highlighted by the authors in their previous studies, is that different turbulence modeling strategies within the differing CFD simulations yield significantly different flow field predictions [47, 46, 28, 84, 30, 27, 31, 32]. This raises some important issues requiring further investigation. To better understand how to best apply a CFD framework to racecar simulations, one must investigate which CFD prediction methodology will result in more accurate representations of real-world conditions. Taking this further, an investigation into which regions in the flow field demonstrate the highest discrepancy will be required. The investigation in the current paper sheds some light as to which flow features around the vehicle may be contributing the most to those discrepancies, directing future studies to those specific areas likely to produce improved simulation accuracy.

To answer these questions, the current paper looks at the correlation between the static pressure data obtained from surface-mounted probes from the wind tunnel experiments of a racecar to the predictions obtained from different CFD simulations of the same geometry. As mentioned earlier, WT data for a Gen-6 NASCAR racecar in three operating conditions was obtained from a sponsor. The three configurations representing the three operating conditions are listed in Table 4.1 below.

Table 4.1 :	Configura	tions of the	e racecar tha	t are consid	lered in	$_{\mathrm{this}}$	study
---------------	-----------	--------------	---------------	--------------	----------	--------------------	-------

Configuration	Yaw (deg)	Splitter Gap
C1	-3.0	Low
C2	-3.0	High
C3	0.0	High

RANS and IDDES CFD simulations of the above three configurations were run

using both incompressible and compressible solvers. Simulations using the RANS and IDDES solvers will be referred to as RAS and DES respectively, and incompressible and compressible solvers will have postfixes of "-I" and "-C", respectively; for example, "RAS-C" will stand for a RANS simulation using a compressible solver.

4.2 Methodology

The current paper contains a further analysis of the CFD experiments published earlier by the authors, with a brief description provided below. However, for further details on the computational setup for the RAS-I and RAS-C setups, the interested reader is directed to a paper by the authors, Misar, A.S., and Uddin, M (2022) [46]. This setup was then used as a baseline for developing the setup for the DES-I and DES-C cases. Again, a brief description is provided in this section, with further details on the computational setup for the IDDES cases available to the interested reader in the following paper by the authors, Misar et al. (2023) [47].

4.2.1 Governing Equations

The Navier-Stokes (N-S) equations are the governing equations for fluid flow. These equations are represent the principles of Conservation of Mass (also referred to as the Continuity Equation) and Conservation of Momentum. For a Newtonian flow these are given by equations 4.1, and 4.2 respectively, using Einstein notation where repeating index variables (i) or (j) imply summation over all possible values, e.g. (i = 1, 2, 3).

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \tag{4.1}$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}$$
(4.2)

where, t represents time and the variables u_i , p, ρ , T, e, K, and τ_{ij} represent the time-dependent values of the velocity in x_i direction, pressure, fluid density, temperature, internal energy, thermal conductivity, and fluid viscous stress tensor, respectively. The viscous stress tensor, τ_{ij} , is defined as:

$$\tau_{ij} = 2\mu s_{ij} \tag{4.3}$$

where μ is the fluid kinematic viscosity and s_{ij} represents instantaneous rate of strain tensor defined as:

$$s_{ij} = \frac{1}{2} \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{4.4}$$

The N-S equations completely and entirely describe the turbulent flow field from the largest to the smallest scales of motion. Their numerical solution requires spatial and temporal resolutions capable of resolving the so-called Kolmogorv scales. A Direct Numerical Simulation (DNS) of the N-S equations can be shown to scale with $\text{Re}^{11/4}$, and is impractical for an engineering application at high Reynolds number. The flow field studied in this paper has a Reynolds number of 2×10^7 and would require about 110 exabytes of memory. Therefore we need to model the turbulent flow.

4.2.1.1 Reynolds Averaged Navier-Stokes (RANS) Approach

The Reynolds Averaged Navier-Stokes (RANS) approach is a commonly used method for solving an engineering problem using CFD. In this approach Reynolds decomposition is used to decompose the instantaneous velocity and pressure fields into mean and fluctuating components, mathematically expressed in the form $a_i = A_i + a'_i$, and followed by ensemble-averaging the original N-S equations. Thus, as an example, in this convention, u_i , U_i , and u'_i represent the time-dependent instantaneous, time averaged, and time-dependent fluctuating parts of the velocity component in *i*-direction respectively. The RANS equations are then expressed by equations 4.5 and 4.6. Here, we describe the turbulent flow statistically in terms of the mean velocity field $U_i(\mathbf{x}, t)$ and mean rate of strain $S_{ij}(\mathbf{x}, t)$ instead of the instantaneous velocity field $u_i(\mathbf{x}, t)$ and instantaneously rate of strain field $s_{ij}(\mathbf{x}, t)$, respectively. These equations are commonly referred to as the Unsteady Reynolds Averaged Navier-Stokes (URANS) equations.

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{4.5}$$

$$\frac{\partial U_i}{\partial t} + \frac{\partial U_j U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left(2\mu S_{ij} - \rho \overline{u'_i u'_j} \right)$$
(4.6)

Let us now consider the terms, $-\rho \overline{u'_i u'_j}$, which is a symmetric tensor known as the Reynolds stresses. These are six additional terms that are challenge introduced into the system of equations as a consequence of the emerges from the Reynolds averaging process. This presents the classical closure problem in fluid mechanics in the fact that the six new independent terms now give us a total of 10 variables to determine using 4 equations. This is often resolved using the turbulent-viscosity hypothesis introduced by Boussinesq in 1877 (see equation 4.7). As per Boussinesq's hypothesis, a relationship is needed between the turbulent stresses and the mean rate of strain, similar to the viscous stress relationship as shown in equation 4.3. However, in this case the constant of proportionality is a fictitious flow variable, called the *turbulent eddy viscosity*, ν_t , shown in equation 4.7.

$$\overline{u_i'u_j'} = \frac{2}{3}k\delta_{ij} - \nu_t \left(\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i}\right)$$
(4.7)

where k is turbulence kinetic energy per unit mass, $k \equiv (1/2) \ \overline{u'_i u'_i}$, and δ_{ij} is Kronecker delta. The determination of this flow variable ν_t is the central element of turbulence modeling approach. All the various eddy viscosity based turbulence models found in literature differ primarily in the way they estimate ν_t . All of the modern turbulence modelling approaches solve additional transport equation(s) to determine ν_t ; this type of modelling approaches are classified on the basis of the number of transports equations involved, and what transport variables are used in the modelled equations. For example, a one-equation turbulence model will involve the solution of one additional transport equations, and a two-equation $k-\omega$ modelling approach will involve transports of turbulence kinetic energy (k), and specific rate of turbulence kinetic energy dissipation (ω) .

The current study uses the SST Menter $k - \omega$ (SST) [19, 20] based IDDES turbulence model. A short description is provided below; however, an interested reader is referred to Zhang et al. (2019) [21] and the original articles of Menter and coworkers [22, 20, 19] for all relevant details.

4.2.1.2 Shear Stress Transport (SST) $k - \omega$ urbulence Model

The $k-\omega$ model replaces the dissipation rate ε used in the $k-\varepsilon$ model of developed by Launder and coworkers (see [88, 89]) with another variable, specific dissipation rate ω , which is defined as $\omega \equiv \frac{\varepsilon}{k}$. This model includes an additional non-conservative cross-diffusion term containing $\Delta k \cdot \Delta \omega$ in the ω transport equation. This crossdiffusion term is used only in regions far from the wall by using a blending function. Thus, the SST model retains the advantages of the $k - \omega$ boundary layer calculation in the near wall region while also retaining the characteristics of the $k-\varepsilon$ model in the far-field freestream flow. The expressions for the eddy viscosity μ_t , and the transport equations are given in Equations 4.8 to 4.15.

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_i} = \widetilde{P_k} - \beta^* k \omega + \frac{\partial}{\partial x_i} \left[\left(v + \sigma_k v_t \frac{\partial k}{\partial x_i} \right) \right]$$
(4.8)

$$\frac{\partial\omega}{\partial t} + U_j \frac{\partial\omega}{\partial x_i} = \alpha \frac{1}{\mu_t} \widetilde{P_k} - \beta \omega^2 + \frac{\partial}{\partial x_i} \left[\left(v + \sigma_k v_t \frac{\partial k}{\partial x_i} \right) \right]
+ 2 \left(1 - F_1 \right) \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial\omega}{\partial x_i}$$
(4.9)

$$v_{\rm t} = \frac{a_1 k}{\max\left(a_1 \omega, SF_2\right)} \tag{4.10}$$

$$S = \sqrt{2S_{ij}S_{ij}} \tag{4.11}$$

$$P_{k} = v_{t} \frac{\partial U_{i}}{\partial x_{j}} \left(\frac{\partial U_{i}}{\partial x_{j}} + \frac{\partial U_{j}}{\partial x_{i}} \right) \to \widetilde{P_{k}} = \min\left(P_{k}, 10\beta^{*}k\omega\right)$$
(4.12)

$$F_{1} = \tanh\left\{\left\{\min\left[\max\left(\frac{\sqrt{k}}{\beta^{*}\omega y}, \frac{500v}{y^{2}\omega}\right), \frac{4\rho\sigma_{\omega^{2}}k}{CD_{k\omega}y^{2}}\right]\right\}^{4}\right\}$$
(4.13)

$$F_2 = \tanh\left[\left[\max\left(\frac{2\sqrt{k}}{\beta^*\omega y}, \frac{500v}{y^2\omega}\right)\right]^2\right]$$
(4.14)

$$CD_{k\omega} = \max\left(2\rho\sigma_{\omega^2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i}, 10^{-10}\right)$$
(4.15)

where $\alpha, \beta, \beta^*, \sigma_k, \sigma_\omega$ and σ_{ω^2} are closure coefficients of the model. These are computed by the blending functions F_1, F_2 and corresponding constants of $k - \varepsilon$ and $k - \omega$ models via the relationships like $\alpha = \alpha_1 F_1 + \alpha_2 (1 - F_1)$, etc. The a_1 , constant in Eq. 4.10 was set to 0.31 per the STAR-CCM+ version 2020.2.1 user manual. A production limiter is used in the SST model to prevent the build-up of turbulence in stagnation regions.

4.2.1.3 Improved Delayed Detached Eddy Simulation (IDDES) Model

As the implementation of the LES approach is computationally expensive for automotive flows, a more practical hybrid RANS/LES approach of DES was proposed by Spalart et al. [15, 16, 17]. Similar in concept to the $k - \omega$ model, a switching function is also implemented by the DES approach to use LES in the regions far from the wall and RANS in the boundary layer regions. The switch between the LES solver and RANS solver is achieved via the computation of two local parameters, a local turbulent length scale, l_T , and a local grid size, ℓ_{LES} .

$$\ell_T \equiv \frac{\sqrt{k}}{\omega} \tag{4.16}$$

$$\ell_{\rm LES} \equiv C_{\rm DES} \ \Delta_{\rm DES} \tag{4.17}$$

A limitation of this hybrid approach is that when the numerical value of ℓ_T and ℓ_{LES} reduces below a critical value, then the LES solver may be erroneously applied inside a boundary layer region. The effect of this local grid size can then be observed as a nonphysical separation being predicted and is thus known as Grid Induced Separation (GIS). GIS is thus a negative consequence of the switching function and is mitigated by modifying the switching function to include a delay based on the wall normal distance and local eddy viscosity [15]. This new approach with the modification to the switching function is called the Delayed DES or DDES. Another version of DES makes a further modification to the switching function between LES and RANS regions with the aim of providing further shielding to the boundary layer regions in high Reynolds number flows [17, 18]. This second modification is called the Improved DDES or IDDES model which has been used for this paper. The IDDES model includes a Sub-Grid SCale (SGS) dependence on the wall-distance that further prevents LES modeling where the wall-distance is much smaller than the boundary-layer thickness.

$$\widetilde{\omega} = \frac{\sqrt{k}}{\ell_{\text{Hybrid}} f_{\beta^* \beta^*}} \tag{4.18}$$

where f_{β^*} is the free-shear modification factor, β^* is an SST $k - \omega$ model constant, and the parameter ℓ_{Hybrid} is defined as:

$$l_{\text{Hybrid}} = \tilde{f}_d \left(1 + f_e\right) \ell_{\text{RANS}} + \left(1 - \tilde{f}_d\right) C_{\text{DES}} \Delta_{\text{IDDES}}$$
(4.19)

4.2.2 Geometry

The geometry used in this study is a full scale Gen-6 NASCAR racecar, previously used in the 2019 NASCAR Cup season. The CAD file consists of a fully-detailed geometry having high resolution descriptions of the external aerodynamic surface, the interior of the driver cabin, NACA cooling ducts, the underhood flow with detailed powertrain and exhaust assemblies, and porous regions for the radiator and gear cooler. This CAD assembly was imported into ANSA v15 and cleaned of all surface tessellation errors. Care was taken to retain all the geometric details. The fully detailed and error-free final surface consisted of 13 million triangles. This surface was then imported into Star-CCM+ and surface-meshed for CFD simulation.

4.2.3 Computational Domain and Boundary Conditions

A sufficiently large computational domain is required to perform an open-road CFD simulation for vehicle aerodynamics. This large domain mitigates the influence of blockage ratio and numerical pressure waves which can occur at boundaries [90, 32, 46]. The test geometry was placed in a Virtual Wind Tunnel (VWT) having dimensions of $211L \times 200W \times 200H$, with the inlet and outlet boundaries being 50L upstream and 160L downstream, where L, W, and H are the respective length, width, and height of the test geometry.

The inlet was placed on the negative x-face of the computational domain and given a velocity of 67.056 m/s (150 mph). A pressure outlet was placed at the positive x-face of the computational domain with a zero-gauge pressure specification. Fu et al. (2019) [29] studied the turbulence modeling effects on the aerodynamic characterizations of a stock racecar subject to yaw. For the crosswind simulations, they used a zero-gradient boundary condition for the side walls. A subsequent study the authors, Misar, A.S., and Uddin, M (2022) [46] shows that such a zero-gradient boundary condition poses nonphysical pressure reflections on the virtual wind tunnel boundaries. Thus, for the crosswind simulations, the upstream side of the VWT was set to a velocity inlet and the downstream side to a pressure outlet. The velocity inlets then had their velocities specified in both x and y components to obtain the correct crosswind angle to simulate the desired yaw, or crosswind conditions. The inlets were given a turbulence intensity specification of 1.0%, and a turbulent length scale specification of 10 mm. In this set of simulations, the inlet velocities were given a constant magnitude. However, a synthetic velocity inlet with an oscillating magnitude, as used by Curley & Uddin [53], has been suggested to more realistically represent open-air turbulence conditions. Due to the computational expense of this approach, and a lack of relevant wind tunnel data to correlate the results, the more simplified fixed magnitude approach was applied throughout. Further discussions of the value of the Curley & Uddin approach will be discussed later.

To cost-effectively emulate a moving ground wind tunnel test scenario, the no-slip floor was given a tangential velocity corresponding to the given freestream velocity, and the wheel rotation was modeled using a local rotation rate for each wheel. A small vertical wall was used to simulate the tire-ground contact patch while maintaining the numerical stability of the simulations [27, 72, 71].

A porous media strategy was developed for modeling the mass flow rates through the condenser, radiator, and fan module (CRFM) to improve underhood flow prediction accuracy. This detail is important because accurate prediction of the underhood airflow was found to be crucial for well-correlated force predictions [77]. The porous media modeling also includes porous baffles to simulate the front and inner grilles of the radiator ducting. Using this approach, the radiator consists of 3 regions: the primary cooling duct, the secondary cooling duct, and the radiator core itself. Porous media modeling was tuned using the RAS-I CFD solver and the C1 configuration in order to achieve a mass flow rate matching with high accuracy to the mass-flow measurements from the wind tunnel [46].

4.2.4 Initialization

The flow field was initialized with the same velocity, pressure and turbulence parameters as the inlet and outlet boundaries, i.e., the freestream velocity, a gauge pressure of zero, a turbulence intensity specification of 1.0%, and a turbulent length scale specification of 10 mm.

4.2.5 Discretization

The computational domain was discretized using the unstructured, hex-dominant "Trimmed cell" meshing algorithm of Star-CCM+. This algorithm takes a reference cell size ("base size" within Star-CCM+), and creates cells whose size is a multiple of 2^n times larger/smaller than the reference cell size where n is an integer. Volume sources were used to refine the cells in regions having high rates of change of the flow field variables. Nine volume sources were placed around the car, and a further eight were placed in regions of interest such as the splitter and spoiler. Prism layers were used on the wetted surfaces to resolve the near wall boundary layers. Eighteen different prism layers were used to ensure that the 1^{st} node height corresponded to a wall $y^+ < 1$. The final RANS and IDDES meshes consisted of 130 and 200 million cells respectively.

4.2.6 Physics Setup

The simulations presented in this paper were performed using the finite volume solver Star-CCM+ version 2020.2.1. All simulations, unless specified otherwise, were performed using segregated flow solver on an unstructured grid using the SIMPLE method. The $k - \omega$ SST-based IDDES turbulence model was used along with its default closure coefficients for all RANS simulations as well as the underlying RANS model of the IDDES simulations. A two-layer all- y^+ wall treatment was used to ensure reasonably accurate boundary layer calculations in complex locations of the geometry where the y^+ was not sufficiently small. A 2^{nd} order discretization scheme was used for the diffusion terms and a 2^{nd} order upwind scheme was used for the convection terms of the momentum equations. For the IDDES cases, the time step was normalized by vehicle length (L) and freestream velocity (U_{∞}) . The non-dimensionalized time step of $\Delta t = 0.00012(L/U_{\infty})$ was used; this corresponds to a nominal CFL (Courant-Friedrichs-Lewy) number of around unity for near-car grids, and even smaller for the far wake regions. This Δt has been reported as a sufficiently small time-step size for automotive IDDES applications [73]; this was also verified through an earlier time-step independence verification study [47]. Six inner iterations were found to be sufficient for all residuals to drop by 3 orders of magnitude within each time-step, see [47].

4.2.7 Stopping Criteria and Data Averaging

The RANS simulations were run for 10,000 iterations and convergence was seen to begin after 4,000 iterations. The IDDES simulations were run for 90 LETOTs, where 1 Large Eddy Turn Over Time (LETOT) = L/U_{∞} , and convergence of force and moment coefficients was seen to begin after roughly 50 LETOTs. Based on this, all RANS results presented in this paper are from an averaging window of the last 4,000 iterations (i.e. averaged between iterations 6,000-10,000), and all IDDES results presented in this paper are from an averaging window of the last 30 LETOTs (i.e. averaged between 60-90 LETOTs).

4.2.8 Computational Resources

The authors have previously observed a significant variation in the aerodynamic coefficient predictions from the CFD of a road vehicle when simulations were carried out using Message Passing Interface (MPI) as the parallelization tool. Thus, care was taken to maintain the same parallelization schemes and hardware consistency throughout this study [23]. All simulations were run on UNC Charlotte's High-Performance Computing clusters using 144 processors across 3 nodes having

4.3 Results and Discussion

The three configurations, examined using the using the four solvers mentioned earlier (RAS-I, RAS-C, DES-I, and DES-C), give a total of twelve simulation and are presented in Table 4.1. Section 4.3.1 presents the percent difference between the force coefficients obtained from the twelve CFD cases and the corresponding WT data. Section 4.3.2 presents the comparison of the accumulated forces between the DES-C and RAS-C solvers. The trends presented in sections 4.3.1 and 4.3.2 were hinted at from the difference in CP prediction on the NASCAR surface from DES-C and RAS-C for configuration C3 in the authors' previous study [47]. The current paper expands the discussion to include all three configurations. And lastly, in order to ascertain which CFD prediction is closer to the WT flow field, section 4.3.3 examines the CP predictions on the surface for C3 with data from DES-C, RAS-C, and WT.

4.3.1 Coefficient Plots

This section will look at the aerodynamic coefficients as predicted from the twelve CFD cases in terms of their percent difference w.r.t the respective WT values. Figure 4.1 reveals the percent difference in CD and CL w.r.t WT values. It can be seen that three general trends emerge. First, both DES solvers have a greater overprediction of CD and CL than their RANS counterparts. This may indicate an increased pressure prediction on the front and rear facing surfaces by the DES solvers as well as an inability to capture the peak suction pressures on the underside of the racecar. Second, both compressible solvers have a slightly reduced percent error as compared to their respective incompressible counterparts. This may indicate a better prediction correlation by the compressible solvers in the regions most susceptible to local compressibility effects such as the splitter suction pressure region. And third, C3 seems to have the largest variance in its predictions between DES-C and RAS-C cases. Thus, C3 is investigated further in section 4.3.3 of this paper. As a reminder of Table 4.1, C3 is the higher splitter gap, zero yaw angle configuration of the racecar.



Figure 4.1: $\%\Delta$ of CD and CL between CFD and WT

Figure 4.2 shows the distribution of front and rear downforce (negative lift) of the racecar. 4.2a shows that generally CLF is overpredicted for all cases. C1 DES-C follows the same trend as all its neighbors but has a negligible difference w.r.t WT. This overprediction in CLF hints at an overprediction in the splitter suction pressure or an overprediction of C_p on the hood, cowl, and windshield surfaces. In 4.2b it is seen that both RANS solvers underpredict CLR, while both DES solvers overpredict CLR. This could indicate an overprediction of C_p on the decklid and spoiler by the RANS solvers and an underprediction of C_p by the DES solvers. This change in trend in the DES solvers w.r.t the RANS solvers may also be attributed to the diffusion of the underbody jet seen in the authors' previous study [47].



Figure 4.2: $\%\Delta$ of CLF and CLR between CFD and WT

Figure 4.3 reports the longitudinal distribution of CL. Racing industry commonly uses the colloquial term "percent front balance" to define the front-to-rear downforce balance of vehicles. As defined earlier, "%_Front" defines this values and is shown in Figure 4.3a. As expected from the previous graphs of 4.1b, and 4.2, the RANS solvers overpredict %_Front. This again points to the C_p predictions on the splitter, hood, decklid, and spoiler surfaces as the possible sources of error as these surfaces play a major role in downforce production. The DES solvers slightly underpredict percent difference in % Front by less than -2%, except for C1 in DES-C, which is closer to -4%. This supports the conjecture that the DES solvers are generally overpredicting ${\cal C}_p$ in an equal proportion in the front and rear parts. In Figure 4.3b, all cases predict L/D around negative 2-3% of WT value, except the two outliers of C3 in RAS-I and C1 in DES-C. Looking specifically at Figure 4.3b, it can be observed that 10 of the 12 simulations report L/D results within a narrow range between -2 to -4%. The specific physics likely producing the 2 outliers is not fully understood, and will be discussed following additional future research pertaining to the effects of crosswind and the effects of splitter gap height.



Figure 4.3: $\%\Delta$ of %_Front and L/D between CFD and WT

4.3.2 Accumulated forces

Next, the accumulated aerodynamic force coefficients along the longitudinal dimension of the vehicle geometry are examined. This will provide a further insight into the development of the pressure field on the vehicle surface. All DES cases are plotted as solid lines and all RANS cases are dashed lines. C1, C2 and C3 are shown in red, blue, and green respectively.

Figure 4.4 shows the accumulated force coefficients. In 4.4(a) it is observed that all DES cases predict higher CD than the respective RANS cases. The differences occur in between location ranges from 0.05 < x/L < 0.30 and 0.70 < x/L < 0.95, corresponding to the hood and decklid regions respectively. Also, C3 has a significant difference between DES-C and RAS-C from 0.30 < x/L < 0.85. This could be a result of the diffused underbody splitter jet in the DES-C case. The diffusion of that jet may indicate higher streamwise wall shear stress in DES-C w.r.t RAS-C and these may contribute towards higher friction drag. In 4.4(b) all RAS-C cases are overpredicting CL in the range 0.05 < x/L < 0.25 and underpredicting CL in the range 0.5 < x/L < 1.0. This is consistent with the observations in Figure 4.2. The front overprediction corresponds to the splitter and front diffuser geometries. The rear underprediction seems to be an effect of the underbody flow. In 4.4(c) C1 and C2 are well matched for both DES-C and RAS-C solvers. The largest difference is observed in the range 0.3 < x/L < 1.0 from C3, which is a zero degree yaw configuration. The NASCAR geometry is inherently asymmetric along the longitudinal axis and this asymmetry is the cause for a non-zero sideforce even in zero yaw configuration. The significant prediction difference of CS between DES-C and RAS-C for C3 indicates a pressure field difference on the side surfaces. The fact that the difference starts just downstream of the front tires suggests an influence of the front wheel wakes.



Figure 4.4: Plot of accumulated force coefficients from each configuration from RAS-C and DES-C solvers. Top: (a) Accumulated CD, middle: (b) Accumulated CL, and bottom: (c) Accumulated CS

Further exploring the effect of both DES-C and RAS-C solvers, Figure 4.5 analyzes the difference in the accumulated force coefficients between DES-C and RAS-C solvers for all 3 configurations. For this, the RAS-C cases were taken as a baseline and the DES-C predictions are reported w.r.t the RAS-C predictions. Thus, all positive differences are overpredictions in the DES-C case and vice versa. C1, C2, and C3 are shown in red, blue, and green respectively. In 4.5(a) C3 has the largest difference in CD between DES-C and RAS-C relative to the differences seen for C1 and C2. This is due to three factors: (i) higher CD contribution from the range 0 < x/L < 0.075corresponding to the splitter and front fascia, (ii) a smaller drop near the cowl region at X/L = 0.325, and (iii) a larger drag contribution from the spoiler located beyond x/L = 0.95.

In 4.5(b) the highest overprediction is observed with DES-C in C1 at x/L = 0.05. C1 is the low splitter gap case and is thus expected to have a higher splitter suction compared to C2 and C3. 4.2a showed that, for C1, DES-C had a lower CLF prediction compared to RAS-C and fig 4.5(b) further indicates that the splitter suction pressures have different predictions. Also, the consistent downward slope of all three cases from x/L = 0.1 to the rear indicates a strong correlation to the underbody splitter jet flow. Thus, it will be important to inspect the static pressure data from the point probes in the underbody region. C1 also seems to have an unphysical spike at x/L = 0.95and may be coming from a numerically induced noise in post-processing the data. This is left for a subsequent investigation.

In 4.5(c) C1 and C2 have differences of less than 10 counts. The difference coming from C3, the zero yaw case, is very significant and highlights the need to study the probes on the sides of the vehicle. It is interesting to note that, for all three cases, the differences in CD appear downstream of x/L = 0.225. This indicates the wake and outwash generated from the front tires may be playing a significant role on the flow prediction over the doors and thus affecting the sideforce predictions. To enhance the understanding of such a phenomenon, more investigation of the flow field in the near vicinity of the vehicle is required. Data of pressure and velocity was collected from this region of the flow field, allowing a more in depth study. This data was generated from a collection of fifty (50) CFD-generated point probes placed in the flow field. From each point probe location five scalars, including the static and total pressure coefficients and the three components of the velocity vector, are collected. Because the corresponding WT data for these point probes is not available, establishment of the overall veracity of the CFD simulations is required prior to analysis of the flow field. The data will be analyzed and presented in a subsequent paper.



Figure 4.5: Plot of differences in accumulated force coefficients from DES-C solver w.r.t RAS-C solver from each configuration. Top: (a) Delta accumulated CD, middle: (b) Delta accumulated CL, and bottom: (c) Delta accumulated CS

4.3.3 Pressure Probe Plots

Figures in this section compare the static pressure data on the vehicle surface for C3 as obtained from DES-C, RAS-C, and WT. Each figure has a plot overlaid on the vehicle geometry. The yellow/gold dots show the physical location of the pressure probes on the vehicle. Each plot has it's own grouping of pressure probes numbered as [P1, P2, ...]. The green circles show the C_p values obtained from the WT. The blue triangles show the CFD predicted C_p values from DES-C. And lastly, the red squares show the CFD predicted C_p values from RAS-C. Examination will begin by first looking at the splitter and underbody regions, then the spoiler region, the hood region, and finally the sides.

4.3.3.1 Splitter and underbody region

Figure 4.6 is a plot of the surface C_p distribution on the splitter of the vehicle for C3. This region of the vehicle has the lowest ground clearance and the strongest suction pressures. Being the most upstream part of the vehicle geometry, this region has the least impact on upstream flow predictions. Towards the sides at locations P1 and P5, the predictions of WT, DES-C, and RAS-C are well correlated. RAS-C continues this good correlation at all interior locations. However DES-C shows a significant underprediction of the suction pressure and fails to capture the peak suction pressure at the central P3 probe. This suggests that, in the DES-C flow field prediction, there may be a local separation bubble slightly downstream of the P3 probe location. Such a flow prediction was indicated in the observations of the authors' previous work [47]. The impact of such a local separation bubble should be seen more clearly in the front diffuser region (also called the splitter extension panel).



Figure 4.6: Plot of C_p distribution at pressure probes on the splitter

Figure 4.7 is a plot of the surface C_p distribution on the trailing edge of the splitter extension panel of the vehicle for C3. Immediately it is seen that the WT seems to be predicting an outlier at P3 having significantly larger suction pressure as compared to the other locations (by about 25%). This seems to be an unphysical phenomenon and requires further flow field data from the WT experiment. Apart from this, a trend similar to that observed in the splitter region is seen. Towards the sides at locations P1 and P4, the values from WT, DES-C, and RAS-C are well correlated. At the interior points of P2 and P3, both DES-C and RAS-C predictions are correlated to each other, but both underpredict w.r.t the WT value. This seems to suggest that neither CFD solver is able to predict the flow acceleration, based upon the divergence from the WT values. Some additional WT pitot tube information from the trailing edge of the splitter extension panel may be required. This would allow more robust comparisons of the streamwise velocity from the different simulations.



Figure 4.7: Plot of C_p distribution at pressure probes on the splitter extension panel

Figure 4.8 is a plot of the surface C_p distribution on the floor of the vehicle for C3. Generally, the floor suction pressure is underpredicted except for RAS-C prediction at P1. A basic understanding of Bernoulli's Principle implies that an underprediction of underbody suction pressure indicates an underprediction of streamwise velocity in the underbody flow. This figure shows us that the RAS-C prediction is better correlated to the WT data than the DES-C prediction. This is consistent with the authors' earlier observations from a scalar of ΔC_P on the underbody surface [47]. It is also wise to remember that the WT in itself is a simulation of open road conditions. The rolling belt used to simulate a moving ground cannot be infinitely rigid and thus may have an induced vertical oscillation due to the vehicle's underbody suction. The pressure and velocimetric data from a coastdown test may be required to ascertain the true state of underbody suction conditions.



Figure 4.8: Plot of C_p distribution at pressure probes on the floor

Figure 4.9 is a plot of the surface C_p distribution on the LHS floor of the vehicle for C3. At these locations, DES-C predictions are closer to WT values, while RAS-C is underpredicting the suction pressure at P1, P2, and P5, and overpredicting the suction pressure at P3 and P4. This is a region of the flow field where air from outside the vehicle's footprint rolls over the edge of the side skirts and enters the underbody flow [47]. Figure 4.9 suggests the transient solver of DES-C is better able to capture the impact of the dynamic flow on the local pressure field.



Figure 4.9: Plot of C_p distribution at pressure probes on the LHS side skirts

Figure 4.10 is a plot of the surface C_p distribution on the RHS floor of the vehicle for C3. In the near wake of the front tires at P1 and P2, and in the region of the rear tire squirt at P5 and P6, suction pressure is underpredicted with the exception of P1 for DES-C. Suction pressure is overpredicted slightly upstream of the exhaust pipes at P3 and P4. Larger discrepancies exist between DES-C and RAS-C predictions at P1, P2, P5, and P6, all of which are within the influence of the tires. The tire squirt and the near wake region contain many flow structures that have a range of frequency and length scales. Accurately predicting these regions is difficult, and the simulation setup may have a significant effect on downstream flow structures, such as those that are being captured in this examination. Different wheel rotation modeling strategies may have a significant impact on these areas [71].



Figure 4.10: Plot of C_p distribution at pressure probes on the RHS side skirts

Figure 4.11 is a plot of the surface C_p distribution on the fuel cell and rear crash structures of the vehicle for C3. All CFD predictions are underpredicting the suction pressures except for P5 in DES-C, which shows good correlation to the WT corresponding value. At P2 and P3, RAS-C seems to be more aligned to WT values, but both solvers have nearly identical predictions at P1 and P4.



Figure 4.11: Plot of C_p distribution at pressure probes on the fuel cell and rear crash structures

4.3.3.2 Spoiler region

Figure 4.12 is a plot of the surface C_p distribution on the rear windshield for the C3 vehicle. DES-C predicts a higher C_p at all three points w.r.t both WT and RAS-C. At P2 and P3, RAS-C and DES-C predictions are very closely matched, but at P1 RAS-C significantly underpredicts compared to both WT and DES-C. It will help the reader to know that P1 is located slightly inboard of the sharkfin. Thus, the RAS-C solver seems to be predicting a much higher tangential velocity along the sharkfin. These CFD predictions also help explain why the DES-C has a higher CS prediction than RAS-C seen in Figures 4.4(c) and 4.5(c).


Figure 4.12: Plot of C_p distribution at pressure probes on the rear windshield

Figure 4.13 is a plot of the surface C_p distribution on the decklid of the vehicle for C3. The P3 value from RAS-C is is best correlated to WT, all other CFD predictions are overpredicted with DES-C having a higher overprediction than RAS-C. The authors have seen in their previous work [47] that RAS-C predicted a smoother flow in this region, whereas DES-C flow field predictions indicated many more localized separation bubbles. In this study RAS-C is better correlated the limited WT surface pressure data available.



Figure 4.13: Plot of C_p distribution at pressure probes on the decklid

Figure 4.14 is a plot of the surface C_p distribution on the Spoiler of the vehicle for C3. The thirty-two (32) physical pressure probes used in the WT are organized into three rows; (a) top, (b) middle, and (c) bottom. The viewpoint is from the rear looking forward, i.e., RHS of the racecar is on RHS of the plot.

On the top row P1, P2, and P3 for both DES-C and RAS-C are well correlated to WT values, and RAS-C has an additional well-correlated prediction at P8. DES-C has a significant overprediction w.r.t WT from P4 to P11. RAS-C predictions share this trend except for locations P6 and P9 which are underpredicted. Both DES-C and RAS-C have similar predictions for the sharkfin side from P1 to P5. From P6 to P11 DES-C significantly overpredicts relative to RAS-C. These locations are directly downstream of the decklid discrepancies seen in Figure 4.13. Thus, it may be that the flow features emerging from the C-Pillar region are being resolved differently in the DES-C and RAS-C methods. As mentioned earlier, the flow field investigation is left for a subsequent paper.

On the middle and bottom rows, again see these general trends continue, particularly the tendency of DES-C to overpredict C_p at locations P5-11 w.r.t both RAS-C and WT values. This helps explain the higher CD and CLR prediction of DES-C seen in Figures 4.1a and 4.2b respectively. However, RAS-C is also generally overpredicting C_p w.r.t WT values in Figures 4.12, 4.13, and 4.14. RAS-C also underpredicts C_p w.r.t WT values in Figure 4.11. These predictions by RAS-C would suggest an overprediction of CLR similar to DES-C, but in fact it can be seen from 4.2b that RAS-C underpredicts CLR. From Figures 4.4(b) and 4.5(b) the underprediction of CLR by RAS-C is mostly from the range of 0.7 < x/L < 1.0. This suggests an influence of the rear wheel wake on the underbody flow affecting CLR. Again, the flow field investigation is left for a subsequent paper.



Figure 4.14: Plot of C_p distribution at pressure probes on the spoiler (a) top row, (b) middle row, and (c) bottom row

4.3.3.3 Hood region

Figure 4.15 is a plot of the surface C_p distribution on the front fascia of the vehicle for C3. Both CFD solvers are overpredicting the surface C_p at all three points w.r.t WT values, with a greater overprediction seen in DES-C relative to RAS-C. These overpredictions contribute towards the higher CD predictions seen in Figure 4.1a. This suggests that CFD is overpredicting the stagnation region and overpredicting mass flow rate through the front bypass ducts, the front grille, and the splitter region. As a reminder, the front grille porosity was tuned using the anemometer data from WT utilizing the RAS-I solver for all three configurations. For C3, the radiator mass flow rate changes by less than 2% across all four CFD solvers. Further understanding and refinement of this region would be possible with enhanced experimental data, collected from additional WT pitot tubes and anemometers located in the front drag ducts. Also, the DNS work of Curley & Uddin (2015) using a surface mounted cube suggests that the use of a steady inlet velocity can result in wake prediction inaccuracies [53]. All the simulations presented in this paper use such a steady inlet velocity and overpredict the drag. The authors thus recommend further investigation of the Curley approach, using a perturbed inlet velocity for better turbulence simulation, to improve the drag prediction of IDDES simulations.



Figure 4.15: Plot of C_p distribution at pressure probes on the front fascia

Figure 4.16 is a plot of the surface C_p distribution on the Hood of the vehicle for C3. At the central P2 location closer to the nose of the vehicle, both CFD solvers are slightly underpredicted in suction pressure. This suggests a slower streamwise velocity as the flow comes over the leading edge of the hood. This may be a consequence of the mass flow redistribution suggested by the front fascia data seen in Figure 4.15. The outward locations P1 and P3, located on the hood flaps near the cowl region, have a significant overprediction of suction pressure. WT has a C_p value close to zero at these locations; this implies a velocity magnitude close to freestream value as the static pressure is very close to ambient air pressure. However, both CFD solvers predict a significant suction pressure at these points. This suggests the cowl stagnation bubble may be underpredicted in CFD.



Figure 4.16: Plot of C_p distribution at pressure probes on the hood

Proper resolution of the mass flow rates through the CRFM module is crucial for accurate and reliable force coefficient predictions [77, 47]. To develop a deeper understanding of the mass flow trends suspected from the analysis of Figures 4.15 and 4.16, observe the mass flow rates through the front grille, the front drag ducts, and underneath the splitter. Figure 4.17 is a scalar showing the Mach number distribution and mass flow rates through these planes. Figure 4.17 indicates that DES-C has a higher mass flow rate through the front grille and front drag ducts by 0.03 kg/s, or 0.9% more than the RAS-C prediction. WT anemometer data reports that the radiator mass flow is exactly in between the RAS-C and DES-C predictions. This small difference, if considered in isolation, has the effect of reduced cooling drag prediction in the DES-C case. Similarly, DES-C has a lesser mass flow rate through the splitter entry region by 0.09 kg/s, 1.6% less than the RAS-C prediction. And again at the splitter throat and exit, DES-C has a lesser mass flow rate prediction by 3.9% and 3.8% respectively. Taken in isolation this would imply a reduced CLF in DES-C. However, DES-C relative to RAS-C has more mass flow through the radiator and front bypass ducts and reduced mass flow through the splitter region. Thus, DES-C is forcing more air around the front fascia, causing the higher C_p prediction on the front fascia and hood regions. This is consistent with observations in Figures 4.15 and 4.16 as well as in the surface C_p observed in authors' earlier study [47].



Figure 4.17: Plot of Mach number and mass flow rate distribution at the splitter and front face geometry (a) top: RAS-C, (b) bottom: DES-C

Figure 4.18 is a plot of the surface C_p distribution on the P1 (engine filter,) P2

(roof front), P3 (cabin filter), and P4 (rear fascia). Consistent with the observations so far, both CFD solvers are underpredicting the suction pressures at all points. The underpredictions at locations P2 and P4 contribute towards a reduced CD prediction. This underprediction of suction pressure at location P4 is an observation consistent with those of Zhang et al. [21]. The most significant discrepancy are seen at P3, suggesting the pipe flow through the cooling ducts may need further validation with anemometer data. It also suggests that DES-C is predicting higher frictional losses through the cooling ducts.



Figure 4.18: Plot of C_p distribution at pressure probes on the P1 (engine filter,) P2 (roof front), P3 (cabin filter), and P4 (rear fascia)

Figure 4.19 is a plot of the surface C_p distribution on the upperbody centerline of the vehicle for C3. P1, P2, P4, and P5 have been seen in Figures 4.12, 4.13, 4.15, and 4.16 respectively. P3, on the front windshield, shows the same trend as the other four points. This suggests that the entire upperbody flow prediction in CFD may have excessive skin friction or wall shear stress. Thus, the wall modeling in terms of both wall roughness as well as boundary layer growth needs to be studied in greater depth. Also, as the locations shown here have a higher C_p prediction relative to WT, they contribute to a higher CL prediction seen in Figure 4.1b, with the predicted CL being higher for DES-C relative to RAS-C.



Figure 4.19: Plot of C_p distribution at pressure probes on the upper body centerline

4.3.3.4 LHS and RHS regions

Figure 4.20 is a plot of the surface C_p distribution on the LHS of the vehicle for C3. DES-C is overpredicted at P1 and P6 but underpredicted at P3-5 and P7-9. RAS-C is overpredicted at P1, P4, and P6 but underpredicted at P9. The P1 overprediction of suction pressure in both CFD solvers again suggests more mass flow coming across the front fascia. Verification of this hypotheses requires anemometer or pitot tube data at appropriate locations. DES-C and RAS-C are in good correlation of each other at P2, and P5-9. At points P3 and P4 the influence of flows over the hood region, and the turbulent wake coming from the front left tire has to be considered. This requires a further study of the effects of wheel rotation modeling and flow field validation data. In general, it was seen that DES-C predictions are further away from WT values than RAS-C predictions. This indicates a greater CS discrepancy in the DES-C solver.



Figure 4.20: Plot of C_p distribution at pressure probes on the vehicle's LHS

Figure 4.21 is a plot of the surface C_p distribution on the RHS of the vehicle for C3. DES-C is overpredicted at P6 and P8 but underpredicted at P2-5, and P9. RAS-C is overpredicted at P4, P5, P7 and P8 but underpredicted at P1 and P9. The trends and effects seen here are similar to those seen in Figure 4.20. In general, DES-C has more positive C_p predictions than RAS-C for points P3-P9 in Figures 4.20 and 4.21. These are contributing to the higher CS prediction of DES-C seen in Figure 4.4(c).



Figure 4.21: Plot of C_p distribution at pressure probes on the vehicle's RHS

4.4 Conclusions

In this paper, the pressure field predictions on the surface of a detailed, full-scale, Gen-6 NASCAR racecar in three configurations using RANS and IDDES turbulence modeling approaches in both incompressible and compressible modes were investigated. The force and moment coefficients were validated against wind tunnel data from WindShear. This facility utilizes an open-jet, closed-return configuration with a rotating belt for moving ground simulation and boundary layer suction to minimize any boundary layer buildup upstream of the rolling belt. It was found that all RANS cases had the drag and lift predictions within 0-5% of wind tunnel predictions, whereas the IDDES cases predicted the drag and lift between 3-13% of wind tunnel predictions. Additionally, it was found that in both turbulence modeling approaches, the compressible solver reduced the prediction discrepancies by up to 3% for both drag and lift predictions relative to the incompressible solver predictions. Hence, in this paper, the C_P predictions between the DES-C and RAS-C turbulence modeling approaches were investigated. A detailed comparison of the predictions between compressible and incompressible solvers is left for a future study by the authors.

As it was found that configuration C3 had the highest discrepancy between the RANS and IDDES turbulence modeling approaches in compressible mode, this configuration was chosen for further investigation. This was done by studying the C_P distribution on the surface of the racecar via data experimentally collected from ninety-five (95) static pressure probes located on a full-scale wind tunnel model. It was found that DES-C is unable to capture the peak suction pressure underneath the splitter. DES-C also predicted higher C_P relative to RAS-C on the front fasica, hood, decklid, RHS of the spoiler, and fuel cell surfaces. These differences contribute to the DES-C solver overpredicting both CD and CL. The uniform DES-C overpredictions of C_P around the racecar resulted in % Front predictions very well correlated to wind tunnel values. In contrast, RAS-C produces a net CL predictions well correlated to wind tunnel values. However, this correlation was a result of cancellation errors in CLF and CLR predictions, with the Front/Rear balance being significantly in error by 4-6%. Further, both DES-C, and RAS-C struggled to predict the correct suction pressure values in the underbody flow. While this helps to explain the overprediction of CLF and CLR by the DES-C solver, the underprediction of CLR by RAS-C is not fully explained from this investigation. For RAS-C, the source of underprediction in CLR may be influenced by the rear tire wakes and the inward flow across the side skirts. Both solvers also had significant discrepancies in predicting the C_P on the decklid and spoiler surfaces; this points to the flow over the rear windshield and C-Pillars being resolved differently. These predictions require a further investigation of the associated flow structures.

Finally, it was found that in some regions both DES-C and RAS-C had C_P predictions well-correlated relative to each other, but with both having discrepancies relative to wind tunnel predictions. These regions were the outer edges of the splitter, the trailing edge of the splitter extension plate, the RHS side skirts around the exhaust manifold, fuel cell, rear windshield, sharkfin side of the spoiler, front fascia, and rear fascia. This suggests that the C_P predictions could benefit from further tuning of the CFD framework such as the closure coefficients of the $k - \omega$ turbulence model.

CHAPTER 5: (ARTICLE 4) INSIGHT INTO THE TURBULENT FLOW AROUND AN IDEALIZED ROAD VEHICLE USING THE DYNAMIC MODE DECOMPOSITION APPROACH

5.1 Introduction

The reduction of the aerodynamic drag force remains a core objective of vehicle aerodynamic development, and is motivated by the desire to reduce fuel consumption [35, 91]. Researchers have attempted to achieve aerodynamic drag reduction through different types of passive flow control devices such as the front bypass ducts [86], rear bypass ducts [92], and various deflector designs [93, 94, 95]. One of the major limitations of passive flow control devices is that, once installed, they can be difficult to remove or modify. Thus, researchers have turned to active flow control devices to achieve flexibility in optimization [34]. Examples of active flow control include a variety of synthetic jet and suction systems [34, 96, 97, 98, 99]; interested readers are refereed to the review articles by [100, 101] for further details.

Recent technological advances in the automotive industry have shifted the focus of transportation research from human-operated-and-controlled fossil-fuel-based vehicles to electrified (EV) connected and automated vehicles (CAVs). As a results there is a growing interest on the prediction of aerodynamic characteristics in adaptive driving conditions. An example is platooning, where drag reduction is desired via vehicle-tovehicle interaction of aerodynamics where one or more trailing cars follow a lead car in close proximity. Active control systems are believed to make platooning feasible for all vehicles as autonomous vehicles allow for closer proximity due to reduced reaction times [33, 102, 103]. Before a control signal can be applied to the moving vehicle, predictions of the future state of aerodynamic forces and moments are required. A Reduced Order Model (ROM) can be used to make future state predictions of the aerodynamic flow field [35]. For adaptive systems, the future state predictions can then be coupled with a control input to obtain the desired performance characteristics [36].

Previous studies on the adaptation of aero-devices largely relied upon time-averaged wind tunnel experiments or Reynolds-Averaged Navier-Stokes (RANS) based numerical methods. Fluid flows around road and race vehicles are highly turbulent and consist of many dynamic coherent structures that are characterized by a wide range of length and time scales. The evolution and convection of these structures gives rise to macroscopic spatio-temporal patterns [12, 104, 35, 47]. Hybrid turbulence modeling simulation approaches such as Improved Delayed Detached Eddy Simulation (IDDES) have shown greater success at elucidating these finer vortical structures in the flow field. The challenge with such Scale-Resolved Simulation (SRS) approaches comes from the grid resolution requirements for the high Reynolds number flow field that they try to resolve. It is seen that the spatio-temporal domain must be resolved to the so-called Taylor scales [42, 5, 47]. Such SRS approaches are resource-prohibitive, since the onboard controller on a moving vehicle would likely not have the processing power and time needed to solve a transient flow field while attempting to implement real-time control of the vehicle's trajectory. Thus, there is a need for a Reduced Order Model (ROM) that can provide fast, accurate, and reliable flow predictions utilizing feasible computational resources; decompose the fluid flow into its constituent parts can be very helpful in this regard. Researchers in this field have largely resorted to methods of modal decomposition to analyze the flow field [39, 38, 35].

Proper Orthogonal Decomposition (POD) has been a popular method for the modal decomposition of fluid flows [105, 106, 107, 108]. However, the POD modes are arranged by energy and not by dynamical importance, contain a mix of frequencies, and have unclear truncation criteria [109]. In recent times researchers have used Dynamic Mode Decomposition (DMD) which is a data-driven linearization algorithm that can decompose a set of data into its constituent modes and extract the associated oscillation frequencies of each mode [37]. These constituent modes and their associated oscillation frequencies can then be used to make future state predictions of the system [110, 111]. DMD has shown success when applied to a variety of fluid dynamic problems including water jets [111], backward-facing step [112], circular cylinder wakes [113, 114, 115], Poiseuille flow, supersonic jet [116], open cavity flows [117], boundary layer flows [114], airfoil, and hydrofoil flows [118, 119]. DMD has been seen to be adaptable and many variants exist. Interested readers are directed to Kutz's book [39] and Schmid's review paper [38] for further details.

All of the studies cited above applied DMD to relatively simple flow fields at low Reynolds numbers. A Ground Vehicle (GV) has an associated flow field that is much more complex and at orders of magnitude higher Reynolds number which implies a larger spread of length and time scales within the flow field [104, 47]. Fewer studies have applied DMD to such separation-dominated flows, high Reynolds number flows. [35] performed DMD on a DrivAer geometry at a Reynolds number of 4.8×10^6 ; note that the DrivAer model, developed by [59], is a simplified rendering of a highly complex vehicle geometry. Ahani's work was primarily focused on comparing the obtained mode shapes from DMD to those obtained from POD. Another study with the DrivAer geometry by [120], performed a low-pass filtering with a cutoff frequency of 10 Hz before the data was processed by the DMD algorithm, and thus, filtered out all the complexities associated with a high Reynolds number flow.

As mentioned earlier, the development of a ROM capable of producing reliable future state predictions will be very useful for the on-road adaptation of the CAVS. However, we needed an engine for this ROM development. Based on the currently available mathematical tools for fluid flow characterization, we anticipated that DMD is a strong candidate. Thus, the objective of this study is to analyze the effectiveness of the DMD methodology in reconstructing the flow field round a moving GV at a high Reynolds number using data generated from an IDDES based CFD simulation. [109] states that the weaknesses of DMD include the requirement to obtain time-resolved data with high resolution, and the metrics to identify dominant modes. In order to obtain such a "reliable" DMD for the high Reynolds number fluid flows, certain parameters pertaining to the DMD requirements must be determined: (a) length of one data sampling window, (b) data sampling frequency, and (c) number of data samles for converged ensemble averaging. Additionally, we need to know whether it is necessary to go through the Singular Value Decomposition (SVD) step, as seen in the existing DMD algorithm, and if so, the truncation criteria for the SVD need to be defined. Also, it is important to know whether the inverse transformation, i.e the reconstruction of the flow field from the DMD modes suffers from the artifacts of spurious high-frequency modes or other noise. As well, we to know how much flow energy is conserved when the flow field is reconstructed, and the minimum energy that must be retained when performing a low-dimensional transformation of the system.

In this paper, we attempted to address these questions by applying DMD to a high Reynolds number, separated flow past an idealized road vehicle, the Ahmed body geometry [58] for which extensive experimental and CFD data are available for correlation and validation. We chose to proceed with the 35° slant angle Ahmed body model variant as the flow over this model shows all the salient features of the flow over a Sport Utility Vehicle (SUV) -type of vehicle that is the subject of the next phase of our work [121].

The work flow of this study involves first perming DMD on a canonical 2D cylinder flow at a low Reynolds number to verify the accuracy of the DMD output. Next, we performed the CFD simulation of the GV and validated the CFD results against published experimental data [58, 122]. We then performed a DMD analysis using the data collected from the CFD simulation. Modifications to both the CFD data generation strategy along with the creation of a filtering process for removing spurious modes obtained from the model decomposition. The subsequent sections further explain the methodology that was used and the results that were obtained. The methodology developed in this paper can be applied to generic vehicle shapes, like the DrivAer, and subsequently to more complex real-life road vehicles to develop ROMs which can predict on-road characteristics of the vehicle subject to changes in vehicle operating conditions, such as the CAVs in a platoon.

5.2 DMD Equations

The first step in the DMD process involves storing the data in a vector form, $X_i^N = \{x_i^1, x_i^2, ..., x_i^N\}$, where the subscript *i* represents the *i*th element of the grid where the snapshots of the flow field were taken, and *N* is the total number of time snapshots collected. Thus, each time snapshot x^n , is a vector containing data from all *m* grid elements at time instant *n*. If we expand the vectors for the grid elements, we can build the complete dataset in matrix form as shown in equation 5.1

$$X = \begin{bmatrix} x_1^1 & \cdots & x_i^N \\ \vdots & \ddots & \vdots \\ x_m^1 & \cdots & x_m^N \end{bmatrix}$$
(5.1)

In DMD approach, the collected data set from a dynamical system is represented as a coupled system of ordinary differential equations, as given in equation 5.2, which itself contains non-linear relations in spatial and temporal domains.

$$dx/dt = f(x,t) \tag{5.2}$$

The idea is to represent data from the non-linear, complex system as a locally linear regression such that $x_{k+1} = Ax_k$, where A is then chosen to minimize $||x_{k+1} - Ax_k||_2$ over k = 1, 2, 3, ..., N-1. Since we have collected the data from the system, x_{k+1} and

 x_k are known, but the function relating them is unknown.

$$x_{k+1} = F(x_k) \tag{5.3}$$

The DMD approach then constructs a locally linear approximation of the dynamical system:

$$\frac{dx}{dt} = Ax \tag{5.4}$$

This ordinary differential equation (ODE) form of the dynamical system is advantageous as, with initial conditions, we have a well known solution

$$x(t) = \sum_{k=1}^{N} \phi_k \, \exp(\omega_k t) b_k = \boldsymbol{\Phi} \, \exp(\boldsymbol{\Omega} t) \, \boldsymbol{b}$$
(5.5)

where b_k is the amplitude of each mode, ϕ_k are the DMD modes (mode shapes involving the eigenvectors of A), and ω_k are the continuous-time eigenvalues of A. The matrix that results as a product of terms, $\exp(\Omega t)$ **b**, is also referred to as the "timedynamics" of the system as it contains the information associated with the frequency, amplitude, and growth rates for all of the modes. Now, when the dimensions of X are large, A becomes impossibly large to mathematically work with. The DMD process circumvents this through its eigen-decomposition of A by considering a rank-reduced representation, \tilde{A} , which has the same non-zero eigenvalues as A, and is obtained by performing SVD of X using the collected data.

$$X \approx U\Sigma V^* \tag{5.6}$$

In equation 5.6, X is a rectangular data matrix of size $m \times n$, U is a complex unitary matrix of size $m \times n$ that contains the left singular vectors which are the POD modes, Σ is a rectangular diagonal matrix of size $m \times n$ having positive real number as its diagonal elements and V^* is a complex unitary matrix of size $n \times n$ and * represents a complex conjugate transform. The diagonal elements σ_i of Σ_{ij} are the singular values of X. Next, the matrix A may be obtained by using the pseudo-inverse of X, shown in equation 5.7

$$A = X' V \Sigma^{-1} U^* \tag{5.7}$$

In practice, since A can be computationally prohibitive to calculate, \tilde{A} is computed by way of a unitary transform of A as shown in equation 5.8

$$\tilde{A} = U^* A U = U^* X' V \Sigma^{-1} \tag{5.8}$$

With \tilde{A} we can now create a low dimensional subspace of A

$$\tilde{x}_{k+1} = \tilde{A}\tilde{x}_k \tag{5.9}$$

an can compute the eigendecomposition of \hat{A}

$$\tilde{A}W = W\Lambda \tag{5.10}$$

where columns of W are the eigenvectors of \tilde{A} and the diagonal elements, λ_k , of Λ are the DMD eigenvalues. Now we can use the eigendecomposition of \tilde{A} to reconstruct the high dimensional DMD modes. The eigenvalues of A, ω_k , are expressed in terms of the diagonal elements λ_k of Λ which are scaled logarithmically according to the relation $\omega_k = ln(\lambda_k)/\Delta t$. The eigenvectors of A are given by equation 5.11

$$\Phi = X' V \Sigma^{-1} W \tag{5.11}$$

The mode amplitudes may be calculated as

$$b = \Phi^{\dagger} x_1 \tag{5.12}$$

where [†] denotes the adjoint operator, ϕ_k , ω_k and b_k may now be used in equation 5.5 to obtain system state predictions. The interested reader is again directed to the original articles and review papers for a more detailed description of the DMD process [123, 124, 38]. Lastly, equation 5.5 can be rewritten as equation 5.13 [125]

$$x_{i} = \sum_{j=1}^{N-1} b_{ij} \Phi_{Norm,j}(x, y)$$
(5.13)

Kou and Zhang [125] then used this representation to extract a new parameter I_j which denotes the influence of a mode on the entire sampling window as opposed to only at the initial condition. I_j is defined as

$$I_j = \int |b_j(t)| \approx \int_{i=1}^N |b_{ij}| dt \qquad (5.14)$$

The parameter I_j was proposed as an improved method of mode selection. This concept was further modified by Ahani and Uddin [35] and the integral term was replaced with a Root Mean Squared(RMS) term. This new RMS method was used for mode selection.

5.3 Methodology

We investigated three cases in this study. Case 1 (C1 here in after) involves a low Reynolds number flow past a 2D circular cylinder which is a simplified and relatively well known case, and was used for initial validation of the DMD process. The second and third cases (C2 and C3, respectively) used the Ahmed body geometry in a high Reynolds number flow. C2 and C3 differ in regards to the extents of the computational domain and the associated boundary conditions. C2 used Ahmed's (1984) [58] wind tunnel dimensions as the computational domain. However, wind tunnel setup of Ahmed results in a blockage ratio of 4% and necessitates blockage ratio corrections. Since, most vehicles are run in an open-air (OA) configuration we considered it is important to switch our Virtual Wind Tunnel (VWT) setup to an OA configuration to be a better resemble real world driving environment. For C3, based upon the authors prior experience, the extents of the computational domain were significantly increased [46].

5.3.1 Solver Settings

All CFD simulations were carried out using a commercial finite volume code STAR-CCM+ version 2020.2. For C1, a laminar, incompressible solver was used. The time step size, Δt , was set to $0.3 \times t^*$, where t^* represents one Large Eddy Turn Over Time (LETOT). A LETOT is defined as the amount of time required for the freestream flow to pass over the characteristic length of the geometry a single time, i.e $t^* = (t \times U_{\infty})/L$, where U_{∞} is the freestream velocity and L is the characteristic length scale.

As flow case C2 and C3 represent turbulent flows, incompressible Improved Delayed Detached Eddy (IDDES) solver was used [17, 18]. The IDDES approach represents extensions of the original Detached Eddy Simulation (DES) approach proposed by Spalart and coworkers [75, 76]. DES is a hybrid approach that combines, for computational efficiency, Large Eddy Simulation (LES) in the regions far away from the wall and Reynolds Averaged Navier Stokes (RANS) in the boundary layer region. The switching between LES and RANS is done by computing a local turbulent length scales, l_T , and a local grid size, l_{LES} . However, existing literature reports that LES may incorrectly be applied inside the boundary layer when l_T and l_{LES} drop below a critical value. This can then cause a phenomenon called Grid Induced Separation (GIS), which is a prediction of nonphysical separation due to the local grid size. In the Delayed DES (DDES) approach, GIS is prevented by introducing a delay in the switching function based on the wall normal distance and local eddy viscosity [15]. IDDES, proposed by [17], is the next extension of the DES which combines DDES and and wall modeled LES (WMLE) [126]. In WMLES, RANS is limited to a much thinner near-wall region where the wall distance y is very small compared to the boundary layer thickness, but $y^+ \equiv y u_\tau / \nu$ is significantly large; note that $u_{\tau} \equiv \sqrt{\tau_w/\rho}$ where τ_w , ρ , and ν represent the wall-shear stress, fluid density and viscosity, respectively. IDDES was reported to resolve the issue of mismatch between the modelled log layer and the resolved log layer and broadens the application area by providing well-balanced simulation approach for high Reynolds number turbulent flows.

The RANS region in the our IDDES is solved using Menter's Shear Stress Transport (SST) $k - \omega$ turbulence model [19, 20]. For brevity, mathematical equations related to the RANS, IDDES and SST models are omitted from this paper as there are plentiful of resources for these. Interested readers are referred to the original articles

by Menter and his coworkers [22, 20, 19] for the development of the $k - \omega$ model, and to automotive external aerodynamics article by [21] for all relevant equations.

In C1 and C2, a two-layer wall treatment was used to ensure accurate boundary layer activities. The time step size of $\Delta t = 1 \times 10^{-4} \times t^*$ was found to be sufficient for this setup [42, 73]; this time-step is same order of magnitude as that within the study of decomposition of flow by DMD relative to POD by [35] where they used a DrivAer geometry, a flow with a Reynolds number of 4.8*E*6, and a time step size of $\Delta t = 5.2 \times 10^{-4} \times t^*$. To minimize the effects of domain decomposition in CFD predictions, all simulations were run on UNC Charlotte High Performance Computing (HPC) clusters using 144 processors across 3 nodes having 48 processors each [23].

5.3.2 Geometry, Domain, and Boundary Conditions

For case C1, the circular cylinder with a diameter D = 0.01m was placed in the simulation domain. The longitudinal extents of the computational domain were 5D upstream and 20D downstream of the object respectively. The cross-stream extents were 5D on both sides. The upstream edge was specified as a velocity inlet having a streamwise velocity set to .15m/s, the down-stream edge was specified as a zero gauge pressure outlet, and the top and bottom edges were specified as zero-gradient boundaries. This setup is found in the user guide of Star-CCM+ version 2020.2 and references the work of Daily et al. [127]. This flow corresponded to a Reynolds number of 75. The C1 case had 15 inner iterations and was run for 120 LETOTs, and the last 80 LETOTs of data were used for analyses.

For case C2, the Ahmed body geometry was placed in a VWT of $8L \times 5H \times 5W$ in the stream-wise, vertical and lateral extents, respectively; here L, W and H represents the length, width, and height of the Ahmed body. These dimensions are similar to the physical wind tunnel used in Ahmed's original experiment. The vehicle body was placed at a distance of 2L from the upstream boundary. Boundary conditions were applied to the computational domain to match the wind tunnel setup of [58] and [122]. These included a velocity inlet of 40 m/s applied to the upstream face with turbulence intensity of 0.25% and a turbulence length scale of 10 mm, a 0 Pa gauge pressure applied as an outlet condition to the downstream face, and all other boundaries were specified as no-slip walls. This configuration results in a blockage ratio of 4%.

For case C3, the domain extents were significantly increased to $31L \times 35H \times 31W$ in the stream-wise, vertical and lateral extents, respectively, with the Ahmed body placed at a distance of 10L from the upstream boundary. The side wall boundaries were changed to a velocity inlet and a pressure outlet to prepare for crosswind simulations which will be the subject of a subsequent study. This setup for the sidewall boundary conditions is different from the one used by [29] in their study the turbulence modeling effects on the aerodynamic characterizations of a stock racecar subject to yaw where in which a zero gradient boundary condition was used for the side walls. However, later studies by [46] shows that zero-gradient boundary condition poses nonphysical pressure reflections unless the virtual tunnel in infinitely wide. Additionally, to imitate a moving-ground simulation, the floor of the tunnel was given a tangential velocity equal to the free-stream velocity, the ceiling of the VWT was set as a zero-gradient boundary. This setup was taken from the authors' experience of performing CFD of crosswind simulations [46]. Both C2 and C3 had a Reynolds number of 2.86×10^6 which is several orders of magnitude larger than the 2D cylinder case.

For Cases C2 and C3, each time step was run for 10 inner iterations to ensure that residuals had reduced by at least 3 orders of magnitude. Last 30 LETOTs of data were used for analyses. C2 simulation was run for 160 LETOTs. It was found that the initial transients subsided after 30 LETOTs. Thus, C3 was run for 130 LETOTs. In both C2 and C3, the last 80 LETOTs were used for averaging and data collection. The last 80 LETOTs correspond to about 2 s of physical time and thus provided the opportunity to capture a lowest possible frequency of 0.5 Hz.

5.3.3 Discretization Scheme

The flow case C1 was discretized using polyhedral cells. It has a near wall cell size of 0.05D near the cylinder surface and grows to 0.1D elsewhere in the domain. It has 5 prism layers on the cylinder boundary resulting in a total cell count of 20,000.

For cases C2 and C3, the simulation domain was discretized using unstructured hexahedral cells. To properly resolve the flow around the GV, five refinement volumes were used around the geometry. The finest mesh was set to a size the order of the expected Taylor length scale, λ [122, 42, 12]. Further, to properly resolve the boundary layer flows on all the surfaces, a prism layer mesher was used to ensure that the wall y^+ values are less than unity. In the final mesh, more than 99% surfaces had a y^+ value less than unity. For C2, the mesh consisted of 15.24 million cells. For C3, meshing parameters are kept the same as Case C2 resulting in a mesh of 21.94 million cells.

5.3.4 DMD Workflow

The step-by-step process required to perform a classical DMD is available in detail in [39], however, a brief summary of these steps is provided below:

- Step 1: Collect multiple time snapshots of the system of interest,
- Step 2: Create a low-dimensional subspace using the SVD or Truncated SVD (TSVD) methods,
- Step 3: Obtain an eigendecomposition of the low-dimensional subspace,
- Step 4: Using the eigendecomposed low-dimensional subspace, assemble the mode shapes and their associated oscillation frequencies, called the 'Time Dynamics' or TD for short,
- Step 5: Use the mode shapes and TD to assemble the DMD output equations,

• Step 6: Use the DMD solution to predict (or reconstruct) the flow field.

5.3.5 Data Collection Strategy

Storing the entire 3D flow field data associated with all of the time-averaging window time steps would impractically require more than 3 petabytes of data. Therefore, since we are more focused on the GV aerodynamic force and moment predictions, we collected the static pressure field on the Ahmed body surface. Additionally, to capture the flow field around the GV, we chose eight 8 reference planes around the Ahmed body. For some of these planes, experimental data are available and may be used for future validation steps, such as the wake planes at x/L = 1.077, 1.192, 1.479[122, 104]. The other chosen planes are anticipated to involve flow patterns in critical flow-regimes when crosswind and vehicle interaction simulations are later performed, such as the planes at y/W = 0.5 (or Y=0 center-plane), 0.88, and 1.27 and $Z = 0.5 \times \text{Ground Clearance}, 0.5 \times H, 1.15 \times H, \text{ and } 1.3 \times H.$ Over each reference plane, seven scalar quantities were collected: the pressure coefficient, three components of velocity, turbulent kinetic energy (TKE) or k, vorticity, and the Q-criterion [128]. Thus, instead of storing the entire 3D flow field, we stored only the data from the Ahmed body surfaces and these eight reference planes. By using this strategy we extract about 3.4 TB of data per GV simulation since STAR-CCM+ exports this data in ASCII format with redundancies in the spatial locations. By converting the data to binary format and removing the redundancies, resulting in about 400 GB of binary data per case, which is deemed to be a feasible approach. This study is limited to the analyses of vehicle surface static pressure data. The CFD simulation time-step size implied that the data is sampled at a rate of 4 kHz, and thus, the Nyquist criterion implies that flow structures having frequency of up to 2 kHz can be captured by the DMD reconstruction.

5.4 Results

5.4.1 CFD Validation

CFD predictions of the drag coefficient (C_D) were validated against the wind tunnel measurements of [58] and was also comapred to the IDDES CFD simulation of [42] as can be seen in Table 5.1, which also contains predictions using the OA configuration. Clearly, our CFD prediction of drag matches very well with the experimental result when the vehicle is placed in a VWT. A 6% reduction in C_D was observed for the OA configuration which has a blockage ratio of < 0.25%; existing literature suggests that up to 12% drop in C_D prediction can be expected [32, 90].



Figure 5.1: Validation of the CFD simulation approach and methodology

5.4.2 Application of DMD to a Canonical Flow Case

As a first learning exercise, we performed the DMD of a canonical flow past a 2D circular cylinder at a Reynolds number of 75, similar to a number of DMD research found in the literature [113, 114, 115]. Figure 5.2 shows scalars of stream-wise velocity normalized by the freestream velocity at the instant $t^* = 120$. The very last time instance was chosen for validation as a test case because the DMD predictions from Eq. 5.5, which is in exponential form, are known and expected to diverge with large values of time. We compared the flow field as obtained from the DMD reconstruction to the CFD simulation. From Figures 5.2(a) and 5.2(b), we can see that the DMD reconstruction was qualitatively similar to the CFD prediction. This was an encouraging sign for the ability of DMD to reconstruct the flow field. Further, in Figure 5.2(c), we plotted the difference in the normalized streamwise velocity prediction between the DMD reconstruction and CFD simulation at the time $t^* = 120$. We saw that the difference between DMD and CFD at this time instance is very small, with the order of magnitude of the differences being 10^{-4} . Thus, we inferred that the DMD reconstruction is well correlated to the CFD prediction.



Figure 5.2: Instantaneous Normalized streamwise velocity for flow past a 2D cylinder: (a) CFD prediction, (b) DMD re-construction and (c) the difference between (b) and (a).

We note that it is pointless to compare two instantaneous turbulent flows. Thus, similar to the Reynolds decomposition approach used in turbulent flows, we will be looking at the mean and fluctuating components of the flow field separately. As an example, using the above 2D cylinder case, in Figure 5.3, we can see a comparison of CFD and DMD results of the normalized mean stream-wise velocity. For both CFD and DMD results, the flow field statistics are taken from the last 30 LETOTs of the simulation data. Similar to the analysis of Figure 5.2, in Figures 5.3(a) and 5.3(b), we saw that the mean of the DMD reconstructed flow-field is qualitatively similar to one obtained from the CFD simulation. Furthermore, in Figure 5.3(c), we can also see that the difference in the mean of the normalized streamwise velocity prediction between DMD reconstruction and CFD simulation is very small, $O(10^{-4})$.

In Figure 5.4, we see a comparison between the RMS of fluctuating components of the normalized stream-wise velocity field as obtained from CFD and DMD. Similar to the previous results, in Figures 5.4(a)-(c), we see a very negligible difference between the RMS values from the CFD simulations and DMD reconstruction of the flow fields. In Figure 5.4(c) we see that the difference is $\mathcal{O}(10^{-3})$, which is one order larger than the difference seen for the mean component. This indicates that the DMD modes associated with the higher frequencies may have more error relative to the modes associated with the lower frequencies. This frequency-based bias in the error of the DMD was explored further using the force and moment time-series data from the Ahmed body CFD simulations.

5.4.3 Ahmed Body Simulations

The Ahmed body simulation of cases C2 and C3 were run with a time step of $\Delta t = 0.000t^*$ which corresponds to a physical time-step of 2.5×10^-4 s implying a sampling frequency of 4 KHz when data was collected from every time step. As an initial exploration of the question, "How much data is required to perform an effective DMD?", we took about 25% of the collected data for DMD analysis. The analysis



Figure 5.3: Mean streamwise velocity for flow past a 2D cylinder: (a) CFD prediction, (b) DMD re-construction and (c) the difference between (b) and (a).

is presented below. To address one of the other fundamental questions pertaining to DMD - "What is the necessary sampling frequency for DMD of high Reynolds's number flows?" - the sampling frequency was increased to 10 kHz for the subsequent sections.



Figure 5.4: RMS of the streamwise velocity fluctuations for flow past a 2D cylinder: (a) CFD prediction, (b) DMD reconstruction and (c) the difference between (b) and (a).

5.4.3.1 Data Sampled at 4 kHz

Figures 5.5(a-d) show the distribution of mean pressure coefficient $C_P \equiv p/(0.5\rho(U_{\infty})^2)$ on the surface of the Ahmed body; here p, and U_{∞} represent pressure and reference free-stream velocity respectively. Note that all sub-figures, unless stated otherwise, are an iso-metric bottom-right view of the GV. The spatial extents of the coordinate system are non-dimensionalized by the length of the Ahmed body, L. Similar to Figure 5.3, we see that the distribution of mean C_p on the GV surface is qualitatively the same in both DMD and CFD. Figures 5.5(c) and (d) show the discrepancy in mean C_p prediction by the DMD relative to the CFD results; in Figure 5.5(d) we changed the camera viewing angle to a bottom-right orientation to accommodate visualization of the hidden portions of Figure 5.5(c). In both Figures 5.5(c) and (d), we noted that the discrepancies were $\mathcal{O}(10^{-3})$. In Figure 5.5(c) we observed that the errors are mostly towards the rear of the GV and around the edges of the front face. From Figure 5.5(d) we observed that the discrepancies are most pronounced on the rear slant, rear fascia, and around the two downstream stilts which are regions with recirculation and where smaller vortical structures can exist, see [121, 104]. We will revisit this when analyzing Figure 5.6



Figure 5.5: Mean of surface C_p as obtained using data sampled at 4 kHz: (a) from DMD; (b) from CFD; (c) difference between (a) and (b); (d) same as (c) but bottom-right isometric view

Figures 5.6(a) and (b) show the RMS of surface C_p fluctuations from DMD reconstruction and CFD calculations, respectively. We see a notable discrepancy in the region immediately downstream of the stilts. Figures 5.6(c) and (d) show the discrepancies between the DMD predicted and CFD simulated values of RMS surface C_p ; note that in Figure 5.6(d) we changed the camera angle to a top-left orientation to accommodate visualization of the hidden portions of Figure 5.6(c). In both Figures 5.6(c) and (d), we noted that the discrepancies were $\mathcal{O}(10^{-2})$ which is an order of magnitude worse compared to the mean-flow DMD predictions in Figures 5.5. Also, in Figure 5.6(d), we observed that the DMD result discrepancy (relative to the CFD results) was due to an underprediction of the fluctuating components along the stilts, rear edges, rear slant and rear face. These were the same regions observed in Figure 5.5(d).



Figure 5.6: RMS of surface C_p fluctuations obtained using data sampled at 4 kHz: (a) from DMD; (b) from CFD; (c) difference between (a) and (b); (d) same as (c) but bottom-right isometric view

To better quantify the implications of these flow field discrepancies, we integrated the surface static pressure field to get the pressure component of force and moment coefficients from both the CFD data and the DMD reconstruction. Figure 5.7(a-f) shows the time-series data of coefficients of drag, lift, sideforce, and pitching, rolling, and yawing moments, respectively. On each subplot, the CFD simulation data-series is shown in blue and the DMD reconstruction data-series shown in red. We can see that DMD reconstruction was able to capture the mean of all the coefficients reasonably well however, the fluctuating components were seen to exist only in the first 10% of the time-series and then dissipated rapidly thereafter. Even a low frequency motion in the CFD data was seen to be initially captured by the DMD, but that too dissipated 5 LETOTS. By investigating the time-dynamics component of Eq. 5.5, we found nonphysical growth rates that caused the eventual dissipation of the higher frequency DMD modes. This is further corroborated by the following frequency analysis.



Figure 5.7: Forces and moments obtained from CFD calculations and DMD reconstructions, sampled at 4kHz; (a) drag, (b) lift, (c) sideforce, (d) pitching moment, (e) rolling moment, and (f) yawing moment

Power Spectral Density (PSD) of all the six force and moment coefficients signals are shown Figure 5.8(a-f) where the CFD data are shown in blue and the DMD reconstructions are shown in red. It can be seen that all of the DMD spectra were missing many of the characteristic frequencies of the flow. The PSD obtained form DMD calculations is underpredicted in the frequencies from 30 Hz to 300Hz, and, except for drag, many of the medium frequency motions from 100-400Hz are entirely missed for all other components of force and moment in Figures 5.8(b-f). In Figures 5.8(a,b,d) the characteristic PSD peaks around 200Hz and, amplitude wise, is significantly underpredicted by the DMD. Thus, we inferred that the present implementation of the DMD process is suffering energy loss due to the nonphysical dampening of mediumto-high frequency motions.



Figure 5.8: PSD of forces and moments obtained from CFD calculations and DMD reconstructions, sampled at 4kHz; (a) drag, (b) lift, (c) sideforce, (d) pitching moment, (e) rolling moment, and (f) yawing moment

5.4.3.2 Data sampled at 10 kHz

To address one of the fundamental questions pertaining to DMD – "What is the necessary sampling frequency for DMD of high Reynolds's number flows?" and to help resolve the issues highlighted in Figures 5.7 and 5.8, we increased our sampling frequency to 10 kHz as used by [35] for a much higher Reynolds number flow. This necessitated a reduction in our CFD time step size by 60% to $\Delta t = 4 \times 10^{-5} t^*$, a re-run
of the CFD simulation, and fresh data collection for DMD at the 10 kHz sampling frequency. To facilitate a consistent comparison of DMD performance between the two CFD runs, the size of the data matrix of Eq. 5.1 was kept constant. Thus, the subsequent plots were generated using about 8 LETOTs of converged CFD data which represents about 0.2s of physical time and a lowest resolved frequency of 4.8 Hz.

In Figure 5.9, we see the distribution of mean C_p on the surface of the Ahmed body. Figures 5.9(a) and (b) show the mean C_p as predicted by DMD and CFD respectively. We see that the distribution of mean C_p on the GV surface is qualitatively the same in both DMD and CFD. Figures 5.9(c) and (d) show the discrepancies in mean C_p prediction of DMD relative to CFD; in Figure 5.9(d) we changed the camera angle to a bottom-right orientation to orientation to help a better visualization of the hidden portions of Figure 5.9(c). In both Figures 5.9(c) and (d), we noted that the discrepancies were $O(10^{-4})$, which is an order of magnitude less than that associated with Figures 5.5(c) and (d). In Figure 5.9(c) we observed that the errors are nearly absent from the upper surface, which shows a marked improvement along the edges of the front face, particularly relative to Figure 5.5(c). From Figure 5.9(d) we observed that the discrepancies were still the most pronounced on the rear slant, rear fascia, and around the two downstream stilts.

In Figure 5.10 we observed the distribution of RMS of C_p fluctuations on the surface of the Ahmed body. Figures 5.10(a) and (b) show the RMS of C_p as predicted by DMD and CFD respectively. In comparison to Figures 5.6(a) and (b), we see a significant improvement in the region immediately downstream of the stilts. Figures 5.10(c) and (d) show the difference in RMS C_p prediction of DMD relative to CFD; like before, in Figure 5.10(d) we changed the camera angle to a top-left orientation to help visualization of the hidden portions of Figure 5.10(c). In both Figures 5.10(c) and (d), we noted that the discrepancies were $O(10^{-2})$, which was similar to the order



Figure 5.9: Mean of surface C_p as obtained using data sampled at 10 kHz. (a) Mean of DMD, (b) Mean of CFD, (c and d) difference between mean of DMD and mean of CFD, where (d) bottom-right isometric view

of magnitude seen in Figures 5.6(c) and (d). In Figure 5.10(d) we observed that the discrepancies in DMD are due to underpredictions of the fluctuating components along the stilts, rear edges, rear slant and rear face. These were the same regions highlighted in Figure 5.6(d). Thus, Figures 5.9 and 5.10 indicate that the low-to-medium frequency response of DMD has improved, but that the high frequencies may yet remain unresolved.

We again integrated the surface static pressure field to get the pressure component of force and moment coefficients from both the CFD simulation and the DMD reconstruction. Figures 5.11(a-f) show the time-series data for coefficients of drag, lift, sideforce, and pitching, rolling, and yawing moments, respectively. We can see in Figures 5.11(a-f) that the DMD reconstruction was now able to capture the moving mean of all the coefficients which is a notable improvement from Figures 5.7(a-f). But in the DMD reconstruction, the higher-frequency fluctuating components are still



Figure 5.10: RMS of the fluctuating component of surface C_p as obtained using data sampled at 10 kHz: (a) DMD; (b) CFD; (c) difference between (a) and (b); (d) same as (c), but bottom-right isometric view

seen to be dissipated. By investigating the time-dynamics component of Eq. 5.5, and plotting the mode amplitudes obtained from Eq. 5.14 vss. their frequency, we found non-physical energies amongst the higher frequency DMD modes. This suggested that some of the time dynamics obtained by DMD reconstruction in Eq. 5.5 involves aspects of frequency, amplitude, and growth rates that are non-physical, which may be due to a consequence of noise in the algorithm. This is further investigated by the following frequency space analysis.

We analyzed the Power Spectral Density (PSD) of all the six force and moment coefficient signals in Figures 5.12(a–f), which shows that the PSD of the coefficients of drag, lift, sideforce, pitching moment, rolling moment, and yawing moment respectively. Each PSD is plotted on the ordinate and the frequency on the abscissa. On each subplot, the CFD simulation data-series is shown in blue and the DMD reconstruction data-series shown in red. Comparing Figures 5.8 and 5.12 we can al-



Figure 5.11: Forces and Moments of CFD vs DMD, sampled at 10 kHz; coefficients of (a) drag, (b) lift, (c) sideforce, (d) pitching moment, (e) rolling moment, and (f) yawing moment

ready tell that the performance of DMD reconstruction when the data is sampled at 10 kHz is better in the medium-frequency range than when the data is sampled at 4 kHz. The high-frequency ranges from 1000 Hz and above are well correlated in Figures 5.12(a,c,e,f), with a minor underprediction in Figures 5.12(b and d). In Figures 5.12(a,b,d,f), we saw that the DMD spectra manifested some underprediction of the characteristic frequencies of the flow in the medium-range frequencies from 30 Hz to 700Hz. Within even these frequency ranges shown in Figures 5.12(c and e), a good correlation between the DMD reconstruction and the CFD simulation result is evident. At a sampling rate of 10 kHz, flow structures involving a frequency upto 1 kHz can be expected have reasonably good anti-aliasing. Thus, the mediumfrequency energies in the DMD reconstructed flow may still be adversely affected by the noise from the decomposition algorithm. Thus, we next explored cleaning the decomposed modes and time dynamics with certain filtering techniques described in the next section.



Figure 5.12: PSD of Forces and Moments of CFD vs DMD, sampled at 10 kHz; coefficients of (a) drag, (b) lift, (c) sideforce, (d) pitching moment, (e) rolling moment, and (f) yawing moment

5.4.3.3 Custom Filtering with Data Sampled at 10 kHz

To circumvent all of the issues discussed above, we made a modification to the commonly used DMD algorithm. Two changes were made. The first involves the removal of the truncation step of the SVD; the second involves the introduction of our own custom filtering of the DMD modes. We filtered the time dynamics based upon their predicted amplitudes, frequencies and contribution towards the total energy using a series of three sequential filters to identify and remove the nonphysical modes. We acknowledge that both the design of the filtration process and the cutoff criteria each require their own methodological optimization which is left for a subsequent investigation. Here we briefly describe the filtration process. However, the objective here is to is to show the concept of filtering out nonphysical modes and how it improved the DMD predictions. After several trails with many strategies and alternatives, we have developed a custom filtering approach as described below:

- The first filter was a low-pass filter applied to the modes identified based upon their maximum instantaneous amplitude in the time dynamics term from Eq. 5.5. The modes having a maximum instantaneous amplitude greater than 50% of the zero-frequency mode were removed.
- The second filter was applied to the modes based upon their frequency and their amplitude, given by the RMS version Eq. 5.14. The second filter was designed to remove high-frequency modes having non-physically excessive energy. To accomplish this, the modes were plotted in frequency space against the amplitudes; among the high frequency-modes (f > 250 Hz), the spurious modes were identified using a clustering-based anomaly detection algorithm. Outliers were defined as modes having an amplitude greater than a moving mean of 10 samples by more than a single local standard deviation. The outliers thus identified had their associated modes removed.
- The third filter was designed to remove modes which contribute negligible energy to the system. The remaining modes were sorted based upon their contribution towards the total cumulative energy in the system. In this example, modes contributing collectively less than 5% to total energy were removed; we suspect that these mode may arise from the numerical noise. However, this aspect and the effects of the mode-cut-off energy limit need to be further investigated.

This modified DMD process was applied to the 10 kHz sampled data. In this example, about 30% of the modes were removed by this filtration process.

Now, let us analyze the same 10 kHz sampled dataset, this time with custom filtering, in the same manner as before. Figures 5.13(a) and (b) show the mean C_p as predicted by DMD reconstruction and CFD prediction, respectively. We see that the distribution of mean C_p on the GV surface is qualitatively the same for both the DMD and CFD results. Figures 5.9(c) and (d) show the discrepancy in the mean C_p prediction of the DMD results relative to CFD results. In both Figures 5.13(c) and (d), we noted that the discrepancies were $\mathcal{O}(10^{-4})$, which is the order of magnitude as in Figures 5.9(c) and (d). But in Figure 5.13(d), we observed that the errors are nearly absent from the rear slant face, reduced on the rear face of the GV, and markedly improved along the edges of the front face - all of, which are are notable improvements relative to Figure 5.9(d).



Figure 5.13: Mean surface C_p distribution, with 10kHz sampling and custom filtering: (a) DMD; (b) CFD (c and d) difference between (a) and (b)

Figures 5.14(a) and (b) show the RMS of the fluctuating component of C_p as obtained from the DMD reconstruction and CFD simulations, respectively. In contrast to Figures 5.10(a) and (b), here the DMD results are virtually identical to the CFD results. Figures 5.14(c) and (d) show the discrepancy in the RMS of the fluctuating component of C_p predicted by DMD reconstruction relative to those predicted by the CFD results; like before in Figure 5.14(d) we changed the camera angle to a top-left orientation to visualize the hidden portion of Figure 5.14(c). In both Figures 5.14(c) and (d), we noted that the discrepancies were $\mathcal{O}(10^{-3})$ which was similar to the order of magnitude seen in Figures 5.10(c) and (d). This indicates a significant improvement in the prediction of the fluctuating component of the flow field. In Figure 5.14(d), we observed that the discrepancies in the DMD reconstruction are concentrated along the rear edges between the stilts. Thus, Figures 5.13 and 5.14 indicate that the medium frequency response of DMD reconstruction has improved through the filtration process. We corroborate this with the subsequent analysis.



Figure 5.14: RMS of fluctuating component of surface C_p with 10kHz sampling and custom filtering: (a) DMD reconstruction, (b) CFD simulation, (c & d) the difference between (b) and (a).

We again integrated the surface static pressure field to get the pressure component of force and moment coefficients from both the CFD simulation and the DMD reconstruction. Figure 5.15 shows the timeseries data for coefficients of (a) drag, (b) lift, (c) sideforce, (d) pitching moment, (e) rolling moment, and (f) yawing moment. On each subplot, the CFD simulation data are shown in blue and the DMD reconstruction data are shown in red. We can see in Figures 5.15(a-f) that the DMD reconstruction and CFD simulation predictions are very well correlated, which is a notable improvement from Figures 5.11(a–f). But in the DMD reconstruction of Figures 5.15(b,c,e, and f), some of the local peaks associated with the CFD simulation are not captured. This could be due to excessive losses in the filtration process.



Figure 5.15: Forces and moments of CFD simulation verses DMD reconstruction, sampled at 10kHz and obtained using custom filters

Figure 5.16(a-f) shows the PSD of coefficients of drag, lift, sideforce, and pitching, rolling, and yawing moments, respectively. Comparing Figures 5.12 and 5.16, we can see that the performance of DMD reconstruction using filtered modes, let's call it fmDMD, is better in the medium frequency range. While there remains some room for improvement within the medium and high frequency ranges, we understand that data collected from an IDDES simulation cannot resolve the very high frequency flow characteristics; in other words, this imitation may be due more to the limitations of the CFD approach and ess to the limitations of the DMD approach.



Figure 5.16: PSD of Forces and moments of CFD simulation verses DMD reconstruction, sampled at 10kHz and obtained using custom filters

5.4.3.4 Coefficients of Aerodynamic Forces and Moments

In order to investigate the effectiveness of DMD to predict force and moment coefficients, a comparison of statistical quantities (mean and rms of the fluctuating component) obtained from the DMD reconstructed reduced order flow field against those from the CFD simulations are presented in Table 5.1. Clearly, the predictions associated with this DMD method match very well to the CFD data. Note that the CFD simulation of 2 seconds of physical time requires 14,400 central processing unit (CPU) hours, where as the DMD averaging over the same period took 15 seconds of CPU time.

5.4.4 Future State Predictions using DMD

We also investigated the effectiveness of DMD to make future state predictions of the aerodynamic forces and moments. This was done by updating the initial condition vector, b in Eq. 5.5, to represent the last time-step shown in Figure 5.15. Then the

	C_D	C_L	C_S	C_{PM}	C_{RM}	C_{YM}
Mean (CFD)	0.220	-0.062	-0.002	0.019	0.000	0.000
Mean (DMD)	0.220	-0.062	-0.002	0.019	0.000	0.000
RMS (CFD)	0.003	0.012	0.006	0.004	0.002	0.001
RMS (DMD)	0.003	0.012	0.006	0.004	0.001	0.002

Table 5.1: Mean of all aerodynamic coefficients and RMS of their fluctuations as obtained from CFD simulation and DMD reconstruction.

same coefficients of matrices Φ and Ω were used in Eq. 5.5 to generate a future state predictions of static pressure distribution on the Ahmed body surface. This pressure distribution was integrated to calculate the aerodynamic forces and moments. Figures 5.17(a-b) show the differences between future prediction by DMD relative to known CFD data. The instantaneous future prediction by DMD has a small and oscillating difference w.r.t known CFD data. This is expected as it is impractical to perfectly recreate an instantaneous snapshot of stochastic processes like a turbulent flow tin these case parameters of interest are the statistical quantities like mean and RMS of fluctuations, and spectral distributions. We presented a comparison of these timeaveraged quantities in Table 5.2. Generally, the predictions from this DMD method well matched to the CFD data. There is a small discrepancy in the mean CFD and mean DMD coefficients of C_D, C_L and C_{PM} of between 2 to 6 counts. This is hypothesized to come from a low frequency oscillation within the flow field that was not captured in the CFD data used to generate the ROM and is, thus, not predicted by DMD. The RMS of the fluctuating components of DMD and CFD shown in Table 5.2 are very well matched. This suggests that the proposed ROM is successfully predicting the medium and high frequency motions.

Figure 5.18 shows the PSD of future predictions of DMD relative to known CFD data. Similar to Figures 5.16, spectra of the DMD future prediction is shown in blue and spectra of the known CFD data is shown in red. Again, the future prediction by DMD is able to capture the PSD of the flow field. Small discrepancies are seen in



Figure 5.17: Differences between future predictions of DMD and CFD data: (a) delta of force coefficients, (b) delta of moment coefficients; delta implies the difference between the DMD predictions and CFD values

Table 5.2: Mean of all aerodynamic coefficients and RMS of their fluctuations as obtained from CFD simulation and a future prediction by a DMD based ROM.

	C_D	C_L	C_S	C_{PM}	C_{RM}	C_{YM}
Mean (CFD)	0.220	-0.059	-0.001	0.022	0.000	-0.001
Mean (DMD)	0.218	-0.065	-0.001	0.018	0.000	-0.001
RMS (CFD)	0.002	0.011	0.005	0.003	0.001	0.001
RMS (DMD)	0.001	0.011	0.005	0.003	0.001	0.001

low-to-medium frequencies in Figure 5.16(e) which indicate that there remains scope for improvement of the proposed ROM. A sensitivity analysis and optimization are left for a future work.

5.4.5 Computational Resources

In Table 5.3 we compare the computational resources required to run the DMD solution shown in Eq. 5.5 against the requirements to run a full blown CFD simulation. We can see that the DMD is able to reduce total CPU time and storage requirements by two orders of magnitude. The resources required by DMD are thus expected to be within the capability of an on-board controller on a moving vehicle.



Figure 5.18: PSD of future predictions of DMD relative to known CFD data; coefficients of (a) drag, (b) lift, (c) sideforce, (d) pitching moment, (e) rolling moment, and (f) yawing moment

The modified DMD process proposed in this paper may thus have the potential to be combined with a control modification, such as the DMD with Control (DMDc) algorithm proposed by [36], and effect real-time control of a moving vehicle. This is a very promising result as a full blown CFD for real-time control by an on-board computer is not possible.

Table 5.3: Computational resources required by DMD and CFD

Parameter	CFD	DMD
Processors	144	1
CPU time for the entire timeseries	100 hrs	$< 15 { m s}$
CPU time for a single time snapshot	$5 \mathrm{s}$	$< 0.01 { m s}$
Storage needed	20 GB	< 0.20 GB

5.5 Conclusion

In this paper, we intended to apply the Dynamic Mode Decomposition (DMD) approach to a high Reynolds number flow around an idealized ground vehicle with an objective of using the DMD as the engine to develop a Reduced Order predictive Model (ROM). We observed that the standard DMD algorithm, as available from the existing literature, can successfully reconstruct the low Reynolds number flow fields past a 2D cylinder. However, when the same algorithm was applied to a high Reynolds number *Re*, separation-dominated complex flow over an idealized ground vehicle, the existing methods failed to accurately reconstruct the flow fields using the derived DMD modes. This implies that a reduced order reconstruction of the flow-field based on the DMD modes obtained using the existing algorithm would be not very reliable for such flows.

It was found that even though a time-step which may be sufficiently small for a CFD simulation to resolve the flow-field accurately, it may be inadequate to generate a well resolved dataset for a well-resolved DMD. A larger than adequate time-step caused nonphysical growth rates of the modes; this caused excessive energy dissipation of the medium to high frequency modes which eventually led to total decay of the higher-frequency DMD modes. Thus, for a high Re flows, data sampling frequency needs to be higher than what may be available on the basis of the time-step size needed for a well resolved IDDES simulation; this implies that the CFD simulation is needed to be run with a much smaller time-step than necessary for a well-resolved IDDES. Though the higher sampling rate improves the observed discrepancies between the DMD reconstructed mean values and the ground truth (values from the CFD simulation in this case) for the mean flow variables, the RMS of the fluctuating quantities still show significant errors. Also, spectral analyses show that the medium frequency motions reconstructed by DMD still show nonphysical dampening. This was hypothesized to be due to the presence of dampening modes emanating from the generation of spurious DMD modes due to the numerical noise present in the CFD training data. Thus, the mode filtration process was developed to remove the offending modes from the DMD reconstruction. This resulted in an order of magnitude improvement in the errors observed in the DMD predictions of the RMS of the fluctuating components when compared to the ground truth. The ROM synthesized via the proposed mode filtration process was able to make a future state prediction that had time averaged quantities and PSD well correlated to known CFD data.

The modified DMD reconstruction algorithm presented in this paper was able to overcome the challenges in the medium-to-high frequency range DMD modes. Thus, we demonstrated that the method, called mfDMD, is capable of flow-field reconstruction that is correct to the accuracy of the CFD modeling scheme used to generate the training data. The computational resources required by the mfDMD algorithm look feasible for the implementation alongside a DMD with Control modification to effect real-time control of a moving vehicle by an on-board controller. Applications of the modified DMD algorithm to aerodynamic interactions between vehicles in close proximity, such as dynamic platooning conditions [121], and NASCAR racecar subject to ride height and crosswind changes [46, 47] are the subjects of future research.

CHAPTER 6: CONCLUSIONS

The summarized conclusions of the four articles are presented here. This dissertation thoroughly examines the aerodynamics of race and idealized road vehicles using scale-resolved and scale-averaged CFD simulations. We have investigated the flow prediction around a full-scale, fully detailed, Gen-6 NASCAR Cup series racecar using RANS and IDDES turbulence modeling approaches. Three configurations of the vehicle, including two ride heights and two yaw angles, were considered. The force and moment coefficients were validated against wind tunnel data from an open-jet, closed-return wind tunnel with a rotating belt and boundary layer suction for moving ground simulation. Both CFD and wind tunnel experiments were setup to simulate open-road conditions.

The first article presented an investigation of the effect of boundary conditions and solver parameters on the flow predictions around a Gen-6 NASCAR using steadystate RANS simulations. Zero-gradient boundaries were compared with inlet and outlet type boundaries for the simulation side walls in crosswind simulation. The effects of realizability coefficient in the SST $k - \omega$ turbulence model and the effects of the compressibility solver were studied. The impact of these changes were observed throughout the racecar geometry using accumulated force coefficient plots. The proposed framework significantly improved the correlation of the CFD predicted forces and moments to within 0 - 4% of wind tunnel values for all three configurations of the vehicle.

The second article investigated the predictive difference between the RANS and ID-DES methods. The drag and lift predictions obtained from the IDDES were within 10% of the wind tunnel predictions. The discrepancy in % Front balance was reduced from 3 - 6% in RANS to 0 - 3% in IDDES. The IDDES approach showed a clear superiority of is its capability to depict a more realistic picture of the vortical structures embedded in the flow, in particular in the wake region. This phenomenon is very important for race teams that intend to optimize aero characteristics of the racecar, especially when the racecar is to be optimized as the trailing car in the pack. And a spectral analysis of the forces showed some potential of being used as a tool in racecar aerodynamic optimization. For such a purpose, an advanced tool of Dynamic Mode Decomposition (DMD) was investigated in Article 4.

The third article scrutinized the correlation, of the static pressure data obtained from surface-mounted pressure probes, between CFD predictions from both RANS and IDDES simulations using wind tunnel reference data. This article sheds some light as to which flow features around the racecar may be contributing the most to the discrepancies, directing future studies to those specific areas likely to produce improved simulation accuracy. It was found that IDDES is unable to capture the peak suction pressure underneath the splitter. IDDES also predicted higher C_p relative to RANS on the front fasica, hood, decklid, RHS of the spoiler, and fuel cell surfaces. These differences contribute to the IDDES solver overpredicting both CD and CL. Further, both IDDES, and RANS struggled to predict the correct suction pressure values in the underbody flow. Both solvers also had significant discrepancies in predicting the C_p on the decklid and spoiler surfaces; this points to the flow over the rear windshield and C-Pillars being resolved differently. Lastly, it was found that in some regions both IDDES and RANS had C_p predictions well-correlated relative to each other, but with both having discrepancies relative to wind tunnel predictions. These regions include the splitter extension plate, rear windshield, and front fascia. This suggests that the C_p predictions could benefit from further improvement of the CFD framework.

The fourth article applied the DMD algorithm to a high Reynolds number flow

over an idealized ground vehicle. The challenge faced was the presence of nonphysical growth rates that caused energy loss of the medium to high frequency modes that led to the eventual dissipation of the higher frequency DMD modes. To address this issue, the the data sampling frequency from the CFD simulation was increased and a preliminary mode filtration process was developed to remove the offending nonphysical modes from the DMD reconstruction. It was demonstrated that the modified DMD reconstruction algorithm was able to overcome the challenges in medium-to-high frequency ranges and thus is capable of the flow-field reconstruction that is correct to the accuracy of the CFD modeling scheme used to generate the training data. The computational resources required by modified DMD algorithm look feasible for the implementation alongside a DMD with Control modification to effect the real-time control of a moving vehicle by an on-board controller.

REFERENCES

- G. Grand, "2007 bmw sauber f1.07." Online, posted 17th Sept 2007, https://www.topspeed.com/racing/f1/f1-cars/2007-bmw-sauber-f1-07/, 2007.
- [2] S. Holt, "Formula for success aerodynamics." Online, posted 12th Nov 2012, https://www.bbc.com/sport/formula1/20264490, 2012.
- [3] GPblog.com, "Wolff enthusiastic about new regulations: 'stopwatch never lies'." Online, posted 16th March 2022, https://www.gpblog.com/en/news/106321/wolff-enthusiastic-about-newregulations-stopwatch-never-lies.html, 2012.
- [4] N. Ashton and A. Revell, "Investigation into the predictive capability of advanced reynolds-averaged navier-stokes models for the DrivAer automotive model," in *The International Vehicle Aerodynamics Conference*, p. 125, Woodhead Publishing, 2014.
- [5] N. Ashton, A. West, S. Lardeau, and A. Revell, "Assessment of RANS and DES methods for realistic automotive models," *Computers & fluids*, vol. 128, pp. 1–15, 2016.
- [6] N. Ashton and A. Revell, "Comparison of RANS and DES methods for the DrivAer automotive body," Tech. Rep. 2015-01-1538, SAE Technical Paper, United States, 2015.
- [7] E. Guilmineau, G. Deng, and J. Wackers, "Numerical simulation with a DES approach for automotive flows," *Journal of Fluids and Structures*, vol. 27, no. 5-6, pp. 807–816, 2011.
- [8] M. Uddin, S. Mallapragada, and A. Misar, "Computational investigations on the aerodynamics of a generic car model in proximity to a side-wall," Tech. Rep. 2018-01-0704, SAE Technical Paper, United States, 2018.
- [9] E. C. J. Gan, M. Fong, and Y. L. Ng, "CFD analysis of slipstreaming and side drafting techniques concerning aerodynamic drag in NASCAR racing," *CFD Letters*, vol. 12, no. 7, pp. 1–16, 2020.
- [10] E. Jacuzzi, A. Barrier, and K. O. Granlund, "NASCAR race vehicle wake modification via passive blown ducts and its effect on trailing vehicle drag," in 2018 AIAA Aerospace Sciences Meeting, p. 0558, 2018.
- [11] E. Jacuzzi, M. Aleman Chona, and K. O. Granlund, "Improvements in NASCAR race vehicle side force and yawing moment stability in race conditions using active or passive blowing," in AIAA Scitech 2019 Forum, p. 0592, 2019.
- [12] H. Tennekes, J. L. Lumley, J. L. Lumley, et al., A first course in turbulence. MIT press, 1972.

- [13] S. Pope, "A perspective on turbulence modeling, modeling complex turbulent flows," 1999.
- [14] S. B. Pope and S. B. Pope, *Turbulent flows*. Cambridge university press, 2000.
- [15] P. R. Spalart, S. Deck, M. L. Shur, K. D. Squires, M. K. Strelets, and A. Travin, "A new version of detached-eddy simulation, resistant to ambiguous grid densities," *Theoretical and computational fluid dynamics*, vol. 20, pp. 181–195, 2006.
- [16] M. Shur, P. Spalart, M. Strelets, and A. Travin, "A hybrid RANS-LES model with delayed DES and wall-modelled LES capabilities," *International Journal* of Heat and Mass Transfer. To be published, 2007.
- [17] M. L. Shur, P. R. Spalart, M. K. Strelets, and A. K. Travin, "A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities," *International journal of heat and fluid flow*, vol. 29, no. 6, pp. 1638–1649, 2008.
- [18] M. S. Gritskevich, A. V. Garbaruk, J. Schütze, F. R. Menter, et al., "Development of DDES and IDDES formulations for the k-ω shear stress transport model," Flow Turbulence and Combustion, vol. 88, no. 3, p. 431, 2012.
- [19] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," AIAA journal, vol. 32, no. 8, pp. 1598–1605, 1994.
- [20] F. Menter, "Zonal two equation $k\omega$ turbulence models for aerodynamic flows," in 23rd fluid dynamics, plasmadynamics, and lasers conference, p. 2906, 1993.
- [21] C. Zhang, C. P. Bounds, L. Foster, and M. Uddin, "Turbulence modeling effects on the CFD predictions of flow over a detailed full-scale sedan vehicle," *Fluids*, vol. 4, no. 3, p. 148, 2019.
- [22] F. R. Menter, M. Kuntz, and R. Langtry, "Ten years of industrial experience with the SST turbulence model," *Turbulence, heat and mass transfer*, vol. 4, no. 1, pp. 625–632, 2003.
- [23] A. S. Misar, C. Bounds, H. Ahani, M. U. Zafar, and M. Uddin, "On the effects of parallelization on the flow prediction around a fastback DrivAer model at different attitudes," Tech. Rep. 2021-01-0965, SAE WCX Technical Paper, United States, 2021.
- [24] J. P. Brzustowicz, T. H. Lounsberry, and J.-M. E. de La Rode, "Experimental & computational simulations utilized during the aerodynamic development of the dodge intrepid r/t race car," *SAE Transactions*, pp. 2387–2403, 2002.
- [25] B. D. Duncan and K. Golsch, "Characterization of separated turbulent flow regions in CFD results for a pontiac NASCAR race car," tech. rep., SAE Technical Paper, 2004.

- [26] R. Singh, "CFD simulation of NASCAR racing car aerodynamics," tech. rep., SAE Technical Paper, 2008.
- [27] C. Fu, M. Uddin, and A. C. Robinson, "Turbulence modeling effects on the CFD predictions of flow over a NASCAR Gen-6 racecar," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 176, pp. 98–111, 2018.
- [28] C. Fu, M. Uddin, C. Robinson, A. Guzman, and D. Bailey, "Turbulence models and model closure coefficients sensitivity of NASCAR racecar RANS CFD aerodynamic predictions," *SAE International Journal of Passenger Cars-Mechanical Systems*, vol. 10, no. 1, pp. 330–345, 2017.
- [29] C. Fu, C. P. Bounds, C. Selent, and M. Uddin, "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*, vol. 233, no. 14, pp. 3600– 3620, 2019.
- [30] C. Fu, M. Uddin, and C. Selent, "The effect of inlet turbulence specifications on the RANS CFD predictions of a NASCAR Gen-6 racecar," tech. rep., SAE Technical Paper, 2018.
- [31] C. Fu, C. Bounds, M. Uddin, and C. Selent, "Fine tuning the SST k ω turbulence model closure coefficients for improved NASCAR cup racecar aerodynamic predictions," SAE International Journal of Advances and Current Practices in Mobility, vol. 1, no. 2019-01-0641, pp. 1226–1232, 2019.
- [32] C. Fu, M. Uddin, and C. Zhang, "Computational analyses of the effects of wind tunnel ground simulation and blockage ratio on the aerodynamic prediction of flow over a passenger vehicle," *Vehicles*, vol. 2, no. 2, pp. 318–341, 2020.
- [33] C. P. Bounds, S. Rajasekar, and M. Uddin, "Development of a numerical investigation framework for ground vehicle platooning," *Fluids*, vol. 6, no. 11, p. 404, 2021.
- [34] B. Zhang, K. Liu, Y. Zhou, S. To, and J. Tu, "Active drag reduction of a high-drag ahmed body based on steady blowing," *Journal of Fluid Mechanics*, vol. 856, pp. 351–396, 2018.
- [35] H. Ahani, J. Nielsen, and M. Uddin, "The proper orthogonal and dynamic mode decomposition of wake behind a fastback DrivAer model," Tech. Rep. 2022-01-0888, SAE Technical Paper, United States, 2022.
- [36] J. L. Proctor, S. L. Brunton, and J. N. Kutz, "Dynamic mode decomposition with control," SIAM Journal on Applied Dynamical Systems, vol. 15, no. 1, pp. 142–161, 2016.
- [37] P. J. Schmid, "Dynamic mode decomposition of numerical and experimental data," *Journal of fluid mechanics*, vol. 656, pp. 5–28, 2010.

- [38] P. J. Schmid, "Dynamic mode decomposition and its variants," Annual Review of Fluid Mechanics, vol. 54, pp. 225–254, 2022.
- [39] J. N. Kutz, S. L. Brunton, B. W. Brunton, and J. L. Proctor, Dynamic mode decomposition: data-driven modeling of complex systems. SIAM, 2016.
- [40] M. Delaney, "Ferrari says extra wind tunnel time worth less than 0.1s." Online, posted 23rd Aug 2021, https://fli.com/news/415285-ferrari-says-extra-wind-tunnel-time-worth-lessthan-0-1s.html, 2021.
- [41] G. Anderson, "Gary Anderson: What Verstappen was really facing in Hungary." Online, posted 4th Aug 2021, https://the-race.com/formula-1/gary-anderson-what-verstappen-was-reallyfacing-in-hungary/, 2012.
- [42] E. Guilmineau, G. Deng, A. Leroyer, P. Queutey, M. Visonneau, and J. Wackers, "Assessment of hybrid RANS-LES formulations for flow simulation around the ahmed body," *Computers & Fluids*, vol. 176, pp. 302–319, 2018.
- [43] M. I. Masouleh and D. J. Limebeer, "Optimizing the aero-suspension interactions in a formula one car," *IEEE Transactions on Control Systems Technology*, vol. 24, no. 3, pp. 912–927, 2015.
- [44] X. Castro and Z. A. Rana, "Aerodynamic and structural design of a 2022 formula one front wing assembly," *Fluids*, vol. 5, no. 4, p. 237, 2020.
- [45] J. Katz, "Aerodynamics of race cars," Annu. Rev. Fluid Mech., vol. 38, pp. 27– 63, 2006.
- [46] A. S. Misar and M. Uddin, "Effects of solver parameters and boundary conditions on RANS CFD flow predictions over a gen-6 NASCAR racecar," Tech. Rep. 2022-01-0372, SAE WCX Technical Paper, United States, 2022.
- [47] A. S. Misar, M. Uddin, T. Pandaleon, and J. Wilson, "Scale-resolved and timeaveraged simulations of the flow over a NASCAR cup series racecar," Tech. Rep. 2023-01-0735, SAE Technical Paper, United States, 2023.
- [48] C. Collin, T. Indinger, and J. Müller, "Moving ground simulation for high performance race cars in an automotive wind tunnel-CFD approach on moving belt dimensions," *International Journal of Automotive Engineering*, vol. 8, no. 1, pp. 15–21, 2017.
- [49] R. F. Soares, K. P. Garry, and J. Holt, "Comparison of the far-field aerodynamic wake development for three DrivAer model configurations using a cost-effective RANS simulation," tech. rep., SAE Technical Paper, 2017.

- [50] M. Elkhoury, "Assessment of turbulence models for the simulation of turbulent flows past bluff bodies," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 154, pp. 10–20, 2016.
- [51] S. M. El-Behery and M. H. Hamed, "A comparative study of turbulence models performance for separating flow in a planar asymmetric diffuser," *Computers & Fluids*, vol. 44, no. 1, pp. 248–257, 2011.
- [52] S. Gorji, M. Seddighi, C. Ariyaratne, A. Vardy, T. O'Donoghue, D. Pokrajac, and S. He, "A comparative study of turbulence models in a transient channel flow," *Computers & Fluids*, vol. 89, pp. 111–123, 2014.
- [53] A. Curley, M. Uddin, and B. Peters, "Direct numerical simulation of turbulent flow around a surface mounted cube," in 22nd AIAA Computational Fluid Dynamics Conference, p. 3431, 2015.
- [54] C. J. Chen, Fundamentals of turbulence modelling. Crc Press, 1997.
- [55] C. D. Argyropoulos and N. Markatos, "Recent advances on the numerical modelling of turbulent flows," *Applied Mathematical Modelling*, vol. 39, no. 2, pp. 693–732, 2015.
- [56] D. C. Wilcox, "Reassessment of the scale-determining equation for advanced turbulence models," AIAA journal, vol. 26, no. 11, pp. 1299–1310, 1988.
- [57] D. C. Wilcox, "Turbulence modeling for CFD. la canada, ca: Dcw industries," Inc, November, vol. 34, 2006.
- [58] S. R. Ahmed, G. Ramm, and G. Faltin, "Some salient features of the timeaveraged ground vehicle wake," SAE Transactions, pp. 473–503, 1984.
- [59] A. I. Heft, T. Indinger, and N. A. Adams, "Experimental and numerical investigation of the DrivAer model," in *Fluids Engineering Division Summer Meeting*, vol. 44755, pp. 41–51, American Society of Mechanical Engineers, 2012.
- [60] C. Heschl, K. Inthavong, W. Sanz, and J. Tu, "Evaluation and improvements of RANS turbulence models for linear diffuser flows," *Computers & fluids*, vol. 71, pp. 272–282, 2013.
- [61] J. Howell, "Aerodynamic drag of passenger cars at yaw," SAE International Journal of Passenger Cars-Mechanical Systems, vol. 8, no. 2015-01-1559, pp. 306-316, 2015.
- [62] F. Bello-Millán, T. Mäkelä, L. Parras, C. Del Pino, and C. Ferrera, "Experimental study on ahmed's body drag coefficient for different yaw angles," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 157, pp. 140–144, 2016.
- [63] J. Keogh, T. Barber, S. Diasinos, and G. Doig, "The aerodynamic effects on a cornering ahmed body," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 154, pp. 34–46, 2016.

- [64] E. Guilmineau and F. Chometon, "Effect of side wind on a simplified car model: Experimental and numerical analysis," *Journal of Fluids Engineering*, vol. 131, no. 2, 2009.
- [65] E. Guilmineau, O. Chikhaoui, G. Deng, and M. Visonneau, "Cross wind effects on a simplified car model by a DES approach," *Computers & Fluids*, vol. 78, pp. 29–40, 2013.
- [66] A. Altinisik, O. Yemenici, and H. Umur, "Aerodynamic analysis of a passenger car at yaw angle and two-vehicle platoon," *Journal of Fluids Engineering*, vol. 137, no. 12, 2015.
- [67] D. Gogel and H. Sakurai, "The effects of end plates on downforce in yaw," tech. rep., SAE Technical Paper, 2006.
- [68] Y. Zhang, C. Yang, Q. Wang, D. Zhan, and Z. Zhang, "Aerodynamics of open wheel racing car in pitching position," tech. rep., SAE Technical Paper, 2018.
- [69] R. G. Dominy, G. LeGood, and G. Aerodynamics, "The use of a bluff body wake generator for wind tunnel studies of NASCAR drafting aerodynamics," SAE International Journal of Passenger Cars-Mechanical Systems, vol. 1, no. 2008-01-2990, pp. 1404–1410, 2008.
- [70] W. JOEL, B. JEFFREY, N. BRIAN, et al., "The windshear rolling road wind tunnel," Sae International Journal of Passenger Cars Mechanical Systems, vol. 5, no. 1, pp. 265–288, 2012.
- [71] A. S. Misar, M. Uddin, A. Robinson, and C. Fu, "Numerical analysis of flow around an isolated rotating wheel using a sliding mesh technique," Tech. Rep. 2020-01-0675, SAE WCX Technical Paper, United States, 2020.
- [72] T. Hobeika and S. Sebben, "CFD investigation on wheel rotation modelling," Journal of Wind Engineering and Industrial Aerodynamics, vol. 174, pp. 241– 251, 2018.
- [73] M. Aultman, Z. Wang, and L. Duan, "Effect of time-step size on flow around generic car models," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 219, p. 104764, 2021.
- [74] F. R. Menter, M. Kuntz, and R. Langtry, "Ten years of industrial experience with the SST turbulence model," *Turbulence, heat and mass transfer*, vol. 4, no. 1, pp. 625–632, 2003.
- [75] P. R. Spalart, "Comments on the feasibility of LES for wings and on the hybrid RANS/LES approach," in *Proceedings of the First AFOSR International Conference on DNS/LES, 1997*, pp. 137–147, 1997.

- [76] P. R. Spalart and C. Streett, "Young-person's guide to detached-eddy simulation grids," tech. rep., NASA, NASA STI Help Desk NASA Center for AeroSpace Information 7121 Standard Drive Hanover, MD 21076-1320, 2001.
- [77] C. Zhang, M. Uddin, X. Song, C. Fu, and L. Foster, "Simultaneous improvement of vehicle under-hood airflow and cooling drag using 3d CFD simulation," tech. rep., SAE Technical Paper, 2016.
- [78] L. Temmerman, M. A. Leschziner, C. P. Mellen, and J. Fröhlich, "Investigation of wall-function approximations and subgrid-scale models in large eddy simulation of separated flow in a channel with streamwise periodic constrictions," *International Journal of Heat and Fluid Flow*, vol. 24, no. 2, pp. 157–180, 2003.
- [79] S. Kubacki, J. Rokicki, and E. Dick, "Hybrid RANS/LES computations of plane impinging jets with DES and PANS models," *International journal of heat and fluid flow*, vol. 44, pp. 596–609, 2013.
- [80] D. Home, M. Lightstone, and M. Hamed, "Validation of DES-SST based turbulence model for a fully developed turbulent channel flow problem," *Numerical Heat Transfer, Part A: Applications*, vol. 55, no. 4, pp. 337–361, 2009.
- [81] M. K. Stoellinger, R. Mokhtarpoor, and S. Heinz, "Hybrid RANS-LES modeling using smooth and rough wall functions," in AIAA Scitech 2019 Forum, p. 1576, 2019.
- [82] C. Fu, M. Uddin, and A. Curley, "Insights derived from CFD studies on the evolution of planar wall jets," *Engineering Applications of Computational Fluid Mechanics*, vol. 10, no. 1, pp. 44–56, 2016.
- [83] A. M. Uddin, A. Perry, and I. Marusic, "On the validity of taylor's hypothesis in wall turbulence," J. Mech. Eng. Res. Dev, vol. 19, no. 20, pp. 57–66, 1997.
- [84] C. Fu, CFD Investigations on the Aerodynamic Characterization of a NASCAR Gen-6 Racecar under Different Operating Conditions. PhD thesis, The University of North Carolina at Charlotte, 2018.
- [85] C. Zhang, M. Uddin, and L. Foster, "Investigation of the turbulence modeling effects on the CFD predictions of passenger vehicle underhood airflow," tech. rep., SAE Technical Paper, 2018.
- [86] E. Jacuzzi and K. Granlund, "Passive flow control for drag reduction in vehicle platoons," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 189, pp. 104–117, 2019.
- [87] C. Collin, S. Mack, T. Indinger, and J. Mueller, "A numerical and experimental evaluation of open jet wind tunnel interferences using the DrivAer reference model," *SAE International Journal of Passenger Cars-Mechanical Systems*, vol. 9, no. 2, pp. 657–680, 2016.

- [88] W. P. Jones and B. E. Launder, "The prediction of laminarization with a twoequation model of turbulence," *International journal of heat and mass transfer*, vol. 15, no. 2, pp. 301–314, 1972.
- [89] B. E. Launder and B. I. Sharma, "Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc," *Letters in heat* and mass transfer, vol. 1, no. 2, pp. 131–137, 1974.
- [90] A. Altinisik, E. Kutukceken, and H. Umur, "Experimental and numerical aerodynamic analysis of a passenger car: Influence of the blockage ratio on drag coefficient," *Journal of Fluids Engineering*, vol. 137, no. 8, 2015.
- [91] J. Ikeda, D. Matsumoto, M. Tsubokura, M. Uchida, T. Hasegawa, and R. Kobayashi, "Dynamic mode decomposition of flow around a full-scale road vehicle using unsteady CFD," in 34th AIAA Applied Aerodynamics Conference, p. 3727, 2016.
- [92] B. Mohammadikalakoo, P. Schito, and M. Mani, "Passive flow control on ahmed body by rear linking tunnels," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 205, p. 104330, 2020.
- [93] W. Hanfeng, Z. Yu, Z. Chao, and H. Xuhui, "Aerodynamic drag reduction of an ahmed body based on deflectors," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 148, pp. 34–44, 2016.
- [94] N. Siddiqui and M. Chaab, "A simple passive device for the drag reduction of an ahmed body," *Journal of Applied Fluid Mechanics*, vol. 14, no. 1, pp. 147–164, 2020.
- [95] J. Tian, Y. Zhang, H. Zhu, and H. Xiao, "Aerodynamic drag reduction and flow control of ahmed body with flaps," *Advances in Mechanical Engineering*, vol. 9, no. 7, p. 1687814017711390, 2017.
- [96] P. Joseph, X. Amandolese, and J.-L. Aider, "Drag reduction on the 25° slant angle ahmed reference body using pulsed jets," *Experiments in fluids*, vol. 52, no. 5, pp. 1169–1185, 2012.
- [97] P. Joseph, X. Amandolese, C. Edouard, and J.-L. Aider, "Flow control using mems pulsed micro-jets on the ahmed body," *Experiments in fluids*, vol. 54, pp. 1–12, 2013.
- [98] Y. Li, W. Cui, Q. Jia, Q. Li, Z. Yang, M. Morzyński, and B. R. Noack, "Explorative gradient method for active drag reduction of the fluidic pinball and slanted ahmed body," *Journal of Fluid Mechanics*, vol. 932, p. A7, 2022.
- [99] C.-H. Bruneau, E. Creusé, D. Depeyras, P. Gilliéron, and I. Mortazavi, "Coupling active and passive techniques to control the flow past the square back ahmed body," *Computers & Fluids*, vol. 39, no. 10, pp. 1875–1892, 2010.

- [100] M. N. Sudin, M. A. Abdullah, S. A. Shamsuddin, F. R. Ramli, and M. M. Tahir, "Review of research on vehicles aerodynamic drag reduction methods," *International Journal of Mechanical and Mechatronics Engineering*, vol. 14, no. 02, pp. 37–47, 2014.
- [101] A. M. I. Mukut and M. Z. Abedin, "Review on aerodynamic drag reduction of vehicles," *International Journal of Engineering Materials and Manufacture*, vol. 4, no. 1, pp. 1–14, 2019.
- [102] J. Zhang, T. Feng, F. Yan, S. Qiao, and X. Wang, "Analysis and design on intervehicle distance control of autonomous vehicle platoons," *ISA transactions*, vol. 100, pp. 446–453, 2020.
- [103] S. Sivanandham and M. Gajanand, "Platooning for sustainable freight transportation: an adoptable practice in the near future?," *Transport Reviews*, vol. 40, no. 5, pp. 581–606, 2020.
- [104] K. Liu, B. Zhang, Y. Zhang, and Y. Zhou, "Flow structure around a low-drag ahmed body," *Journal of Fluid Mechanics*, vol. 913, p. A21, 2021.
- [105] O. T. Schmidt and T. Colonius, "Guide to spectral proper orthogonal decomposition," Aiaa journal, vol. 58, no. 3, pp. 1023–1033, 2020.
- [106] T. W. Muld, G. Efraimsson, and D. S. Henningson, "Flow structures around a high-speed train extracted using proper orthogonal decomposition and dynamic mode decomposition," *Computers & Fluids*, vol. 57, pp. 87–97, 2012.
- [107] M. Sieber, C. O. Paschereit, and K. Oberleithner, "Spectral proper orthogonal decomposition," *Journal of Fluid Mechanics*, vol. 792, pp. 798–828, 2016.
- [108] G. Berkooz, P. Holmes, and J. L. Lumley, "The proper orthogonal decomposition in the analysis of turbulent flows," *Annual review of fluid mechanics*, vol. 25, no. 1, pp. 539–575, 1993.
- [109] K. Taira, S. L. Brunton, S. T. Dawson, C. W. Rowley, T. Colonius, B. J. McKeon, O. T. Schmidt, S. Gordeyev, V. Theofilis, and L. S. Ukeiley, "Modal analysis of fluid flows: An overview," *Aiaa Journal*, vol. 55, no. 12, pp. 4013–4041, 2017.
- [110] P. J. Schmid, L. Li, M. P. Juniper, and O. Pust, "Applications of the dynamic mode decomposition," *Theoretical and Computational Fluid Dynamics*, vol. 25, no. 1, pp. 249–259, 2011.
- [111] P. J. Schmid, D. Violato, and F. Scarano, "Decomposition of time-resolved tomographic PIV," *Experiments in fluids*, vol. 52, no. 6, pp. 1567–1579, 2012.
- [112] A. Wynn, D. Pearson, B. Ganapathisubramani, and P. J. Goulart, "Optimal mode decomposition for unsteady flows," *Journal of Fluid Mechanics*, vol. 733, pp. 473–503, 2013.

- [113] M. Sakai, Y. Sunada, T. Imamura, and K. Rinoie, "Experimental and numerical flow analysis around circular cylinders using POD and DMD," in 44th AIAA Fluid Dynamics Conference, p. 3325, 2014.
- [114] M. S. Hemati, C. W. Rowley, E. A. Deem, and L. N. Cattafesta, "De-biasing the dynamic mode decomposition for applied koopman spectral analysis of noisy datasets," *Theoretical and Computational Fluid Dynamics*, vol. 31, no. 4, pp. 349–368, 2017.
- [115] N. B. Erichson, L. Mathelin, J. N. Kutz, and S. L. Brunton, "Randomized dynamic mode decomposition," *SIAM Journal on Applied Dynamical Systems*, vol. 18, no. 4, pp. 1867–1891, 2019.
- [116] M. R. Jovanović, P. J. Schmid, and J. W. Nichols, "Sparsity-promoting dynamic mode decomposition," *Physics of Fluids*, vol. 26, no. 2, p. 024103, 2014.
- [117] F. Guéniat, L. Mathelin, and L. R. Pastur, "A dynamic mode decomposition approach for large and arbitrarily sampled systems," *Physics of Fluids*, vol. 27, no. 2, p. 025113, 2015.
- [118] W. Mengmeng, H. Zhonghua, N. Han, S. Wenping, S. Le Clainche, and E. Ferrer, "A transition prediction method for flow over airfoils based on high-order dynamic mode decomposition," *Chinese Journal of Aeronautics*, vol. 32, no. 11, pp. 2408–2421, 2019.
- [119] R. Qiu, R. Huang, Y. Wang, and C. Huang, "Dynamic mode decomposition and reconstruction of transient cavitating flows around a clark-y hydrofoil," *Theoretical and Applied Mechanics Letters*, vol. 10, no. 5, pp. 327–332, 2020.
- [120] D. Matsumoto, L. Haag, and T. Indinger, "Investigation of the unsteady external and underhood airflow of the DrivAer model by dynamic mode decomposition methods," *International Journal of Automotive Engineering*, vol. 8, no. 2, pp. 55–62, 2017.
- [121] M. Uddin, S. Nichols, C. Hahn, A. Misar, S. Desai, N. Tison, and V. Korivi, "Aerodynamics of landing maneuvering of an unmanned aerial vehicle in close proximity to a ground vehicle," Tech. Rep. 2023-01-0118, SAE Technical Paper, United States, 2023.
- [122] H. Lienhart and S. Becker, "Flow and turbulence structure in the wake of a simplified car model," SAE transactions, pp. 785–796, 2003.
- [123] D. Dylewsky, M. Tao, and J. N. Kutz, "Dynamic mode decomposition for multiscale nonlinear physics," *Physical Review E*, vol. 99, no. 6, p. 063311, 2019.
- [124] J. H. Tu, Dynamic mode decomposition: Theory and applications. PhD thesis, Princeton University, 2013.

- [125] J. Kou and W. Zhang, "An improved criterion to select dominant modes from dynamic mode decomposition," *European Journal of Mechanics-B/Fluids*, vol. 62, pp. 109–129, 2017.
- [126] U. Piomelli and E. Balaras, "Wall-layer models for large-eddy simulations," Annual review of fluid mechanics, vol. 34, no. 1, pp. 349–374, 2002.
- [127] J. W. Daily, W. James, D. R. Harleman, et al., Fluid dynamics. Addison-Wesley, 1966.
- [128] J. C. Hunt, A. A. Wray, and P. Moin, "Eddies, streams, and convergence zones in turbulent flows," *Studying turbulence using numerical simulation databases*, 2. Proceedings of the 1988 summer program, 1988.