

Integrated Numerical and Experimental Workflow for High Performance Vehicle Aerodynamics

Steven Rijns^a, Tom-Robin Teschner^{b,*}, Kim Blackburn^a, James Brighton^a

^aAdvanced Vehicle Engineering Centre, Cranfield University, Cranfield, MK43 0AL, United Kingdom

^bCentre for Computational Engineering Sciences, Cranfield University, Cranfield, MK43 0AL, United Kingdom

Abstract

The high performance and motorsport vehicle sectors are pushing the performance frontiers of aerodynamically efficient vehicles. Well-balanced use of accurate and consistent numerical simulation tools in combination with wind tunnel experiments is crucial for cost-effective aerodynamic research and development processes. Therefore, this study assesses the simulation performance of four Reynolds-Averaged Navier-Stokes (RANS) turbulence models in relation to experimental and high fidelity Delayed Detached Eddy Simulation (DDES) data for the aerodynamic assessment of a high performance variant of the DrivAer model (*DrivAer hp-F*). The influences of predominant wind tunnel conditions on the vehicle's aerodynamic force coefficients and flow field are also investigated. Additionally, a novel CFD-based blockage correction method is introduced and applied to evaluate the accuracy of conventional blockage correction methods. Among the RANS models, the $k-\omega$ SST model exhibited notable relative accuracy in the prediction of force coefficients and demonstrated generally the best correlation with detailed DDES flow field data. The wind tunnel blockage effect caused a 9% increase in downforce and 16% increase in drag, whereas the interference effects from the overhead measurement system reduced downforce by 4% and drag by 8%. The novel CFD-based blockage correction method confirmed that conventional blockage correction methods adequately estimate the dynamic pressure in proximity of a wind tunnel model ($< 3\%$), but do not consider local effects on downforce and drag individually. Overall, the research extends beyond prior work on automotive applications, contributing to the advancement of aerodynamic research methodologies suitable for the complex flow fields of high performance vehicles.

Keywords: Aerodynamics, High Performance Vehicles, DrivAer, CFD, RANS, DDES, Wind Tunnel

1. Introduction

The competitive nature of motorsport creates an environment which encourages fast development of new technologies. It acts as a platform of leading innovations from which many other sectors like the automotive sector adopt technology from. One

of the central themes in the motorsport and automotive sectors is the deployment of more efficient aerodynamic designs to optimise vehicle performance and energy efficiency. The increasing complexity of these designs requires the combined usage of Computational Fluid Dynamics (CFD) simulations and wind tunnel experiments. However, the highly competitive nature of this sector restricts the public sharing of (industrial) knowledge on high performance vehicle aerodynamics and the associated combined numerical and experimental development workflows.

*Corresponding author at: Centre for Computational Engineering Sciences, Cranfield University, Cranfield, MK43 0AL, United Kingdom.
Email address: tom.teschner@cranfield.ac.uk

A desirable numerical tool for an aerodynamic development process should deliver consistent and robust results with minimal computational time, while achieving adequate absolute accuracy [1]. The selection of a turbulence model has a large impact on this trade-off between accuracy and computational time. Reynolds-Averaged Navier-Stokes (RANS) turbulence models are widely used in motorsport and automotive industries because of their good computational efficiency. RANS models have shown adequate accuracy in drag predictions for automotive vehicles, but typically struggle to capture unsteady flow behaviour [2, 3, 4]. Scale-Resolving Simulations (SRS) have demonstrated to capture the flow field on automotive vehicles with more detail, especially in areas of large flow separation such as the wake [5, 6, 7]. However, SRS methods require significant computational resources, which can therefore often not meet the cost and turnover time requirements in fast developing sectors.

Besides the numerical setup, correlation between CFD simulations and wind tunnel experiments is a crucial process in aerodynamic research and development. Exact replication of wind tunnel geometries in CFD simulations is often not feasible, due to either the unavailability of full specification sets or time constraints. Hence, most CFD simulations are conducted on large domains, which represent open road conditions. However, wind tunnel measurements are subject to effects caused by the experimental setup and could therefore vary substantially from CFD simulations in simplified open road conditions. The main wind tunnel conditions to consider are the blockage effect and interference effects. The blockage effect is imposed by test section walls, which restrict airflow displacement around the model and consequently create higher flow velocities in proximity of the model [8]. Blockage effects associated to solid elements like the vehicle model and measurement systems are called solid blockage, whereas effects caused by the vehicle's wake displacement are called wake blockage [9, 10]. Various blockage correction methods are available which use geometrical relationships to account for the blockage effect on experimental force coefficient data [9, 11]. In addition to blockage, airflow obstruction from measurement systems creates interference effects which influence the aerodynamic performance of a vehicle [12, 13, 14].

Published work on the assessment of turbulence models has been primarily conducted on passenger vehicles for automotive applications [2, 3, 4, 5, 6, 7]. Since lift is not a dominant performance parameter for most automotive applications, these studies often only consider accuracy in drag predictions. However, high performance vehicles are equipped with aggressive aerodynamic devices that generate large amounts of downforce (negative lift), which is the primary performance indicator for such vehicles. Their flow fields exhibit far more intricate flow phenomena like steep adverse pressure gradients, large flow separations and strong vortex flows, which need to be accurately captured to assess the vehicle's aerodynamic performance. Therefore, findings on the applicability of turbulence models on automotive applications cannot be directly applied to high performance vehicles. Previous work on wind tunnel

conditions and blockage correction methods often only consider the effect on drag predictions for passenger vehicles and trucks [11, 15, 16, 17, 18]. Findings from these studies generally do not report on the impact of the blockage effect and accuracy of blockage correction methods on downforce, which can therefore also not be directly applied to high performance vehicles. Studies on interference effects from measurement systems generally considered the impact on both force coefficients and have reported varying magnitudes of interference, dependent on the function and flow sensitivity of downstream components [12, 13, 14]. High performance vehicles are equipped with downforce generating aerodynamic devices which are typically highly sensitive to oncoming flow conditions. The vehicle's overall aerodynamic performance is reliant on individual performances of these devices, and therefore expected to be more affected by interference effects than passenger vehicles.

In this paper, the applicability of four RANS turbulence models for aerodynamic simulations on high performance vehicle is evaluated. A comprehensive comparative analysis is provided using experimental force coefficients and high fidelity SRS results of the flow field as reference data. In addition, numerical representations of the wind tunnel blockage and interference effects are created to study their influence on the flow field and aerodynamic performance of a high performance vehicles. Moreover, the numerical representation of the blockage effect is used to investigate the accuracy of various blockage correction methods, including a newly introduced CFD method. By these means, the paper aims to provide detailed insights of the flow field and aerodynamic performance of high performance vehicles. Furthermore, the paper aims to provide a better understanding of the key differences between wind tunnel experiments and CFD simulations, to support the combined numerical and experimental workflow for industrial and academic research on high performance vehicle aerodynamics.

2. Methodology

2.1. Vehicle Model

Aerodynamic research on passenger vehicles in the automotive industry is commonly conducted on reference vehicle models like the DrivAer [19]. This study uses the *DrivAer hp-F* reference model [20, 21], which is a high performance variant of the DrivAer Fastback model (Figure 1). The *DrivAer hp-F* model is a 35% scale model which consists of a smooth underbody, side mirrors and enclosed wheel cavities. It is equipped with additional aerodynamic devices including a front bumper splitter, forebody strakes, rear spoiler and an underbody diffuser. The vehicle model's main parametric dimensions are listed in Table 1 and more geometrical details are available in [20].

Table 1: Parametric dimensions of the *DrivAer hp-F* model

Parameter	Value	
Car scale (CS)	35	%
Length (L)	1.6144	m
Ride height (h_o)	0.0455	m
Width (W)	0.7012	m
Area (A_{ref})	0.2556	m ²

2.2. Experimental Data

Wind tunnel experiments conducted by Renan et al. [20] [22] on the *DrivAer hp-F* model in the 8x6 Wind Tunnel at Cranfield University are used as experimental reference data in this paper. The experiments were performed in a transformed test section including a moving ground and two-stage boundary layer suction system. Aerodynamic forces were measured at eight different ride heights at velocity of 40 m/s, using an overhead balance with a maximum full-scale measurement uncertainty of ± 0.90 N for lift and ± 0.18 N for drag.

2.3. Computational Domains

Numerical simulations on three different computational domains are performed throughout this study. A computational wind tunnel domain is created for improved correlation with experimental data (Section 2.3.1), and two open road computational domains are modelled to investigate the effects of wind tunnel conditions on the vehicle's flow field (Section 2.3.2). Straight-line conditions from the experiments are considered in this study, where flow patterns on a symmetric car can be assumed to be symmetric. This study therefore uses half-car cases, which can reduce mesh sizes by over 40% while providing less than 1% variance in aerodynamic coefficients compared

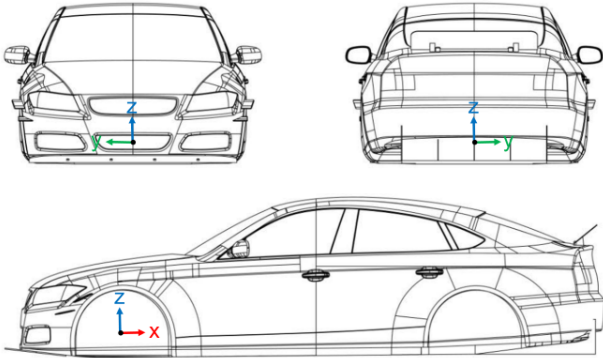
Figure 1: *DrivAer hp-F* model

Table 2: Wind tunnel air properties

Parameter	Value	
Air Density	1.1678	kg/m ³
Air Dynamic Viscosity	18.377e-6	kg/m s
Atmospheric Pressure	99982	Pa
Dynamic Pressure	934.18	Pa
Tunnel Temperature	25.1	°C
Turbulent Intensity	0.052	%

to a full car model [23, 24]. The usage of the symmetry condition is supported by experiments on multichannel diffusers, like the one on this *DrivAer hp-F* model, which have shown to eliminate asymmetric flow conditions otherwise observed at very low ride heights in plane diffusers [25, 26]. The resolved unsteady turbulence in the DDES simulations on the other hand does not obey symmetry instantaneously [27], and the time-dependent behaviour could consequently be influenced by the symmetry simplification. However, this study only uses time-averaged flow quantities from the DDES simulations, which are minimally affected by small turbulent fluctuations along the symmetry plane. This is also apparent from the high accuracy of the DDES results to the experimental data in Section 3.1.1. Hence, the symmetry simplification is a cost-effective approach for this study and allows for qualitative learning from the analyses.

2.3.1. Wind Tunnel Installation

The computational wind tunnel domain (*WT* installation) is designed to resemble the experimental setup in the 8x6 Wind Tunnel at Cranfield University (Section 2.2). The wind tunnel domain is modelled as a straight tunnel without a contraction and diffuser section, due to the unavailability of more detailed specifications. The half domain has a cross-section height of 1787 mm and a width of 1219 mm, including 431.05 mm long corner fillets. Following best practices for automotive external aerodynamics [28], the velocity inlet is placed 3L in front and the pressure outlet 5L behind the vehicle model. This domain size allows adequate space for the development of the flow's turbulent characteristics in front of the vehicle model and avoids outflow conditions affecting the vehicle's near-wake. The top and side walls of the numerical wind tunnel domain are modelled as no-slip stationary walls to allow boundary layer growth on them. The moving ground system used in the experiments is modelled as a no-slip translational moving wall. Furthermore, the *WT* installation matches the model configuration in the experiments, consisting of the vehicle with overhead measurement system. The simulations are performed with a moving wall and inlet velocity of $U_\infty = 40$ m/s, and air properties listed in Table 2.

2.3.2. Open Road Installations

Two open road installations are created which both use the same computational domain but different model configurations. Again, following best practices for automotive external aerodynamics [28], the velocity inlet is placed 3L in front and the pressure outlet 5L behind the vehicle model. The top and side walls are positioned 1.5L distant from the vehicle model, leading to a much smaller blockage ratio compared to the wind tunnel of approximately 1.3%. To resemble open road conditions, the top and side walls are modelled as symmetry walls, which prevent boundary layer growth on them. The moving ground and inlet velocities match the experimental wind speed of $U_\infty = 40$ m/s, and the air properties are kept constant between the wind tunnel and open road domains to allow for consistent relative comparisons (Table 2). The first open road installation is called

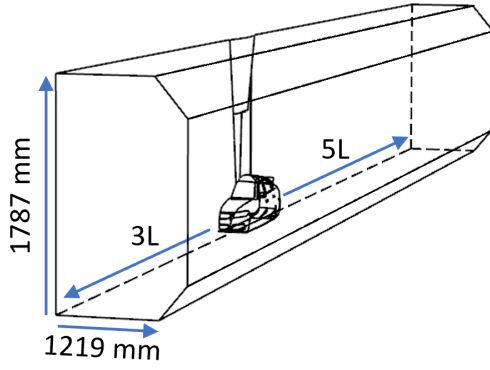


Figure 2: WT installation: vehicle model with top strut in wind tunnel computational domain

the *wTS* installation (with Top Strut) and consists of the vehicle including the overhead measurement system (Figure 3a). The second open road installation is called the *woTS* installation (without Top Strut) and consists of the vehicle without the overhead measurement system (Figure 3b).

2.4. Meshing

2.4.1. Description

Unstructured poly-hexcore meshes are created in ANSYS Fluent Meshing for each installation. The poly-hexcore meshing technique has shown to reduce the total mesh size by about 40% compared to a conventional hexcore technique, and decrease the solve time by approximately 14% for simulations on a standard DrivAer model [29]. A medium-sized mesh is created using optimised element sizes for scaled vehicle models [23]. Element sizes of about 7.25% and 0.45% of the car's length are used as base mesh size and vehicle mesh size respectively. Furthermore, the element sizes on the vehicle are locally reduced to adequately capture the complex geometries of aerodynamic add-on devices and small features. The first layer thickness is also locally controlled, aimed to capture the turbulent boundary layer with a practical near-wall resolution of $y^+ \approx 30-300$ [30]. Lastly, refinement zones based on the 1st Automotive CFD Prediction workshop [31] are created to provide additional detail in the vehicle's flow field (Figure 4).

2.4.2. Grid Convergence

A grid convergence study using the Grid Convergence Index (GCI) [32] is conducted. The systematic refinement approach for unstructured meshes from Celik et al. [33] is adopted to create two additional meshes with varying levels of grid refinement (Table 3). This method suggests the use of refinement factors r greater than 1.3 based on the average cell size h (mm). The simulations for the grid convergence study are performed with the $k-\omega$ SST turbulence model using the standard settings in ANSYS Fluent and ran on two AMD EPYC 7543 CPUs using a total of 64 cores.

The sensitivity of the numerical solution to further grid refinement is also demonstrated by following Celik et al.'s [33]

Table 3: Mesh properties

Case	Coarse (3)	Medium (2)	Fine (1)
No. of Cells [N (-)]	2,489,238	5,928,362	13,624,327
Average wall y^+ [y^+ (-)]	208.11	167.96	49.11
Average cell size [h (mm)]	36.95	27.67	20.97
Refinement Factor [r_{32}, r_{21} (-)]	-	1.34	1.32
Computation Time [t (h)]	0.36	1.22	3.08

approach for unstructured grids. This method uses the deviation of a numerical solution to an asymptotic solution, which is estimated through Richardson Extrapolation. Even though the GCI has less of a firm basis for unstructured grids compared to structured grids, it provides more comprehensive insights over simply reporting errors [32] and has proven to be successful in similar cases for automotive vehicle aerodynamics [34, 35, 36].

The convergence study results in Table 4 show GCI values between the fine and medium meshes (GCI_{21}) which are much smaller than GCI values between the medium and coarse meshes (GCI_{32}) for both force coefficients. This indicates that the increased refinement levels have reduced the grid dependency. Similar to results from [34, 37], different grid convergence rates are observed between both force coefficients, depicted by the higher order of accuracy and smaller GCI values for the lift coefficient. Roache [32] has reported that unstructured grids with non-integer refinement ratios can result in above average orders of accuracy, dependent on refinement levels in critical areas. On the DrivAer *hp-F* model, lift is dominated by the surface pressure distribution, whereas drag is dominated by pressure drag in the vehicle's wake [38, 39, 40]. Hence, unstructured meshing approaches with unequal refinement in these critical areas cause variation in the grid convergence rates between both force coefficients. Nonetheless, the asymptotic range condition [41]; $GCI_{32} = r^p GCI_{21}$, provides values of approximately one for both force coefficients. This indicates that both solutions are within their respective asymptotic range. Considering grid dependence levels, computational time and that lift will be used as the primary performance indicator in this study, the medium mesh strategy is used for all RANS simulations. This mesh demonstrates a relative error to the Richardson extrapolated asymptotic value for lift of 0.38% and for drag of 8.40%.

2.4.3. DDES - High Near-Wall Resolution Mesh

The mesh strategy described above is adopted to create high near-wall resolution meshes for the DDES simulations on the computational wind tunnel domain (WT installation). The high near-wall resolution meshes use a lower y^+ treatment of $y^+ \approx 0.7$ with 15 inflation layers. Moreover, a smoother transition with the small elements in the near-wall region is created by reducing the element size of the refinement zone around the car from 6.25% to 2% (Figure 4). Accordingly, the high near-wall resolution meshes for DDES simulations on the WT installation contain about 22 million cells, whereas the meshes for RANS simulations on the WT installation contain only about 6 million cells.

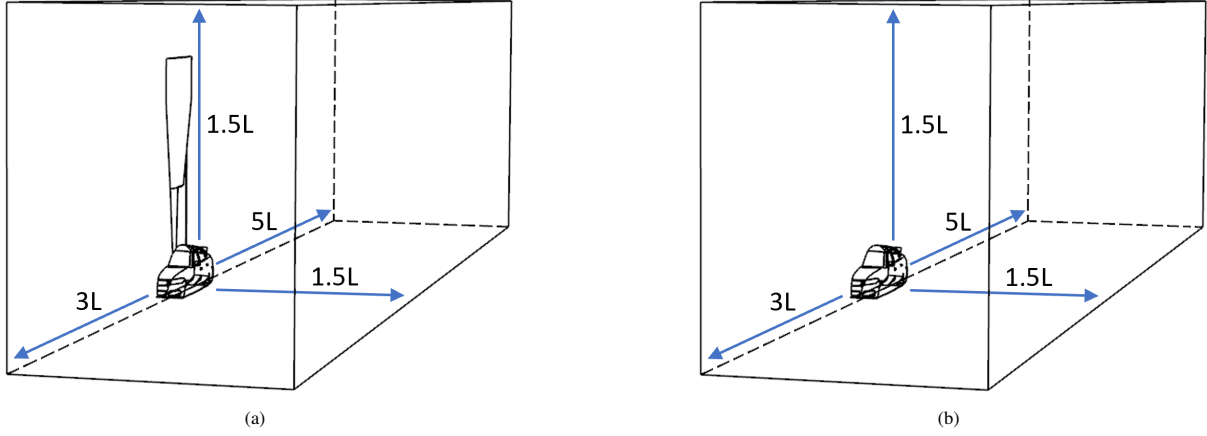


Figure 3: (a) *wTS* installation: vehicle model with top strut in open road computational domain and (b) *woTS* installation: vehicle model without top strut in open road computational domain

2.5. Turbulence Models

2.5.1. RANS

Steady-state simulations are performed with the Spalart Allmaras [42], $k-\epsilon$ Realizable [43], $k-\omega$ SST [44] and Reynolds Stress [45] turbulence models using the standard settings in ANSYS Fluent. Each RANS simulations is carried out for 1000 iterations and force coefficients were averaged over the last 500 iterations. The RANS simulations ran on two AMD EPYC 7543 CPUs using a total of 64 cores which resulted in an average computation time of about 1h for the Spalart Allmaras model, 1.2 hours for the $k-\epsilon$ Realizable and $k-\omega$ SST models, and 1.58 hours for the Reynolds Stress model.

2.5.2. DDES

Scale resolving simulations using the DDES model [46] with the standard settings in ANSYS Fluent and the $k-\omega$ SST RANS background model are performed on the *WT* installation to

serve as numerical reference data. The DDES simulations are initialised with RANS simulations on the same high near-wall resolution mesh using the $k-\omega$ SST model. The time-step for the DDES simulations is based on an approach suggested by Aultman et al. [47]. This method defines the smallest non-dimensional time-step of 1χ which resembles a specific case with an average Courant number of around one. Aultman et al. indicated a progressive blending of small scale turbulent structures into larger scale structures with increasing time-step sizes. To achieve adequate fidelity in the turbulent structures, a conservative time-step size of $t = 3.125 \cdot 10^{-4}s$ (5χ) with 10 inner iterations is used in this study. Each DDES simulations is performed for a total of 2s or 49.6 convective time units (CTUs) of flow time. The first 1s (24.8 CTUs) is used to wash out the initial RANS condition and reach statistical convergence, followed by the remaining 1s (24.8 CTUs) used to average the flow field and collect unsteady statistics. The DDES simula-

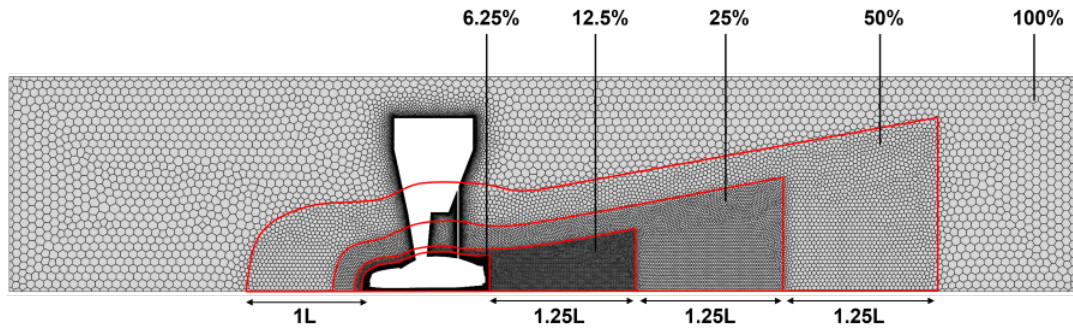


Figure 4: Side view of the medium mesh on the *wTS* installation at the symmetry plane with highlighted refinement zones

Table 4: Grid convergence study results

Case	coarse (f_3 [-])	medium (f_2 [-])	fine (f_1 [-])	Richardson extrapolation (f_{RE} [-])	p (-)	GCI_{32} (%)	GCI_{21} (%)	$\frac{GCI_{32}}{r^p GCI_{21}}$ [-]
C_L	-0.8597	-0.8214	-0.8185	-0.8183	8.92	0.47	0.04	0.9966
C_D	0.2756	0.2575	0.2482	0.2374	2.23	9.68	5.39	0.9642

tions were performed on four AMD EPYC 7543 CPUs using a total of 128 cores, which led to an average computational time of around 90 hours per DDES simulation.

2.6. Wind Tunnel Correlation Process

This paper presents a step-wise approach to numerically investigate the influence and significance of the blockage and measurement interference effects. Wind tunnel experiments and CFD simulations are complementary techniques which can be used to gain a comprehensive understanding of high performance vehicle aerodynamics. While physical tests in a wind tunnel are crucial for validation, they are also subject to conditions like the blockage effect and obstruction of measurement systems. A thorough understanding of the influence and significance of these effects on a vehicle's flow field is therefore crucial to improve correlation processes between wind tunnel experiments and numerical simulations.

2.7. Blockage Effect

A numerical representation of the blockage effect is created by simulations on the computational wind tunnel domain (*WT* installation) and open road domain (*wTS* installation). Both domains use the vehicle including the overhead measurement system and thereby only differentiate in the amount of blockage. The isolation of the blockage effect is used to investigate local changes on the vehicle's flow field and quantify the overall impact on aerodynamic forces.

2.8. Blockage Correction Methods

The numerical representation of the blockage effect is also used to evaluate the applicability of the Maskell, DNW Maskell, Extended Maskell III, Thom & Herriot, and Mercker blockage correction methods [9, 11] for corrections on high performance vehicle models. Each correction method is applied to the results of simulations on the computational wind tunnel domain (*WT* installation) and compared to results of simulations on the open road domain (*wTS* installation). The correction methods vary in complexity, where the DNW Maskell and Extended Maskell III methods offer incremental extensions to the correction methodology of the basic Maskell method. The Thom & Herriot and Mercker methods additionally consider solid and wake blockage separately. Yet, each correction method provides an estimation of the corrected dynamic pressure q_c (Pa) in proximity of the vehicle, based on measured force coefficients and geometrical relationships. Additionally, a reference dynamic pressure q (Pa), which is often measured at the wind tunnel inlet, is used in Eq.1 to compute corrected force coefficients C_{Dc} (-) and C_{Lc} (-). These corrected force coefficients are estimations of the vehicle's aerodynamic performance in open road conditions without a blockage effect imposed by wind tunnel walls.

$$\frac{C_D}{C_{Dc}} = \frac{q_c}{q} \quad \text{and} \quad \frac{C_L}{C_{Lc}} = \frac{q_c}{q} \quad (1)$$

In addition to the conventional correction methods, a CFD based correction method is introduced in this paper. Similarly

to the Maskell methods, the CFD method considers the dynamic pressure increase in proximity of the vehicle model. However, the dynamic pressure is not estimated based on geometric relationships, but extracted from representative simulation data. The CFD based correction method uses results from the numerical representation of the blockage effect. Here, the average dynamic pressure on a frontal plane in the wind tunnel domain and in the open road domain are computed (Figure 5). The frontal plane should be located at a longitudinal position which is subject to the largest blockage ratio to avoid under correction. Furthermore, the dynamic pressure within the projected wind tunnel area, indicated by the black borders, should be considered in the open road domain to prevent influence of low dynamic pressure in the far-field. The corrected dynamic pressure q_c is represented by results from the wind tunnel domain, and the reference dynamic pressure q by results from the open road domain. By these means, the CFD method extracts both q_c and q in proximity of the vehicle model and thereby only considers the dynamic pressure increase imposed by the blockage effect. The essence of the CFD based correction method is the use of representative simulation data in proximity of the vehicle model, as opposed to the use of conventional geometrical relationships. Since a numerical representation of the blockage effect is provided, the CFD method can also be used to evaluate dynamic pressure estimations from the conventional methods.

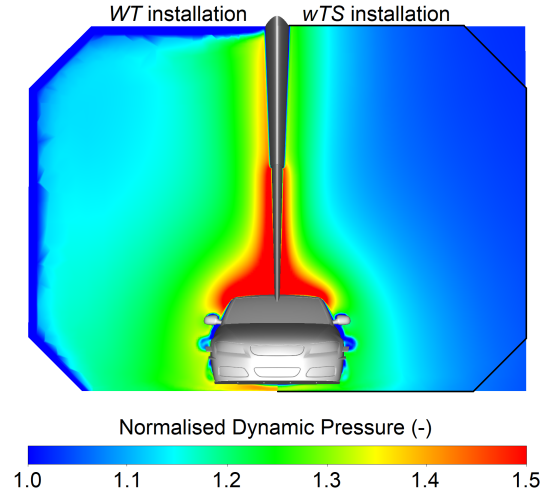


Figure 5: Comparison of the dynamic pressure on a frontal cross-section (at $x = 0.45\text{m}$) in the *WT* and *wTS* installations with black borders highlighting the area for integration

2.8.1. Overhead Measurement System Interference Effect

The presence of measurements systems in wind tunnel experiments cause interference effects which influence the aerodynamic performance of a vehicle. The interference effects created by an overhead measurement system are numerically investigated through simulations in the open road domain on

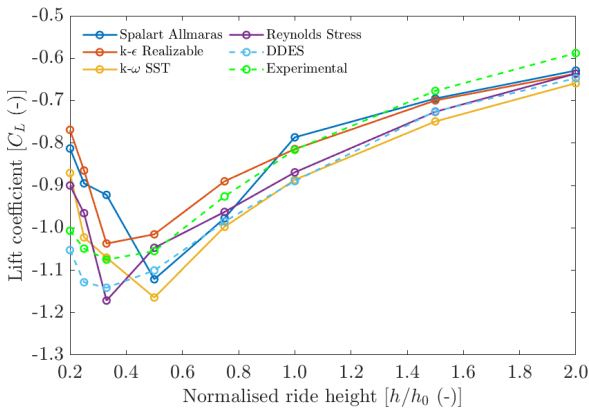
the vehicle with overhead measurement system (*wTS* installation) and without overhead measurement system (*woTS* installation). These installations are not subject to a wind tunnel blockage effect and only differentiate in the vehicle model configuration. By these means, the effects imposed by the overhead measurement system are isolated.

3. Results and Discussion

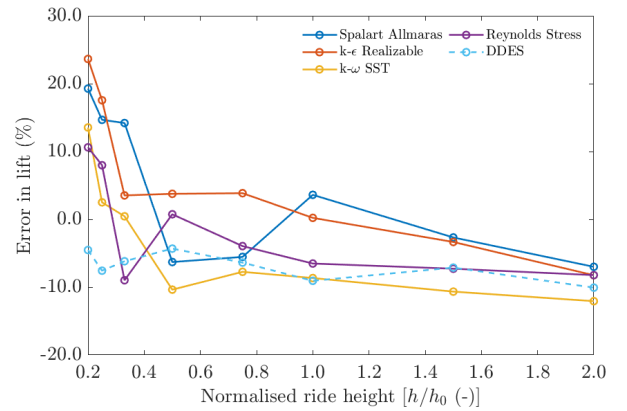
The aerodynamic performance of high performance vehicles is typically highly sensitive to operating conditions like ride height. The first subsection discusses this ride height sensitivity using experimental force coefficients data. Additionally, the predictive capabilities of four practical RANS turbulence models are assessed by relative comparison to the experimental data and absolute comparison to high fidelity DDES flow field data. The following subsection outlines the main sources of variation between wind tunnel experiments and open road simulations on high performance vehicles. Here, the process to numerically quantify the blockage effect, assess the applicability of correction methods and investigate interference effects of measurement systems is discussed and applied to the *DrivAer hp-F* experimental setup in the 8x6 Wind Tunnel at Cranfield University.

3.1. Turbulence Models Assessment

It is important to note that discrepancies between CFD and wind tunnel results have to be anticipated due to factors like numerical approximations, geometric variability and measurements uncertainties. However, through careful consideration of such factors, both techniques can be used as complementary methods for aerodynamic research. For this reason, this paper uses experimental data to investigate aerodynamic performance trends, whereas high fidelity DDES results serve as the best feasible approximation of absolute quantities in the computational wind tunnel domain (*WT* installation). Moreover, wind tunnel data for high performance vehicles is extremely scarce and the available dataset for the *DrivAer hp-F* model is limited to force coefficients data. For that reason, high fidelity DDES results are also presented as a complementary method for a more detailed analysis of the *DrivAer hp-F* model's flow field. Accordingly, the predictive capabilities of various RANS models for high performance vehicle aerodynamics are assessed by their relative accuracy to experimental force coefficients data and their absolute accuracy to DDES flow field data in the same computational wind tunnel domain.

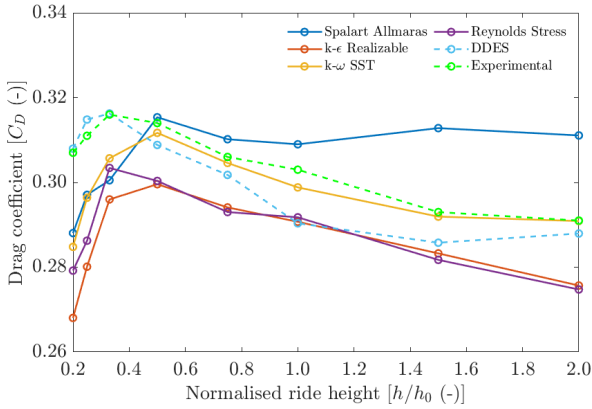


(a)

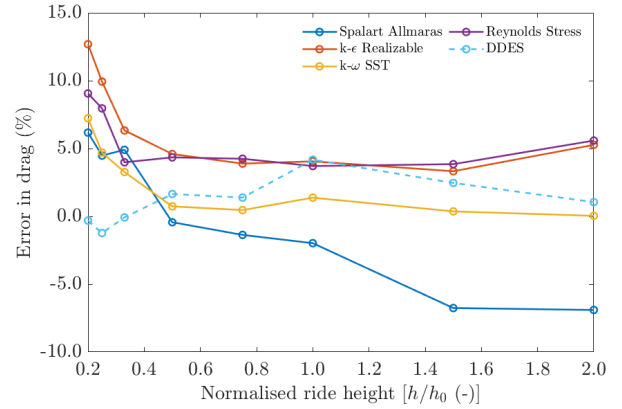


(b)

Figure 6: (a) Lift coefficients of the wind tunnel experiments and numerical simulations on the *WT* installation, and (b) error in lift coefficients between the wind tunnel experiments and numerical simulations on the *WT* installation



(a)



(b)

Figure 7: (a) Drag coefficients of the wind tunnel experiments and numerical simulations on the WT installation, and (b) error in drag coefficients between the wind tunnel experiments and numerical simulations on the WT installation

3.1.1. Force Coefficients Correlation

Figures 6a and 6b depict the vehicle's lift and drag coefficients as a function of ride height, where h_0 represents the normalised ride height of 45.5 mm. The experimental data shows a force enhancement region over the $2h_0 - 0.33h_0$ range, where downforce increases with decreasing ride height due to stronger airflow acceleration underneath the vehicle. As a consequence of increased downforce production, more induced drag is also created along the $2h_0 - 0.33h_0$ range. Significant downforce losses and a steep drop in drag are observed below the critical ride height of $0.33h_0$. At such low ride heights, the airflow underneath the vehicle becomes increasingly unsteady, which negatively affects the aerodynamic performance [48, 49].

The DDES model demonstrates good correlation with the experimental data. It provides an averaged overestimation of 7% in downforce and underestimation of 2% in drag. Moreover, the DDES results are highly consistent, seen by the relatively invariant error which has a standard deviation of approximately 2% across the ride height range. However, following the interrelation between downforce and induced drag, a modest overestimation in drag was expected. This interrelation is likely affected by the varying impact of discrepancy sources and turbulence model accuracy on each individual force coefficient. For example, downforce is dominated by the vehicle's near-wall flow, whereas drag is dominated by unsteady flow in the vehicle's turbulent wake [24, 40]. Overall, the DDES model has demonstrated its ability to consistently reproduce both experimental force coefficient trends across the entire ride height range. This provides confidence in its capabilities to adequately capture detailed flow field data. The DDES results of the vehicle's flow field will therefore be used as numerical reference data to further assess the accuracy of the RANS models in the WT installation.

A common limitation of time-averaged methods applied by RANS turbulence models is their inherent inability to accurately predict highly unsteady flow conditions [50]. This limitation is also apparent in Figures 6 and 7, where each RANS

model is unable to adequately predict the vehicle's aerodynamic performance at low ride heights of $0.33h_0 - 0.2h_0$. This assessment therefore focusses on the high to medium ride height range of $2h_0 - 0.5h_0$, where the aerodynamic performance is less critically determined by highly unsteady flow behaviour. The downforce enhancement region over the $2h_0 - 0.5h_0$ range is captured by each RANS model. The Spalart Allmaras, k-epsilon Realizable and Reynolds Stress models underpredict downforce by about 4%, 8% and 2% respectively, whereas the k-omega SST model overestimates downforce by around 2% compared to the DDES results (Figure 6a). The k-omega SST model provides the most consistent downforce predictions throughout the ride height range in relation to the experimental data, with less than 2% relative deviation in its downforce error (Figure 6b). The other RANS models provide less consistent downforce predictions, with about 4-5% relative deviation in their downforce error.

Considering drag, the k-epsilon Realizable and Reynolds Stress models provide an average underestimation of about 2% whereas the k-omega SST model overestimates drag by around 2% compared to the DDES results (Figure 7a). All three of these RANS models provide consistent drag predictions across the considered ride height range, shown by the less than 1% relative deviation in their drag error (Figure 7b). Contrary, the Spalart Allmaras model fails to adequately capture drag as it depicts a nearly ride height insensitive drag trend over the $2h_0 - 0.5h_0$ ride height range, likely caused by the model's inability to accurately predict decay of free-stream turbulence [51].

3.1.2. Pressure Distribution

The pressure distribution along the upper surface is depicted in Figures 8 and 9. Here, a high pressure region at the splitter and nose is followed by a low pressure region as the airflow accelerates around the curvature of the nose. Along the bonnet, a pressure recovery occurs towards the front of the windscreen, which displays a high pressure region. Airflow is again accelerated over the roof, creating a low pressure region that gradually

recovers towards the trailing edge of the roof. Moderate pressures are seen on the slant due to flow separation behind the roof. Finally, a high pressure region is again created as the airflow interacts with the spoiler.

The RANS models generally show good agreement with the DDES results on the upper surface. The $k-\omega$ SST and Reynolds Stress models provide pressures within 1% of the DDES predictions on the bonnet. The $k-\epsilon$ Realizable and Spalart Allmaras models show discrepancies of about 2% and 4% with the DDES results in this region. Downstream of the bonnet, the RANS models generally predict a larger re-circulation zone in front of the windscreen. As a result, the airflow is redirected higher onto the windscreen compared to the DDES results. Therefore, an 18% smaller pressure peak is locally observed at the windscreen for the RANS models. Along the roof, the RANS models depict a less gradual initial pressure recovery. This leads to larger discrepancies of around 10% towards the trailing edge of the roof. Further downstream at the slant, the RANS models underpredict separation, which consequently impacts the pressure distribution on the spoiler. Here, the DDES model depicts more distinct high pressure concentrations on the spoiler, which can locally reach about a 50% higher peak pressure compared to the RANS models. Discrepancies along the slant and spoiler due to flow separation predictions will be investigated in more detail in Section 3.1.3.

Downforce is predominately created by the airflow underneath the vehicle and should therefore be accurately captured. Figures 10 and 11 depict the pressure distribution along the underbody. Here, an initial low pressure plateau followed by a steep pressure recovery is observed underneath the splitter. This indicates the presence of a separation bubble and subsequent flow reattachment point. A small depression in pressure is created at the floor's leading edge as airflow is accelerated between the floor and ground. The pressure recovers towards the middle section of the floor before airflow is extracted through the diffuser inlet, creating a large depression in pressure. Following this depression, a pressure recovery occurs throughout the diffuser as the airflow is gradually slowed down. In addition, low pressure concentrations are present at the sides of each diffuser channel due to strong vortex formation at the side plates. These

vortices induce relatively high energy airflow onto the diffuser ramp, which supports downforce generation.

While the general pressure distribution along the underbody is adequately captured by the RANS models, local discrepancies are seen around the splitter, diffuser inlet and diffuser ramp. The RANS models predict a smaller separation bubble and earlier flow reattachment underneath the splitter compared to the DDES model. The $k-\omega$ SST model predicts the smallest separation bubble, resulting in a 15% shorter reattachment length. The Reynolds Stress model estimates the largest separation bubble of the RANS models, but still underestimates the reattachment length by about 5%. Despite variation along the splitter, each RANS model provides accurate pressure predictions within 4% of the DDES data around the suction peak and initial pressure recovery behind the floor's leading edge. Towards the latter half of the floor, the $k-\omega$ SST and Reynolds Stress models perform particularly well, with pressure estimations within 1% of the DDES results. The $k-\epsilon$ Realizable model still provides adequate estimations within 4% in this region, whereas the Spalart Allmaras model significantly underpredicts the suction by approximately 20%. The pressure along this section of the floor and the diffuser inlet is primarily affected by the flow extraction created through the expansion in the diffuser. The Spalart Allmaras is the only model which displays a large pressure fluctuation on the diffuser ramp. This is indicative of a separation bubble, which causes the above-mentioned decreased suction on the floor. The other RANS models show better general agreement to the DDES results on the diffuser ramp. Here, the pressure recovery mainly occurs along the centre of the ramp and low pressure concentrations are present at the sides of each diffuser channel. Still, the $k-\epsilon$ Realizable and Reynolds Stress models depict a steeper pressure recovery throughout the diffuser, with surface pressures about 20% higher compared to the DDES results. The $k-\omega$ SST model demonstrates most agreement with the DDES data in this region. The model depicts nearly identical pressure estimations downstream of the inlet, and discrepancies towards the diffuser outlet stay within 10%.

3.1.3. Slant & Spoiler

airflow along the slant and spoiler is primarily affected by the prediction of vortex flow and flow separation along the roof pillars, roof trailing edge and slant. Here, the DDES results show a gradual airflow retardation along the slant with faster moving airflow along the centre and sides of the slant (Figure 13). These high velocity airflow traces are a result of high energy vortices from the overhead measurement system and roof pillars. The vortices propagate along the slant and create high pressure concentrations on the spoiler, as previously identified in Section 3.1.2. Smaller circular high velocity airflow traces at the most upstream plane indicate that the RANS models predict considerably smaller vortices from the overhead measurement system. Furthermore, the vortex structures are captured with less fidelity, illustrated by the blending of two distinct circular traces into one trace as the vortices traverse downstream. Despite this reduced fidelity, the $k-\epsilon$ Realizable model predicts a comparable trajectory of the vortices along the centre of the

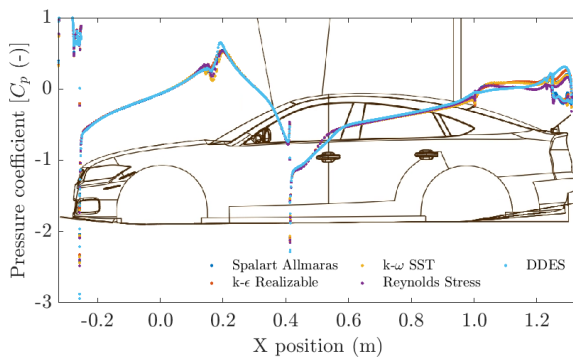


Figure 8: Pressure coefficient distributions on a line close to symmetry ($y = -50$ mm) at the upper body surface in the WT installation at $1h_0$

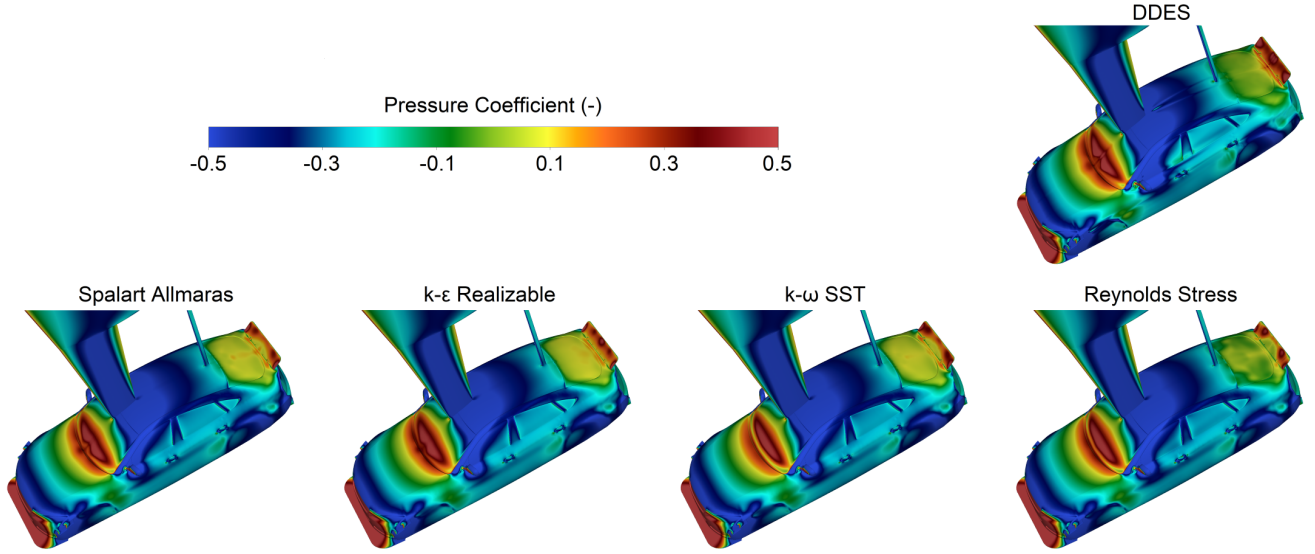


Figure 9: Pressure coefficient contour plots at the upper body surface in the *WT* installation at $1h_0$

slant. This is demonstrated by the height of the high velocity airflow traces and consequent pressure concentrations at the centre of the spoiler (Figure 9). Contrary, the Spalart Allmaras, $k-\omega$ SST and Reynolds Stress models predict a more upwards trajectory, seen by the increased distance of the high velocity airflow traces to the slant surface. Moreover, these models display reduced or even lack high pressure concentrations around the centre of the spoiler (Figure 9). In addition to the discrepancies in vortex predictions from the overhead measurement system, a more inwards trajectory of the vortices generated along the vehicle's roof-line is observed for the DDES results. This is indicated by the wider span of the high velocity airflow traces at the sides of the slant and the resultant larger high pressure concentrations at the sides of the spoiler (Figure 9).

To investigate separation predictions along the slant, velocity profiles at approximately one quarter of the slant's width are extracted from each simulation (Figure 12). Typical behaviour of the boundary layer in an adverse pressure gradient is observed for each model, where the near-wall flow is decel-

erated and starts to separate. Yet, discrepancies in the velocity profiles are observed due to variation of each model's adverse pressure gradient sensitivity and flow predictions at the roof. The DDES results depict near-wall flow reversal at the most upstream velocity profile, which indicates the airflow immediately separates from the roof's trailing edge. The airflow is further decelerated along the slant and reaches a flow reversal velocity of around 25% of the free-stream velocity at the most downstream velocity profile. The RANS models initially demonstrate a more energised boundary layer that reattaches behind the roof, demonstrated by the positive velocity in the most upstream velocity profile. Even though the RANS models depict a noticeable boundary layer deceleration in the subsequent velocity profile, the Reynolds Stress model is the only model which shows near-wall flow reversal. Better agreement is observed at the most downstream velocity profile, where each RANS model depicts flow separation. The $k-\epsilon$ Realizable and $k-\omega$ SST models predict a flow reversal velocity of about 20% of the free-stream velocity and a nearly identical back flow height compared to the DDES results. The Spalart Allmaras and Reynolds Stress models predict a flow reversal velocity of about 15% of the free-stream velocity and underpredict the back flow height by approximately 2%. In addition, higher streamwise velocities are observed towards the top of the boundary layer. This is a result of the initial increased boundary layer momentum prediction further upstream on the slant. Despite the discrepancies in flow prediction along the slant, no significant performance variations are present on the *DrivAer hp-F* model. This is due to the spoiler's small contribution to total downforce of approximately 2% and the insensitivity of its efficiency to oncoming flow conditions.

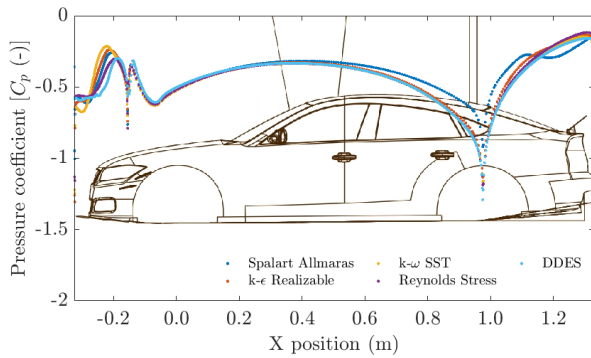


Figure 10: Pressure coefficient distributions on a line close to symmetry ($y = -50$ mm) at the underbody surface in the *WT* installation at $1h_0$

3.1.4. Diffuser

The diffuser is a crucial device on a high performance vehicle, as it typically generates over one third of the vehicle's total

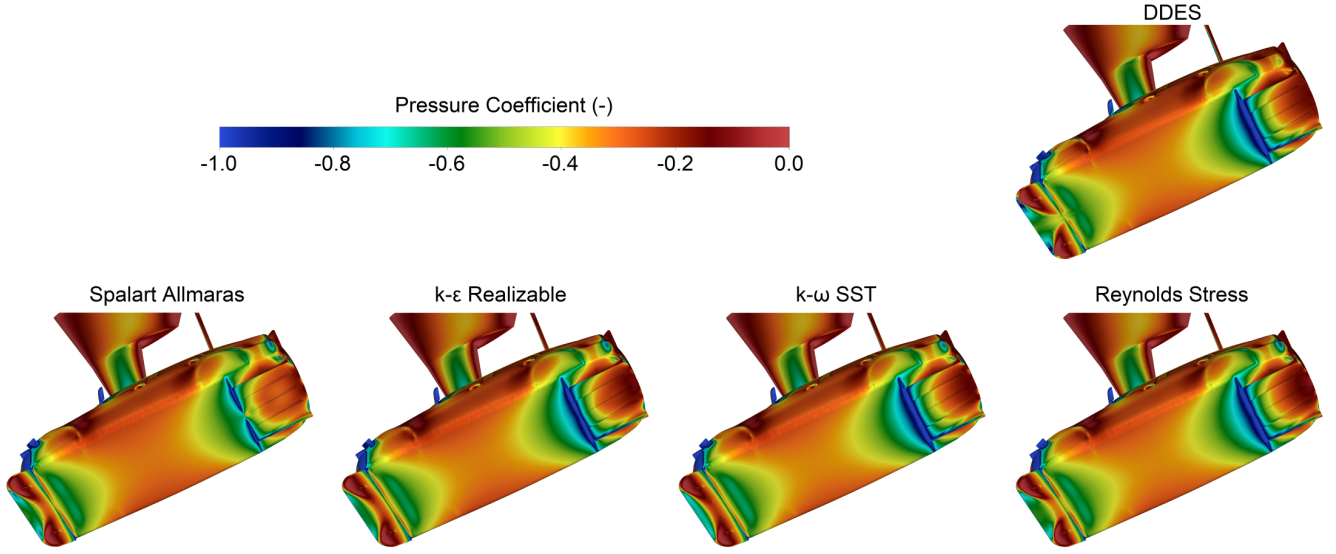


Figure 11: Pressure coefficient contour plots at the underbody surface in the *WT* installation at $1h_0$

downforce. Accurate diffuser flow predictions are therefore im-
 portant, especially when assessing a vehicle's ride height sen-
 sitivity. As discussed in Section 3.1.2, a large depression in
 pressure is created at the diffuser inlet, which is followed by
 a gradual pressure recovery along the diverging diffuser ramp.
 The DDES results depict a narrow region of zero shear stress
 enclosed by a region of low shear stress towards the middle of
 the diffuser (Figure 14). Low shear stress regions are indicative
 of regions which experience considerable near-wall flow decel-
 eration and even separation, as observed in Figure 15. Traces
 of moderate streamwise velocity are also seen underneath the
 symmetry separator of the diffuser, resulting from flow sepa-
 ration in this region. In addition, vortices are generated at the
 diffuser side plates and separators due to the pressure differ-
 ence with the free-stream air. These vortices induce relatively
 high energy airflow from outside the vehicle towards the dif-
 fuser ramp. This phenomenon supports flow attachment in the
 diffuser channels, indicated by the high shear stress concentra-
 tions and high near-wall flow velocities in these areas.

As previously identified, the Spalart Allmaras model signif-
 icantly overpredicts the separation around the centre of the
 diffuser, as shown by the large low shear stress and streamwise
 velocity regions. The $k-\epsilon$ Realizable model provides less over-
 estimation of flow separation in this region, but more agree-
 ment is observed for the $k-\omega$ SST and Reynolds Stress models.
 These models depict a narrower region of low shear stress and
 decelerated airflow along the centre of the diffuser, similar to
 the DDES results. However, none of the RANS models cap-
 ture the traces of moderate streamwise velocity further distant
 from the diffuser ramp. Nonetheless, comparable flow enter-
 tainment from the side plate vortices in the outside channels is
 observed for each RANS model. The $k-\omega$ SST model shows
 most agreement in flow entertainment from the vortices at the
 separators, followed by the $k-\epsilon$ Realizable and Reynolds Stress
 models. The Spalart Allmaras model noticeably underpredicts
 the flow entertainment from the separator vortices, likely af-
 fected by its significant flow separation overprediction. Over-
 all, the $k-\omega$ SST model demonstrates most agreement to the
 diffuser flow predictions from the DDES simulations and con-
 sequently provides diffuser downforce predictions within about
 3% of the DDES results. The other RANS models underpredict
 the diffuser's downforce by up to approximately 8%.

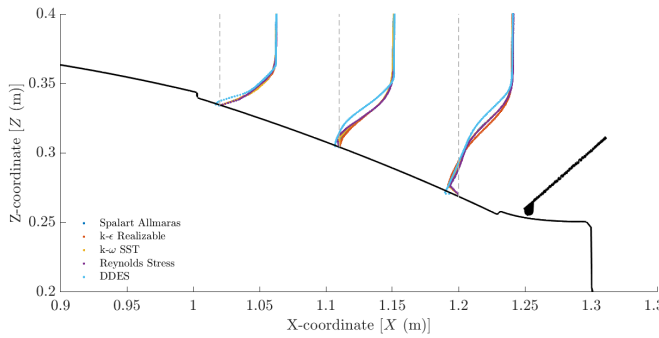


Figure 12: Streamwise velocity profiles at three positions ($x = 1.02$ m, $x = 1.11$ m & $x = 1.2$ m) on a longitudinal line ($y = -85$ mm) on the slant in the *WT* installation at $1h_0$

3.1.5. Wake

airflow separation from the vehicle's body and spoiler cre-
 ates a wake of relatively low energy airflow behind the vehi-
 cle, indicated by regions of low total pressure (Figure 16). The
 DDES results show a squared near-wake region with severe
 pressure losses just behind the vehicle. Moderate total pressure
 traces are seen around the top of this primary wake region, cre-
 ated by flow separation at the slant, spoiler, and overhead mea-
 surement system. Furthermore, moderate total pressure wake
 traces from the mirrors and strakes are observed along the pe-
 riphery of the primary wake region. Larger pressure losses

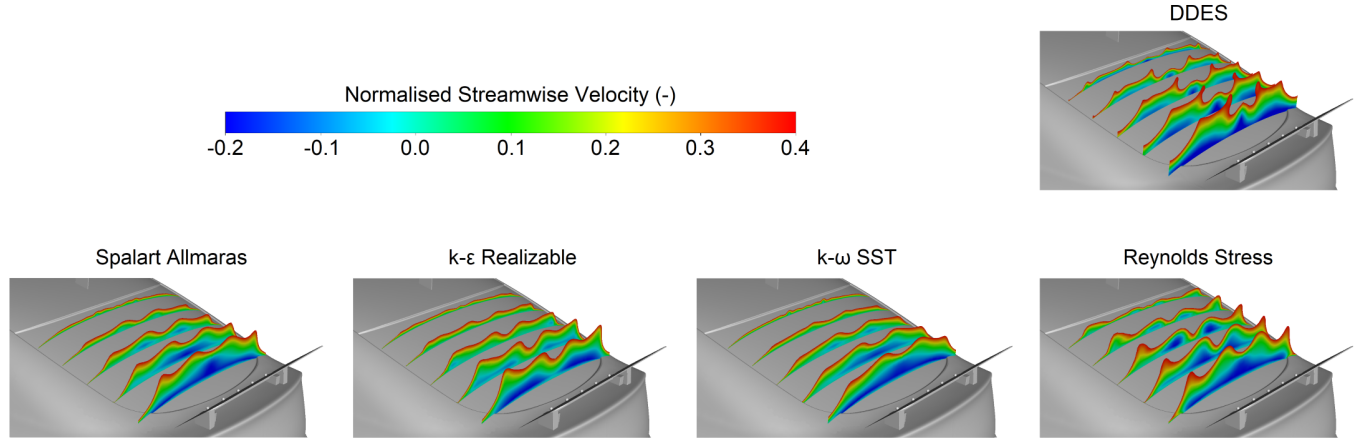


Figure 13: Normalised streamwise velocity contour plots on frontal planes along the slant ($x = 1.02$ m, $x = 1.065$ m, $x = 1.11$ m, $x = 1.155$ m & $x = 1.2$ m) on the slant in the *WT* installation at $1h_0$

are seen in the lower region of the near-wake, resulting from the above-mentioned flow separation and vortices in the diffuser. Further downstream, the wake is dominated by a pair of counter-rotating vortices, indicated by the two oval-shaped low total pressure concentrations. To investigate the dissipation and vertical trajectory of the wake, the area-weighted average total pressure coefficient and pressure loss weighted average vertical position are determined for each wake-plane (Figure 17). These results show that the pressure losses in the wake decay exponentially with distance behind the vehicle and demonstrate a predominately upwards trajectory due to interaction between the counter-rotating vortex pair (Figure 17).

The RANS models demonstrate a squared near-wake region with high pressure losses, similar to the DDES results. However, the intensity of the pressure losses are underpredicted, especially around the top of the primary wake region. The Spalart Allmaras and $k-\epsilon$ Realizable models depict more pressure losses in the lower region of the near-wake as a result of their overprediction of flow separation in the diffuser. The location and intensity of the wake traces from the spoiler and strakes are adequately captured by each RANS model. However, wake traces from the mirrors are generally located further outboard and captured with less intensity. Focusing on the wake development, all RANS models demonstrate the propagation of two counter-rotating vortices, but with variation in intensity and vertical trajectory (Figure 17). The dissipation of pressure losses in the wake is most accurately captured by the Spalart Allmaras and $k-\epsilon$ Realizable models, with an average deviation of about 3% to the DDES results. The $k-\omega$ SST and Reynolds Stress models demonstrate about 7% less wake dissipation throughout the majority of the wake. Despite the adequate agreement in area-weighted average total pressure across the wake planes, the Reynolds Stress model clearly shows a distinct wake propagation with very little vortex diffusion. Consequently, a strong vortex interaction is present as the wake develops downstream, which causes an overprediction of about 10% in the wake's vertical trajectory. The vertical trajectory of the wake is better captured by the $k-\epsilon$ Realizable and $k-\omega$ SST models, with verti-

cal position estimations within 2% of the DDES results at the most downstream plane. Contrary, the Spalart Allmaras model fails to adequately capture the interaction between the counter-rotating vortices. This leads to an underprediction of about 18% of the wake's vertical position at the most downstream plane.

3.1.6. Domain Sensitivity

An advantage of CFD simulations is its control over conditions, which allows the aerodynamic performance of a vehicle to be studied in different operating conditions and domains. A desired turbulence model should provide robust and consistent results to perform such cross-domain analyses. To investigate the sensitivity of each RANS model to domain modifications, simulations are conducted on the computational wind tunnel (*WT* installation) and open road (*wTS* installation) domains. Results from these simulations provide a numerical representation of the wind tunnel blockage effect, which is expected to be minimally affected by small ride height changes. Hence, a relatively invariant offset between the force coefficients from the domains is anticipated across the ride height range. The variability in the offset between the results from both domains can therefore be used to assess the robustness of each RANS models for cross-domain analyses. Figure 18 shows the force coefficients from the *WT* installation normalised by the force coefficients from the *wTS* installation. The $k-\omega$ SST and Reynolds Stress models demonstrate a nearly constant offset in downforce across the ride height range. Contrary, deviations of about 4-5% are observed in the downforce offset from the Spalart Allmaras and $k-\epsilon$ Realizable models. The Spalart Allmaras model also depicts deviations of about 2% on its drag offset, whereas the other RANS models demonstrate fluctuations smaller than 1%.

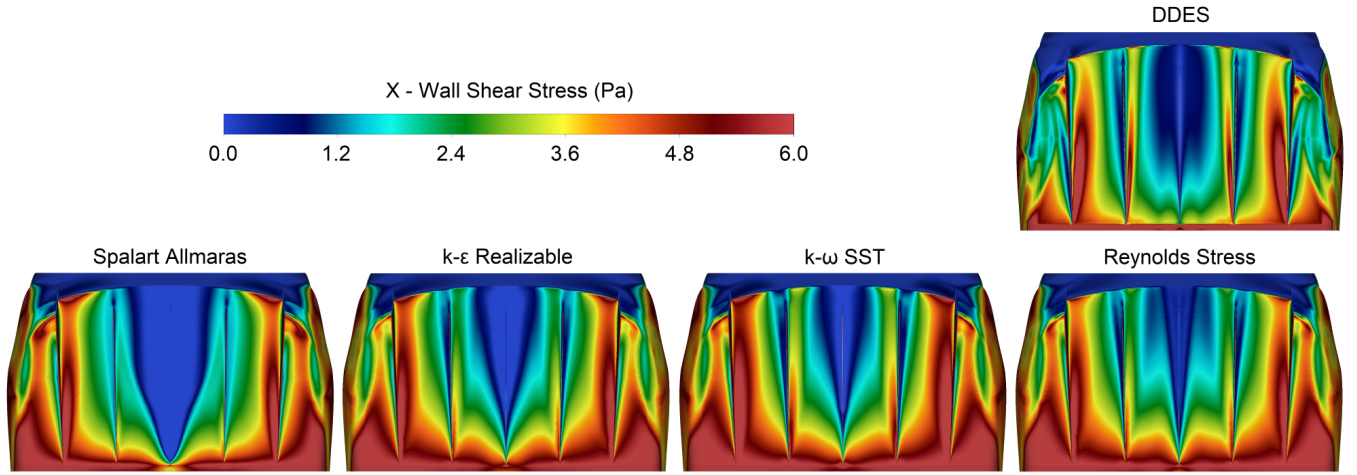


Figure 14: Streamwise shear stress contour plots on the diffuser in the *WT* installation at $1h_0$

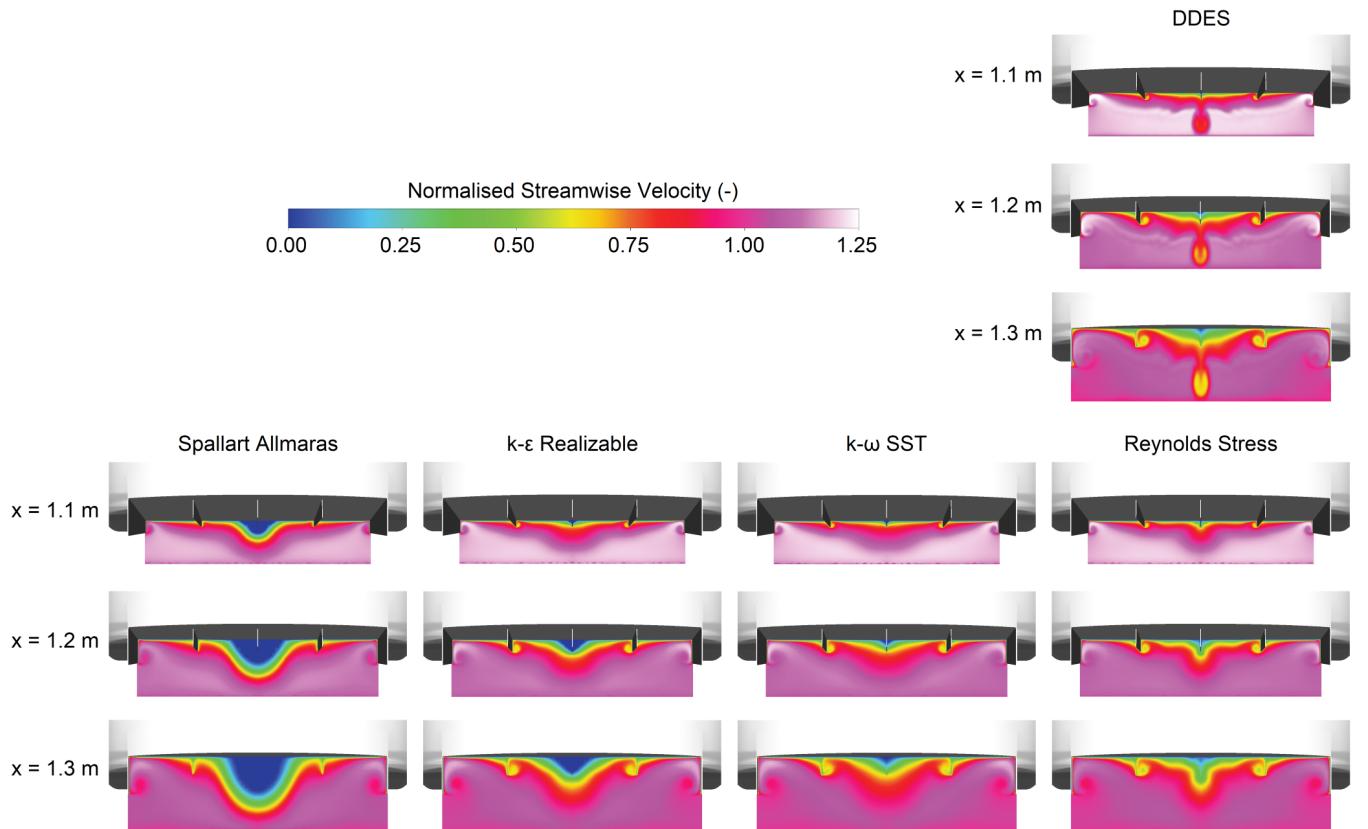


Figure 15: Normalised streamwise velocity contour plots on frontal planes along the diffuser ($x = 1.1$ m, $x = 1.2$ m & $x = 1.3$ m) in the *WT* installation at $1h_0$

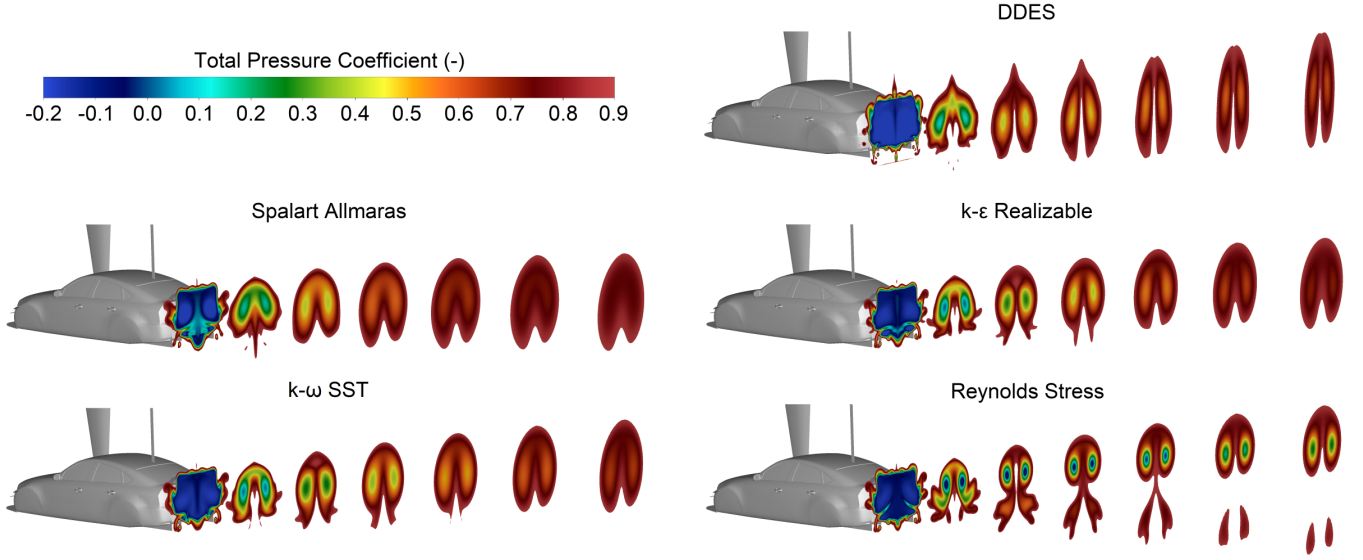


Figure 16: Total pressure coefficient contour plots on vertical planes in the wake ($x = 1.35$ m, $x = 1.85$ m, $x = 2.35$ m, $x = 2.85$ m, $x = 3.35$ m, $x = 3.85$ m & $x = 4.35$ m) in the *WT* installation at $1h_0$

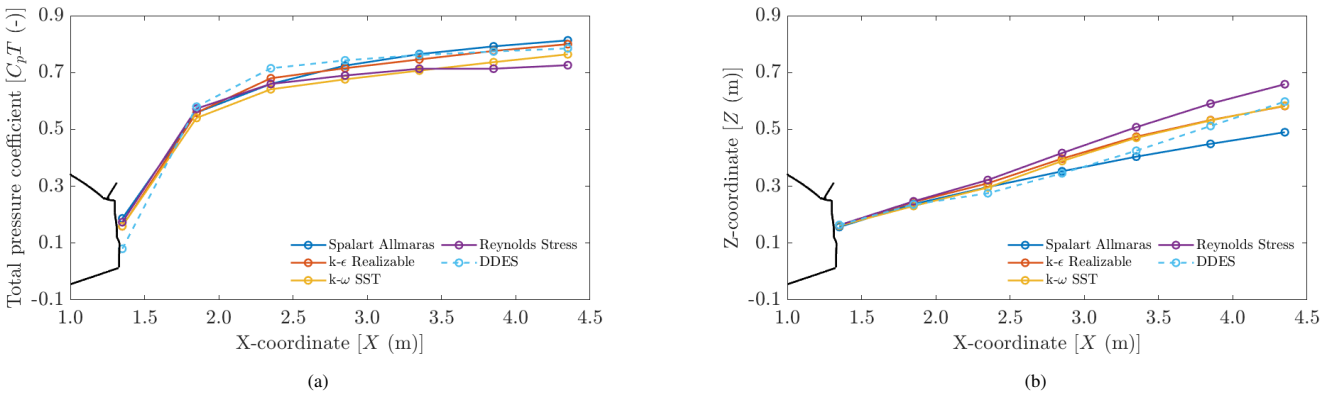


Figure 17: (a) Area-weighted average total pressure coefficient and (b) total pressure loss weighted average vertical position of the wake behind the vehicle in the *WT* installation at $1h_0$

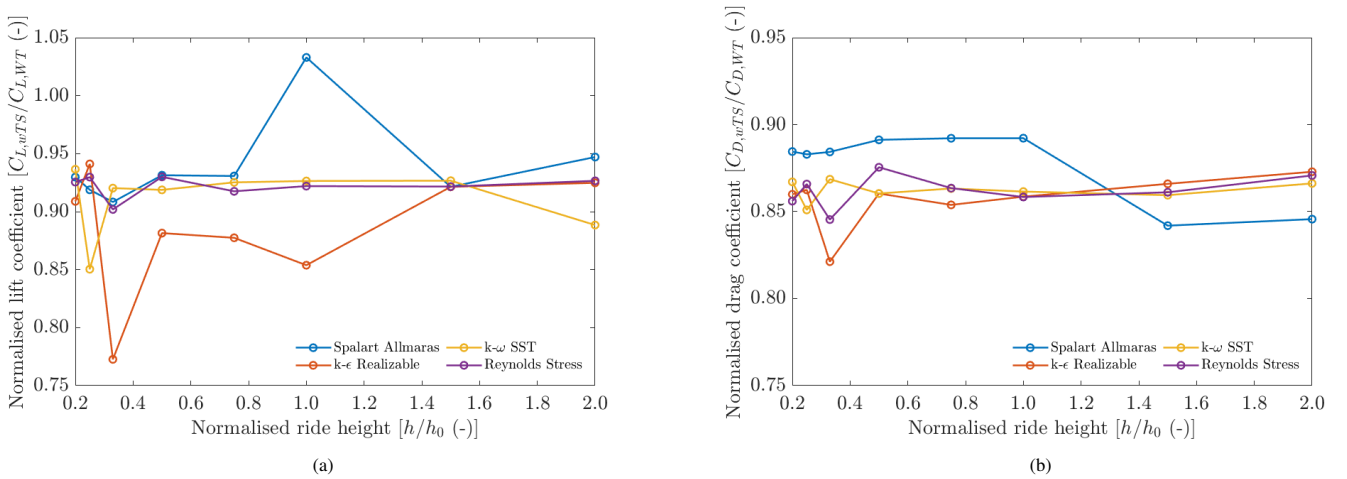


Figure 18: The (a) lift coefficient and (b) drag coefficient from the *wTS* installation normalised by the force coefficients from the *WT* installation

3.1.7. Assessment Findings

The $k-\omega$ SST model has demonstrated the highest accuracy and consistency of the considered RANS models in this assessment. Aerodynamic force trends were adequately captured with estimations within 2% of high fidelity DDES results and comparable relative accuracy to experimental data. The surface pressure distribution on the upper surface was sufficiently estimated. However, discrepancies in the order of 10% developed towards the separated flow region across the slant and spoiler, due to noticeable differences in the adverse pressure gradient sensitivity of the RANS and DDES models. Yet, the $k-\omega$ SST model performed relatively well compared to other RANS models. Predictions of the near-wall flow reversal velocity and back flow height downstream on the slant were respectively within 5% and 1% of the DDES results.

Furthermore, the $k-\omega$ SST model captured the pressure distribution along the underbody generally within 4% of the DDES data. Notable qualitative agreement was achieved for the low pressure concentrations, shear stress distribution and streamwise velocity distribution throughout the diffuser. Consequently, the downforce generation by the diffuser was predicted within 3% of the DDES results. The $k-\omega$ SST model also adequately captured the main wake characteristics, like the counter-rotating vortex pair and wake traces from upstream components. Wake dissipation was underpredicted by about 7%, while the wake's vertical trajectory was estimated within

2% of the DDES results. Additionally, the $k-\omega$ SST model demonstrated a low sensitivity to domain modifications, with deviations smaller than 1% on a theoretical invariant offset between results from two computational domains.

Overall, the $k-\omega$ SST model provides sufficient accuracy and detail for generic aerodynamic analyses on the performance and flow field of high performance vehicles. It requires 75 times less computational time while using half the amount of computational resources compared to the DDES simulations. Hence, the $k-\omega$ SST model is a practical and cost-effective simulation tool which will be used throughout the remainder of this paper.

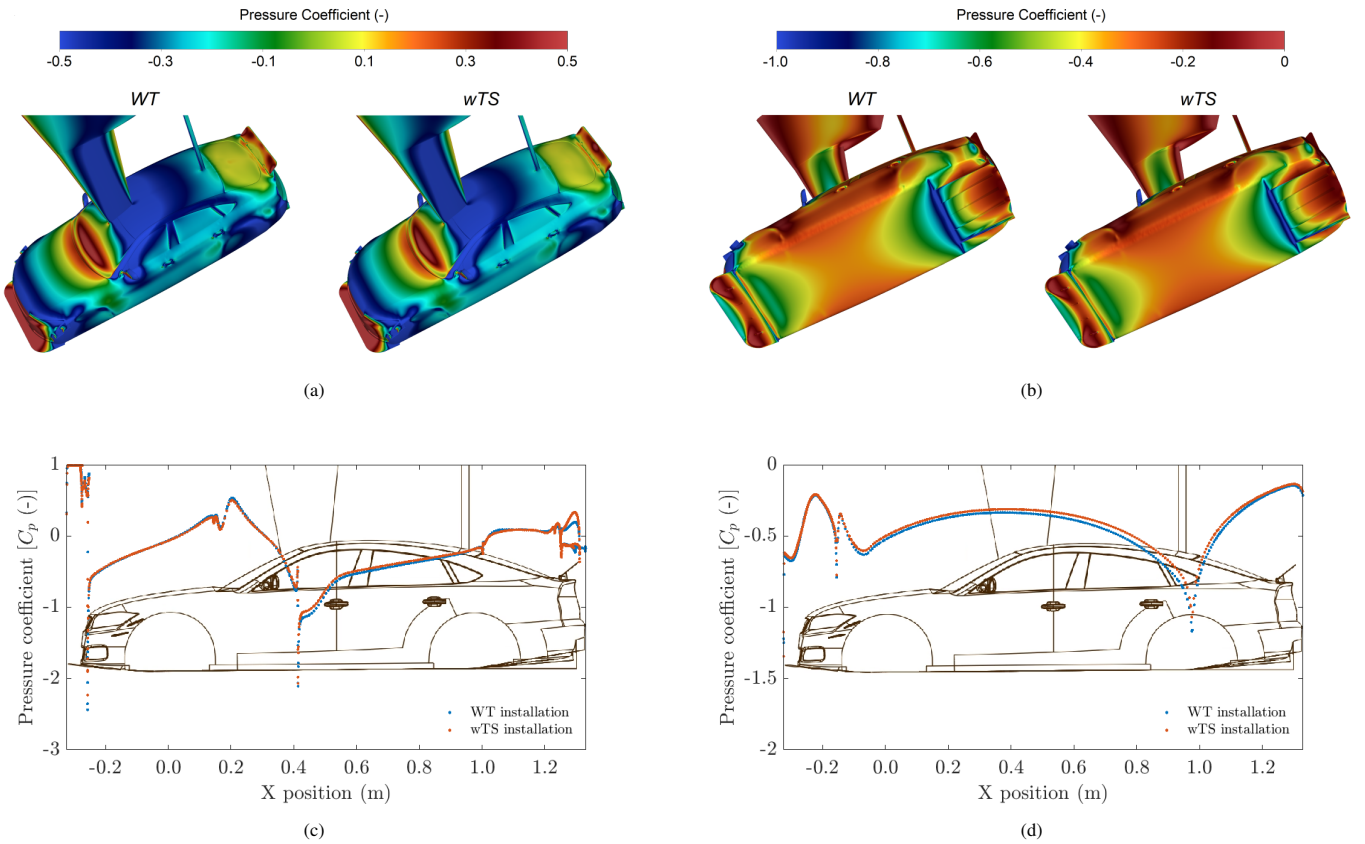


Figure 19: Pressure coefficient contour plots on the (a) upper body surface and (b) under body surface, and pressure coefficient distributions on a line close to symmetry ($y = -50$ mm) at the (c) upper body and (d) underbody surface in the WT and wTS installations at $1h_0$

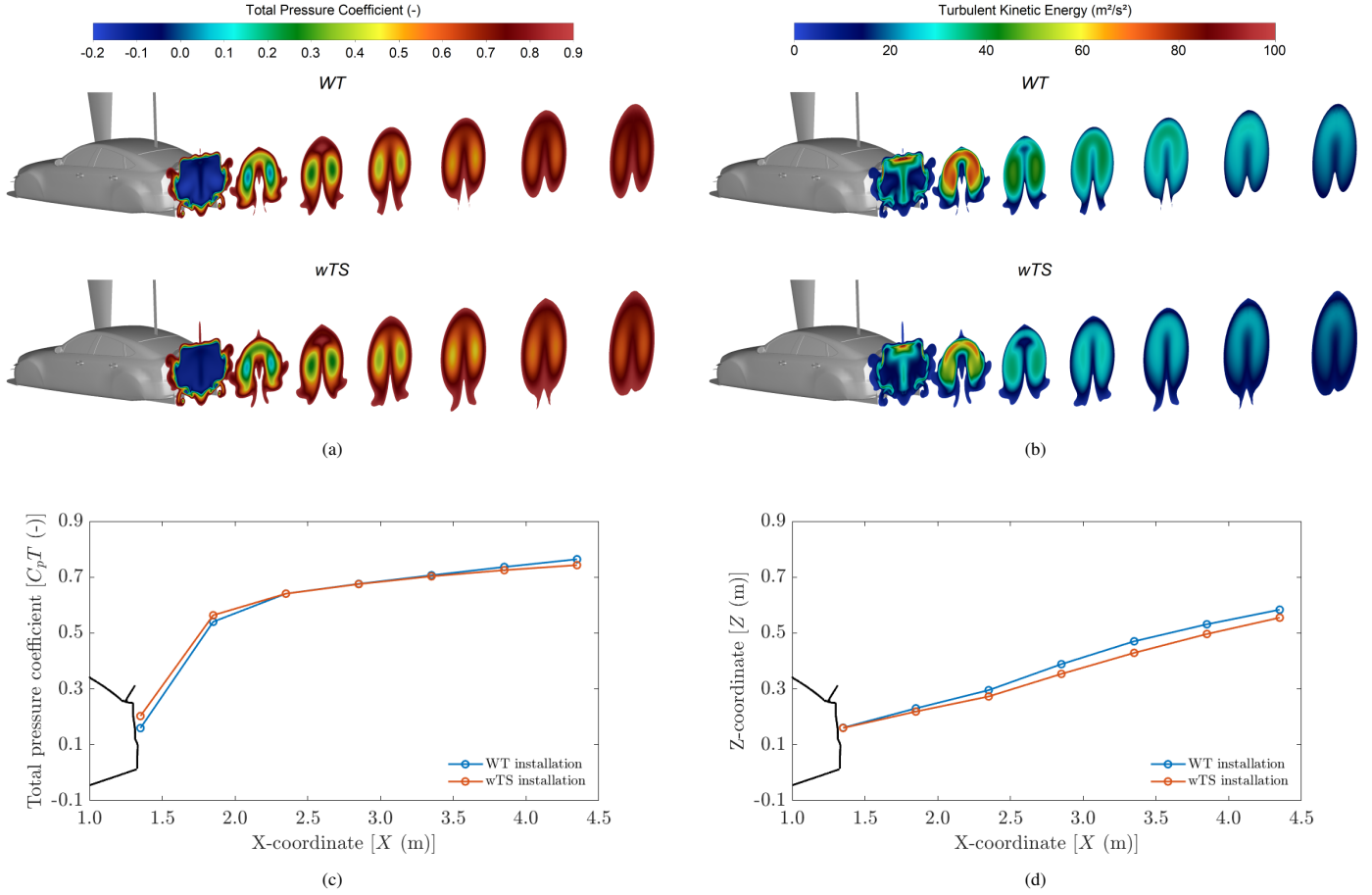


Figure 20: (a) Total pressure coefficient contour plots and (b) turbulent kinetic energy contour plots on vertical planes in the wake ($x = 1.35$ m, $x = 1.85$ m, $x = 2.35$ m, $x = 3.35$ m, $x = 3.85$ m & $x = 4.35$ m), and (c) area-weighted average total pressure coefficient and (d) total pressure loss weighted average vertical position of the wake in the WT and wTS installations at $1h_0$

3.2. Wind Tunnel Correlation Process

3.2.1. Blockage Effect

The presence of test section walls in wind tunnel experiments restricts airflow displacement in proximity of a vehicle model and consequently increases dynamic pressure locally. A numerical representation of this blockage effect is created by simulations on a computational wind tunnel domain (WT installation) and open road domain (wTS installation) domain. Figure 21 shows that the wind tunnel blockage effect induces approximately a 9% increase on downforce and a 16% increase on drag compared to open road conditions. As previously identified, downforce production is dominated by near-wall flow along the vehicle's surface, whereas drag is dominated by pressure drag from the vehicle's wake. Therefore, local flow field changes imposed by the blockage effect should be considered to understand the unequal impact on downforce and drag production.

To investigate local effects on downforce production, a side-by-side comparison of the vehicle's surface pressure distribution in the wind tunnel and open road domain is provided in Figure 19. Low pressure regions found on areas like the roof and underbody in the open domain depict even lower pressures in the wind tunnel domain. This indicates that local fast moving airflow is even further accelerated due to the blockage effect,

coinciding with findings from [52, 53]. Increased airflow velocity along the roof creates locally nearly 8% more lift compared to the open road conditions. This also affects downstream flow conditions, resulting in an approximately 3% increase in downforce generation on the spoiler. Furthermore, the faster moving airflow along the underbody in the wind tunnel domain increases downforce generation of the floor and diffuser by about 7% and 9% respectively.

Apart from the predominant local effects on downforce, an increased pressure build-up at the vehicle's nose is present in the wind tunnel domain, which increases drag by around 11% on this component. Moreover, flow separation of the vehicle's body is promoted and results in about 22% more pressure drag at the vehicle's base compared to the open road conditions. The higher flow velocities in proximity of the vehicle model in the wind tunnel domain also impose a larger velocity gradient at the vehicle's surface, which increases skin friction drag by over 7%.

A side-by-side wake comparison between the wind tunnel and open road domains (Figure 20) shows that the main wake characteristics are unaffected by the blockage effect. Nonetheless, the above-mentioned increased flow separation in the wind

tunnel domain also translates into about 20% higher average pressure losses in the near-wake at $x = 1.35\text{m}$. Yet, more wake dissipation is observed in the wind tunnel domain, resulting in around 3% less pressure losses further downstream at $x = 4.35\text{m}$. This is a result of the faster moving airflow imposed by the blockage effect, which creates a stronger counter-rotating vortex pair that contains more turbulent kinetic energy. Consequently, energy dissipation and mixing with airflow from outside the wake is promoted in the wind tunnel domain. The blockage effect also causes higher energy airflow to be present in proximity of the wake compared to the open road domain. Mixing higher energy airflow into the low energy wake promotes the faster wake dissipation observed in the wind tunnel domain as well. Additionally, the wake's trajectory is about 5% higher in the wind tunnel domain compared to the open road domain. This is a result of the intensified interaction between the stronger vortex pair in the wind tunnel domain.

3.2.2. Blockage Correction Methods

As previously described, a numerical representation of the blockage effect is created by simulations on a computational wind tunnel domain (WT installation) and open road domain (wTS installation). Various blockage correction methods are applied to the results from simulations on the computational wind tunnel domain, to assess their accuracy (Figure 21). The corrected force coefficients depict a maximum relative spread of about 5% between methods. For the considered case, Maskell based correction methods demonstrate average under corrections of about 3% in downforce and 9% in drag. The more complex Thom & Herriot and Mercker methods over correct downforce by around 2% and under correct drag by approximately 5%. The newly introduced CFD correction method provides corrected force coefficients close to the average of the conventional methods, with an under correction of about 1% in downforce and 8% in drag.

The CFD and Maskell based correction methods both apply a simple approach to estimate the dynamic pressure increase in proximity of the vehicle model, which is used for the correction of the force coefficients. However, the CFD correction uses representative simulation data whereas the Maskell methods use basic geometric relationships. The agreement of the Maskell methods with the CFD method therefore indicates that these geometric relationships provide an adequate estimation of the dynamic pressure increase that is induced by the numerical representation of the blockage effect. Nonetheless, the global correction applied by each correction method does not consider the above-mentioned local effects on lift and drag separately. Such correction methods should therefore be used with careful consideration, especially for drag corrections. More comprehensive approaches like the one conducted in this paper would be desirable, and are even essential for more detailed analyses.

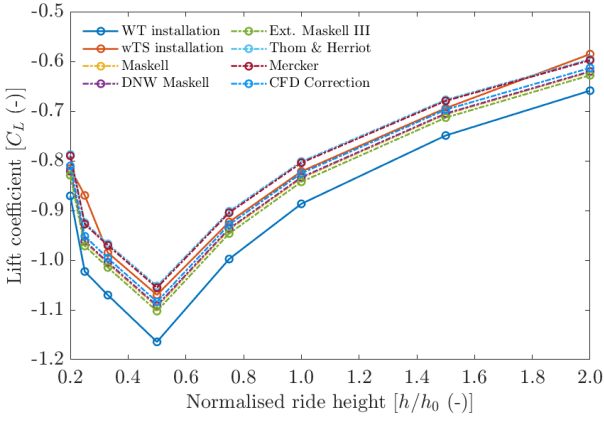
3.2.3. Overhead Measurement System Interference

Interference effects from the overhead measurement system are numerically investigated through simulations in the open road domain on the vehicle with overhead measurement system

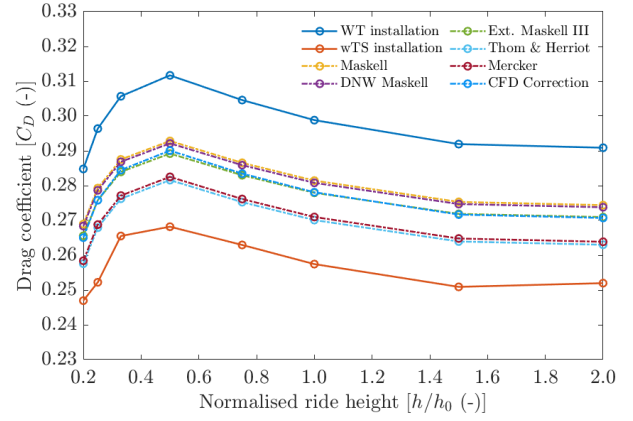
(wTS installation) and without overhead measurement system (woTS installation). Similar to findings on a race car [14], the presence of the overhead measurement system reduces downforce by about 4% and drag by 8%.

The pressure distribution on both vehicle configurations shows that the stagnation at the leading edge of the overhead measurement system affects the re-circulation zone in front of the windscreen, which increases the pressure on the windscreen itself (Figure 22). Local airflow acceleration around the aerodynamic shape of the overhead measurement system increases lift generation on the roof by about 6%. Furthermore, relatively high energy streamwise vortices are formed downstream, which energise airflow along the centre of the slant. The presence of the overhead measurement system also affects the formation of vortices along the roofline. These roof-line vortices propagate further inboard on the vehicle without the overhead measurement system, depicted by the wider high velocity airflow traces at the sides of the slant.

Consequently, a more evenly distributed high pressure zone is created on the spoiler of the vehicle without the overhead measurement system. The vehicle with the overhead measurement system only displays high pressure concentrations around the centre and periphery of the spoiler, resulting in approximately 30% less downforce generation by the spoiler. The reduction in pressure build-up on the spoiler of the vehicle with the overhead measurement system also causes about 30% less drag production by the spoiler. Moreover, the reduced airflow interaction with the spoiler decreases local flow separation, contributing to the approximate 11% reduction in pressure drag around the rear-end of the vehicle with the overhead measurement system.



(a)



(b)

Figure 21: Correction methods applied to the (a) lift and (b) drag coefficients from the simulations in the WT installation

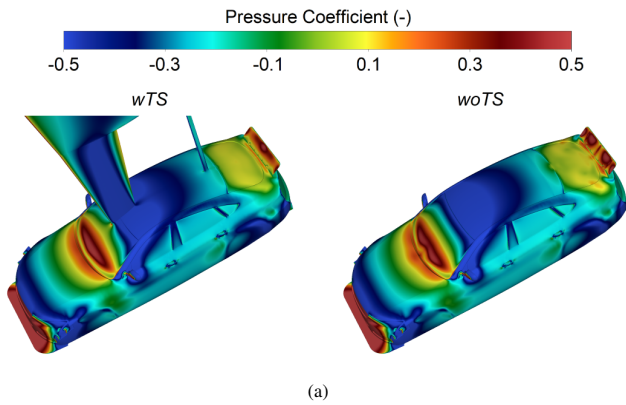
4. Conclusions

The simulation performances of four RANS turbulence models on the complex flow field of a high performance vehicle model (*DrivAer hp-F*) are investigated in relation to experimental force coefficients data and high fidelity DDES results of the flow field in Section 3.1. Subsequently, a step-wise numerical analysis of the wind tunnel blockage effect and overhead measurement system interference effects is performed in Section 3.2. The applicability of various blockage correction methods for corrections on high performance vehicles is also studied in this section.

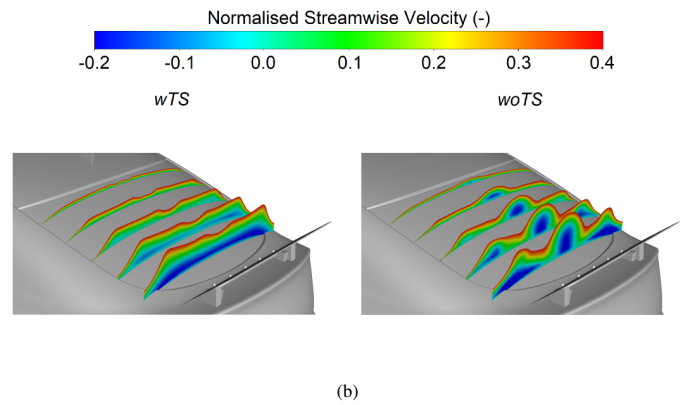
The presented experimental force coefficients data shows that downforce and drag increase as the vehicle is lowered across the $2h_0 - 0.33h_0$ ride height range. Below the critical ride height of $0.33h_0$, the airflow along the underbody becomes increasingly unsteady, which negatively affects downforce and drag production. While the RANS models are generally able to reproduce force coefficient trends at high to medium ride heights, their inherent inability to predict highly unsteady flow conditions is apparent at very low ride heights. From the considered RANS models, the $k-\omega$ SST model was found to be the most applica-

ble for aerodynamic simulations on high performance vehicles. Aerodynamic force predictions were provided within 2% of the DDES results with comparable relative accuracy to the experimental data. However, the $k-\omega$ SST model used 75 times less computational time on half the number of CPU cores compared to the DDES model. Moreover, the $k-\omega$ SST model demonstrated the best overall agreement with more detailed flow field data from the DDES simulations. Estimations of the general surface pressure distribution and dominant wake characteristics were closely matched. Larger discrepancies were observed in the separated flow region on the slant, due to variation in the adverse pressure gradient sensitivity of the models.

The numerical representation of the wind tunnel blockage effect showed that the increased dynamic pressure in proximity of the vehicle model had unequal effects on downforce and drag. Regions with fast moving airflow like the roof and underbody experienced an additional flow acceleration, which increased downforce by 9% for the considered setup. Furthermore, the stagnation pressure, pressure drag and skin friction drag were increased by the blockage effect, causing a 16% increase in total drag.



(a)



(b)

Figure 22: (a) Pressure coefficient contour plot on the upper body surface and (b) Normalised streamwise velocity contour plots on vertical planes along the slant ($x = 1.02$ m, $x = 1.065$ m, $x = 1.11$ m, $x = 1.155$ m & $x = 1.2$ m) in the *wTS* and *woTS* installations at $1h_0$

The newly introduced CFD correction method confirmed that geometric relationships used by conventional correction methods provide an adequate estimation of the dynamic pressure increase imposed by the blockage effect. However, the global corrections based on this estimated dynamic pressure were only applicable for downforce corrections, since local blockage effects on drag are larger. In addition, the most dominant effects of the presence of the overhead measurement system showed to be the local flow acceleration and downstream interference effects, which combined reduced downforce by 4% and drag by 8% for the considered setup.

Overall, the $k-\omega$ SST model is recommended as a cost-effective numerical tool for comparative design processes or aerodynamic analyses of general flow fields and performance trends of high performance vehicles at medium to high ride heights. For conditions dominated by highly unsteady flow behaviour such as low ride heights or in-depth analyses of specific flow features as vortex propagation, more complex approaches like the DDES method are recommended when viable, considering the substantial computational costs. Furthermore, wind tunnel correlation processes would benefit from a numerical blockage and interference effect quantification as performed in this paper over the use of conventional blockage correction methods, since these provide unsatisfactory corrections for drag.

Data Availability Statement

All data supporting this study are included within the article

References

- [1] R. F. Soares, F. J. de Souza, Influence of cfd setup and brief analysis of flow over a 3d realistic car model, SAE Technical Paper 2015-36-0513 (11 2015). doi:10.4271/2015-36-0513.
- [2] C. Zhang, C. P. Bounds, L. Foster, M. Uddin, Turbulence modeling effects on the cfd predictions of flow over a detailed full-scale sedan vehicle, Fluids 4 (2019). doi:10.3390/fluids4030148.
- [3] A. Heft, T. Indinger, N. Adams, Investigation of unsteady flow structures in the wake of a realistic generic car model, 29th AIAA Applied Aerodynamics Conference 2011 (2011). doi:10.2514/6.2011-3669.
- [4] D. Igali, O. Mukhmetov, M. Zhao, S. Fok, L. Teh, Comparative analysis of turbulence models for automotive aerodynamic simulation and design, International Journal of Automotive Technology 20 (04 2019). doi:10.1007/s12239-019-0107-7.
- [5] D. Aljure, J. Calafell, A. Baez, A. Oliva, Flow over a realistic car model: Wall modeled large eddy simulations assessment and unsteady effects, Journal of Wind Engineering and Industrial Aerodynamics 174 (2018) 225–240. doi:10.1016/j.jweia.2017.12.027.
- [6] S. Jakirlic, L. Kutej, P. Unterlechner, C. Tropea, Critical assessment of some popular scale-resolving turbulence models for vehicle aerodynamics, SAE International Journal of Passenger Cars - Mechanical Systems 10 (03 2017). doi:10.4271/2017-01-1532.
- [7] N. Ashton, A. West, S. Lardeau, A. Revell, Assessment of rans and ddes methods for realistic automotive models, Computer and Fluids 128 (2016). doi:10.1016/j.compfluid.2016.01.008.
- [8] E. Mercker, A blockage correction for automotive testing in a wind tunnel with closed test section, Journal of Wind Engineering and Industrial Aerodynamics 22 (2), special Issue 6th Colloquium on Industrial Aerodynamics Vehicle Aerodynamics (1986). doi:10.1016/0167-6105(86)90080-2.
- [9] U. Bin-Khalid, J. Suda, Comparative study of wind tunnel blockage correction methods for bluff body aerodynamics (swimmer's hand model), M.sc thesis, Budapest University of Technology and Economics (07 2020).
- [10] G. Wickern, Wake related wind tunnel corrections for closed wall test sections, SAE Technical Paper 2006-01-0567 (2006). doi:10.4271/2006-01-0567.
- [11] A. Hyvarinen, Investigation of blockage correction methods for full-scale wind tunnel testing of trucks, M.sc thesis, KTH, Royal Institute of Technology (2015).
- [12] B. Hetherington, D. Sims-Williams, Wind tunnel model support strut interference, SAE Technical Paper 2004-01-0806 (03 2004). doi:10.4271/2004-01-0806.
- [13] B. Hetherington, Interference of supports used for ground vehicle wind tunnel testing, Ph.D. thesis, available at Durham E-Thesis Online: <http://etheses.dur.ac.uk/2671/> (2006).
- [14] B. Hetherington, D. Sims-Williams, Support strut interference effects on passenger and racing car wind tunnel models, SAE Technical Paper 2006-01-0565 (04 2006). doi:10.4271/2006-01-0565.
- [15] G. Carr, W. Stapleford, Blockage effects in automotive wind-tunnel testing, SAE Technical Paper 860093 (1986). doi:10.4271/860093.
- [16] S. Gupta, V. J. Shah, Numerical study on the effects of wind-tunnel conditions on a passenger vehicle in crosswinds, M.sc thesis, KTH Royal Institute of Technology (10 2020).
- [17] Z. Yang, M. Schenkel, Assessment of closed-wall wind tunnel blockage using cfd, SAE Technical Paper 2004-01-0672 (2004). doi:10.4271/2004-01-0672.
- [18] Z. Yang, A. Nastov, M. Schenkel, Further assessment of closed-wall wind tunnel blockage using cfd, SAE Technical Paper 2005-01-0868 (2005). doi:10.4271/2005-01-0868.
- [19] A. Heft, T. Indinger, N. Adams, Introduction of a new realistic generic car model for aerodynamic investigations, SAE Technical Paper 2012-01-0168 (04 2012). doi:10.4271/2012-01-0168.
- [20] R. F. Soares, A. Knowles, S. G. Olives, K. Garry, J. C. Holt, On the aerodynamics of an enclosed-wheel racing car: An assessment and proposal of add-on devices for a fourth, high-performance configuration of the driver model, SAE Technical Paper 2018-01-0725 (April 2018). doi:10.4271/2018-01-0725.
- [21] R. F. Soares, A. Knowles, S. G. Olives, K. Garry, J. C. Holt, Driver hp-f: the cad geometry pack, Cranfield Online Research Data (CORD), 2018. doi:10.17862/cranfield.rd.c.3969120.v1.
- [22] R. F. Soares, A. Knowles, S. G. Olives, K. Garry, J. C. Holt, Dataset for sae 2018-01-0725 "on the aerodynamics of an enclosed-wheel racing car: An assessment and proposal of add-on devices for a fourth, high-performance configuration of the driver model", Cranfield Online Research Data (CORD), 2018. doi:10.17862/cranfield.rd.c.3969120.
- [23] N. Ahmad, E. Abo-Serie, A. Gaylard, Mesh optimization for ground vehicle aerodynamics, CFD Letters 2 (03 2010).
- [24] R. F. Soares, F. Souza, Influence of cfd setup and brief analysis of flow over a 3d realistic car model, SAE Technical Paper 2015-36-0513 (09 2015). doi:10.4271/2015-36-0513.
- [25] L. Jowsey, M. Passmore, Experimental study of multiple-channel automotive underbody diffusers, Proceedings of the Institution of Mechanical Engineers, Part D: The Journal of Automobile Engineering (May 2010). doi:10.1243/09544070JAuto1339.
- [26] L. Jowsey, An experimental study of automotive underbody diffusers, Ph.D. thesis (2013).
- [27] Best practice: Scale resolving simulations in ansys cfd, ANSYS, Inc., 2015.
- [28] M. Lanfrit, Best practice guidelines for handling automotive external aerodynamics with fluent, Fluent Deutschland GmbH (2005).
- [29] K. Zore, D. Caridi, I. Lockley, Fast and accurate prediction of vehicle aerodynamics using ansys mosaic mesh, SAE Technical Paper 2020-01-5011 (2020). doi:10.4271/2020-01-5011.
- [30] Modeling turbulent flows, introductory FLUENT Training (2006).
- [31] V. Skaperdas, Meshing for the 1st automotive cfd prediction workshop, <https://autocfd1.s3.eu-west-1.amazonaws.com/Presentations/BETA-CAE-Case2aMeshing.pdf> (December 2019).
- [32] P. J. Roache, Perspective: A method for uniform reporting of grid refinement studies, Journal of Fluids Engineering; (United States) 116:3 (9 1994). doi:10.1115/1.2910291.

- [33] I. Celik, U. Ghia, P. Roache, C. Freitas, H. Coloman, P. Raad, Procedure of estimation and reporting of uncertainty due to discretization in cfd applications, *J. Fluids Eng.* 130 (07 2008). doi:10.1115/1.2960953.
- [34] N. Haziqah, M. Sukri, M. S. Mat Ali, S. Mansor, S. Muhamad, A. Zaki, Numerical study for flow over a realistic generic model, driver, using urans, *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* (08 2018).
- [35] P. Kumar, E. Guilmineau, Urans simulation of a realistic road vehicle with k- ω sst and ssg/lrr- ω model as turbulence closure, Fourth international conference in numerical and experimental aerodynamics of road vehicles and trains (Aerovehicles4) (07 2020).
- [36] A. Guerrero, R. Castilla, Aerodynamic study of the wake effects on a formula 1 car, *Energies* 13 (2020). doi:10.3390/en13195183.
- [37] R. D. Lotz, B. E. Thompson, C. A. Konings, Numerical uncertainties in transonic flow calculations for airfoils with blunt trailing edges, *International Journal For Numerical Methods in Fluids* 24 (1997).
- [38] J. Katz, Aerodynamics of race cars, *Annu. Rev. Fluid Mech* 38 (01 2006). doi:10.1146/annurev.fluid.38.050304.092016.
- [39] R. Himeno, K. Fujitani, Numerical analysis and visualization of flow in automobile aerodynamics development, in: S. Murakami (Ed.), *Computational Wind Engineering 1*, Elsevier, 1993. doi:10.1016/B978-0-444-81688-7.50088-7.
- [40] S. R. Ahmed, G. Ramm, G. Faitin, Some salient features of the time-averaged ground vehicle wake, *SAE Technical Paper 840300* (1 1984). doi:10.4271/840300.
- [41] J. W. Slater, Examining spatial (grid) convergence, <https://www.grc.nasa.gov/WWW/wind/valid/tutorial/spatconv.html> (February 2021).
- [42] P. Spalart, S. Allmaras, A one-equation turbulence model for aerodynamic flows, *AIAA* 439 (01 1992). doi:10.2514/6.1992-439.
- [43] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, J. Zhu, A new k-epsilon eddy viscosity model for high reynolds number turbulent flows, *Computers & Fluids* 24 (3) (1995). doi:10.1016/0045-7930(94)00032-T.
- [44] F. R. Menter, Two-equation eddy-viscosity turbulence models for engineering applications, *AIAA Journal* 32 (1994).
- [45] B. Launder, G. Reece, W. Rodi, Progress in the development of a reynolds stress turbulence closure, *Journal of Fluid Mechanics* 68 (04 1975). doi:10.1017/S0022112075001814.
- [46] F. Menter, M. Kuntz, R. Langtry, Ten years of industrial experience with the sst turbulence model, *Heat and Mass Transfer* 4 (01 2003).
- [47] M. Aultman, Z. Wang, L. Duan, Effect of time-step size on flow around generic car models, *Journal of Wind Engineering and Industrial Aerodynamics* 219 (2021). doi:10.1016/j.jweia.2021.104764.
- [48] L. Porcar, W. Toet, J. Gamez-Montero, Study of the effect of vertical airfoil endplates on diffusers in vehicle aerodynamics, *Designs* 5 (2021). doi:10.3390/designs5030045.
- [49] X. Zhang, A. Senior, A. Ruhrmann, Vortices behind a bluff body with an upswept aft section in ground effect, *International Journal of Heat and Fluid Flow* 25 (1) (2004). doi:10.1016/j.ijheatfluidflow.2003.11.002.
- [50] M. Breuer, N. Jovičić, K. M. Mazaev, Comparison of des, rans and les for the separated flow around a flat plate at high incidence, *International Journal for Numerical Methods in Fluids* 41 (2003). doi:10.1002/flid.445.
- [51] F. R. Menter, R. Lechner, A. Matyushenko, Best practice: Rans turbulence modeling in ansys cfd, Tech. rep., Ansys Germany GmbH, <https://www.ansys.com/resource-center/technical-paper/best-practice-rans-turbulence-modeling-in-ansys-cfd> (2021).
- [52] E. Ljungskog, S. Sebben, A. Broniewicz, Inclusion of the physical wind tunnel in vehicle cfd simulations for improved prediction quality, *Journal of Wind Engineering and Industrial Aerodynamics* 197 (02 2020). doi:10.1016/j.jweia.2019.104055.
- [53] S. Gupta, V. J. Shah, Numerical study of the effects of wind-tunnel conditions on a passenger vehicle in crosswinds (dissertation), Ph.D. thesis (2020).



DSPACE

<https://dspace.org/>

Integrated numerical and experimental workflow for high-performance vehicle aerodynamics

Rijns, Steven; Teschner, Tom-Robin; Blackburn, Kim; Brighton, James

2024-02-06

Society of Automotive Engineers (SAE)

<https://doi.org/10.4271/2024-01-5016>

Rijns S, Teschner TR, Blackburn K, Brighton J. (2024) Integrated numerical and experimental workflow for high-performance vehicle aerodynamics. Available online 6 February 2024, SAE TP 2024-01-5016