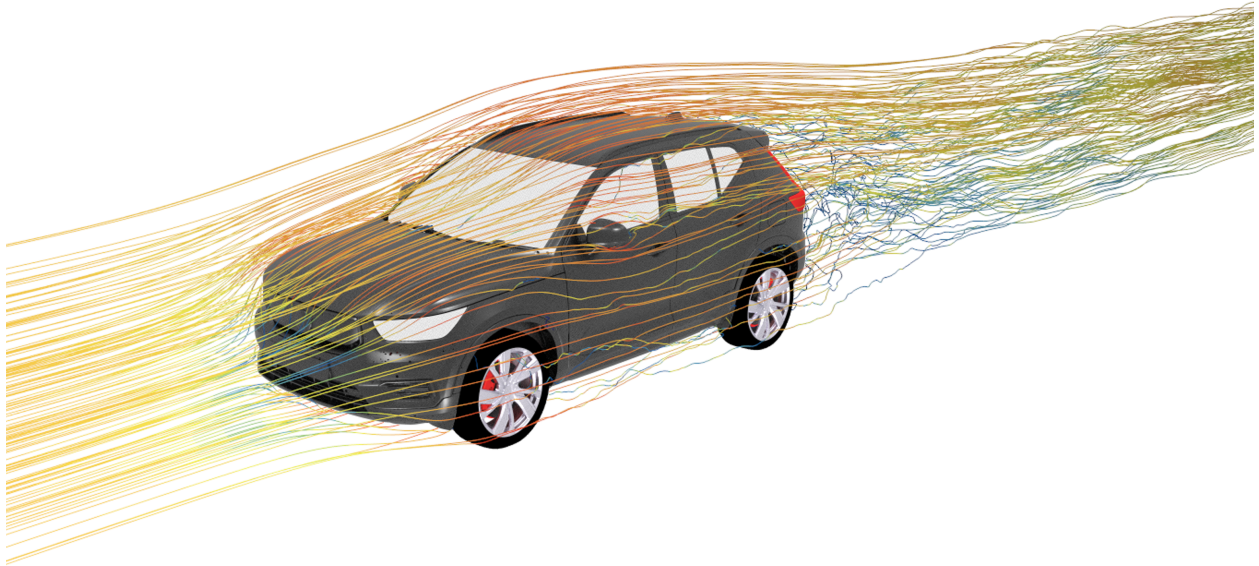




CHALMERS
UNIVERSITY OF TECHNOLOGY



Towards on-road aerodynamics

Evaluation and implementation of real-world conditions

Master's thesis in Automotive Engineering

AGAM SADAN

VICENTE SARTOR POLONI

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES (M2)

CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2022
www.chalmers.se

MASTER'S THESIS 2022

Towards on-road aerodynamics

Evaluation and implementation of real-world conditions

Agam Sadan, Vicente Sartor Poloni



Department of Mechanics and Maritime Sciences
Division of Vehicle Engineering and Autonomous Systems (VEAS)
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden 2022

Towards on-road aerodynamics
Evaluation and implementation of real-world conditions
Agam Sadan, Vicente Sartor Poloni

© Agam Sadan, Vicente Sartor Poloni, 2022.

Supervisor: Erik Sällström and Guglielmo Minelli, Volvo Cars
Examiner: Alexey Vdovin, Department of Mechanics and Maritime Sciences

Master's Thesis 2022:12
Department of Mechanics and Maritime Sciences
Division of Vehicle Engineering and Autonomous Systems (VEAS)
Chalmers University of Technology
SE-412 96 Gothenburg
Telephone +46 31 772 1000

Cover: Streamlines over a Volvo XC40 under on-road turbulence.

Typeset in L^AT_EX
Printed by Chalmers Reproservice
Gothenburg, Sweden 2022

Towards on-road aerodynamics
Evaluation and implementation of real-world conditions
Agam Sadan, Vicente Sartor Poloni
Department of Mechanics and Maritime Sciences
Chalmers University of Technology

Abstract

As newer regulations require improved energy efficiency for road vehicles to reduce their environmental impact, the need to study the effect of real-world conditions becomes a strategic step for improving the on-road performance and range of new vehicles. The aerodynamic performance of a road vehicle is sensitive to the turbulence it undergoes in realistic conditions, making the introduction of turbulence to simulations a must. It also provides a better base for an early understanding of a car's behaviour and optimizations that will perform better in the high turbulence conditions experienced on-road. Adding turbulence levels experienced on the road to aerodynamic simulations will ultimately benefit the end-users, with improved high-speed stability and decreased fuel or energy consumption.

The study starts by evaluating literature based on related topics to gather common findings and recommendations. Based on the literature review, the simulations are run with different turbulence settings and further bench-marked to standard quasi-steady simulations in the Computational Fluid Dynamics (CFD) software StarCCM+. Furthermore, coast-down test data is used to extract the aerodynamic load from the total road load using a proprietary method. This serves as a comparison to find what settings are the most representative of realistic conditions.

Based on the findings from the comparison between real-world data and in-house simulations, an optimized set of parameters is proposed for a possible standardized method of computing on-road conditions in CFD.

Keywords: Vehicle aerodynamics, On-road conditions, CFD, Turbulence, Transient simulations, IDDES

Preface

The study includes the investigation of the effects of on-road turbulence on a passenger vehicle. This 30 ECTS master thesis project has been conducted at the Department of Mechanics and Maritime Sciences at Chalmers University of Technology, with the work carried out at the Aerodynamics Department at Volvo Car Corporation in Torslanda, Sweden. This work serves as a starting point for the future implementation of on-road flow in computational fluid dynamics.

The focus of this thesis is on building a model in STAR-CCM+ with synthetic turbulence that has an energy spectrum similar to what is seen on the road. The idea is to compare the differences in results and spectra to represent these real-world conditions in simulations. This will help optimise new vehicles for better performance in terms of energy efficiency as well as stability. The end goal is to have CFD simulations that closely represent the on-road environment.

Firstly, we would like to thank our supervisors Erik Sällström, Guglielmo Minelli and examiner Alexey Vdovin for guiding us in the right manner to accomplish the objectives of the thesis. We would also like to thank Magnus Urquhart, Per Abrahamsson and everyone at the Aerodynamics division, Volvo Car Corporation, for consistently supporting us throughout the duration of the thesis. A special thanks to Emil Ljungskog from Siemens Digital Industry Software for constantly making time and showing his interest to support the progress of the thesis.

Göteborg

List of Acronyms

Below is the list of acronyms that has been used throughout this thesis listed in alphabetical order:

CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
DES	Detached Eddy Simulation
IDDES	Improved Delayed Detached Eddy Simulation
LES	Large Eddy Simulation
LS	Length Scale
MRF	Moving Reference Frame
PSD	Power Spectral Density
RANS	Reynold's Averaged Navier-Stokes
RMS	Root Mean Squared
SST	Shear Stress Transport
TI	Turbulence Intensity
VCC	Volvo Car Corporation

Nomenclature

Below is the nomenclature of parameters and definitions used in this thesis.

Parameters

ρ	Air Density	kg/m ²
A	Vehicle Frontal Area	m ²
g	Gravity	m/s ²
ν	Kinematic Viscosity	m ² /s
μ	Coefficient of Friction	
α	Road Inclination	°
C_d	Coefficient of Drag	
C_l	Coefficient of Lift	
C_{lf}	Front Coefficient of Lift	
C_{lr}	Rear Coefficient of Lift	
F_D	Drag Force	N
U	Mean Velocity	m/s
I	Turbulence Intensity	
U'	RMS of Turbulent Velocity Fluctuations	m/s
l	Turbulent Length Scale	m
k	Turbulent Kinetic Energy	J/kg
C_μ	Turbulence Model Constant	m/s
ϵ	Turbulence Dissipation	J/kg·s
ω	Specific Turbulence Dissipation	1/s
PSD	Power Spectral Density	g ² /Hz

Contents

List of Acronyms	ix
Nomenclature	xi
List of Figures	xv
List of Tables	xvii
1 Introduction	1
1.1 Background	1
1.1.1 The on-road environment	1
1.1.2 Vehicle aerodynamics	1
1.1.3 Wind tunnels and turbulence generators	2
1.1.4 Computational fluid dynamics(CFD)	2
1.2 Purpose	3
1.3 Project scope and limitations	3
2 Theory	5
2.1 Computational fluid dynamics (CFD)	5
2.1.1 Turbulence modelling	5
2.1.2 Turbulence kinetic energy	6
2.1.3 Turbulence intensity	6
2.1.3.1 Turbulence intensity estimation	7
2.1.4 Turbulence length scale	7
2.1.4.1 Length scale estimation	8
3 Methods	9
3.1 Boundary conditions	10
3.2 Simple car model	10
3.3 DrivAer model	11
3.3.1 Improvements to the DrivAer simulations	14
3.4 Volvo XC40 Recharge model	14
3.4.1 Discretization scheme	18
4 Results	19
4.1 DrivAer model	19
4.1.1 Low turbulence simulation	20

Contents

4.1.2	High turbulence simulation	22
4.2	Volvo XC40 Recharge model	24
4.2.1	Low turbulence simulation	25
4.2.2	High turbulence simulation	27
4.2.3	Comparisons between low and high turbulence models	29
4.2.4	Coast-down and wind-tunnel test analysis	33
5	Conclusion	35
	Bibliography	37
A	Appendix	I
A.1	Power spectral density (PSD)	I

List of Figures

3.1	Flowchart for the general CFD phases that are undertaken in a simulation	9
3.2	Vector scene showing velocity streamlines around the simple car model	11
3.3	DrivAer in notch-back configuration, side-view	11
3.4	The DrivAer's flat floor is shown, bottom-view	12
3.5	Snapshot of velocity and line integral convolutions streamlines at $y = 0$ around the DrivAer model with 0.1% TI	12
3.6	DrivAer run with 10% TI and 4.6m LS	13
3.7	DrivAer run with 8% TI and 2m LS	13
3.8	Volvo XC40 geometry model and dimensions, side-view	15
3.9	Volvo XC40 geometry model and dimensions, front and rear views . .	15
3.10	The fine mesh from the inlet domain can be seen, side-view	16
3.11	The refined mesh around the XC40, side-view	16
3.12	Figure shows distance of the point probe placed in front of the car, side-view	17
3.13	Figure shows the line probe and its resolution placed in line with the Volvo XC40, side-view	17
4.1	An example image from a low turbulence simulation on the DrivAer notch-back model showing streamlines around the test object	19
4.2	A mid-sectional scalar image of the low turbulence simulation on the DrivAer notchback represents the turbulence intensity at different regions, side-view	20
4.3	Scalar scene of the low turbulence simulation showing the mean velocity magnitude at the center plane of the DrivAer notchback, side-view	21
4.4	The turbulence variation across the DrivAer domain along a line probe	21
4.5	A mid-sectional scalar image of the high turbulence simulation on the DrivAer notchback represents the turbulence intensity at different regions, side-view	22
4.6	Scalar scene of the high turbulence simulation showing the mean velocity magnitude of the DrivAer notchback, side-view	23
4.7	The mean coefficient pressure on the rear end of the DrivAer between the low (left) and high (right) turbulence models respectively is shown, rear-view	23
4.8	The turbulence variation across the domain by a line probe	24

List of Figures

4.9	An example image of a low turbulence simulation on the XC40 showing streamlines around the test object	25
4.10	A mid-sectional scalar image of the low turbulence simulation on the XC40 represents the turbulence intensity at different regions, side-view	25
4.11	Scalar scene of the low turbulence simulation showing the mean velocity magnitude of the XC40, side-view	26
4.12	Accumulated drag and lift coefficients of the XC40 low turbulence simulation	27
4.13	A mid-sectional scalar image of the high turbulence simulation on the XC40 representing the turbulence intensity at different regions, side-view	28
4.14	Scalar scene of the high turbulence simulation showing the mean velocity magnitude of the XC40, side-view	28
4.15	An accumulated drag and lift curves of the high turbulence simulation on the XC40 is shown	29
4.16	The delta accumulated drag and lift curves between the two simulations are represented	30
4.17	The vector plots show the mean velocity on the wake end of the car between the low (left) and high (right) turbulence models respectively, side-view	30
4.18	The vector plots show the mean velocity on the wake end of the car between the low (left) and high (right) turbulence models respectively, top-view	31
4.19	The mean pressure coefficient on the rear end of the car between the low (left) and high (right) turbulence models respectively is shown, rear-view	31
4.20	The iso-surfaces of the car between the low (left) and high (right) turbulence models respectively	32
A.1	The dimensional power spectra for turbulence intensity 8% of length scale 2.0m(top) and dimensional wind component(U, V and W) power spectra(bottom)	II

List of Tables

4.1	Tabulation of delta results from high to low turbulence scenarios for the DrivAer simulations	24
4.2	Tabulation of delta results from low to high turbulence scenarios for the Volvo XC40 simulations	32
4.3	Differences in C_d values between the simulation results from the wind tunnel and coast-down respectively	33

List of Tables

1

Introduction

1.1 Background

1.1.1 The on-road environment

From the beginning of aerodynamics studies, most simulations have focused on maintaining low levels of turbulence, which have allowed controlled experiments and repeatability. However, these low turbulence environments hardly occur in real-world conditions, where wind, gusts or the wakes of passing cars prevails. These effects can generate turbulence intensities of up to 15%, while in most simulations and wind tunnel tests the turbulence intensities are 0.1% [1], thereby indicating a need for simulations that can represent the on-road environment better.

On-road aerodynamics has been gaining relevance in recent years due to more stringent requirements for lower emissions and efficiency of passenger vehicles. Experiments were performed by Wordley and Saunders ([1] and [2]) to obtain data on turbulence intensities and length scales in different on-road scenarios. Their results serve as a starting point for this study. A turbulent environment can have adverse effects on a vehicle's performance, usually causing higher drag, which directly influences energy consumption. Turbulence can also change the vehicle's characteristics regarding drivability by affecting its response to the variation of wind.

1.1.2 Vehicle aerodynamics

As emission regulations and environmental requirements become more strict, the energy efficiency of road vehicles needs to improve to comply with these new rules. Aerodynamics plays a big role in the energy consumption for automobiles, usually at speeds above 70 to 80 km/h [10], when the aerodynamic drag force becomes the dominant force opposing the motion of a vehicle. The equations for driving force are given below:

$$F_x = F_A + F_G + F_R + F_D \quad (1.1)$$

Where F_A is the resistance from acceleration, F_G is the climbing resistance, F_R is the rolling resistance and F_D is the drag force. Expanding the terms above we obtain the following equation [11]:

$$F_x = \gamma m \frac{dv}{dt} + mg \sin(\alpha) + \mu mg \cos(\alpha) + \frac{1}{2} \rho C_D A v^2, \quad (1.2)$$

1. Introduction

where γ is a factor that accounts for the resistance caused by moving parts in a vehicle, m is the vehicle's mass, g is the gravitational acceleration, α is the road inclination, μ is the coefficient of friction, C_D is the drag coefficient, A is the frontal area of the vehicle and v is its velocity. To obtain the power requirement, i.e. the power needed to drive at a certain velocity with given acceleration and road inclination, the driving force is multiplied by velocity:

$$P_x = vF_x \quad (1.3)$$

The power requirement is related to the energy consumption of the vehicle, and velocity has a great impact on it. At high speeds, the energy consumption per distance is the asymptote of F_x and proportional to the square of velocity, while P_x is proportional to the cube of velocity. At speeds above 70 to 80 km/h [10], F_D is the main force acting on the vehicle at a constant velocity and small road inclinations, causing optimizations in aerodynamics to have a great effect on the energy consumption of road vehicles.

1.1.3 Wind tunnels and turbulence generators

Most wind tunnels today have structures that reduce turbulence to have quasi-steady air flow conditions in their test sections. Only a small number of wind tunnels have turbulence generation systems. There are different methods for generating turbulence in wind tunnels, with both passive and active methods. For passive turbulence generators, grids are used to increase the unsteadiness of air, whilst for active generators, turning vanes working at pre-determined frequencies are used as is the case for the Pininfarina and FKFS wind tunnels. Commonly, active turbulence generators are capable of generating high turbulence intensity levels of up to 45%, whereas passive generate up to only 3% turbulence intensities.[2]

1.1.4 Computational fluid dynamics(CFD)

To save time and cut costs, most new vehicle projects have their aerodynamics tests starting with CFD. The benefits are that the virtual model of the car can be tested before physical models are assembled, which is more cost-effective and allows for early improvements in the shape of the vehicle. Also, it gives the added benefits of predicting the vehicle behaviour and aerodynamics which can serve as a base for initial predictions on energy consumption, as well as the need for add-ons to rectify possible stability problems.

For CFD, a virtual environment is created where all physics are defined by the user, and in conjunction with a CAD model of a vehicle. Generally, most CFD simulations use low levels of turbulence intensity of 1% or less. However, for this study, the turbulence intensities will be varied along with a more complex physical modelling to represent realistic road conditions. The software used in the following study is STAR-CCM+ developed by Siemens Digital Industries.

1.2 Purpose

The purpose of this project is to show the importance of implementing more realistic, turbulent conditions to the current aerodynamics simulation methods used for passenger vehicles, to realistically represent their behaviour on the road. As most simulations in the Vehicle Aerodynamics industry run low turbulence simulations and prioritize the reduction of C_d , the computational costs and time spent on these fine optimizations may not have the intended effect when the vehicle is finally on-road where it faces different and unpredictable conditions.

Low turbulence simulations are commonplace in CFD and wind tunnel simulations, an effective way to improve the aerodynamics of a car. However, the addition of real-world conditions with higher levels of turbulence in simulations, would facilitate early predictions and improvements of a vehicle's on-road performance. This allows for optimization in the early stages of the design of a new vehicle, offering better estimates for its aerodynamic efficiency and range, which are important factors for new electric vehicles due to limitations in range.

1.3 Project scope and limitations

The scope of this project is to implement close to real-world flow conditions that vehicles face on-road, in CFD simulations. Standard procedure for testing in CFD is done using ideal conditions, with lower levels of turbulence than seen on-road, showing an under prediction of drag force that a vehicle will experience under real-world flow.

To accomplish the scope of the study, a turbulence model will be implemented using the STAR-CCM+ software, by varying the turbulence intensities and length scales to match the trend of energy spectra of turbulence to what has been found in the literature studies of on-road tests. A limitation of STAR-CCM+ is the inability to differentiate the turbulence intensities and length scales along each of the three components (U, V or W). Other smaller limitations and solutions are discussed in the Methodology. Due to the limited time for the thesis, experiments were only run in CFD, where real-world flow conditions can be modelled and simulated.

To further validate the results, coast-down data provided by the Vehicle Energy Efficiency department at Volvo Cars is used for comparisons to high turbulence simulations. The coast-down tests are performed by recording the forces opposing a vehicle, starting from high speed and then coasting to a halt. The tests are done on a flat straight road, with minor external disturbances, to obtain the loss-inducing forces affecting the vehicle. As the energy consumption data is not gathered specifically for the thesis, the exact vehicle specifications may vary from the model used in simulations and thus could cause tiny discrepancies.

The studies are conducted on two defined car models, the 'DrivAer' model in notch-

1. Introduction

back configuration and the Volvo XC40 Recharge model. The DrivAer model is chosen due to its simplicity, whilst the Volvo XC40 Recharge model to see the full effect of turbulence on a complex vehicle model. The other reason for choosing the Volvo XC40 model is due to the availability of coast-down test data from VCC. More details of these models are mentioned later.

2

Theory

In this section, the intricate process of setting up the computational fluid dynamics models as well as turbulence and how it influences flow will be explained. The goal of this section is to cover the basics of what has been used in the rest of this thesis.

2.1 Computational fluid dynamics (CFD)

By breaking down a computational domain into finite volume cells, it is possible to numerically analyse the fluid flow within the domain's environment quickly and accurately with the means of Multiphysics computational software. The thesis will be using STAR-CCM+ to emulate the different turbulence models/setups. The CFD theory will thereby focus on the modelling of turbulence. It will also focus on the characterization of turbulence and the different synthetic turbulence parameters, which are required by STAR-CCM+ to implement a turbulent inlet.

2.1.1 Turbulence modelling

As the fluid flow around a passenger car is very complex while also consisting of a very turbulent wake, the Improved Delayed Detached Eddy Simulation (IDDES) which is a complex hybrid model between the standard Reynold's Averaged Navier-Stokes (RANS) and Large Eddy Simulation (LES) is chosen. It is proven to be more accurate for vehicle aerodynamic simulations. [4]

The RANS model shows poor performances with high unsteadiness and in areas where flow separation occurs, and can thus be omitted from consideration. LES on the other hand cannot effectively resolve the near-wall regions where turbulence length scales reach minimalistic ranges but have shown to be reliable for giving better results with a free shear flow that consists of large eddies.

Detached Eddy Simulations (DES) is a hybrid between the previous 2 models specified, where the RANS model aims to resolve the inner boundary layer and LES the detached regions. This effectively provides increased accuracy and cuts computational costs. The SST IDDES however minimizes the losses at the boundary layer formation areas near the walls and simulates instabilities.[5] It further resolves the log layer mismatch between wall-resolved and wall-modelled regions that is present in DES simulations.

2. Theory

The SST k-omega, a two-equation model of turbulent kinetic energy and dissipation, is easier to solve in comparison to the k-omega model. Another reason for choosing the SST model is because it blends between the k-epsilon model in the far-field parts of the domain and the k-omega model at the near wall, thereby saving computational power. [6]

2.1.2 Turbulence kinetic energy

Turbulence kinetic energy (TKE or k) concerning transient fluid flows is defined as the kinetic energy per unit mass in relation to turbulent eddies:

$$k = \frac{1}{2}[\overline{(u')^2} + \overline{(v')^2} + \overline{(w')^2}] \quad (2.1)$$

where the fluctuating part of the velocity components,

$$u' = u - \bar{u} \quad (2.2)$$

$$v' = v - \bar{v} \quad (2.3)$$

$$w' = w - \bar{w} \quad (2.4)$$

are the instantaneous velocity minus the average velocity for each component. The fluctuating velocity components are squared, time-averaged and halved to produce the TKE.

2.1.3 Turbulence intensity

Turbulence which is the irregular motion of a fluid or air resulting from eddies can be characterized as turbulence intensity or turbulence levels. It can be derived from the turbulence velocity fluctuations in the 3 different components and the mean free stream velocity (U_∞). As mentioned previously in the introduction, it is very important to get a good understanding of the turbulence intensity values that need to be at the inlet of the domain to represent accurate real-world conditions.

To determine turbulence intensity (TI),

$$TI = \frac{U'}{U_\infty} \quad (2.5)$$

where, U' is the root mean square of turbulent velocity fluctuations that can be determined from,

$$U' = \sqrt{\frac{1}{3}(\overline{(u')^2} + \overline{(v')^2} + \overline{(w')^2})} \quad (2.6)$$

or,

$$U' = \sqrt{\frac{2}{3}k} \quad (2.7)$$

where, k is represented as the turbulent kinetic energy.

The mean velocity or U_∞ can be computed as,

$$U_\infty = \sqrt{(u^2 + v^2 + w^2)} \quad (2.8)$$

The mean velocity equation as shown in equation 2.8 can be negated altogether when used as a field function for calculating turbulence intensity in equation 2.5 in a fluid simulation. The absolute value of the velocity specified at the inlet can be considered.

2.1.3.1 Turbulence intensity estimation

To accurately mimic real-world conditions, based on previous studies and on-road tests it is important to have a good understanding of the estimated turbulence intensities that a car undergoes in various conditions. As wind-tunnel simulations are made in the most controllable manner with the main focus of achieving steady flow in the most ideal conditions, they represent turbulence intensities between 0.1-0.3% and sometimes even lesser. Although there are means of using passive turbulence generators to achieve up to 3% intensities, this remains outside the scope of the thesis. As for the estimations taken into consideration based on the on-road aspects of flow conditions, turbulence intensities vary between 1-5% in natural wind and 5-15% while driving in the upstream wake of another car according to Alajbegovic et al. [8].

2.1.4 Turbulence length scale

To determine the turbulence kinetic energy, turbulence dissipation and specific turbulence dissipation based on the synthetic turbulence usage, the turbulence length scale is an important parameter that is required along with intensity levels. The turbulence length scale is used to define the size of the eddies induced at the inlet of a turbulent flow. It's a required parameter to gauge and define the synthetic turbulence flow properties.

The turbulence length scale can be calculated from the derived TKE as,

$$l = \tau \cdot k^{\frac{1}{2}} \quad (2.9)$$

where τ is the turbulent time scale which is determined as,

$$\tau = (\beta^* \cdot \omega)^{-1} \quad (2.10)$$

In equation 2.10 the β^* coefficient is obtained by the k-omega model in STAR-CCM+, and ω represents the specific dissipation rate.[9] The k-omega SST model is a mix of the k-epsilon and k-omega models. The SST model switches between the k-epsilon and k-omega between free-stream and near-wall situations respectively, making it the primary choice for its usage in the following study.

2. Theory

2.1.4.1 Length scale estimation

It is important to get an accurate estimation of the length scale as this can provide a good flow prediction. Estimating a precise turbulence length scale can be very difficult as they vary from case to case and only an approximation can be made. These eddy lengths vary constantly with temperature, speed, altitude and the definition of the object/problem itself.

A simple prediction of the turbulent length scale can be the characteristic length scale of the problem(or model). This characteristic length scale is used as the inlet parameter while defining the synthetic turbulence. The characteristic length scale in the case of vehicle aerodynamics can be considered as the width or height of the vehicle, or even the square root of the frontal area of the vehicle.

3

Methods

The standard procedure for running a CFD simulation can be divided into five main stages, as seen in Fig. 3.1. Starting with the "Preparation" stage which consists of importing the cleaned CAD geometry files of the vehicle parts, creating the domain and positioning them accordingly. Further in this step, the mesh refinements areas are defined by adding "boxes" in the areas of interest. It also includes defining the continuum region, the physics solvers, meshers, initialization of boundary conditions and finally, reports and monitors for recording the data that will be used for post-processing.

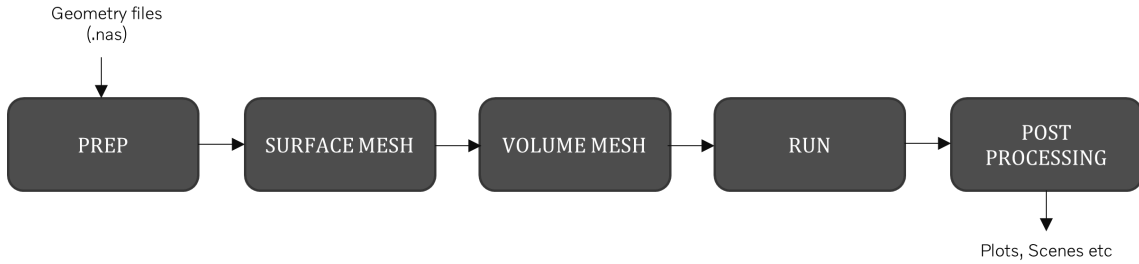


Figure 3.1: Flowchart for the general CFD phases that are undertaken in a simulation

In the simulations conducted during the course of the thesis, an open road scenario is followed. The large domain size is defined keeping in mind that no blockage effects must occur. The vehicle model is placed closer to the inlet, unlike standard CFD processes where the models are placed further away to help with the development of close to a steady flow. This step gives an added benefit during meshing by reducing the number of cells in the domain. A large refinement box is implemented in the domain that consists of a fine mesh from the inlet until the rear end of the car. The refinement box helps in conserving the turbulent kinetic energy provided from the inlet when synthetic turbulence is induced, as well as conserving turbulence intensities and length scales up to the vehicle.

The surface meshers help generate a triangulated surface mesh over the car body and the outer surface of the domain boundaries. Using the surface mesh as a starting point, the volume meshers generate a tetrahedral volume mesh in the 'Air' regions.

3. Methods

Before setting the simulation to run, the stopping criteria and other important safety criteria are specified to regulate the model. An appropriate time step is set in relation to the 2nd order implicit solver specified along with the total physical time that the simulation will run.

The final stage of the CFD process involves the "Post-processing" phase. This is where the analysis of the results is made through the means of different scenes, reports, and data obtained in the form of graphs and animations.

3.1 Boundary conditions

Boundary conditions help define the constraints of the domain like the initial, inlet and wall boundaries etc. The walls of the open road domain used for this thesis are set to have a symmetry plane condition. To remove boundary layer formations on the ground, the ground is set to a wall type and a relative velocity the same as that of the inlet velocity is implemented to replicate a moving floor, which is to emulate real road relation to the vehicle model. The inlet is defined with a velocity inlet condition specifying the velocity magnitude along with the turbulence settings, while the outlet is a pressure outlet with a constant condition. The DrivAer model uses a moving boundary condition at the wheels to simplify the fluid flow around this region. In the case of the Volvo XC40 model, a sliding mesh is used at the region where the rims are present to realistically visualize fluid flow around this region.

3.2 Simple car model

The first iterations of simulations were run using a simplistic car model provided by Siemens during their certification program. The goal was to try various physics settings as well as strategies to see what turbulence models suited best for the thesis, enabling multiple quick iterations to measure turbulence with probes and plots while having simulations with simple geometry and meshes. It helps not only save time, but also computational costs during the iterating stage.

During these runs, a lot could be learnt about the different solvers and how they affected the end solution. After running a few simulations, the domain was found to be too small for turbulence to fully develop, due to the length scales being similar to the width and height, so it was updated accordingly to avoid blockage effects. Any data collected from these experiments were disregarded, as their sole purpose was to check the functionality of the different parameters set while the model used in itself was very bluff to provide adequate results. An example image of the simple car model is shown in Fig. 3.2.

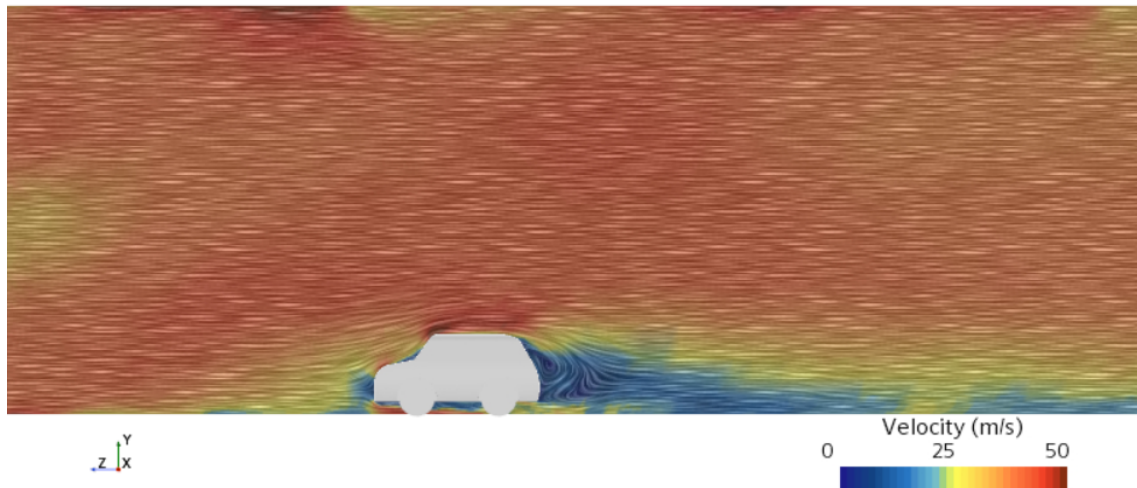


Figure 3.2: Vector scene showing velocity streamlines around the simple car model

3.3 DrivAer model

The next step was to try the different parameters for turbulence with a more realistic car model. The model of choice was the DrivAer in notch-back configuration with a flat floor and simplified wheels.[12]

The DrivAer model is a generic passenger vehicle model, which offers three different exchangeable rear-ends, offering three different body styles. It serves its purpose to bridge the gap between highly complex car models from manufacturers and very simplified ones, to provide a subject of study for aerodynamic projects. Due to its simplified but representative geometry of real passenger vehicles, it has been chosen as the start point for the simulations including turbulence in this study. The model has been created by the Chair of Aerodynamics and Fluid Mechanic at the Technical University of Munich, and it is available for download for individual studies.



Figure 3.3: DrivAer in notch-back configuration, side-view

3. Methods

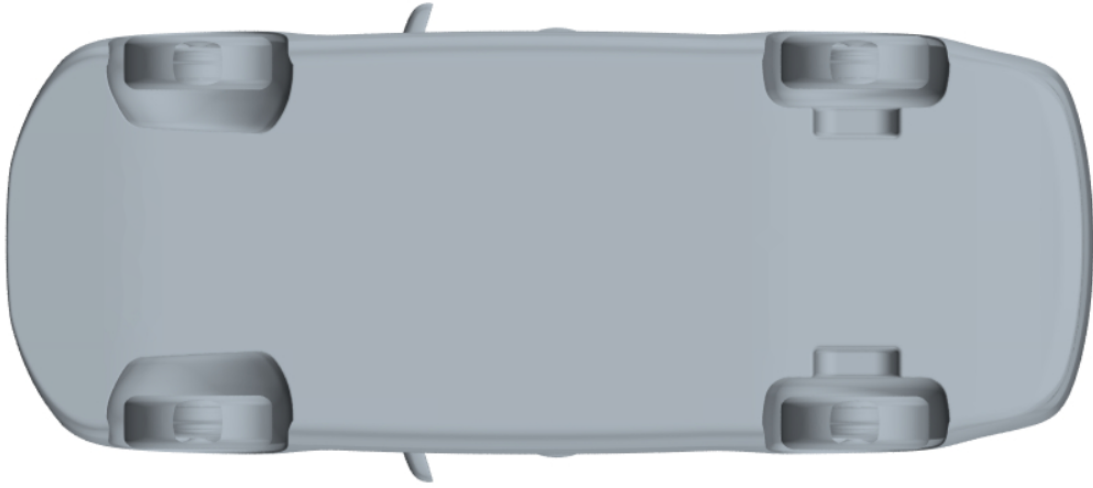


Figure 3.4: The DrivAer’s flat floor is shown, bottom-view

The initial simulations were run with the following inlet conditions only to observe the fluctuations in the domain caused by the varying turbulence parameters. Three different turbulence settings were set, the first with 0.1% TI and no synthetic turbulence, a second with 10% TI and 4.6m LS and a final setting trying to replicate the power spectral density curves from the on-road experiments by Wordley and Saunders [2], with 8% TI and 2m LS. The inlet velocity was set to 140 km/h for all simulations. In these initial simulations, both ground and wheels were stationary. The physical time was 3.7 seconds, with 1 second of averaging time.

On the first run with 0.1% of turbulence intensity, it is clear from Fig. 3.5 that the flow in the domain is relatively steady. The C_d was found to be 0.226, which is relatively low but is explained by the simplicity of the simulation and model used.

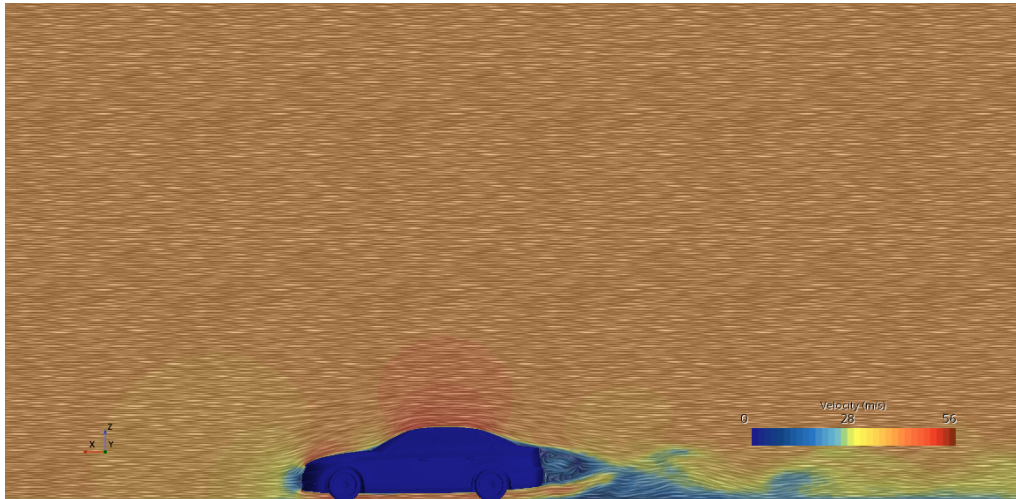


Figure 3.5: Snapshot of velocity and line integral convolutions streamlines at $y = 0$ around the DrivAer model with 0.1% TI

For the second run, the pattern throughout the domain changes with the addition of

turbulence. The large length scales can be seen by the change in colour in Fig. 3.6, which illustrates the variations in velocity within the domain. Also, the stagnation point in front of the vehicle has changed, as well as the rear wake, with a more pronounced down-wash likely causing a higher rear lift in comparison to the low turbulence case. Usually higher turbulence comes with smaller length scales, but following what was found in a paper by A. Gaylard et al. [7], a characteristic dimension for length scale is used in this case for experimentation. The paper mentions that length scales similar to the size of the vehicle often have a greater effect on the flow around it.

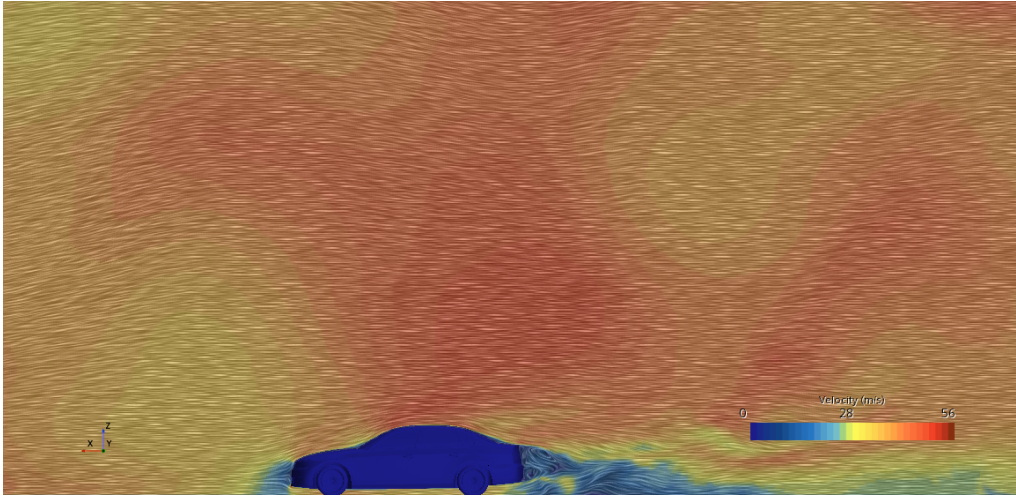


Figure 3.6: DrivAer run with 10% TI and 4.6m LS

For the third run, a smaller length scale was used, and it is again visible in the domain seen in Fig. 3.7. The parameters of 8% TI and 2m LS are used to replicate the turbulent energy spectra that is found previously. In this case, the stagnation up front is similar to the first run, but the down-wash is similar to the second run. These parameters seem to replicate the on-road environment better than the other runs, so they will be used as a baseline for more advanced simulations.

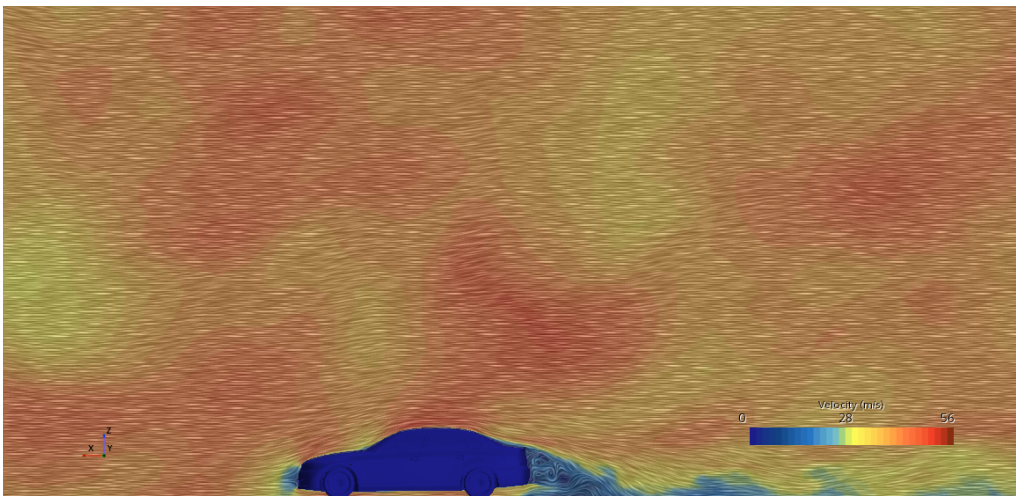


Figure 3.7: DrivAer run with 8% TI and 2m LS

3. Methods

3.3.1 Improvements to the DrivAer simulations

In order to obtain significant scientific data for this model, more complex simulations are run and its results will be presented in the next chapter. At first, a moving ground is added, matching the inlet speed of 140 km/h. This is done by setting a tangential velocity to the ground region in the simulation. It improves the results of the simulation by removing the boundary layer formed on the ground due to its roughness, as the region is defined with a no-slip condition.

To further improve the model, the rotation of wheels is implemented using a rotating wall boundary condition. Rotating wheels play an important part in the aerodynamics of a vehicle, as they modify flow structures and are a great source of drag. By finding the center of each wheel, its rotational axis can be determined and accordingly the wheel speed is set to match the ground speed.

Furthermore, the time of simulation is increased to 3 and 5 seconds for low and high turbulence simulations respectively, with time steps of $2.5 \cdot 10^{-4}$ seconds, and averaging of 1 and 3 seconds of the simulations respectively. Since the simulations have high turbulence intensities, the longer simulation time allows for the turbulence structures within the domain to fully develop, and the longer averaging allows for more accurate results in transient conditions. In addition to these improvements was to add the side mirrors to the DrivAer, in order to further improve the fidelity of the results.

3.4 Volvo XC40 Recharge model

The Volvo XC40 Recharge is chosen as the main test object for the thesis due to the detailed complexity of the model and the availability of coast-down data from Volvo Cars. The purpose of this is to use the road load from coast-down data which is ideally meant for energy consumption tests, to split this obtained road load appropriately into aerodynamic drag force, rolling resistance forces and other losses using a proprietary polynomial equation. The aerodynamic drag force from the polynomial model can be correlated with the results obtained from the thesis's simulations. With this information, the ideal workflow of new aerodynamic part development during the CFD stages can be verified if these parts provide an added benefit in real conditions as well. This way the time and cost spent on tiny optimizations that may provide small improvements in drag counts and perform well in low turbulence conditions may indeed reflect poor results during the high turbulence test runs.

A sample of the geometry image of the XC40 model used is shown along with its dimensions in Fig. 3.8 and Fig. 3.9.

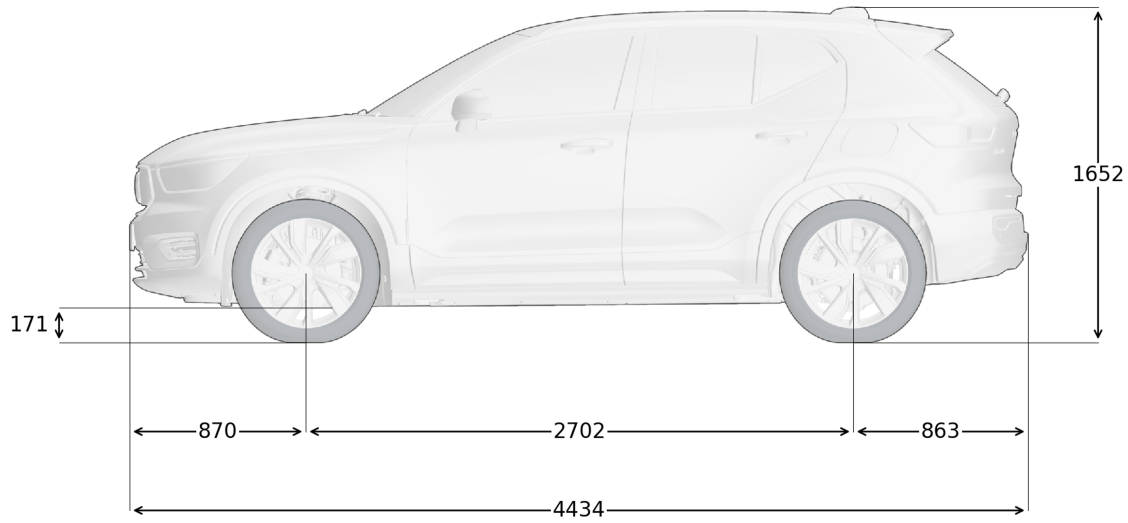


Figure 3.8: Volvo XC40 geometry model and dimensions, side-view

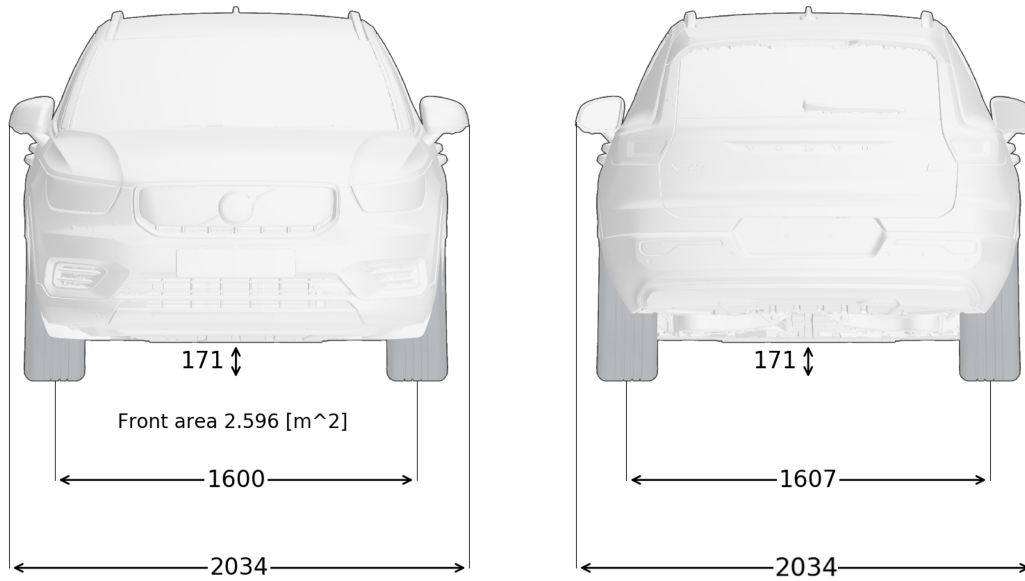


Figure 3.9: Volvo XC40 geometry model and dimensions, front and rear views

Although the simulations conducted with the XC40 consisted of numerous iterations, it can be boiled down to mainly two standard types of simulations, again into low and high turbulence types. The low turbulence simulations also run SST IDDES solvers with a very fine time-step with averaging results for 1 second since it consists of very low turbulence (0.1%). Although the same physics models were used for the later simulations, the introduction of higher turbulence required a longer averaging time, as the short 1-second time span will cause highly inaccurate results. It can be difficult to predict an accurate estimation of the averaging period, however excluding the flushing period and flow formation, an averaging time of 5 seconds has been

3. Methods

chosen.

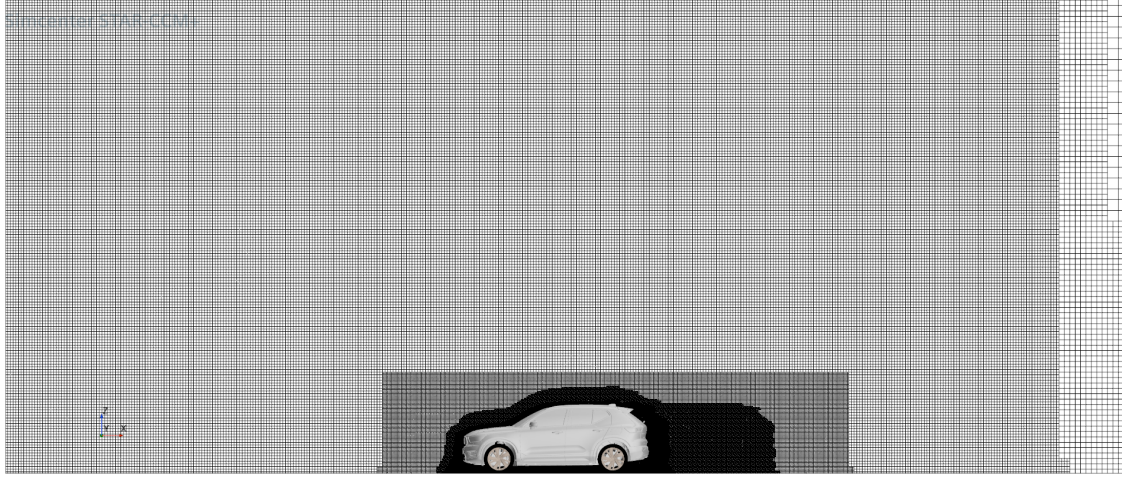


Figure 3.10: The fine mesh from the inlet domain can be seen, side-view

A standard method of meshing conducted at VCC is followed for the low turbulence simulation. As mentioned earlier the meshing method had to be altered in the cases where higher turbulence levels had been introduced. In order to retain the turbulent kinetic energy and structures of the vortices, which could be lost in the presence of large cells, a special refinement box is made from the inlet to the end of the vehicle's wake. A base size of 0.064m was specified for the special refinement box.

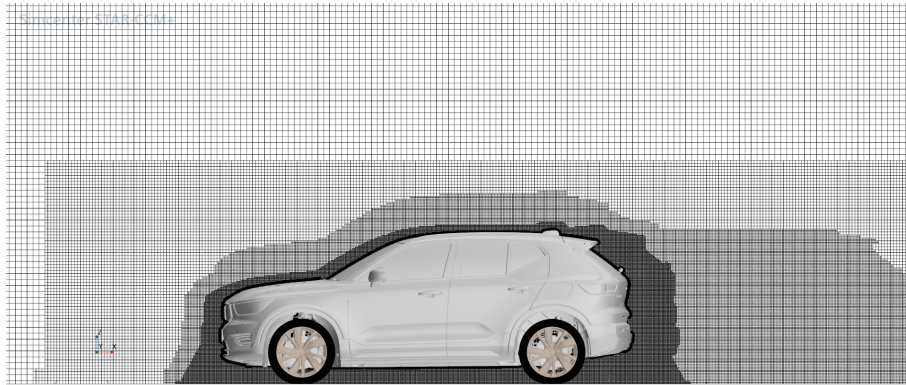


Figure 3.11: The refined mesh around the XC40, side-view

This resulted in a total cell count up to ≈ 460 million as opposed to the low turbulence model which consisted of ≈ 300 million cells. Such high cell counts could result in high computational requirements and time, which is why it is very important to have a reduced number of cells without the expense of deviated results. By moving the domain inlet closer to the vehicle not only were the total cells reduced by an approximate cell count of ≈ 50 million, but it also helped retain the turbulent length scales better. A profound parameterized method is repeated to remove invalid cells

in the air stream, prior to the run until none exists. This is done to ensure the numerical stability of the solution.

Simcenter STAR-CCM+

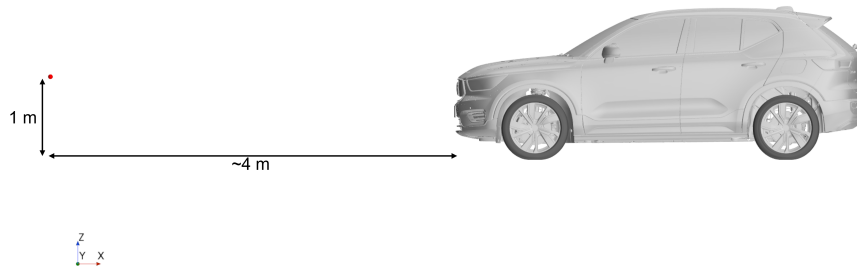


Figure 3.12: Figure shows distance of the point probe placed in front of the car, side-view

To analyze the Power spectral density curves for the different velocity components and the turbulence intensity, a point probe was used and placed at approximately one car length of the XC40 model and a height of 1m as shown in Fig. 3.12, similar to the On-Road Turbulence tests but which had it a lot closer to the test object. The reason for doing so is to avoid the point probe from recording data in these zones where there could be stagnation or low-pressure areas and where the velocity of the flow could be affected. Wordley and Saunders [1] goes on to explain the difficulties they face during the correction of data as their on-road experimental probes tend to undergo pitching and other issues which required accelerometers to get a good estimation of the vertical component. However, this is not the case in CFD simulations as the model remains stationary and such corrections are not required. By checking the pressure scenes, one can identify the closest point a probe can be placed outside this affected region.



Figure 3.13: Figure shows the line probe and its resolution placed in line with the Volvo XC40, side-view

To further verify the numbers mentioned by Adrian Gaylard's [7] research of how much turbulence is expected in the wake of a car, and how the intensities diminish over distance from the domain's inlet, a line probe has been implemented over 70 points as shown in Fig. 3.13. This helps determine the intensity the following car

3. Methods

undergoes with varying distances.

As for the PSD curves a point time Fourier transform function is used for measuring the turbulence intensity fluctuations and the three different velocity fluctuation components (U-Longitudinal component, V-Lateral component, W-Vertical component) at the very same place the point probe is placed.

3.4.1 Discretization scheme

For a complex simulation of this order, it is very hard to determine the initial conditions as this could be significantly different from the final solution. Therefore, the simulation of the low turbulence model is started with the first-order steady solver for larger time steps to obtain better convergence for the initial solution's run-time. After which the temporal discretization is switched to a second-order steady solver for smaller time steps up until the domain has been entirely flushed and the flow patterns develop. The flow also needs to be aligned parallel to the domain's mesh and must avoid oblique travelling between the different mesh cells which is why this step is necessary.

Next, the unsteady solver's second-order discretization is implemented with a very fine time-step of $2.5 \cdot 10^{-4}$ s and a maximum of 6 inner iterations to obtain faster convergence. The flow pattern is redeveloped after which the averaging of the low turbulence solution can begin for calculations.

As for the high turbulence model, it is continued from the low turbulence simulation, but with the required mesh changes where the last saved iteration's solution is interpolated into the new cells. A repetition of the invalid cell removal function is required to be used as the mesh has been recreated. With the same fine time steps and inner iterations, the physical time of the simulation is changed to a total of 10 seconds as specified earlier.

The averaging period starts from the 5th second, to allow the flow patterns caused by the high turbulence inlet to be recreated. An initial jump in residuals can generally be noticed when the discretization order is changed, which is normal so long as the averaging period consists of a convergent uniform residual fluctuation.

4

Results

In this chapter, the results of the simulations run using StarCCM+ are presented and analyzed. The two models presented with results are the DrivAer notch-back model and the Volvo XC40 Recharge, and the simulations were run with both low and high turbulence intensity levels for each of the cases. Finally, it is followed with a comparison of these simulation results to the data obtained from on-road tests and the wind tunnel, to evaluate the importance of implementing higher levels of turbulence in regular road aerodynamic simulations.

4.1 DrivAer model

Different settings were run with the DrivAer model, ranging from simpler to more complex simulations. For comparison purposes, only the longer and more complex simulations will be analyzed in this section. This comprises the simulations using the DrivAer model with side mirrors (as seen in Fig. 4.1), rotating wheels and a moving ground plane.

Simcenter STAR-CCM+

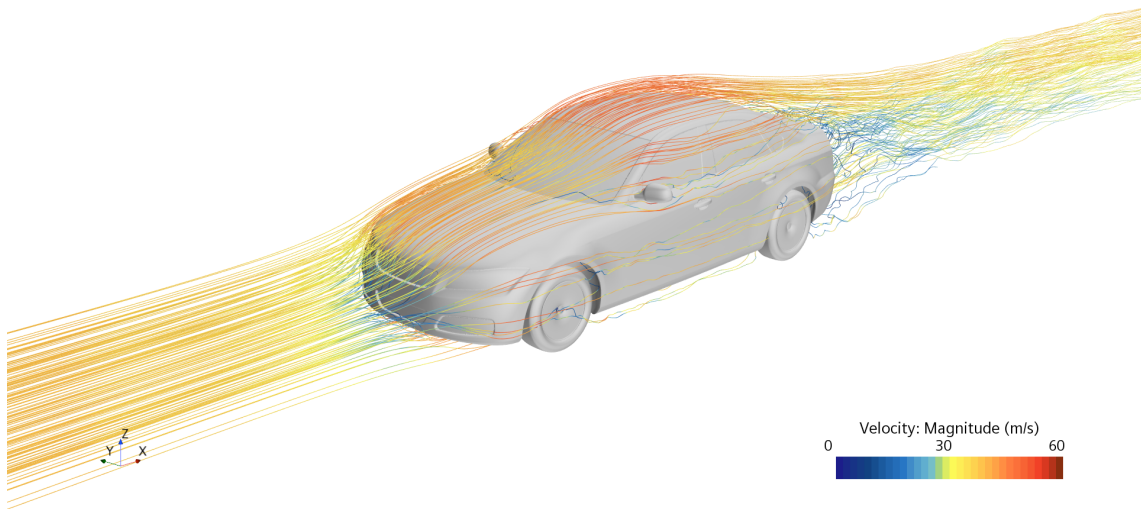


Figure 4.1: An example image from a low turbulence simulation on the DrivAer notch-back model showing streamlines around the test object

4. Results

4.1.1 Low turbulence simulation

For the low turbulence scenario, TI is set to 0.1%, similar to what is generally assumed while running standard simulations or in wind tunnel tests. Interestingly, the scalar plots from this simulation showed the airflow was detaching in the transition between the roof and the rear window, then reattaching at the boot lid before separating again at its end. This shows quite a big difference from the earlier simulations discussed in the Methodology section, indicating the importance of having smaller time steps and a higher number of inner iterations when running CFD simulations with IDDES.

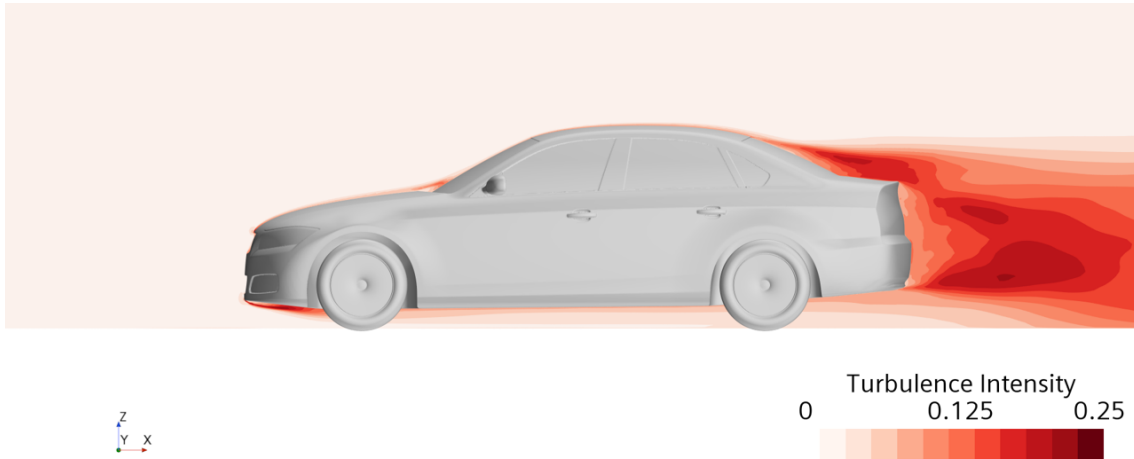


Figure 4.2: A mid-sectional scalar image of the low turbulence simulation on the DrivAer notchback represents the turbulence intensity at different regions, side-view

Visible in Fig. 4.2 is how steady the flow is due to the low turbulence. As seen in the theory section, in the case of driving in the wake of upstream vehicles, the turbulence intensities are much higher behind the vehicle, reaching 15% or more, as seen in Wordley and Saunders[1].

Flow separations in the cowl area, as well as the rear window, can be seen in Fig. 4.3. The rear wake seems to be small but its shape is well defined, curving downwards due to a higher downwash in comparison to the upwash coming from the floor of the vehicle. This indicates lift on the rear axle, but a low drag coefficient due to the well-defined wake, although the flow detaches at the roof. An indication of the model's flat floor is how smooth the airflow is under it, despite a slight separation at the front overhang, likely due to a small radius at the front bumper.

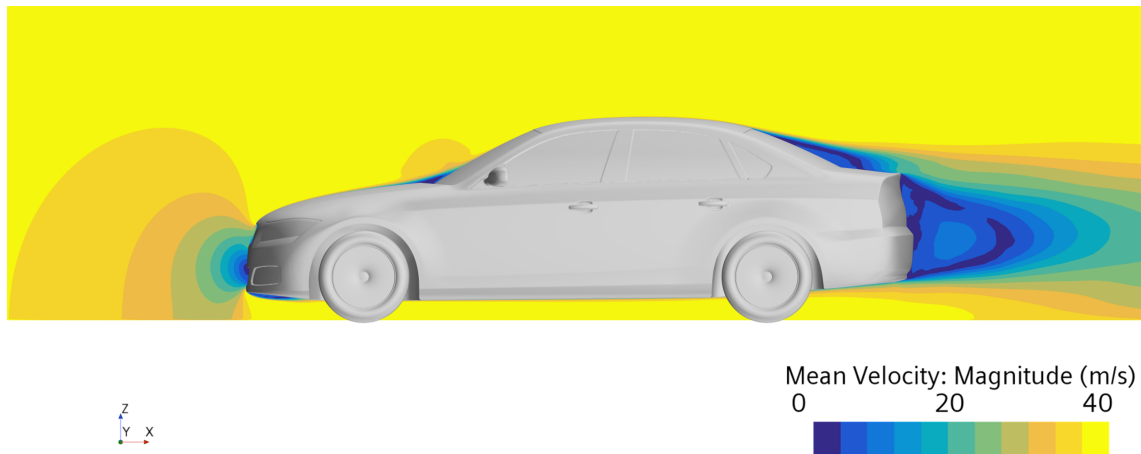


Figure 4.3: Scalar scene of the low turbulence simulation showing the mean velocity magnitude at the center plane of the DrivAer notchback, side-view

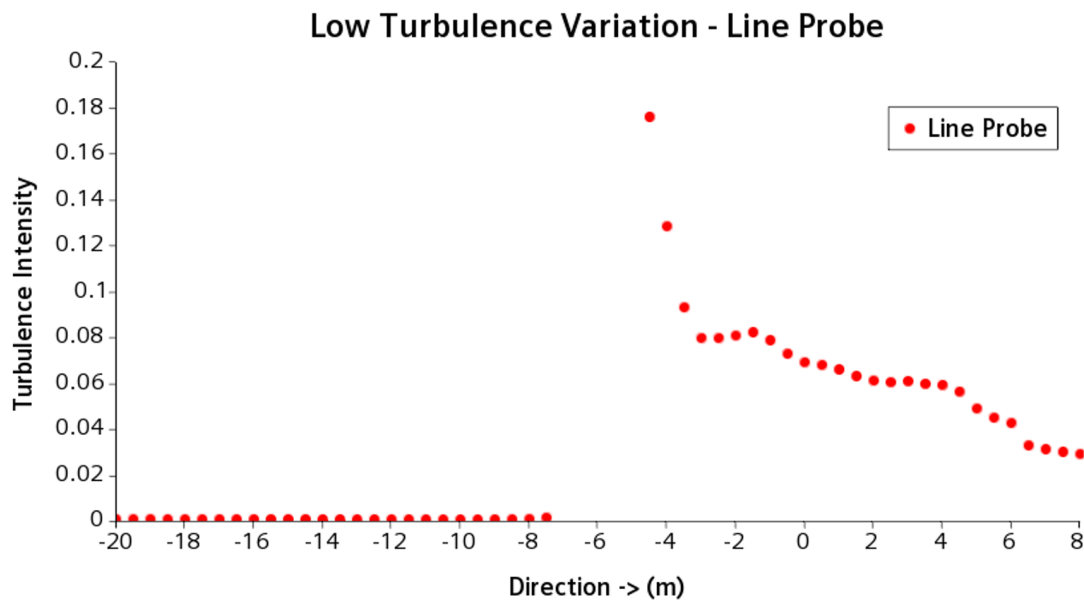


Figure 4.4: The turbulence variation across the DrivAer domain along a line probe

The turbulence varies across the domain and to determine the different intensities around it, a line probe is placed at a height of 1 meter, along the x-axis, in line with the centre of the car. This also helps to gauge the turbulence intensities at different distances that the following car would undergo. Fig. 4.4 shows the varying TI levels for the low turbulence simulation. From the inlet to the vehicle it can be noticed that a near to 0% turbulence is maintained as the intensity is specified to be 0.1%. With the vehicle model being placed between -8.8m to -4m, the initial 3 probes show low intensities over the hood of the car. The probe point showing a high 18% TI is

4. Results

due to its placement being the closest point to the wake. At distances of up to 10m behind this vehicle it can be seen that the intensities are between the range of 4-8% which is similar to what had been assumed during the simulations conducted by A. Gaylard et al. [7]. It can also be seen that the turbulence intensity drops off slowly as the wake is dispersed and the flow stabilises.

4.1.2 High turbulence simulation

For the high turbulence scenario, TI is set to 8% and the length scale to 2 meters. In Fig. 4.5, the change in turbulence intensity is visible within the entire domain. Also, noticeable areas of increased turbulence are in front of the bonnet, the cowl area and the start of the roof. These were not seen in the low turbulence simulation, like in Fig. 4.2.

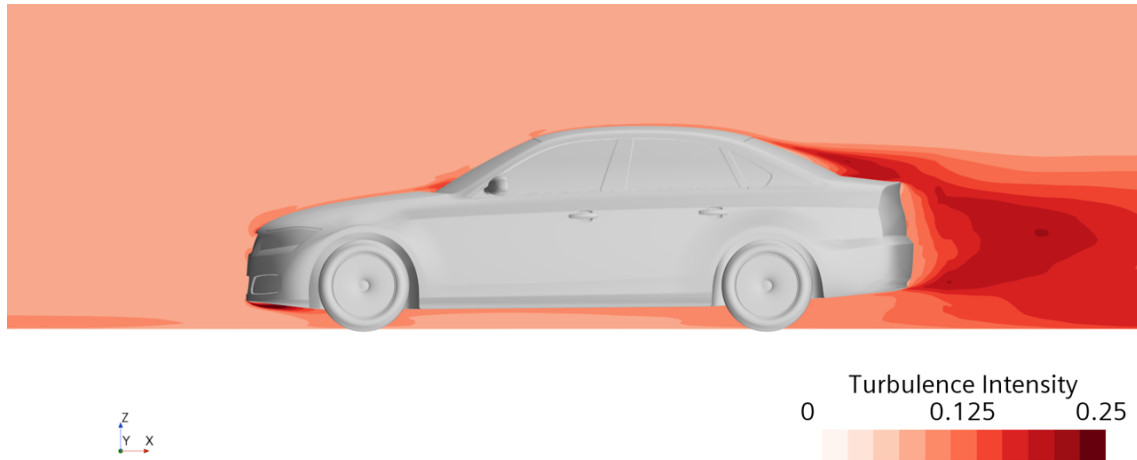


Figure 4.5: A mid-sectional scalar image of the high turbulence simulation on the DrivAer notchback represents the turbulence intensity at different regions, side-view

The velocity magnitude around the DrivAer model under high turbulence seen in Fig. 4.6 shows some key differences that can be noticed in the wake region. Here, the wake appears to be smaller and better balanced in comparison to the low turbulence simulation. The velocity within the wake also appears to have higher percentages of quicker flow within this small region which helps with a more balanced recovery. But this smaller region may not be sufficient to decrease the drag count or increase pressure as a larger region of backflow would be better as seen in the previous case.

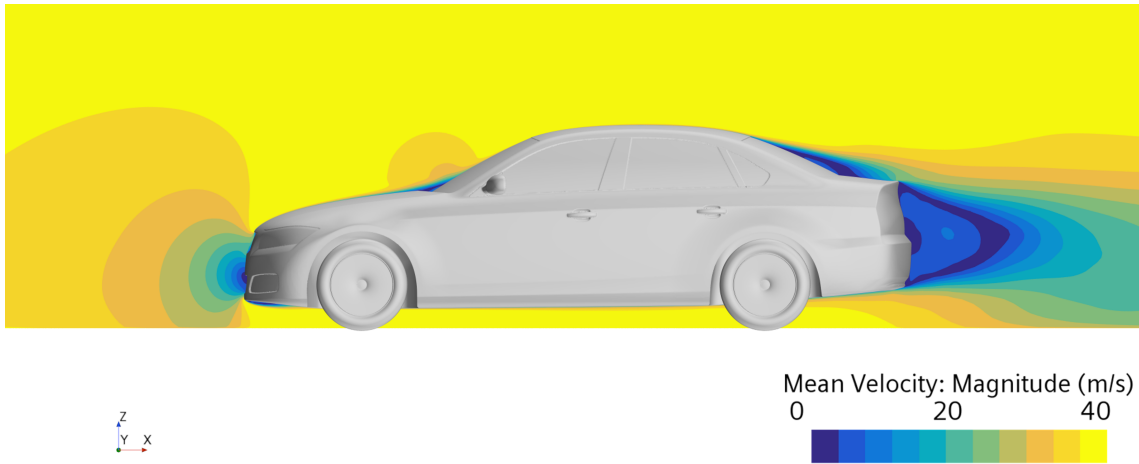


Figure 4.6: Scalar scene of the high turbulence simulation showing the mean velocity magnitude of the DrivAer notchback, side-view

The pressure recovery is visible at the rear of the DrivAer notchback model in Fig. 4.7. The mean coefficient of pressure is defined on the rear surface of the model. The higher pressure recovery is concise to the middle region in both cases, however, the higher mean coefficient of pressure can be noticed in the lower turbulence simulation which is caused due to the larger wake which consists of higher reversed velocity thereby increasing the pressure at the base. The opposite of this occurs due to the smaller wake in the high turbulence case and thereby has a lower pressure recovery and worsens the drag coefficient. Due to a visibly more balanced wake in the high turbulence case, the pressure recovery appears to be more symmetric as seen in the mean coefficient of pressure plots. The high turbulence case also shows comparatively higher pressures on the area between the rear windshield and base which perhaps contributes to the generation of the 16 counts of downforce between the two cases at the rear(C_{lr}).

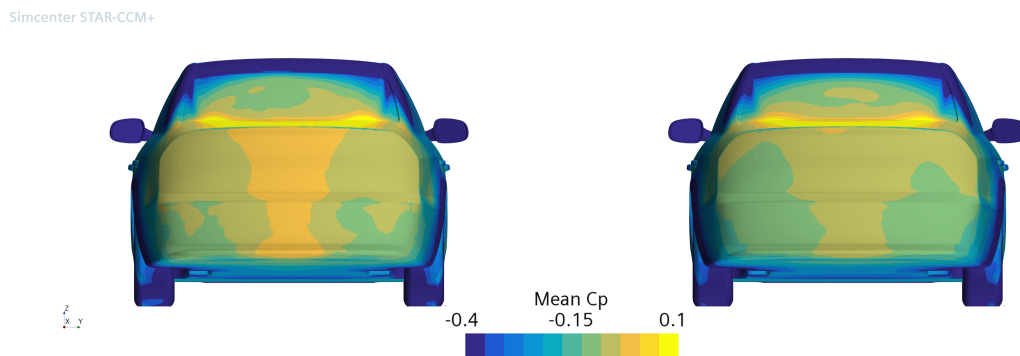


Figure 4.7: The mean coefficient pressure on the rear end of the DrivAer between the low (left) and high (right) turbulence models respectively is shown, rear-view

4. Results

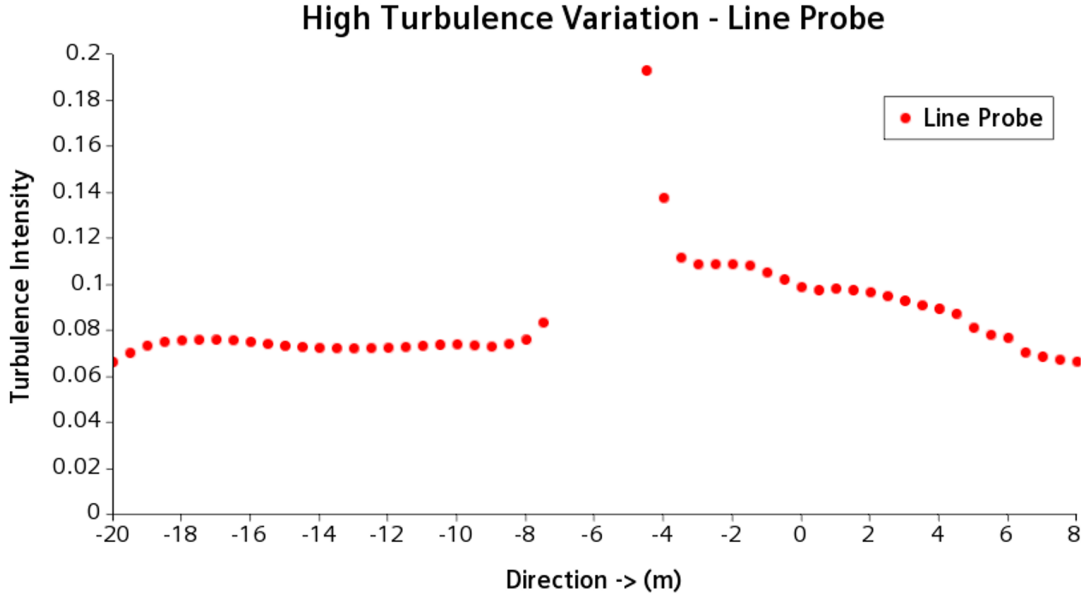


Figure 4.8: The turbulence variation across the domain by a line probe

Fig. 4.8 shows the varying TI levels for the high turbulence simulation. From the inlet to the vehicle it can be noticed that the synthetic turbulence intensity specified to be 8% is nearly maintained. Disregarding the points within the car as previously stated, the values behind the car now in the presence of high turbulence, show a higher range of 7-11% TI levels which will be further faced in the case of the following vehicle.

A summation of the DrivAer results has been tabulated in Table 4.1 showing the increase in drag coefficient from low to high turbulence cases and in the lift. A loss of downforce is noticed on the front end of the sedan notchback model while the rear lift decreases.

	ΔC_d	ΔC_l	ΔC_{lf}	ΔC_{lr}
DrivAer	0.009	0.003	0.019	-0.016

Table 4.1: Tabulation of delta results from high to low turbulence scenarios for the DrivAer simulations

4.2 Volvo XC40 Recharge model

In this section, the Volvo XC40 Recharge simulation results are presented with more focus on the high turbulence simulations and the comparison between them. Although the computing resources required are high, a full car simulation was necessary to run both the low and high turbulence intensity cases for comparison reasons. For further comparison and to bridge the gap between on-road tests and simulation,

data from the coast down and wind tunnel tests are correlated to the results obtained in CFD simulations. An example image of the streamlines around the Volvo XC40 is shown in Fig. 4.9.

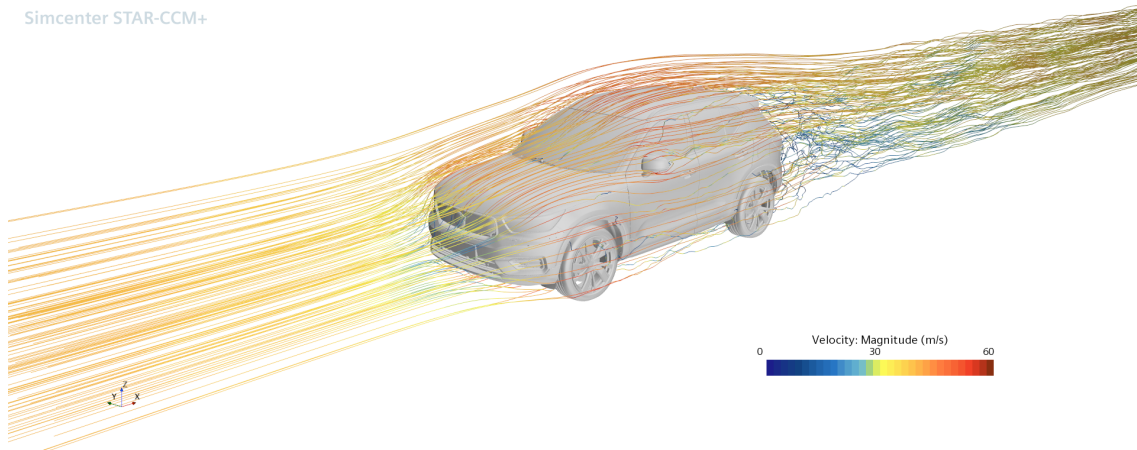


Figure 4.9: An example image of a low turbulence simulation on the XC40 showing streamlines around the test object

4.2.1 Low turbulence simulation

The low turbulence simulation is run with no synthetic turbulence introduced from the inlet. However, as the specification of the IDDES solver is defined, the turbulence intensity parameter needs to be specified, which is accordingly set to 0.1%.

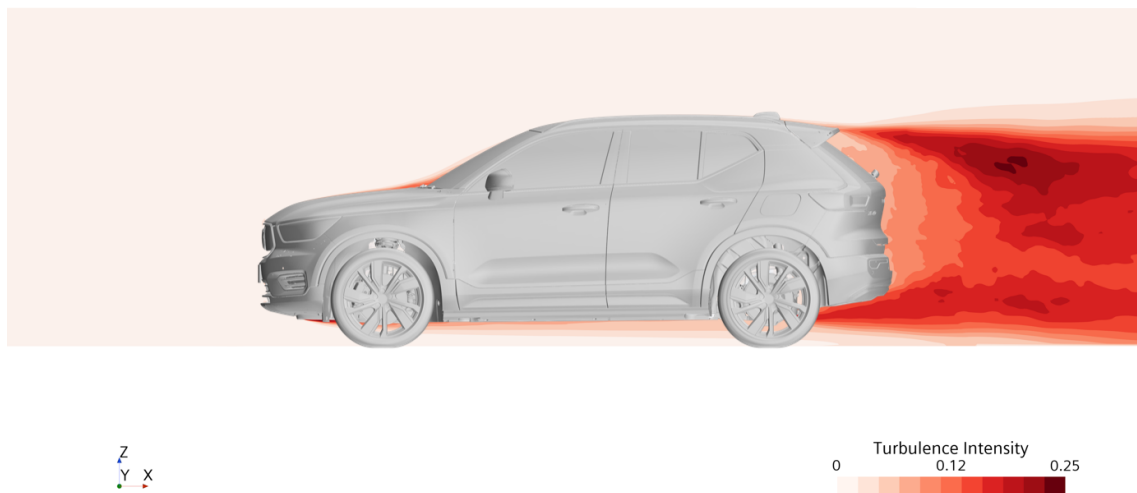


Figure 4.10: A mid-sectional scalar image of the low turbulence simulation on the XC40 represents the turbulence intensity at different regions, side-view

On the scalar scene shown in Fig. 4.10, the varying turbulence intensity levels at different regions are shown, where fluctuations at places of flow discontinuity and

4. Results

differences in pressure. As expected, there is a wide variation in turbulence intensity in the wake region. This is due to fluctuations in fluid flow velocities in this region of low pressure concerning its surroundings. The scene also shows a clear near to 0% TI level ahead of the vehicle as a result of no synthetic turbulence being implemented.

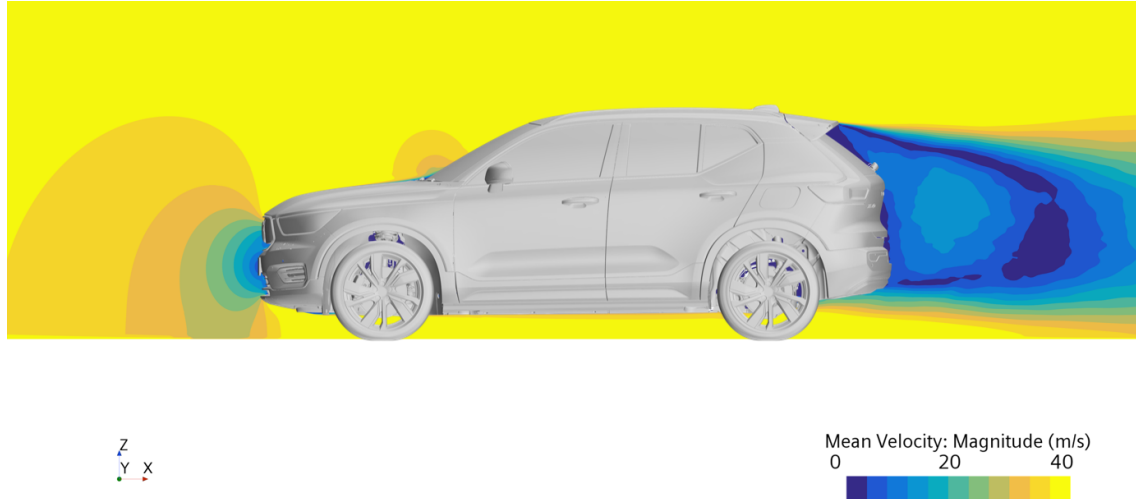


Figure 4.11: Scalar scene of the low turbulence simulation showing the mean velocity magnitude of the XC40, side-view

The mean velocity scalar scene is shown in Fig. 4.11, where some specific zones like the stagnation zone and the rear wake can be noted for comparison to the turbulent simulations later on. On the low turbulence run, the rear wake of the vehicle is seen to be balanced, although the downwash from the rear hatch spoiler is slightly more than the upwash coming from the floor of the car. The spoiler shows to be effective in decreasing the downwash, delaying flow separation from the roof and helping with pressure recovery. The stagnation point, the highest pressure point, where there is zero relative velocity, is well pronounced and around the front number plate.

For the drag coefficient area of the vehicle, $C_d A$, the two main areas of interest are the front of the car and the rear. Up front, the stagnation area is responsible for a substantial spike in drag seen in Fig. 4.12, caused by the high resistance force against the car. As the flow passes over the hood, the drag slightly decreases due to flow reattachment to the body. The $C_d A$ trend next rises due to the increased drag forces caused by the wheels and decreases over the car as the flow goes over the roof before reaching the rear wheels. An upward trend is noticed at the rear wheel and due to the SUV body. The hatchback's rear-end causes higher drag as it doesn't recover pressure well, despite the rear spoiler which is implemented to reduce drag by delaying flow separation and balancing the wake.

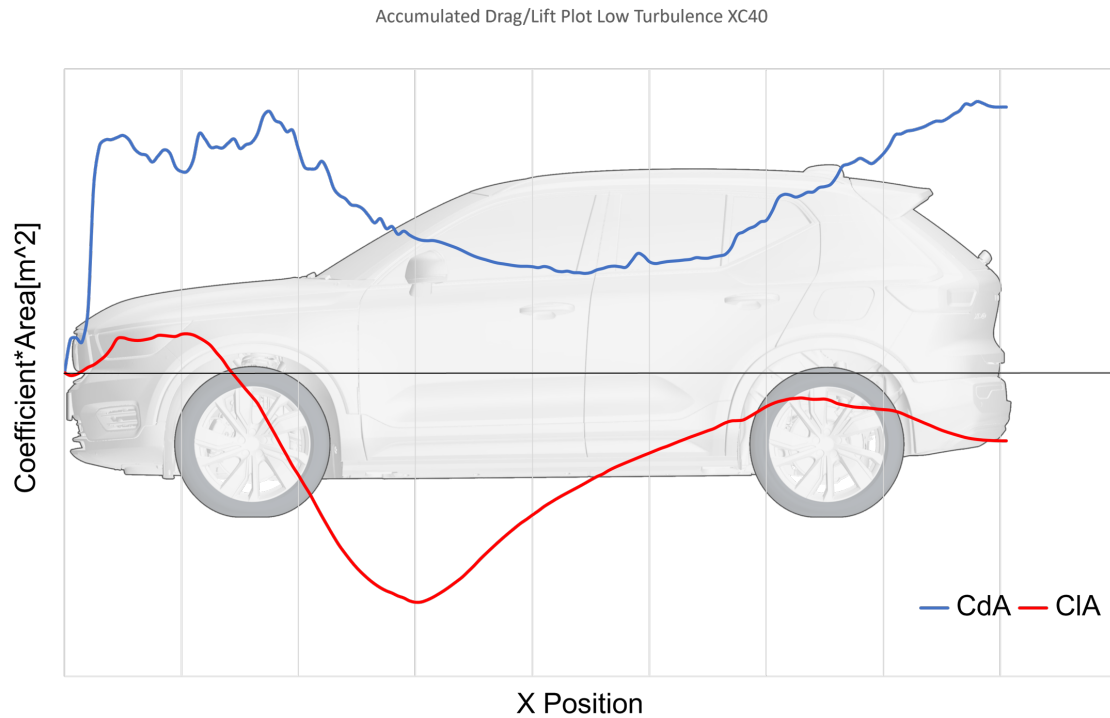


Figure 4.12: Accumulated drag and lift coefficients of the XC40 low turbulence simulation

For the vehicle's lift, $C_l A$, the first section of the car starts by generating lift, but as the flow re-attaches to the vehicle's bonnet and passes the cowl area, the curve turns down as the vehicle starts generating downforce. As the flow accelerates over the roof of the car, a low-pressure zone is formed, pushing the car upward, thereby decreasing downforce. The rear spoiler of the car helps control the down-wash, making the flow separate backwards. This helps the car to generate overall downforce.

4.2.2 High turbulence simulation

The high turbulence simulation is run with the introduction of synthetic turbulence in the inlet of the open road domain. From the literature survey, suggestions by Gaylard et al. [7] in the event of following a car and from Wordley and Saunders [1, 2], to match the PSD curves, a common stance was made to choose 8% turbulence intensity with a 2m length scale. As for the characteristic length scale, the car's length was avoided as it was unrealistically large and had a great effect on the flow over the vehicle, which would cause inconsistency in the final results. However, the 2m length scale is chosen as it is quite close to the height or width, or even the square root of the frontal area of the vehicle which can all be considered as the characteristic length scales. The high turbulence intensity is also chosen to give a clear depiction of which region or part shows the biggest differences in comparison to the low turbulence simulations.

From Fig. 4.13, the high turbulence along with its varying length scales can be

4. Results

noticed in the domain. An increased effect of TI levels can also be observed in the wake region due to the unsteady environment.

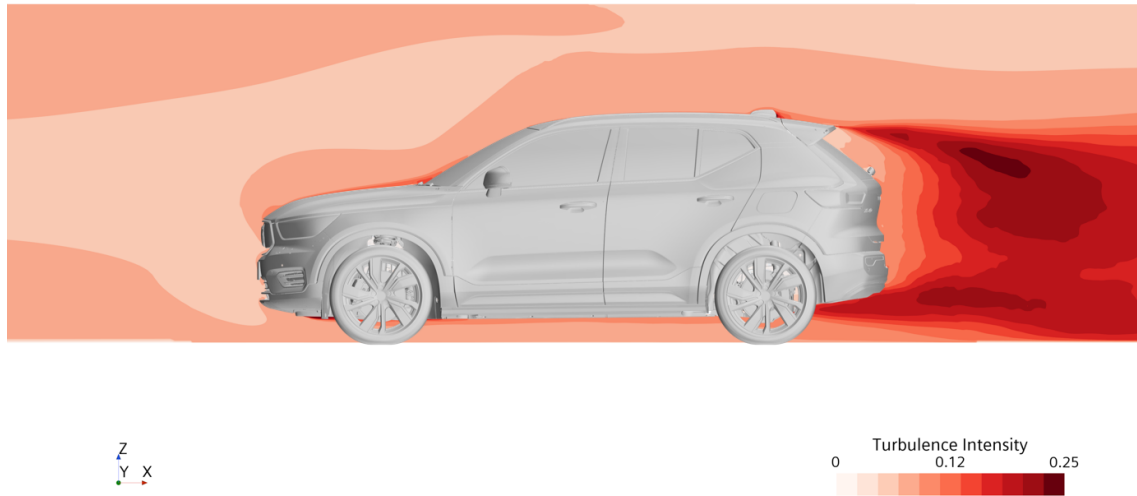


Figure 4.13: A mid-sectional scalar image of the high turbulence simulation on the XC40 representing the turbulence intensity at different regions, side-view



Figure 4.14: Scalar scene of the high turbulence simulation showing the mean velocity magnitude of the XC40, side-view

Due to increased flow fluctuations, it can be seen that the wake of the vehicle has changed in Fig. 4.14. It appears that the size of the wake is smaller in the higher turbulence simulation. This could be due to the lesser noise caused by the longer averaging time of 5 seconds.

The stagnation region in front of the car can also be compared to the low turbulence model as it shows a decrease in size for the very same reason that the randomness

in turbulent flow gives rise to increased pressure losses.

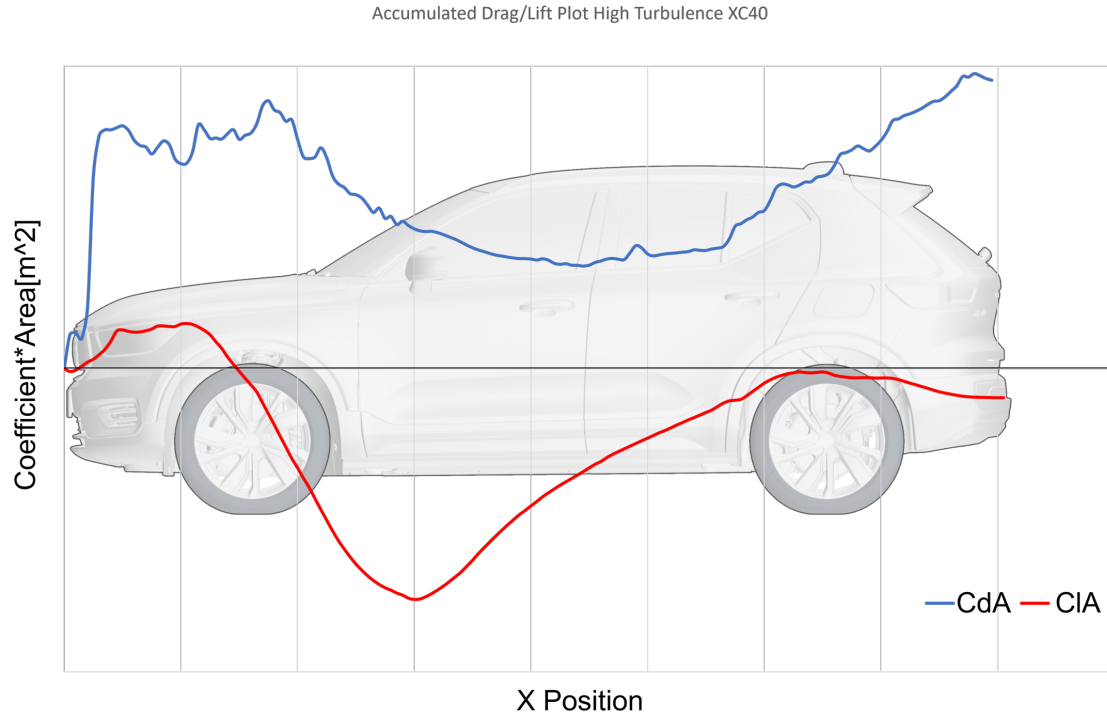


Figure 4.15: An accumulated drag and lift curves of the high turbulence simulation on the XC40 is shown

The trends from Fig. 4.12 and Fig. 4.15 are similar for both C_dA and C_lA . The main changes are evident at the rear of the vehicle, as drag is higher and downforce is lower. Comparisons between the delta drag and lift coefficients are discussed further in the next section.

4.2.3 Comparisons between low and high turbulence models

This section gives a deeper comparison between the low and high turbulence models. By focusing more on what is happening in the delta accumulated drag/lift curves, the behaviour of the wake from different views and how that affects the recovery of pressure at the rear of the vehicle. The iso-surfaces are also compared to find the various regions where the pressure of 0.0 Pa is present.

From Fig. 4.16, it is visible how the implementation of higher turbulence modifies the trends of both drag and lift coefficients. At first, for drag, the variability of the flow helps in reducing the stagnation zone in the front of the car, as seen in Fig. 4.14 in comparison to Fig. 4.11, with the front of the car having a trend of lower drag. The trend completely switches towards the rear of the vehicle, with a significant increase in drag.

4. Results

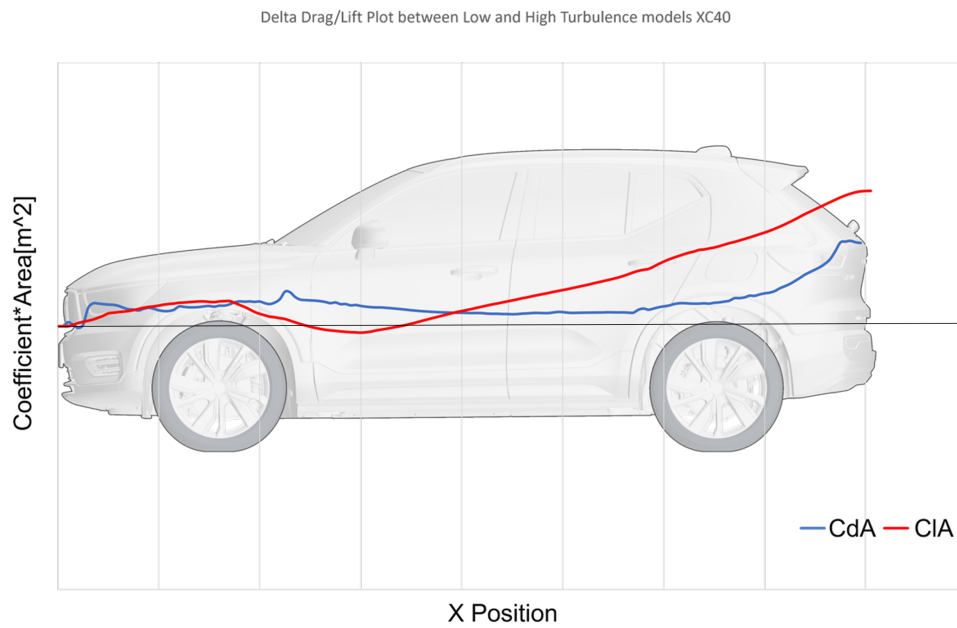


Figure 4.16: The delta accumulated drag and lift curves between the two simulations are represented

According to other studies [7], the increase in drag could be caused by the fact that under turbulent flow, the front wheel deflectors lose their effect and aren't effectively shielding the rear wheels from the flow. Also, there are changes in the wake structures, with a less balanced wake due to the increased down-wash on the high turbulence case. The higher down-wash is noticeable as the rear lift is increased, with the front lift also increasing, but to a smaller degree. Higher lift is preferred on the front axle instead of the rear, increasing stability as the car becomes less "nervous" to steering inputs due to lesser loads. The opposite is preferred for the rear axle, as it needs to support side loads well to resist rotational moments, requiring higher loads and increased downforce.

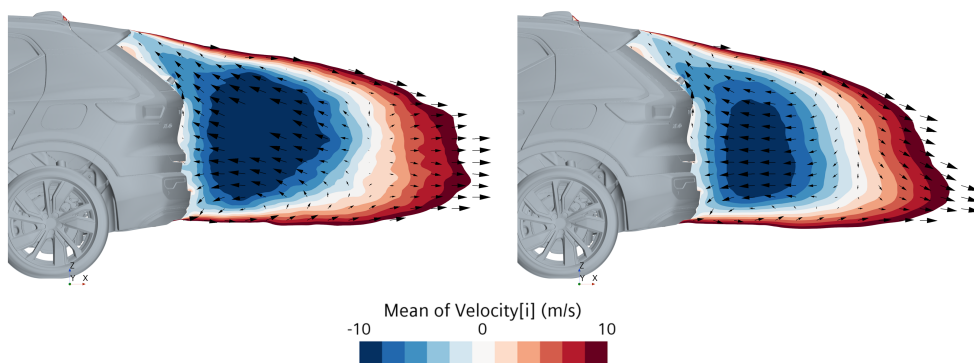


Figure 4.17: The vector plots show the mean velocity on the wake end of the car between the low (left) and high (right) turbulence models respectively, side-view

A clear comparison between the wakes can be seen in Fig 4.17. It can also be seen

that there is a larger region where a higher magnitude of velocity is formed in the low turbulence situation. This larger region should result in better pressure recovery. Although the wake is smaller in the higher turbulence image, it shows a smaller region of high negative velocity resulting in higher drag. The velocity arrows in the blue zone pointing towards the base help with base pressure recovery, but since they are lesser in quantity, the drag is increased in the high turbulence case. The wake, however, obtains a similar angle to the slanting roof in higher turbulence which is the primary purpose of the roof shape during design, while in low turbulence the wake tends to become more parallel to the X-axis.

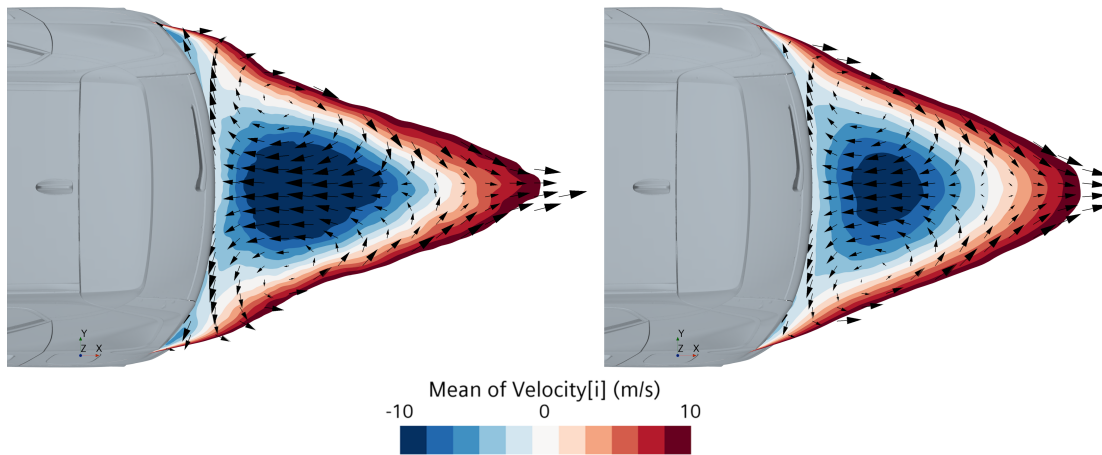


Figure 4.18: The vector plots show the mean velocity on the wake end of the car between the low (left) and high (right) turbulence models respectively, top-view

From the high turbulence scene in Fig. 4.18, the top view of the rear wake shows how the wake seems to be more homogeneously balanced. In the low turbulence case, the wake is longer, and the vortices from its side seem to aid with more reversed airflow, causing it to be compressed. The reverse airflow means there is more negative velocity towards the rear end of the vehicle, aiding with base pressure recovery.

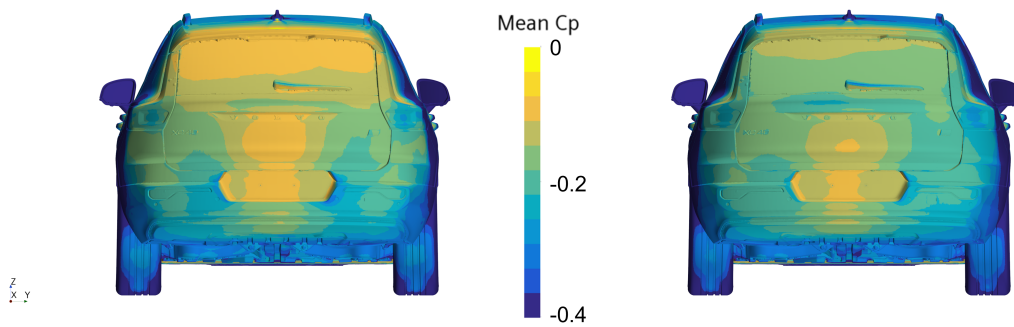


Figure 4.19: The mean pressure coefficient on the rear end of the car between the low (left) and high (right) turbulence models respectively is shown, rear-view

4. Results

From Fig. 4.19, the changes in the pressure coefficient are visible in the rear end of the Volvo XC40. Although it seems like the flow has not changed much underneath the vehicle, the large drop in counts of downforce can be due to the poor recovery of pressure at the roof of the vehicle as seen in the high turbulence case. Following the above Fig. 4.17 and Fig. 4.18, the low turbulence case has visibly more base pressure as specified earlier, and in the high turbulence case, there is a big base pressure deficit due to the vehicle's modified wake. This means that some of the solutions to reduce drag in low turbulence conditions are not as effective under on-road turbulence.

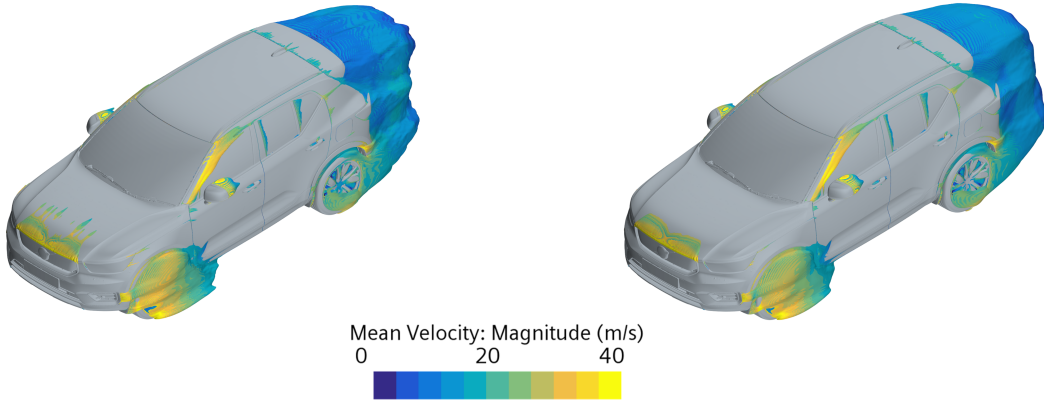


Figure 4.20: The iso-surfaces of the car between the low (left) and high (right) turbulence models respectively

The total pressure coefficient iso-surfaces show regions where lesser energy in the flow is present. From Fig. 4.20 the major differences between the two simulations are noticed in the wake. The high turbulence causes more mixing due to the increased fluctuations of velocity in the V and W directions, which transports high energy flow to the wheels and the wake. This results in larger gradients and hence higher losses. Thereby, showing that the smaller wakes very much relate to higher losses as opposed to when the shape becomes slender and the wake size reduces in the same flow conditions.

A summation of the standard differences in coefficients of drag and lift is shown in Table 4.2.

	ΔC_d	ΔC_l	ΔC_{lf}	ΔC_{lr}
Volvo XC40	0.028	0.049	0.015	0.034

Table 4.2: Tabulation of delta results from low to high turbulence scenarios for the Volvo XC40 simulations

In Table 4.2, we can see how drag and lift coefficients have increased for the high turbulence simulation. Similar to what has been found in [7], there has been a significant increase in rear lift, and front lift also increased, but to a smaller degree.

This is an indication of imbalance for the lift coefficients, as the rear lift increase by over double the increase of the front lift. The averaged coefficient of drag showed an increase of $\Delta C_d = 0.028$ (28 counts) and an increase in lift of $\Delta C_l = 0.049$ (49 counts) in comparison to the low turbulence simulation. The increased lift is also noticed to be higher on the rear axle in comparison to the front, with $\Delta C_{lf} = 0.015$ (15 counts) and $\Delta C_{lr} = 0.034$ (34 counts). This shows how large the deviation can be when comparing high to low turbulence simulations. These values represent a more realistic overview of the vehicle, as traditional methods for aerodynamics testing don't reproduce on-road airflow.

4.2.4 Coast-down and wind-tunnel test analysis

To bring the results in the previous section closer to on-road studies, we compare them to coast-down and wind tunnel tests performed on the Volvo XC40 model. This gives a platform to compare the deviation between a single similar vehicle model between the 3 stages, as coast-down tests are a step towards the on-road conditions. To estimate the drag coefficient, a proprietary method is used to extract the drag-coefficient area ($C_d A$) from the weight and weather compensated data. It consists of a polynomial equation with coefficients that represent conversion factors which have been obtained by matching coast-down and wind-tunnel data.

		Wind-tunnel estimation (ΔC_d)	Coast-down estimation (ΔC_d)
Simulation	Low TI	0.027	0.024
	High TI	-0.001	-0.004

Table 4.3: Differences in C_d values between the simulation results from the wind tunnel and coast-down respectively

When comparing both low and high turbulence CFD simulations to wind tunnel results, the wind tunnel's estimated value from calculations shows 1 count lesser than what was found in the averaged C_d of high turbulence simulation. On the other hand, when compared to the low turbulence simulation, a large difference of 27 counts was found, resulting in a significant difference. This difference is unexpected as the turbulence intensities in a wind tunnel are generally very low, and the ΔC_d between low TI simulation and wind tunnel should be closer than the ΔC_d between high TI simulations and wind tunnel. The reason is that the extraction polynomial has been fitted to match C_d seen from coast-down data to the wind tunnel, causing an overestimation.

Similar to what was obtained for the wind tunnel tests, the high turbulence simulation obtained an averaged C_d that is remarkably close to the coast-down test values.

4. Results

In this case, the coast-down test was 4 counts lesser than the high turbulence simulation, meaning the CFD simulation here tends to over-predict the C_d value slightly. As for the low turbulence, the difference in C_d value is 24 counts and although it is a smaller difference with the comparison between simulation and wind tunnel, it is still a significant number to neglect. It shows that the low turbulence simulation here tends to under-predict the real C_d value that a vehicle experiences on-road.

This under-prediction of C_d from the low turbulence simulations is a clear depiction showing the importance of not entirely expecting the small drag count improvements from simulations to occur in reality. It is also important to remember that turbulence causes a lack of repeatability, meaning, should the coast-down test be repeated, the new estimations could result in slight differences. The δC_d could get smaller or larger based on the conditions of when the coast down tests is conducted. This is why coast-down tests are generally conducted where there is minimal variation in the environment considering air temperature, density and crosswind. It is also highly recommended to match the specifications of the vehicle including its rims and tyres to conduct such analysis.

Repeated comparisons between coast-down tests and simulations can help determine the standard deviation seen in the drag coefficient, to assess the reliability of conducting these high turbulence simulations. If the synthetic turbulence parameters are accordingly adjusted in simulations, based on the +/- differences of drag coefficients, an estimate can be made as to how much turbulence on an average the coast-down tests undergo. These can perhaps provide a better TI and length scale than the ones used in the scope of the thesis for future standardised high turbulence simulations.

5

Conclusion

The scope of the thesis project was based on the importance of considering the effects of real-life flow conditions and the impact they incur on drag and other vehicle parameters. As aerodynamics in recent years has gained significant relevance to the role it plays in energy consumption, the study focuses on bridging the gap caused between the standardized low turbulence simulations conducted by vehicle manufacturers to the results seen on-road. This is done to show the importance of introducing turbulence in the simulations to predict the vehicle's aerodynamic characteristics in reality. A significant amount of time and computational power is spent to achieve the lowest drag coefficient for a vehicle in low turbulence tests, while these improvements can not always be expected in real-life conditions.

The thesis takes a CFD approach as it is generally the first stage of development and testing in the product cycle. By introducing synthetic turbulence generated in STAR-CCM+ an iterative approach is made to analyze the effects that turbulence have on the test objects. Upon investigating the differences between low and high turbulence simulations, it can be noticed that using higher turbulence intensities is more representative of on-road flow conditions. As higher turbulence disturbs the flow structures around the vehicle, they give a deeper insight into what the vehicles undergo during gusty, crosswind and in cases of driving in the wake of another car.

The main complication with high turbulence simulations is their natural variability and unsteadiness, which can cause an issue when it comes to repeatability in results. This makes standard methods of low turbulence work better in terms of absolute values for drag and lift coefficients, which are useful for standard comparisons of vehicles. In the case of actual fuel economy predictions, add-on parts for improving performance can use high turbulence methods to analyze their actual effect. This would help manufacturers improve their cars right from the CFD stage, rather than having to fine-tune their specifications from on-road tests.

For future studies, it can also be recommended that longer averaging periods are required for analyzing high turbulence simulations based on the complexity of the test object, a good time would be between 3 to 5 seconds. An in-depth analysis of the convergence of different results can better help in predicting averaging periods. Subsequently, in the area of on-road aerodynamics, more on-road data especially those consisting of velocity fluctuations in different directions should be collected and directly compared with CFD simulations. The velocity fluctuation data can directly be fed into the STAR-CCM+ through the means of a table, to mimic the

5. Conclusion

inlet conditions exactly as recorded in reality. This would allow for the development of a workflow that includes high turbulence CFD simulations early on in the development of new vehicles, thereby saving time and money in the process benefiting car companies and the final customers with vehicles that are more efficient and safer to drive on the road.

Bibliography

- [1] Wordley, S. and Saunders, J., "On-road Turbulence," SAE Int. J. Passeng. Cars - Mech. Syst. 1(1):341–360, 2009, doi:10.4271/2008-01-0475.
- [2] Wordley, S. and Saunders, J., "On-road Turbulence: Part 2," SAE Int. J. Passeng. Cars - Mech. Syst. 2(1):111–137, 2009, doi:10.4271/2009-01-0002.
- [3] Dalessio, L., Duncan, B., Chang, C., Gargoloff, J. et al., "Accurate Fuel Economy Prediction via a Realistic Wind Averaged Drag Coefficient," SAE Int. J. Passeng. Cars - Mech. Syst. 10(1):2017, doi:10.4271/2017-01-1535.
- [4] M .S Gritskevich et al., 'Development of DDES and IDDES Formulations for the $k-\omega$ Shear Stress Transport Model'. In: *Flow, Turbulence Combust* (2012).
- [5] T. Avraham., *Understanding The Detached Eddy Simulation From DES to IDDES*, URL: <https://cfdisrael.blog/2020/06/23/understanding-the-detached-eddy-simulation-from-des-to-iddes/> (visited on 2022-05-24).
- [6] Menter, F.R. 1994. "Two-equation eddy-viscosity turbulence modeling for engineering applications", AIAA Journal, 32(8), pp. 1598-1605.
- [7] Gaylard, A., Oettle, N., Gargoloff, J., and Duncan, B., "Evaluation of Non-Uniform Upstream Flow Effects on Vehicle Aerodynamics," SAE Int. J. Passeng. Cars - Mech. Syst. 7(2):2014, doi:10.4271/2014-01-0614.
- [8] Alajbegovic, A., Gaylard, A., Gargoloff, J., and Duncan, B., "Vehicle Aerodynamic Effects of Realistic Transient Wind Conditions" 11th World Congress on Computational Mechanics (WCCMXI), 5th European Conference on Computational Mechanics (ECCM V), 6th European Conference on Computational Fluid Dynamics (ECFD VI), July 20 -25, 2014, Barcelona, Spain.
- [9] Siemens. *STAR-CCM+ User Guide*. Version: 2021.2
- [10] T. C. Schuetz. *Aerodynamics of Road Vehicles*, Fifth Edition. English. 5th edition. Warrendale, Pennsylvania: SAE International, Dec. 2015.isbn: 978-0-7680-7977-7.
- [11] Sebben, S. Introduction to RVAD, Chalmers University of Technology, Gothenburg.
- [12] Heft, A., Indinger, T., and Adams, N., "Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations," SAE Technical Paper 2012-01-0168, 2012, URL: <https://doi.org/10.4271/2012-01-0168> (visited on 2022-05-24).

Bibliography

A

Appendix

A.1 Power spectral density (PSD)

Comparing and validating results to previous on-road tests can be a very convoluted process as the turbulence intensities are constantly varying. Assuming the turbulence intensity to be a random signal, and with the help of a probe to measure the turbulence intensity value, a power spectral density can be used to measure the power content of the recorded signal against its frequency at any given stationary point, where the energy content of the respective signal is characterized.

The PSD function normalizes the magnitude spectrum based on the resolution of the frequency. The number of samples recorded determines the number of data points of the PSD. As the recorded data points are lower the amplitude will appear to be higher. The RMS sum of the spectrum is depicted as the total value of the signal and this can be used to compare if two or more signals are similar.

$$PSD = \frac{G_{xx}}{\Delta f} \quad (\text{A.1})$$

where G_{xx} is the power spectrum which is the squared magnitude of the frequency spectrum.

This way a PSD shows that a random signal can be laid out independent of the resolution used in any other experiments/tests, from which the Root-Mean-Squared(RMS) value can be determined as,

$$RMS = \sqrt{\int_{f1}^{f2} PSD(f) \cdot df} \quad (\text{A.2})$$

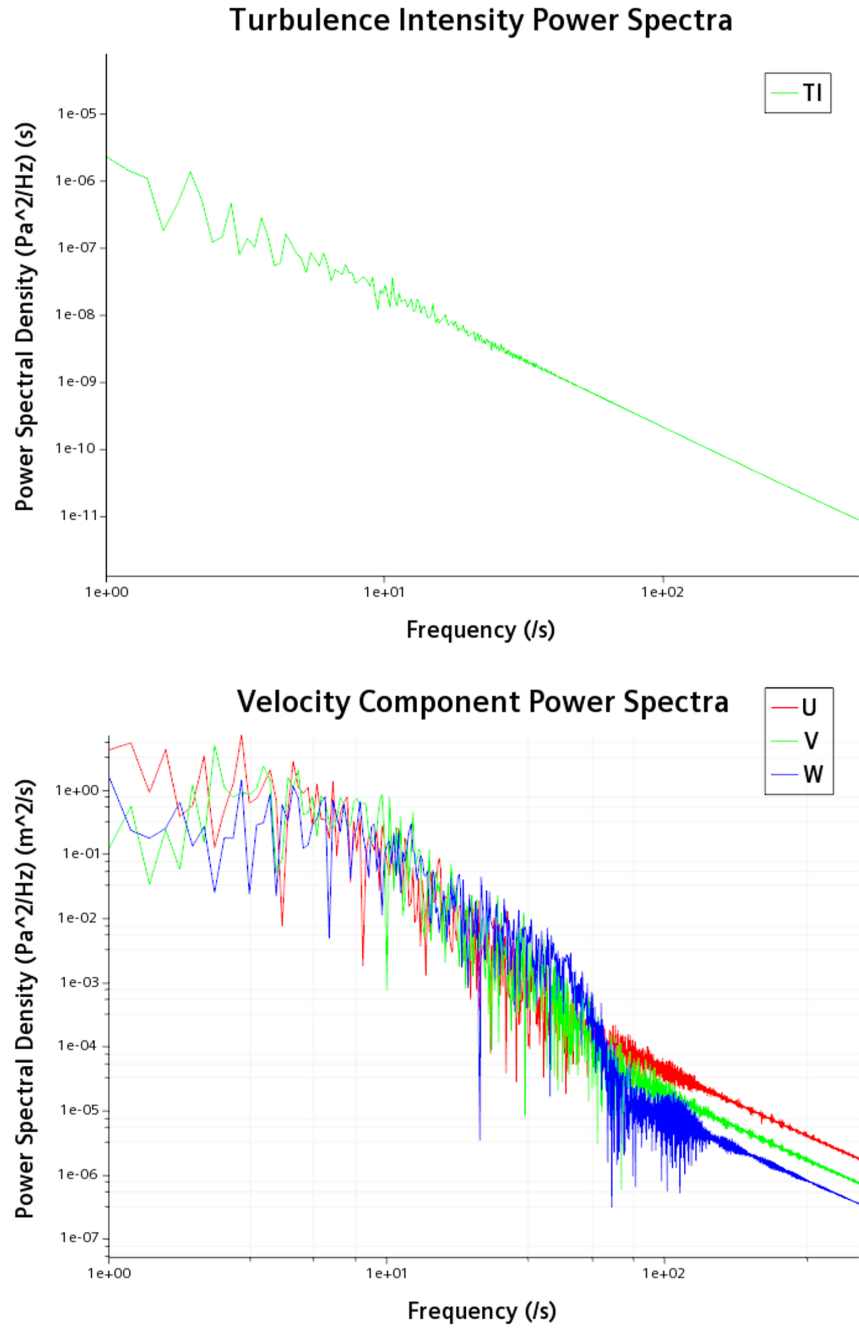


Figure A.1: The dimensional power spectra for turbulence intensity 8% of length scale 2.0m(top) and dimensional wind component(U, V and W) power spectra(bottom)

Fig. A.1 shows the power spectral density curves for the turbulence intensity and the velocity components in each of the three directions. The data has been recorded from the point probe used in the simulations. To further compare the trend and amplitudes of the power spectra, it's crucial to have similar recording frequencies. The data points shown in Fig. A.1 have been recorded at a frequency of 1 for every time step, meaning 4000Hz.

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden
www.chalmers.se



CHALMERS
UNIVERSITY OF TECHNOLOGY