



Modelling of door cavities and simulations to assess flow properties in door cavities for dirt/dust applications

Master's thesis in Automotive Engineering

SAGAR MAHESH KUKREJA

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES DIVISION OF VEHICLE ENGINEERING AND AUTONOMOUS SYSTEMS

CHALMERS UNIVERSITY OF TECHNOLOGY Gothenburg, Sweden 2020 www.chalmers.se

MASTER'S THESIS IN AUTOMOTIVE ENGINEERING

Modelling of door cavities and simulations to assess flow properties in door cavities for dirt/dust applications

SAGAR MAHESH KUKREJA

Department of Mechanics and Maritime Sciences Division of Vehicle Engineering and Autonomous Systems CHALMERS UNIVERSITY OF TECHNOLOGY

Göteborg, Sweden 2021

Modelling of door cavities and simulations to assess flow properties in door cavities for dirt/dust applications SAGAR MAHESH KUKREJA

© SAGAR MAHESH KUKREJA, 2021

Master's thesis 2021:02 Department of Mechanics and Maritime Sciences Division of Vehicle Engineering and Autonomous Systems Chalmers University of Technology SE-412 96 Göteborg Sweden Telephone: +46 (0)31-772 1000

Cover: Isometric view of all the raw CAD models imported in ANSA, to prepare the geometry.

Chalmers Reproservice Göteborg, Sweden 2021 Modelling of door cavities and simulations to assess flow properties in door cavities for dirt/dust applications Master's thesis in Automotive Engineering SAGAR MAHESH KUKREJA Department of Mechanics and Maritime Sciences Division of Vehicle Engineering and Autonomous Systems Chalmers University of Technology

Abstract

In the automotive industry, where lead times are long and prototypes expensive, simulation driven designs play a vital role to reduce the overall cost and development time. Various tools utilizing Computational Fluid Dynamics (CFD) are used to e.g. evaluate the aerodynamic performance of vehicles. Firstly, the automotive industry is in the midst of a switch from combustion to electrified vehicles, hence it is essential to improve vehicle efficiency in order to increase the range of the battery. Secondly, modern vehicles are equipped with multiple cameras and sensors connected to driving aid and safety systems. For these systems to function well, the regions where cameras and sensors are mounted cannot be contaminated by e.g. snow or dust. Finally, preventing dirt/dust intrusion to the door-cavities of a car is desirable, as dirt/dust present in this region will likely cause soiling of the occupants when exiting/entering the vehicle.

The Contamination and Core CFD department is responsible for the experimental and virtual assessment of contamination performance at Volvo Car. In prior tests the in-cavity air flow at the tailgate cavities have been evaluated. However, the cavities at the doors were not analysed for dirt/dust intrusions. The objectives of this thesis is to use previous knowledge and data on how to predict flow fields in cavities, and from that formulate a multiphase approach to also include dust/dirt distribution in the simulations by means of experiments and CFD. The vehicle geometry used for experimentation and CFD analysis is a Volvo V90 D4 AWD. The geometry to capture the cavities was prepared using ANSA and the following simulations were conducted in StarCCM+. The vehicle was driven on gravel roads to analyse, compare and validate dust patterns in the door cavities. The conclusions from the thesis is that it is possible to resolve the door cavities and subsequently simulate the in-cavity flow pattern in these cavities between the fender, the front door and the sill moulding. Second, at cavities between the front door, rear door and the sill moulding.

Keywords: Multiphase modelling, aerodynamics, in-cavity flow, dirt/dust intrusions, door cavities, passenger vehicles, CFD simulations, StarCCM+, ANSA, contamination, self-contamination.

Preface

This Master's thesis work has been conducted in collaboration with Volvo Car Corporation with support and guidance from Chalmers University of Technology. The Master's thesis concludes a two-year education for a degree in Master of Science in Automotive Engineering at Chalmers University of Technology. The Master's thesis project corresponds to 30 credit points and has been conducted from late August 2020 to late February 2021. This Master's thesis is titled 'Modelling of door cavities and simulations to assess flow properties in door cavities for dirt/dust applications'. The Master's thesis is written and submitted by Sagar Mahesh Kukreja. The Master's thesis is supervised by Torbjörn Virdung, PhD. Technical Expert, Complete Vehicle CFD at Volvo Car Corporation and Professor Simone Sebben from Chalmers University of Technology.

ACKNOWLEDGEMENTS

Foremost, I would like to thank Volvo Car Corporation for giving me the opportunity to conduct my master thesis at the department of Contamination & Core CFD. I have thoroughly enjoyed my time at VCC even though the majority of the thesis was spent working from home due to the situation regarding COVID-19. I would like to thank my industrial supervisor at VCC, Torbjörn Virdung, for your support and guidance during the entirety of this project. I would also like to thank Professor Simone Sebben, the examiner of the thesis, for your input and support to improve the outcome of the project in spite of having certain technical difficulties. Finally, I would like to thank the support team at Siemens and particularly Emil Ljungskog, Siemens Digital Industry Software, for the help and guidance with mesh and simulation related matters.

I would like to thank my family for the constant support and care that they have showered on me at all times. I would also like to thank all my friends back in India and all around the world. Additionally, I would like to thank the friends that I met here in Sweden during my Masters, they made me feel like home away from home.

0.1 Abbreviations and Acronyms

Abbreviation	Meaning
2D	Two-dimensional
3D	Three-dimensional
AWD	All Wheel Drive
BLC	Boundary Layer Control
CAD	Computer Aided Design
CD	Cross Diffusion
CFD	Computational Fluid Dynamics
$\rm CO_2$	Carbon Dioxide
CV	Control Volume
DDES	Detached Delayed Eddy Simulation
DES	Detached Eddy Simulation
DNS	Direct Numerical Simulation
FVM	Finite Volume Method
HVAC	Heating, Ventilation, and Air Conditioning
HPC	High-Performance Computing
IDDES	Improved Detached Delayed Eddy Simulation
LES	Large Eddy Simulation
LIC	Line Integral Convolution
MRF	Moving Reference Frame
PID	Property Identification
PLM	Product Lifecycle Management
RANS	Reynolds Averaged Navier-Stokes
RSTM	Reynolds Stress Transport Models
SDR	Specific Dissipation Rate
SIET	Solver Iteration Elapsed Time
SIMPLE	Semi-Implicit Method for Pressure Linked Equations
\mathbf{SRS}	Scale Resolving Simulation
STET	Solver Time Elapsed Per Time-Step
SST	Shear-Stress Transport
TKE	Turbulent Kinetic Energy
URANS	Unsteady Reynolds Averaged Navier-Stokes
VCC	Volvo Car Corporation
WMLES	Wall-Modeled Large Eddy Simulation

0.2 Latin Symbols

Symbol	Description	Units
A	Area	$[m^2]$
B	Wall Smoothness Constant	[-]
C_d	Coefficient of Drag	[-]
C_{DES}	DES Calibration constant	[-]
C_f	Skin Friction Coefficient	[-]
C_p	Coefficient of Pressure	[-]
$C_{p,tot}$	Coefficient of Total Pressure	[-]
C_{μ}	Dimensionless constant	[-]
f	Body force	[N]
F_d	Drag Force	[N]

Ι	Identity matrix	[-]
k	Turbulent kinetic energy	$[m^2/s^2]$
L	Characteristic length	[m]
L_{DES}	DES length scale	[-]
L_{DDES}	DDES length scale	[-]
L_{IDDES}	IDDES length scale	[-]
L_{RANS}	Turbulent length scale	[-]
L_{LES}	LES length scale	[-]
\overline{n}	Normal vector	[m]
p	Static pressure	[Pa]
p_t	Pressure term	[Pa]
q	Dynamic pressure	[-]
Re	Reynolds number	[-]
S_{ij}	Mean strain rate	[1/s]
$S_{ heta}$	Source term	[-]
t	Time	$[\mathbf{s}]$
$u_{ au}$	Frictional velocity	[m/s]
u, v, w	Velocity components	[m/s]
u^+	Dimensionless velocity	[-]
V	Volume	$[m^{3}]$
x, y, z	Cartesian coordinates	[m]
y	Wall distance	[m]
y^+	Dimensionless wall distance	[-]

0.3 Greek Symbols

Symbol	Description	Units
β^*	Modeling Constant	[m]
δ	Boundary Layer Thickness	[m]
δ_{ij}	Kronecker delta	[-]
ϵ	Turbulent dissipation rate	$[m^2/s^3]$
Γ	Diffusion coefficient	[-]
κ	von Karman constant	[-]
u	Kinematic viscosity	$[m^2/s]$
μ	Dynamic viscosity	[kg/ms]
μ_t	Turbulent viscosity	$[m^{2}/s]$
ω	Turbulence frequency	[1/s]
ϕ	Scalar property	[-]
ϕ'	Fluctuating scalar property	[-]
$\overline{\phi}$	Time averaged scalar property	[-]
$ ilde{\phi}$	Filtered scalar property	[-]
ρ	Density	$[kg/m^3]$
σ_{ij}	Stress tensor	[Pa]
$ au_{ij}$	Shear stress tensor	[Pa]
$ au_{\omega}$	Wall shear stress	[Pa]

0.4 Subscripts and superscripts

Abbreviation	Meaning
CS	Cross-section
i,j,k	x,y,z components
tr	Transition point
W	Wall

Free-stream value

Contents

Abst	ract	i
Pref	ace	iii
Ack	nowledgements	iv
Non 0.1	enclature Abbreviations and Acronyms	v v
0.2	Latin Symbols	v
0.3	Greek Symbols	vi
0.4	Subscripts and superscripts	vi
Con	tents	ix
1 Iı	ntroduction	1
1.1	Background	1
1.2	Problem statement	1
1.3	Aims & Objectives	2
1.4	Delimitation	2
2 T	heory	3
2.1	Fundamental Fluid Mechanics	3
2.1.1	Governing equations	4
2.1.2	Multiphase flow	5
2.1.3	Flow descriptors	6
2.1.4	Streamlines	6
2.1.5	Line Integral Convolution	7
2.2	Computational Fluid Dynamics	7
2.2.1	Near-Wall Treatment	7
2.2.2	Numerical Flow Solution	8
2.2.3	Turbulence Modelling	9
2.2.4	Reynolds Averaged Navier-Stokes	9
2.2.5	Two-equation Models	10
2.2.6	Large Eddy Simulations	10
2.2.7	Hybrid Turbulence Modelling	11
2.3	Road Vehicle Aerodynamics	12
2.3.1	Vehicle soiling or Contamination	13
3 N	fethod	15
3.1	Vehicle Geometry	15
3.1.1	Overview of the Vehicle	15
3.1.2	Geometrical simplifications	15
3.1.3	Vehicle cavities	16
3.1.4	Cavity sealings	16
3.2	Experimental study	17
3.2.1	Wind tunnel test	17
3.2.2		19
პ.პ ეექ	OFD study	22
3.3.1	Geometry Clean-up	22
3.3.2	Computational Domain	22
3.3.3	Numerical Grid and Surface mesning	23
3.3.4	Boundary Conditions	27
3.3.5	Souver settings	28

4 Results	30
4.1 Numerical Grid and CFD results	30
4.1.1 Numerical Grid	30
4.1.2 CFD results	32
4.2 Experimental Study	33
4.2.1 Wind tunnel test results	33
4.2.2 On road driving test results	35
5 Discussion	39
5.1 Virtual assessments	39
5.1.1 Geometry, numerical grid and Computational domain	39
5.1.2 CFD workflow	40
5.2 Experimental Method	40
5.2.1 Wind tunnel tests	40
5.2.2 On road driving tests	40
6 Conclusion	42
7 Future Work	43
References	44

1 Introduction

Volvo Car Corporation (VCC), founded in Gothenburg, Sweden in the year 1927, is a subsidiary luxury automotive maker currently owned by the Chinese company Zhejiang Geely Holding. In 2019 Volvo Car had, for the sixth consecutive year, achieved a new global sales record, surpassing 705, 000 sold cars. During 2020, VCC sold 661, 713 cars. Furthermore, rather than just building and selling premium cars, VCC aims to provide their customers with the freedom to move in a personal, sustainable and safe way. Over the next five years, VCC will launch a fully electric car every year as an aim to make all-electric cars 50 per cent of global sales by 2025, with the rest hybrids. This emphasises the importance of sustainability within the company. The research and development is keen on exploring the possibilities to improve user experience in all possible ways. Hence, evaluating dust accumulation on vehicle surfaces is vital not just for user experience but also for safety and functionality.

This thesis is performed within a research project between the Contamination & Core CFD section at Volvo Car and Vehicle Engineering and Autonomous Systems Division at Chalmers University of technology. The Contamination & Core CFD section at Volvo Car is within the department of Sustainability Centre and is responsible for testing and development within water, dirt and snow contamination as well as cleaning.

1.1 Background

Traditionally, a whole lot of work has been put into recognising the forces and moments a vehicle experiences while being operated in different driving conditions. Modern vehicles are much more efficient and safe compared to its predecessors and it is expected of the vehicle to maintain drivability regardless of the driving and weather conditions. A newly developed vehicle would undergo multiple tests and experiments to evaluate its performance in regular and extreme case scenarios. The vehicles are rigorously tested in wind tunnels, climate tunnels and field expeditions, all of these methods are expensive and time consuming, it is still crucial. For an automotive company from a financial standpoint there is a need to reduce the amount of physical testing conducted; advanced modeling and simulation tools and methods, facilitates this. Furthermore, unnecessary seals are removed to reduce overall costs of the vehicle. Due to the lack of proper investigation of in-cavity flows and subsequent cavity contamination related issues, automotive companies find it challenging to justify which seals could be removed without affecting the sealing capacity.

Computational fluid dynamics (CFD) is an important tool in solving various aerodynamic problems for e.g. clearing rain from the wind screens. Any foreign substance like water droplets, dust or snow that influences visibility, performance or aesthetic appeal when in contact with the vehicle, is know as a contaminant. In modern vehicles contaminants are evaluated not only for hindrance in visibility but also to evaluate deposition on critical areas like sensors and camera lenses. Reduction of deposition of contaminants in these critical areas can in turn improve the reliability of these systems and reduce risks of system malfunctions, possibly even save lives.

1.2 Problem statement

Volvo Car Corporation develops a wide variety of passenger cars in different configurations and sizes. During an ongoing project accumulation of contaminants was observed at the tailgate cavities. This led to observing contamination at all cavities in the vehicle and consequently the door cavities. The rubber sealing present at these cavities in vehicle models currently in the market might not be sufficient and an air flow in the door cavities might be present.

Preventing dirt/dust intrusion to the door-cavities of a car is desirable, as dirt/dust present in this region will likely cause soiling of the occupants when exiting/entering the vehicle. In order to prevent this from happening, computational methods to predict the flow field and dirt/dust intrusion is needed, so that predictions can be made very early in a car projects development cycle. The department of Contamination & Core CFD is responsible for the experimental and virtual assessment of contamination performance at Volvo Car. The aim of this master thesis and the department is to use previous knowledge on how to predict flow fields in cavities, and from that formulate a multiphase approach to also include dust/dirt distribution in the simulations.

1.3 Aims & Objectives

Essentially, the overall aim for the thesis project is to investigate the possibilities to virtually assess the air flow within the side door cavities with the help of CFD modeling. Data from previous wind tunnel experiments will be used as validation to said CFD computations. Pressure measurements from a previous study have proved that evaluation of flow properties is possible in certain cavities. The aim is to extend that to the remaining door cavities. Currently, door cavities are not considered in the computational domain and therefore are not evaluated in the vehicle development process. If the thesis is successful in creating a workflow, cavities can be evaluated in an early stage of a project to assess the sealing performance, and therefore reduce the contaminant accumulation.

The key objectives of the thesis are listed as follows:

- Perform physical experiments to assess dirt/dust accumulation in selected door cavities. The experiments will be performed on a non-confidential production vehicle, a Volvo V90.
- Clean up and prepare a geometry that captures the door cavities for CFD simulations from raw CAD models
- Perform CFD simulations to virtually assess the flow field and pressure distribution on the basis of previously experimentally(in the Volvo Car Aerodynamic Wind Tunnel) evaluated cavities. In order to formulate a multiphase CFD methodology/workflow, enabling prediction of dirt/dust distribution in door cavities.

1.4 Delimitation

Delimitation of this study are listed and justified as follows:

- Experiments: Previous Experimental tests and their respective delimitations stand for pressure measurements performed at Volvo Car Aerodynamic Wind Tunnel. The respective pressure measurements are used to save time and cost of repeating similar tests. In addition to this physical experiments will be conducted with goal of visual analysis of accumulation of dirt/dust. Road and weather conditions of these tests will be specific for this test only since it is hard to test the vehicle on a specific track with the required conditions during the period in which the thesis is carried out.
- Geometry: Test vehicle used for experiments is a Volvo V90, hence the geometry used for CFD will be likewise. The geometry of interest is the cavities around the front and rear doors hence their respective pressure measurements will be considered.
- Computer resources: ANSA will be used for geometry clean up and surface preparation, and STAR-CCM+ will be used for meshing and as CFD solver due to already existing methodology for external aerodynamic simulations. Computations will be performed, and limited to, Volvo Car High-Performance Computing (HPC) cluster.

2 Theory

The theory section of this master thesis aims to expose the reader to the underlying necessary ideas by giving an introduction to the fundamentals of fluid mechanics (in particular to road vehicle aerodynamics) and an in depth explanation of the theory related to Computational Fluid Dynamics (CFD).

2.1 Fundamental Fluid Mechanics

The field of fluid mechanics is the study of fluids and forces on them. It can be divided into fluid statics, the study of fluids at rest; and fluid dynamics, the study of the effect of forces on fluid motion. Fluid dynamics is the study of how a fluid flows around an object and its interaction with the object. Fluid dynamics, is an active field of research and aerodynamics is a sub-discipline of fluid dynamics. The field of aerodynamics is the study of gas motion, how a gas flows around an object and its interaction with the object. The modern concept of aerodynamics has been around since the 18th century, during the time studies were targeted on how to design a flying aircraft. Since then, the concept of aerodynamics has been applied to various engineering fields. The physics behind aerodynamics is the forces generated due to the interaction between a fluid and a solid object. The physics behind the interaction between two solid objects have been studied and are well established. When two solid objects collide, the forces are applied at the point of contact. However, the forces acting on a solid body when interacting with a fluid are more complex and rather hard to predict. Multiple parameters need to be taken into account for e.g. the shape and velocity of the solid object, additionally the flow properties such as density and viscosity of the fluid also play a significant role during this interaction. Research shows that the flow characteristics and aerodynamic forces are heavily dependent on the viscosity of the fluid. [29] As a fluid flows past an object, the fluid sticks to the interacting surface due to the stickiness property of the fluid, commonly known as Viscosity. Fluid molecules that are in direct contact with the interacting surface are heavily influenced and subsequently have a velocity equal to zero, typically called the no-slip condition. Considering the fluid as a series of layers, this influence of the surface decreases as we move away from the surface thus creating a very thin region close to the wall commonly know as the boundary layer. In this region the velocity of the fluid ranges from zero at the wall surface to almost free stream velocity. [29] Despite this very thin region known as the boundary layer, the viscous flow properties have a strong influence on how the overall flow field develops. The boundary layer development along a thin flat plate is depicted in Fig. 2.1, where the free stream velocity is denoted as U_{∞} , $\delta(x)$ is the boundary layer thickness as a function of the streamwise coordinate x, and x_{tr} the transition point between the laminar state and the turbulent state of the boundary layer.



Figure 2.1: Boundary layer along a flat plate

The boundary layer flow is separated in two regions. In the initial region the flow is steady and more or less in line with the surface of the plate, this region and the state of flow until the transition point is knows as laminar. The flow state after the transition point is turbulent, where the flow is rather unsteady. The Reynolds number, Re, of a fluid flow is a dimensionless quantity of the flow that determines the state of the flow. It is defined by the ratio of inertial and viscous forces present in the flow:

$$Re = \frac{\rho U_{\infty} L}{\mu} \tag{2.1}$$

In eq.2.1 U_{∞} is the free stream velocity, L is the characteristic length of the geometry and μ is the dynamic viscosity of the fluid. Based on the type of flow, either external or internal, and the geometry itself, there exists a threshold value, known as the critical Reynolds number, where a fluid flow transitions from a laminar state to turbulent state. For external aerodynamic applications, the critical Reynolds number is approximately 5×10^5 .[11]

2.1.1 Governing equations

On a macroscopic level, where length scales are significantly larger than the spacing between atoms, the discrete structures of a material can be neglected and modeled as a continua. The mathematical models describing the physics of a continua are derived from fundamental conservative laws. The conservation laws of a continuum are applied to a control volume based two different approaches, the Eulerian or the Lagrangian. In the Eulerian approach, the volume in consideration represents a portion of time and space where a fluid is allowed to flow through. On the other hand, in the Lagrangian approach the volume in consideration represents a portion of a fluid in the specimen, such that an observer follows the fluid as it moves through time and space. The governing equations of fluid motion are based on the following laws of conversation [1]:

- Conservation of mass
- Conservation of linear momentum
- Conservation of energy

The conservation of mass of a fluid states that the rate of change of mass equals the net in flow of mass through a control volume. Written in differential, conservative form, the three-dimensional continuity equation is as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} \left(\rho v_i \right) = 0 \tag{2.2}$$

where ρ denotes the density of the fluid, and v_i the fluid velocity vector. Aforementioned continuity equation includes compressibility effects by varying the density, ρ , over time t. However, in cases of fluid flows when the Mach number is less than 0.3, compressibility effects can be neglected. [1] Assuming incompressibile flow, and with the velocity components expanded, the continuity equation thus reads as:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{2.3}$$

Newton's second law of motion gives the basis for the law of conservation of linear momentum. In short the law states that, if the rate of change of linear momentum is equal to the sum of forces acting on a fluid, the linear momentum is said to be conserved. Forces acting on a fluid can be divided into two categories, henceforth denoted as surface forces and body forces. Pressure forces and viscous forces constitutes surface forces, whereas gravitational and centrifugal forces constitutes body forces. The conservative, differential form of the conservation of linear momentum is given as:

$$\frac{\partial}{\partial t} \left(\rho v_i\right) + \frac{\partial}{\partial x_i} \left(\rho v_i v_j\right) = \frac{\partial \sigma_{ij}}{\partial x_i} + \rho f_i \tag{2.4}$$

where σ_{ij} denotes the stress tensor, and f_i the body forces. For fluids, the stress tensor can be written as a sum of normal stresses and shear stresses, $\sigma_{ij} = -pI + \tau_{ij}$, where p denotes pressure, I the identity matrix, and τ_{ij} the shear stress tensor, i.e., both components of the aforementioned surface forces. Furthermore, with the assumptions of neglecting body forces (i.e. $f_i = 0$), a fluid flow that is incompressible (constant density), and having a Newtonian fluid with constant viscosity, the Navier-Stokes equations, or momentum equations, can be expressed as:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)$$
(2.5)

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial y} + \nu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right)$$
(2.6)

$$\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + \nu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right)$$
(2.7)

the equations 2.5, 2.6 and 2.7 are the equations of linear momentum in x, y and z directions respectively.

The final governing equation for fluid motion is the energy equation that follows the conservation of energy law or the first law of thermodynamics. Since the influence of the energy equation is neglected in the thesis work it is not further discussed.

2.1.2 Multiphase flow

The flow of particles and droplets in fluids is a subcategory of multicomponent, multiphase flows. The flow of multicomponent, multiphase mixtures covers a wide spectrum of flow conditions and applications. A component is a chemical species such as nitrogen, oxygen, water or Freon. A phase refers to the solid, liquid or vapor state of the matter. Examples of single and multicomponent, multiphase flows are provided in Table 2.1.

Table 2.1: Examples of different types of flows based on components and phase.

	Single component	Multicomponent
Single phase	Water flow	Air flow
Single phase	Nitrogen flow	Flow of Emulsions
Multiphage	Steam-water flow	Air-water flow
Multipliase	Freon-Freon vapor flow	Slurry flow

The flow of air, which is composed of a mixture of gases (nitrogen, oxygen,etc.), is the best example of a single-phase multicomponent flow. It is common to treat these types of flows as the flow of a single component with a viscosity and thermal conductivity which represents the mixture. This kind of approach is practical unless the major constituents of the component gases have significantly different molecular weights. In this case the momentum associated with the diffusional velocities may be important. Also, the multicomponent nature of air will be important at high temperatures where dissociation occurs, or at very low temperatures where some species may condense out. Multiphase flows can be subdivided into four categories: gas-liquid, gas-solid, liquid-solid and three-phase flows.

Gas-solid flows are usually considered to be a gas with suspended solid particles. This category includes pneumatic transport as well as fluidized beds. Another example of a gas-solid flow would be the motion of particles down a chute or inclined plane. These are known as granular flows. Particle-particle and particle-wall interactions are much more important than the forces due to the interstitial gas. If the particles become motionless, the problem reduces to flow through a porous medium in which the viscous force on the particle surfaces is the primary mechanism affecting the gas flow. An example is a pebble-bed heat exchanger. It is not appropriate to refer to flow in a porous medium as a gas-solid flow since the solid phase is not in motion. Gas-solid flow is another example of a dispersed phase flow since the particles constitute the dispersed phase and the gas is the continuous phase.

For many years, the design of systems with particle/droplet flows was based primarily on empiricism. However, more sophisticated measurement techniques have led to improved process control and evaluation of fundamental parameters. Increased computational capability has enabled the development of numerical models that can be used to complement engineering system design. The improved understanding of this is a rapidly growing field of technology which will have far-reaching benefits in upgrading the operation and efficiency of current processes and in supporting the development of new and innovative approaches.

The idea for this thesis was to set up injectors at various locations of the inlet to evaluate with the help of Lagrangian particle tracking how the particles would behave once they come in contact with the vehicle surfaces. StarCCM+ currently supports the use of Lagrangian particle tracking along with the existing physics models. The numerical model will assess this behaviour based on particle size distribution, mass of the particles and the various interactions between each phase. Due to time constraints this was not conducted during this thesis. However, further details are mentioned in the book cited here, if interested the reader can take a look. [6]

2.1.3 Flow descriptors

In order to observe the flow in different cases various flow related parameters can be observed and visualised in many different ways. It is typical to observe a non-dimensional quantity when comparing results between different cases. For aerodynamic applications, flow descriptors such as pressure coefficient and skin friction coefficient are commonly used to evaluate different design configurations.

Pressure coefficient

The pressure coefficient is defined as:

$$C_p = \frac{p - p_\infty}{q_\infty} \tag{2.8}$$

where p is the static pressure, p_{∞} is the reference pressure (ex. wind tunnel pressure) or atmospheric pressure depending on the experimental setup, and q_{∞} the free-stream dynamic pressure(dynamic pressure under free stream conditions)[11]. The total pressure coefficient is to be defined as the amount of energy in the flow with respect to the vehicle and is given by:

$$C_{p,tot} = \frac{p+q}{q_{\infty}} \tag{2.9}$$

where p is the static pressure, q is the local dynamic pressure and q_{∞} is the free-stream dynamic pressure. [11]

Skin friction coefficient

The skin friction coefficient is generally used to evaluate areas of separation, where an area with low or no wall shear indicates a separated flow. This parameter is widely used in the automotive field as separated flow increases the aerodynamic drag. The skin friction coefficient is given by:

$$C_f = \frac{\tau_\omega}{q_\infty} \tag{2.10}$$

where τ_{ω} is the wall shear stress and q_{∞} is the free-stream dynamic pressure. [11]

2.1.4 Streamlines

Another way to evaluate aerodynamic designs is by visualising the flow around a vehicle. In order to visually approximate the path of a fluid particle as it flows around an object, streamlines can be used. A streamline is the path traced by a no-mass particle as it flows in the domain. The streamline curve is created by taking into account the instantaneous velocity vector as the streamline curve is tangent to the velocity vector at all positions. In other words, a streamline indicates in which direction a fluid particle would travel at any given time.[10]



Figure 2.2: Streamlines showing airflow around the tail of Volvo V90.

2.1.5 Line Integral Convolution

Another way of visualizing an approximate fluid motion, or path, is by visualizing the vector field using Line Integral Convolution (LIC). Typically, LIC is used in the e.g. automotive industry to visualize in-plane flow paths around an object, e.g. the flow around a vehicle at a symmetry plane in the streamwise direction. In comparison to other similar techniques where flow field curves are computed using an integration based technique, LIC displays all structural features of the vector field without needing to adapt the two end points of a flow field curve in relation to the vector field itself.[4]



Figure 2.3: LIC of velocity field vector on a cut plane situated in the cavity.

2.2 Computational Fluid Dynamics

During early development stages, simulation software are used to virtually analyze different design choices in relation to air resistance, and therefore fuel economy. Computational Fluid Dynamics (CFD) is the field of fluid dynamics that incorporates numerical analysis to simulate and solve problems involving fluid flows. In modern times, the automotive industry and academia have widely used CFD as a tool for research and development purposes. In the automotive industry the major areas of application of CFD are typically external aerodynamics, internal flows and thermodynamic problems. The complexity of computing fluid flows has historically been an issue. Nevertheless, as affordable High-Performance Computing (HPC) cluster solutions are becoming available to an increasing number of organisations, along with open-source CFD solvers, the development and fields of application continues to expand.

2.2.1 Near-Wall Treatment

The boundary layer is a significant region and can be further divided into smaller regions based on the distance from the wall. Typically, y+, the non dimensional wall distance defines these boundary layer regions. The non dimensional wall distance y+ is given as:

$$y + = \frac{yu_{\tau}}{\nu} \tag{2.11}$$

where y is the absolute distance from the wall, u_{τ} the frictional velocity and ν the kinematic viscosity.[11]. Here, the frictional velocity is defined as the square root of the ratio of wall shear stress to fluid density, i.e.

$$u_{\tau} = \sqrt{\frac{\tau_{\omega}}{\rho}}; \tau_{\omega} = \mu \frac{\partial v}{\partial y}$$
(2.12)

here μ is the dynamic viscosity, and $\frac{\partial v}{\partial y}$ the velocity gradient where the velocity v is parallel to the wall, and y the normal distance to the wall.[11] Furthermore, an additional useful dimensionless variable is u+, interpreted as a dimensionless velocity, is given by[10]:

$$U + = \frac{U_{\infty}}{u_{\tau}} \tag{2.13}$$

The relationship between y+ and u+ is commonly known as the logarithmic law of the wall or simply log-law, which emphasises that the dependency of the fluid flow velocity in a region is dependent on the absolute wall distance, fluid density, fluid viscosity and the wall shear stress in the near wall region i.e. in short U+=f(y+).



Figure 2.4: Law of the wall that describes the various regions within the boundary layer based on the relationship between y + and U +. DNS data is based on experimental data of a fully turbulent flow around a flat plate.[9]

In Fig. 2.4, the influence of the wall on the flow properties is depicted in terms of y+ and U+. Typically, three regions are observed:

- Viscous sub-layer
- Buffer layer
- Log-law layer

The viscous sub-layer is a very thin layer that is closest to the wall, in this layer, the flow is dominated by the viscous forces between the fluid and the solid wall, hence making the flow laminar in nature. In terms of y+ the viscous sub layer lies between the wall surface and up to y+=5. The flow in this region can be represented by the linear model U+=y+. On the other hand the flow in the outer layer i.e. the log-law layer the flow is dominated by turbulent effects and hence known as the turbulent layer. The velocity of the flow varies based on a logarithmic function of y+, given by:

$$U + = \frac{1}{\kappa} \ln(y^{+}) + B$$
 (2.14)

where $\kappa \approx 0.42$ and $B \approx 5.0$ (κ is the von Kármán constant and B is a wall smoothness calibration constant). This region is observed at y+ values equal to or higher than 30. The second layer i.e. 5 < y+ < 30, as the name suggest is the buffer layer where the flow transitions from laminar to turbulent flow, modelling of this layer is not easy and is not as accurate to predict when compared to the other two layers. When compared to the DNS data from experiments conducted on a fully turbulent flow around a flat plate, a valid numerical correlation exists.[1]

2.2.2 Numerical Flow Solution

For a given specific flow case a computational domain encloses a finite number of cells where the governing equations are solved to generate a solution that will depict the physics of the flow. In Finite Volume Method (FVM), the computational domain is divided into smaller finite Control Volumes (CV), the numerical method

discretizes the underlying mathematical model into a system of algebraic equations.

The aforementioned conservation equations can be written in terms of a generic transport equation. By integrating the generic transport equation over a volume V and applying Gauss's divergence theorem, the following integral form of the general transport equation is obtained [29][1]:

$$\frac{\partial}{\partial t} \left(\int_{V} \rho \phi \mathrm{d}V \right)_{TransientTerm} + \underbrace{\int_{A} \bar{n} \cdot (\rho v_{i}\phi) \,\mathrm{d}A}_{ConvectiveTerm} = \underbrace{\int_{A} \bar{n} \cdot (\Gamma \nabla \phi) \,\mathrm{d}A}_{DiffusiveFlux} + \underbrace{\int_{V} S_{\phi} \mathrm{d}V}_{SourceTerm}$$
(2.15)

where ϕ denotes a scalar property, A the surface area of the specific volume, Γ the diffusion coefficient and \bar{n} the normal vector. To obtain a solvable set of the governing equations, the integral form of the transport equation is discretized for every control volume of the computational domain. The accuracy of the solutions are highly effected by the choice of discretization methods, or schemes based on the nature of discretization. A higher order scheme implies a more accurate solution, whilst having a downside of being computational expensive.[1] The discretized equations can be solved in two different ways, one being a segregated flow solver, and the other a coupled flow solver.

2.2.3 Turbulence Modelling

Generally, fluid motion in the field of CFD is considered to be as irregular and random with chaotic behaviors. Essentially, this state of a fluid experiencing irregular behavior is know as being turbulent. Accurately modelling turbulence using CFD is one of the major challenges for most engineering applications. If a fluid flow is not turbulent, it is known to be laminar given the assumption of neglecting the transition interval between the two. As previously mentioned, a common method of estimating if a flow is laminar or turbulent is by calculating the Reynolds number.

For turbulent flow cases, the size of the fluctuating quantities are small and fluctuates with high frequencies. Hence, resolving the fluctuations in time and space comes at an excessive computational costs. By Direct Numerical Simulations (DNS), all turbulence is resolved. However, solving for the exact governing equations is not always necessary and averaged or filtered quantities can approximate the fluctuations to reduce the complexity and therefore the computational cost. Throughout the years, multiple turbulence models have been formulated and continuously improved as new discoveries are made. Commonly, turbulence models is separated into two groups, one being Reynolds Average Navier-Stokes (RANS) turbulence models, and the other being Scale Resolving Simulations (SRS).

2.2.4 Reynolds Averaged Navier-Stokes

To reduce the computational cost associated with resolving turbulence, turbulent behavior can be modeled instead of being resolved. To do so, each solution variable ϕ in the instantaneous Navier-Stokes equations is decomposed (commonly known as Reynold's decomposition) into its time averaged mean value $\bar{\phi}$ and its fluctuating component ϕ' :

$$\phi = \bar{\phi} + \phi' \tag{2.16}$$

here ϕ is to denote the velocity components, pressure, energy or species concentration. In Eq.2.16, $\bar{\phi}$ and ϕ' are functions of position and time. $\bar{\phi}$ is a time averaged quantity and is given by:

$$\bar{\phi} = \frac{1}{\Delta t} \int_{0}^{\Delta t} \phi(t) \,\mathrm{d}t \tag{2.17}$$

On inserting the Reynold's decomposition for the pressure and velocity components into the incompressible momentum equations, i.e. Eqs. 2.5, 2.6 and 2.7, the so called RANS equations are obtained:

$$\bar{v_j}\frac{\partial\bar{v_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial\bar{p}}{\partial x_i} + \frac{1}{\rho}\frac{\partial}{\partial x_j}\left(2\mu S_{ij} + \tau_{ij}\right)$$
(2.18)

where S_{ij} is the mean strain-rate tensor and μ is the dynamic viscosity. τ_{ij} , is called the Reynolds stress tensor. The tensor is symmetric, hence consists of six components, unfortunately all unknown at the time. Hence, to be able to solve the problem, additional equations are needed. Commonly, this is known as the Closure problem. However, by applying Boussinesq hypothesis, which states that there is a correlation between Reynolds stresses and viscous stresses, the unknown stresses from the Reynolds stress tensor are approximated by the eddy viscosity. This effectively eliminates the Reynolds stress tensor from the equations, the Boussinesq hypothesis in equation form is as follows:

$$\tau_{ij} = -\rho \overline{v_i' v_j'} = 2\mu_t S_{ij} - \frac{2}{3}\rho k \delta_{ij}$$
(2.19)

where μ_t denotes the turbulent (or eddy) viscosity, $k = 1/2(\bar{u'^2} + \bar{v'^2} + \bar{w'^2})$ is the turbulent kinetic energy (TKE), and δ_{ij} the Kronecker delta. To ensure that the equation outputs the correct results for the normal Reynolds stresses, the Kronecker delta is formulated as $\delta_{ij} = 1$ if i = j, and $\delta_{ij} = 0$ if $i \neq j$. Thus the RANS equations writes as follows:

$$\bar{v_j}\frac{\partial\bar{v_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial\bar{p_t}}{\partial x_i} + \frac{1}{\rho}\frac{\partial}{\partial x_j}\left(2S_{ij}\left(\mu + \mu_t\right)\right)$$
(2.20)

here the turbulent kinetic energy, k, has been incorporated into the pressure term as $\bar{p}_t = \bar{p} + 2k/3$. The two viscosity terms, μ and μ_t , are to be interpreted as that μ is fluid dependent and is a function of e.g. temperature, whereas μ_t is location dependent within the flow itself. The eddy viscosity, however, adds an extra unknown variable, but the equations are closed by relating μ_t to the turbulent kinetic energy k, and either to the turbulence frequency ω or to the turbulent dissipation rate ϵ by [1]:

- $\mu_t = \rho k / \omega$, commonly known as the $k \omega$ model, or
- $\mu_t = \rho C_{\mu} k^2 / \epsilon$, the $k \epsilon$ model, respectively, where C_{μ} is a dimensionless constant.

The two mentioned models are categorized as Eddy Viscosity Models and applies the approximation made by Boussinesq and solves the system of equations by introducing additional transport equations. Since the eddy viscosity is related to two additional variables, k and ω or k and ϵ , both are what is commonly known as two-equation turbulence models. Furthermore, other approaches exists, such as Reynolds Stress Transport Models (RSTM), where the individual Reynolds stresses are solved for one transport equation each, thus six in total. However, for this thesis work, the Eddy Viscosity Model $k - \omega$ will be implemented and thus RSTM will not be further discussed.

2.2.5 Two-equation Models

The $k - \omega$ turbulence model was originally developed by Wilcox [32] in order to improve the already existing $k - \epsilon$ by Jones and Launder. [14] Wilcox claimed his model was superior over the $k - \epsilon$ model as it improved the performance for modelling of boundary layers under adverse pressure gradients. Additionally, Wilcox's $k - \omega$ model without any further modifications has the possibility to be used throughout the boundary layer, including the viscous dominated region, making the model appropriate for external aerodynamic applications. Wilcox later revised his work in 1998 [33] and in 2008 [31] further developing the turbulence model. In its original formulation, the model was sensitive to changes in the free stream, ω values in the boundary layer computations, leading to an extreme sensitivity to changes in the inlet boundary conditions. Issues were mainly experienced for internal flow cases, which was unheard of for $k - \epsilon$. The obvious complications of Wilcox's model was addressed by Menter [17] who realized that the ϵ transport equation of of the $k - \epsilon$ model could be transformed to an ω transport equation by a variable substitution. Menter later developed his own variant of the $k - \omega$ model called the $k - \omega$ SST (Shear-Stress Transport) turbulence model. The $k - \omega$ SST model uses the $k - \omega$ formulation close to the wall and $k - \epsilon$ in the free stream by applying a blending function. Moreover, a second blending function is incorporated in the $k - \omega$ SST model which limits the turbulent dissipation. [1]

2.2.6 Large Eddy Simulations

In most engineering operations, results from a RANS solution of a turbulent flow case is often considered satisfactory. However, due to the inherently random and time dependent behaviour of turbulence, a statistical time averaged-approach where instantaneous fluctuations of the fluid flow are resolved might sometimes be necessary to obtain the true characteristics of flow properties. The main problem in simulating high-Reynolds-number flows is the presence of very small length scales and timescales, a logical solution is to filter the equations, thus resolving only intermediate-to-large turbulence scales. Large-eddy simulation (LES) extends the usefulness of DNS for practical engineering applications by intentionally leaving the smallest turbulence scales unresolved but instead modelling them. In LES the dynamics of the large scales are computed explicitly.

Therefore, LES needs to be 3D and transient. The advantages of LES arise from the fact that the large eddies, which are hard to model in a universal way since they are anisotropic, are simulated directly. In contrast, small eddies are more easily modelled since they are closer to isotropy and adapt quickly to maintain a dynamic equilibrium with the energy-transfer rate imposed by the large eddies.[1] In relation to the RANS equations, the equations solved for LES are obtained by a spatial filtering rather than an averaging process. Each solution variable ϕ is decomposed into a filtered value $\tilde{\phi}$ and a sub-filtered, or sub-grid, value ϕ' :

$$\phi = \dot{\phi} + \phi' \tag{2.21}$$

where ϕ denotes either the velocity components, pressure, energy, or species concentration.

In order to resolve the crucial turbulent structures close to the wall, LES requires an excessively detailed grid resolution in the wall boundary layer, not only in the direction normal to the wall but also in the in-plane direction. The computational cost for LES is high in comparison with those of other turbulence models. The high computational cost stems from a very fine grid, short time steps and long computational time taken to obtain reliable statistics. Thus, LES is used mainly for academic research or for flows with low Reynolds numbers.

To overcome the prohibitive computational cost of LES, several models have been developed which seek to take advantage of the benefits of RANS when possible while still resolving the largest turbulent structures. Detached eddy simulation (DES) is an approach which alternate between RANS and LES, but instead uses the grid resolution to determine which mode to use. If the maximum length of the cell edges is larger than the turbulent length scale, RANS mode is used, else the LES formulation is applied. That requires the mesh to be of LES quality in those regions. The intention is to use RANS to solve the attached boundary layers, but since the grid is fine near the wall, LES mode might be unintentionally activated. [1][24]

2.2.7 Hybrid Turbulence Modelling

A Detached Eddy Simulation (DES) is a hybrid RANS-LES model that applies a combination to capture the fluid flow, it resolves turbulent structures in the core flow by conventional LES and treats the wall boundary layers with a conventional RANS model. Introduced by Spalart et al. in 1997 [25], DES uses the benefits of RANS modeling in combination with LES to avoid the extensive mesh requirements otherwise necessary for LES-only models. Fluid flow in the near wall boundary layer is heavily influenced by the unsteady nature of LES, hence the boundary layer flow will also be unsteady thus modelled by the unsteady RANS (URANS) turbulence model.

A significant element in DES methodology is to make a decision on how and when to segregate between the RANS and LES portions of the computational domain. Arbitrarily, two critical length scales are of interest. The first is the effective turbulence length scale implied by the model, denoted as L_{RANS} , and the second is the maximum local grid size, Δ . [19] The larger the cell size, the further from away the wall the switch happens. The technique of adopting a threshold value to distinguish between appropriate length scales, thus turbulence models, is commonly know as blending, much similar to the functionality of the $k - \omega$ model.

The main difference between SST $k - \omega$ DES and SST $k - \omega$ is essentially how the dissipation term of the k equation is integrated in the model. The DES transport equation of the k equation is given as:

$$\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_j} \left(\bar{v}_j k \right) = \tilde{P}_k - \frac{k^{3/2}}{L_{DES}} + \frac{\partial}{\partial x_j} \left[\left(v + v_t \sigma_k \right) \frac{\partial k}{\partial x_j} \right]$$
(2.22)

where $L_{DES} = min(L_{RANS}, L_{LES})$ with $L_{RANS} = \sqrt{k}/\beta^*\omega$ being the turbulent length scale with $\beta^* = 0.09$ as a modelling constant, $L_{LES} = C_{DES}\Delta$ is the LES length scale and C_{DES} is a calibration constant. The maximum local grid spacing is given by $\Delta = max(\Delta x, \Delta y, \Delta z)$. [19] Whilst DES is signi

cantly more computational expensive in relation to RANS and cheaper than LES, the results reveal roughly the same information about the flow dynamics as a typical LES model. However, major concerns have been targeted towards the inference between RANS and LES in DES models which partly depends on grid spacing. Grid sizes in streamwise and spanwise direction less than the thickness of the boundary layer, δ , may cause premature switches between the two models, thus treating the boundary layer using LES on a, in theory, to coarse grid resulting in inaccurate predictions. To avoid said issue of resolving the boundary layer using LES, recommendations by Davidson states that streamwise and spanwise grid spacing is to be, expressed in wall units, $\Delta x \simeq 100$ and $\Delta z \simeq 30$, where one wall unit equals the near-wall spacing in wall normal direction (Δy). [7] Moreover, multiple approaches have been developed to, more or less, shield the boundary layer from adapting LES. [18] [27] Extensive research has lead to the development of the Delayed Detached Eddy Simulation (DDES) model, or the Delayed DES model. The governing transport equations are more or less identical to the two used for DES with a minor change to the dissipation term in the k-equation. The length scale term for DDES reads as:

$$L_{DDES} = L_{RANS} - f_d min \left(0, L_{RANS} - L_{LES}\right)$$

$$(2.23)$$

For length scales of $L_{RANS} = L_{LES}$, the formulation reverts to the original $k - \omega$ SST model. The variable f_d is a generic shielding function. As a further development of DES and DDES, the Improved Delayed Detached Eddy Simulation (IDDES) model was discussed by Shur et al. [23] in 2008. The model combines the aforementioned DDES model and a model developed for Wall-Modeled LES (WMLES) into one. The distinction between the two is if the simulation contains in flow turbulent contents or not. In the case of present in flow turbulent content, the model resolves the turbulence except for the near-wall region, i.e the model uses the WMLES branch. A major improvement in relation to the DES is how the resolved turbulence is adjusted to the modelled equivalent in the logarithmic layer, hence solving the previously observed issues of log-layer mismatch (LLM) for DES and other WMLES models. [23] A significant parameter of making this possible was the introduction of the sub-grid length scale that not only depends on the grid spacing in normal, spanwise and streamwise direction, but also on the distance from the wall. Furthermore, for cases with no turbulent content present in the in flow, the model behaves as a DDES model, i.e. models the near-wall, uses RANS for attached flows and resolves the massively separated regions using a DES. The switch between the two branches, i.e. the switch between DDES and WMLES is controlled via a blending function. In a similar manner to how a DDES model is incorporated in the transport equations, the dissipation term of the k-equation for IDDES is divided by the IDDES length scale as:

$$L_{IDDES} = \tilde{f}_d \left(1 + f_e\right) L_{RANS} + \left(1 - \tilde{f}_d\right) L_{DES}$$

$$(2.24)$$

where the length scale previously denoted as Δ in $L_{LES} = C_{DES}\Delta$, now is expressed as $\Delta = min[C_{\omega}max[d, h_{max}], h_{max}]$, where h_{max} is the maximum edge length of the cell of interest, and C_{ω} is a model constant. Additional information regard IDDES can be found in the paper by Shur et al. [23]

2.3 Road Vehicle Aerodynamics

When road vehicles are taken into consideration, typically two fundamental forces play a major role, i.e. lift and drag forces. Aerodynamic drag acts in the opposite direction of the vehicle velocity and thus resists the motion of the vehicle. To overcome this resistive force, additional energy will be required for the propulsion system further increasing the consumption of energy. Road vehicle transportation has been known to be a major contributor to green house emissions, in 2013, road vehicles contributed up to 17% of the total emissions.[20] Due to this, authorities have pressured vehicle manufacturers to reduce the CO_2 emissions from their vehicle by placing legislation and mandates. Reducing the drag of a vehicle would lead to the reduction of energy consumption and subsequently reduce the CO_2 emissions of the vehicle, this is one of the common approaches used by modern vehicles to reduce the aerodynamic drag of the vehicle. Aerodynamic drag is given by:

$$F_d = \frac{1}{2} C_d \rho A v^2 \tag{2.25}$$

where C_d is the coefficient of drag, and is a dimensionless quantity that is used to quantify the drag or resistance of an object in a fluid environment, A is the reference area or the frontal area of road vehicles, ρ is the fluid density and v is the free stream velocity

As observed in Eq.2.25 the aerodynamic drag F_d has a quadratic dependence on the velocity of the vehicle, hence at high velocities the drag force has a significant influence on the energy consumption. Furthermore, aerodynamic drag can be divided into two components, pressure drag and friction drag. [11] The physical phenomenon behind the pressure drag is due to the pressure differences of the vehicle when in motion. Typically, a high pressure zone, also known as stagnation region, is seen at the front of a vehicle. The pressure decreases downstream of the vehicle hence creating a net pressure force in the direction opposite to vehicle motion. Furthermore, the pressure can divided into static and dynamic pressure. Static pressure is based on the state of the fluid and is affected by various factors like altitude over sea level and so on. The dynamic pressure depends on the fluid properties and can be defined by $0.5\rho v^2$, where ρ is the density of the fluid and v the fluid velocity. The sum of the two pressures i.e. static pressure + dynamic pressure is defined as the total pressure. On the other hand, friction drag originates from the shear forces generated between the fluid and the vehicle surfaces due to surface resistance.

It has been observed that approximately 45 % of a passenger cars aerodynamic drag is generated by the upper body shape, while the underbody sums up for around 30 %.[10] The remainder 25 % of the aerodynamic drag force originates from the rotating wheels and the tyre wake created.[21] Additionally, an important aspect of tyre wake is the interaction with the underbody flow and with the larger wake downstream of the vehicle, hence it is significant to take into consideration the underbody, rotating or stationary wheels and moving ground.

2.3.1 Vehicle soiling or Contamination

In addition to the traditional topics of aerodynamics such as airflows, vehicle soiling also represents one of the aerodynamic design tasks tackled during vehicle development. The most important aspect in this context is perceptual safety. The relevant vehicle surfaces should also remain free of contaminants so that customers do not become dirty when entering and exiting the vehicle. In modern development processes a substantial amount of resources are spent to evaluate the effects of contamination due to its significance on visibility primarily, additionally it's interaction with sensors and cameras in modern vehicles. The contamination of a vehicle can be caused by two factors:

- External contamination: This is the result of contaminants thrown up by surrounding traffic or surrounding air that carries the contaminants, e.g. dust storms.
- Self induced contamination: This is referred to the contamination particles propagated by the vehicle itself; especially with regard to the spinning wheels.

Reducing external contamination is the main area of focus when optimizing vehicle soiling characteristics in passenger car development e.g. rain droplets. However, in modern development processes more effort is being spent in the context in vehicle-induced contamination to improve user experience. External contamination strategies predominantly deal with the contamination due to rain and water droplets. On day to day basis the likelihood of driving in a dust storm is comparatively low. Hence, the details regarding external contamination are not discussed here. The reader can find the details in the book cited here. [11]

Causes of Vehicle-induced contamination

The main cause of self contamination has to do with the water droplets and contamination particles whirled up by a rotating wheel. The following highlights how this process takes place at a wheel and the effects it has. The contaminants whirled up by a spinning/rotating wheel are broken down into two categories:

- Splash contamination: Larger particles that move in parabolic curve formation close to the ground and at relatively high speeds.
- Spray contamination: A fine mist of smaller particles that are carried downwind by the turbulent airflow

Figure 2.5 shows the spray pattern of a free-rolling, uncovered wheel as Koessler [15], Braun [2], and Clarke [5] have described. The spinning wheel with profiled tire can transport only some of the water or contaminants on the road in the profile tread ribs toward the outside or in the direction of rotation. The remaining portion is displaced in front of the tire contact patch, similar to a bow wave created by a boat. Splash water then takes shape as relatively large droplets, which are dispersed forwards and to the side at an angle of up to 45 degrees. In the profile tread ribs, water rotates along with the tire due to the adhesion forces it exerts. When the circumferential speed reaches a critical threshold, however, this adhesion is no longer sufficient in resisting the centrifugal forces. Small water droplets are cast away in a tangential way in the form of spray water.



Figure 2.5: Schematic display of water/contaminant separation at a rotating wheel.

Two independent spray zones are created: At the end of the contact area, the local circumferential wheel speed rapidly increases. Some of the water below the tire contact patch sprays away in the form of fine droplets at an angle of between 20 and 30 degrees. Included in this spray mist is splash water from the profile tread ribs that is thrown outward at a shallower angle. A splash-free zone now forms in the tire circumference direction. The adhesion forces of the flat water layer remaining in the profile tread ribs exceed the centrifugal forces in the surrounding area. Only in the upper third area can the centrifugal forces overcome the adhesion, and water droplets are cast away vertically and counter to the travel direction of the vehicle. When a wheel is covered by a fender, the droplets contact the wheel well liner and disperse. Some of the water can collect there, travel downward, and combine to form large droplets at the end of the liner. The remaining water is transported out of the wheel well by air circulating in the upper rear wheel well area and transitions into a spray mist. In general, a portion of the spray mist at the wheel is channeled into the outside airflow by the outward flow coming out of the wheel well and in the center area of the underfloor. This mist either accumulates on the side of the vehicle and the door handle or it is transported to the rear wake and considerably contributes to vehicle-induced contamination. Here the contaminant is considered to be water droplets but a similar pattern can be seen for dust particles based on the size of the dust particles.

Vehicle-induced contamination primarily refers to soiling of the rear of the vehicle and its side quarters. The sides and door handles are contaminated as a result of the splash contamination created by the spinning front wheels and the spray mist ejected from the front wheel wells, whereas the trunk lid, rear lights, and trunk lid handle are mainly contaminated by the rear wheels. Adding to these phenomena is the contaminants that comes from the turbulent underfloor airflows. Fine particles, some of which have dispersed a second time here, enter the wake of the vehicle and land on the rear panels.

3 Method

The method section is to summarise the development process, methods and settings that were used in order to create a workflow for the simulations. Additionally it will also describe the method used to carry out the physical experiments and give a description of the wind tunnel tests conducted prior to this thesis. The first section focuses on Geometry clean up and CFD related matters, the second highlights the experimental test conducted and third the wind tunnel testing performed.

3.1 Vehicle Geometry

It can not be stressed enough about the high importance of details required for CFD simulations. Low quality CAD models, poor meshing method or unsuitable choice of turbulence modeling may lead to a converged solution but can be most likely unrelated to real-world physics. At the beginning of the thesis the Author was made aware that it was tricky to completely resolve the side door cavities. How the geometry of the vehicle was cleaned up and prepared, including solver and turbulence settings to reach a valid CFD simulation, are further presented below.

3.1.1 Overview of the Vehicle

A complete vehicle model consists of various parts. Hence it is key to maintain the high number of parts in an organised manner. At Volvo Car, each vehicle model is divided into 30 or more smaller focus groups. Subsequently these groups arrange their developed parts on a PLM system used by Volvo Car called Teamcenter. From teamcenter a geometrical model of the Volvo V90 could be exported. The choice of this model was made during a prior study and hence the same was made in order to keep continuity. Moreover, a physical vehicle of the V90 was accessible for road and wind tunnel testing. The basic configuration of this vehicle is an in-line four cylinder diesel engine with an automatic gearbox and all wheel drive (AWD). The vehicle is illustrated in the figure below. The V90 model has a long history of being developed and manufactured in the Torslanda plant in Gothenburg, thus one can expect a high level of details in the CAD geometry. Typically, all details were present in the export to depict the physical vehicle model in an accurate way. Since this project deals with the external aerodynamics of the vehicle and would not require the interiors details of the vehicle, parts like seats, interior trim, infotainment systems and so on within the cabin were excluded during the export to save time and disk space. However, all parts that would most like influence or come in contact with the external airflow were retained, for example powertrain systems, underbody and suspension assembly. Figure [3.2] shows a detailed view of the geometry.

3.1.2 Geometrical simplifications

Managing disk space and grid size is vital in CFD simulations to ease the computational power required for each simulation. Hence no internal flow is modelled in the cooling and HVAC systems. Additionally, the air intake system is closed.

In the wind tunnel tests the drive shafts were removed to allow the use of rotating wheels and avoid overheating of the vehicle. During this test the cardan shaft (part of the AWD system) connecting the front and rear wheel axle was disconnected as well. Consequently, these parts were removed even in the CFD simulations. Curb weight (American English) or kerb weight (British English) is the total mass of a vehicle with standard equipment and all necessary operating consumables such as motor oil, transmission oil, brake fluid, coolant, air conditioning refrigerant, and sometimes a full tank of fuel, while not loaded with either passengers or cargo. The gross vehicle weight is larger and includes the maximum payload of passengers and cargo. This is an important criteria from an aerodynamic perspective as the weight of the vehicle affects the ride height of the vehicle and that in-turn has an effect on the drag and lift forces experienced by the road vehicle. In the automotive industry the base curb weight is donated as curb+0. A setup of curb+2 was used for this thesis which means the weight of two passengers were added. This would affect the ride height, a more compressed suspension and eventually a more deformed tyre. These were the basic simplifications and manipulations done to the geometry of the vehicle in order to depict real world physical conditions.[3.3]





(b) Isometric view Figure 3.1: Volvo V90 AWD

3.1.3 Vehicle cavities

By definition cavities are empty spaces within solid objects. Cavities are created when objects are designed to open or close with respect to each other. In a vehicle these can be the doors, bonnet, gas lids or in modern vehicles even a sun roof. A small tight space is created in between the interior of a door and the chassis when the door is closed, this is specifically termed as a cavity in the automotive industry. The cavity of interest for this thesis are the cavities between all the four passenger doors, the front and back fenders, the sill moulding and parts mounted on the vehicle chassis like the inner sealings, the inner trim details and the chassis itself. This specific cavity is depicted in figure 3.4. The cavity of interest is highlighted in green. In order to capture all the activities in the cavity, no modifications were made in the region, neither in the physical tests nor in CFD simulations. In an ideal scenario this cavity would be closed in the best way possible. However, to accommodate for various properties like mechanical tolerances, flexibility and expansion due to change in temperatures, it is not possible to have an idealistic fit in the real world. Furthermore, modern automotive companies pay a lot of attention to user experience, hence these cavities are tried to be minimized to avoid the accumulation of dirt/dust and snow to eventually provide a feeling of luxury.

3.1.4 Cavity sealings

The cabin of a passenger car is expected to be completely sealed from contaminants despite tough road and weather conditions. To maintain high user experience car manufacturers apply different types of sealants to enclose the cabin. A rubber sealant is the most common choice due to its elasticity and compressible properties. The sealing is usually glued on the chassis side of the cabin and the ends are snapped together by a slight overlap to complete the circle without leaving any gaps. A good sealing would completely close the gap between the door and the chassis when the the door is in closed position. Another type of sealing exists in cars, the one that is used to prevent particles from the entering the cavities. These are present at regions usually created at intersections of two different parts, for e.g. the gap between the exterior panels of the front fender and the



Figure 3.2: Top view and bottom view of the vehicle. The bonnet in the top view has been removed to visualize detailing in the engine compartment.

front door. Typically such gaps are known as split lines and a split line sealing is used to seal these gaps in order to avoid intrusions by unnecessary contaminants like dirt/dust. However, little or no research has been done on the performance of this sealing and no virtual simulations have been previously conducted within the global automotive industry. The only means of testing the performance has been through physical tests like open road and climate wind tunnel tests.

3.2 Experimental study

To evaluate, validate and compare the results of CFD simulations to real world physics, experimental data is essential. For this thesis two types of experimental data was used to compare and visualise the intrusion of dust in the door cavities. First, pre existing wind tunnel data from a previous test and secondly, open road physical test.

3.2.1 Wind tunnel test

This section describes the method used during a test that was previously conducted[3], the results achieved during this test are presented further on. Furthermore, these results are of interest for this thesis and have not been presented before. The main reason behind using the aero wind tunnel is that the climate wind tunnel currently at Volvo car can be not used for testing of dust particles. Since, the contaminant of interest is dust for this thesis and dust is assumed to be light enough to be carried by air, the air flow around the vehicle can be used to get a basis of how the dust+air would flow.

The wind tunnel at Volvo Car commonly known as PVT was built in the start of year 1980 and was fully operational in the year 1986.[26] PVT is a horizontal closed loop wind tunnel that utilizes a 5.0 MW fan with a



(a) Drive shafts removed for simplification



(b) Front view of type deformation under curb+02



(c) Side view of tyre deformation under curb+02Figure 3.3: Geometrical Modifications



Figure 3.4: The side(left) door cavities, highlighted in green.

diameter as big as 8.2m to stimulate the air around the loop. The wind tunnel has a capacity to generate wind speeds up to 250 kph. The test section has a cross-sectional area of 27 m², ranging from a width of 6.6 m and a height of 4.1 m. The key aspects to simulate driving conditions, are the 5-belt moving ground system as well as a boundary layer control (BLC) system. The 5-belt moving ground system consists a centre belt under the vehicle and four wheel driving belts to simulate rotating wheels. The BLC system comprises of a suction scoop close to the test section inlet, two suction zones upstream of the centre belt, and tangential blowers located downstream of the centre belt and rear tyres. These two systems are essential in replicating realistic driving conditions to maximise the correlation between results from the wind tunnel test and open road driving performance or tests.

Further details and performance evaluations regarding the Volvo Car wind tunnel can be found in the work by Sternéus et. al. [26]

Forces and pressure measurements

This section gives a brief summary of how forces and pressure acting on the vehicle were measured in the aforementioned pre-existing wind tunnel test.

The 5-belt moving ground system is a combination of belts and struts. The four struts, two mounted downstream of the front wheels and two mounted upstream of the rear wheels, keep the vehicle in place while the belts allow each individual wheel to be controlled independently. The struts are connected to the under-floor balance and used to measure the aerodynamic loads present. One of the important forces for road vehicles is the drag force, F_d , which can be easily measure by the balance system. Subsequently from this the coefficient of drag, C_d can



Figure 3.5: Highlighted in white is the splitline cavity at the B and C pillar.

be easily calculated. However, for this thesis the aerodynamic drag coefficient is not the primary validation variable inspite of existing experimental data being at hand. Complimentary to force measurements, various other parameters can be used as a validation variable, in particular pressure measurements are of interest for this thesis as it would depict the regions where dust could be most likely accumulated. Since pressure can be measured at different locations on the surface of the vehicle using pressure sensors, this data can be used as a promising CFD validation variable for surface specific properties. Pressure data can be used to calculate the pressure gradients, pressure coefficients or the total pressure values. During this test in house transient pressure sensors that could be placed at the surface of interest were used. These sensors recorded samples of instantaneous pressure values and time averaged values over a fixed time duration.





(a) Vehicle positioned in the wind tunnel.[3]
 (b) Pressure sensors around the vehicle.[3]
 Figure 3.6: Volvo Wind tunnel(PVT)

3.2.2 On road driving test

The key advantage of a wind tunnel test is the repeatibility. One can almost recreate the test to finest of details. However, when it comes to testing the intrusion of dirt/dust contaminants in the cavities, the wind tunnel can not be used due to the presence of dust particles that would cause damage to the tunnel and the fan. Hence, engineers use test tracks and expeditions to test for contamination kind of applications. Automotive companies usually have tracks that are specific for certain particles. Expeditions are set up in the North of Europe to test for snow contamination and in desert regions for dirt and dust applications. Unfortunately, due to restrictions and limitations during the period of the thesis it was not feasible to set up an ideal driving test in a desert region. Nevertheless, the idea was to drive the vehicle physically in areas that have an abundance of gravel roads. The location chosen for this was a few kilometers south of Gothenburg. After, researching for gravel road types around Gothenburg on trafikverket.se website, a few locations were selected. This test was

conducted without any control over the weather conditions. The vehicle was cleaned and set up to visualise the dust pattern in the cavities during driving conditions. Each test was repeated on each track to account for repeatibility and an attempt was made to maintain similarity between each test on a particular track. The details of the test are as follows: The test on Road1 was conducted during slight rainy conditions. The road had

	Distance (km)	Avg. speed (km/h)	Duration (min)
Test 1	7.4	37	12
Test 2	7.4	47	11

Table 3.1: Details of road test on Road1

a lot of gravel and dust. The road was through the woods and looked like it would yield in good dirt patterns. Before each trip the cavity regions of interest were thoroughly cleaned. Figure 3.7 shows an illustration of Road1, the kind of terrain can be noted in the figure.

Google Maps Varberg Ö, 430 16 Rolfstorp till Köpmansvägen 97, 430 17 Kör 6,2 km, 11 min Skällinge



Bilder ©2021 TerraMetrics, Kartdata ©2021 1 km I

Figure 3.7: Terrain Map of Road1

The details of the second session are as follows: The second road session was conducted during no rain but wet road conditions. The road was around a lake and consisted of a combination of gravel and asphalt road. In this test as well, the cavities of interest were thoroughly cleaned in order to capture clear dust patterns. Figure 3.8 illustrates the road and terrain in the region.

Road type	Distance (km)	Avg. speed (km/h)
Asphalt	4.7	40
Gravel	5.7	38
Asphalt	1	42
Gravel	71	37

Table 3.2: Details of road test on Road2

Google Maps

Veddigevägen, 430 20 Veddige till Backen, 439 64 Frillesås

Kör 8,3 km, 15 min



Bilder ©2021 TerraMetrics, Kartdata ©2021 2 km ⊨

Figure 3.8: Terrain Map of Road1

3.3 CFD study

In this section, details about the CFD study will be presented. The computational domain, mesh strategy, solver settings and boundary conditions will be summarised to give the reader a broad insight of the work.

3.3.1 Geometry Clean-up

Raw CAD geometries are seldom suitable to be used in CFD simulations without any modifications. The surface quality might be unsatisfactory, or additional property id's (PID) are needed. The CAD geometries were set up based on the in-house guidelines provided by the aerodynamic department of VCC. This was done to keep the complete CFD workflow in-line with how typical aerodynamic CFD simulations are performed within the company. As mentioned earlier, 30 smaller groups make up a complete vehicle model. ANSA, the pre processing software developed by BETA CAE Systems is the main tool used at Volvo Car to prepare geometric models for further evaluations in a digital simulation environment. Each of these smaller groups were imported in ANSA to further categorize them based on different requirements like the number of prism layers that were going to be used. In other words, what would be the apt y+ value for those surfaces. This was done to assign different mesh settings to surfaces, based on the significance of the surface or part with respect to external aerodynamic flow. The author selected three subgroups :

- Significant surface: this comprised of external surfaces that would come in contact with air flow and dust particles. This included parts like exterior panels of the doors, roof, the external hood. Here, a low y+ value is required along with multiple prism layers.
- Less significant surface: these are surfaces that are interior surfaces where the air flow is not of interest for this thesis. Surfaces such as the engine bay, wheel hubs. Here a few prism layers are sufficient taking into consideration the quality(low y+) of the mesh.
- Thesis specific surfaces: these are surfaces in the cavity regions. Surfaces where it is most likely for dust or contaminants to accumulate i.e. cavities enclosing all the four doors and the tailgate. Here, like the significant surfaces, a sufficiently low y+ values along with multiple prism layers are required. This category was created to accommodate the additional mesh settings that would be required in the simulation

Various approaches exists to capture the physics behind rotating wheels in CFD simulations. Moving reference frame (MRF) was used for this study. The user can set a rotating coordinate system on specific surfaces in this approach instead of rotating the wheels itself. In Fig.3.9 the closed volumes between the rims and the spokes are know as the MRF zones. The importance of correctly modelling the rotation of the wheels is to appropriately capture the self-induced contamination effect of the wheels as mentioned in the theory section.



Figure 3.9: Wheels without and with MRF zones. Air is rotated in these zones instead of rotating the wheels itself.

3.3.2 Computational Domain

The size of the computational domain in CFD is significant to avoid unnecessary computational time or influence of the domain walls itself. The vehicle length is approximately 5.0 m, further denoted as L, and is placed 4L downstream of the inlet of the domain. The overall length of the domain is 14L, hence 2/3 of the domain is downstream of the vehicle. The frontal area of the vehicle, A_v , equals 2.3 m^2 , and the cross sectional area of the domain, A_{cs} , equals 1200 m^2 . The ratio between the two, commonly known as the blockage ratio, is used as an indicator to ensure that the computational domain is sufficiently large in relation to the object of interest, thus not affected by the presence of applied boundary conditions. To reduce overall grid size, thus reducing the computational cost, mesh refinement regions are used. By applying this meshing strategy, grid size settings of the domain and the different refinement regions can be individually controlled. The refinement regions are added to accurately resolve the fluid flow around the vehicle, but also to properly capture the downstream wake region. An additional aspect of using refinement region is the have a smoother transition in the volume mesh. Illustrated in Fig. X, the vehicle in true relation to the domain and the refinement regions are seen. The reason of having a sufficiently large domain is to avoid interference with used boundary conditions. A summary of the dimensions of the different regions are found in Tab. 3.3.

Table 3.3: Domain and mesh refinement sizes

Object	Length (m)	Width (m)	Height (m)
Small	11.0	3.5	2.5
Medium	18.0	5.0	3.5
Large	35.0	10.0	6.0
Full domain	70.0	40.0	30.0



Figure 3.10: Computational domain with mesh refinement areas

3.3.3 Numerical Grid and Surface meshing

To numerically solve a problem using CFD, a computational grid is a must. Different techniques have been developed throughout the years and most often are there multiple approaches available. To essentially build your computational domain, the geometry surface of interest needs to be defined. Commonly this is known as the surface mesh or surface grid. A high-quality surface mesh is essential for a successful CFD simulation. To establish a high-quality surface mesh, a technique commonly known as surface wrapping was used. A surface wrapper is suitable to be used when imported CAD geometry quality is low, or when having an overly detailed geometry to complex to manually correct for errors. As one can imagine, generating a surface mesh to a complete vehicle ends up in the category of an overly detailed geometry. A surface wrapper works, essentially, in the way of creating a skin around the geometry, hence a key benefit of using a surface wrapper is that the geometry becomes watertight, i.e. no volume mesh will be created within the geometry itself. However, the surface quality of a surface wrapper itself may be insufficient and additional work to improve the quality as mentioned in the next section is needed.

To establish a high-detailed geometry, the majority of the vehicle was wrapped with a minimum target mesh size of 1.0 mm. To prevent bridging between cells in sharp corners or angles, a technique called contact prevention was utilised. Furthermore, the surface mesh created by the surface wrapper is not the actual surface mesh, but a rather way of creating a watertight geometry that is suitable for surface meshing. The new surfaces created by the surface wrapper were used to perform the surface mesh. As for the surface mesh, the minimum target mesh size spread between the interval of 1.0 mm to 4.0 mm depending on the location and relevance of specific surfaces. Typically, all external surface were meshed with a size of 1.0 mm while surfaces within the engine bay were meshed with 4.0 mm.

Surface meshing

All simulations were carried out using the commercial CFD solver STAR-CCM+ developed by CD-adapco who in 2016 was acquired by Siemens. Star CCM+ is the main tool used at Volvo Car to run CFD simulations. The in built mesher are used to create the surface and volume mesh. A major part of this thesis was to get a surface mesh that resolved the cavities of interest in a satisfactory manner as this would affect the volume mesh generated and consequently the simulations. Through previous works it was observed that the cavities at the door were not resolved well by the in built surface wrapper in Star CCM+. The wrapper was capable to resolve the tailgate cavities but would close the cavities at the door, inspite of having the appropriate mesh settings. Traditionally during an external aero simulation the wrapper closes all the cavities as it is not the area of interest. However, for this thesis it was an important region and depended on it. The settings in Star CCM+ that handled these were :

- Gap closure sizes
- Contact prevention
- Seed points

The surfaces at the edges of the door panels were further split to allow for the right settings to be applied in order to resolve the cavities and avoid the closure of the door cavities. Star CCM+ allows the user to set up custom controls at different surfaces in order to manipulate the mesh settings at those surfaces. In fig.3.11, the changes that were made to the surfaces of the front fender and the door (blue surfaces) in order to resolve the door cavities.



Figure 3.11: Changes in the surfaces of the door and fender to resolve door cavities

This allowed the cavities to be resolved well and remained open. However, the consequences of this were that the surface wrap had holes in them and would thus leak in to the cabin and create a mesh in the cabin area as well. This created a surface mesh with a huge number of cells that would be computationally expensive. The specific options and controls used during this method are as follows: Surface Repair

1. Split the edges of fender panel and door panel into two new surfaces. Hence, created one new surface in the fender cavity and one new surface in the door cavity.

Default controls

- 2. Set Gap Closure Method to Both
- 3. Enable Use Enhanced Size-Based Method
- 4. Set Default Control, Gap Closure Size to 20mm

Surface Controls

- 5. Surface Control _cavity set Target Size, Minimum Size and Gap Closure Size to 2mm and Surface Curvature at 200.
- 6. a new Surface Control and add the two new Surfaces created (the edge of the panel and the door). Set Target Size and Minimum Size to 1mm, and Gap Closure Size to 0.5mm.

Contact prevention

7. Create new contact prevention on the two new surfaces created from Surface Repair. Minimum Size = $1e^{-4}m$

Thinking these settings would be sufficient the surface mesh was generated. Unfortunately, this did not help either and a lot of holes were created in the wrap. In fig. 3.12 & 3.13 the defects in the surface wrap can be visualised.



Figure 3.12: Holes in the surface wrap created in StarCCM+

During the time of the thesis the author contacted Siemens regarding this issue and was notified that an underlying bug in the software was causing this problem and this was being fixed by the development team at Siemens. As a workaround the surface wrap was exported and ANSA was used to manually patch the holes in the cabin wrap. Furthermore, this was a time consuming and laborious method to crate a surface mesh.

This underlying bug was finally fixed by Siemens during the end period of this thesis. Along with this fix, a few minor tweaks in the surface wrap controls and a manual fill up of few holes in the CAD surfaces were required to resolve this issue. Eventually, a satisfactory surface mesh was generated that could resolve the door cavities in the right manner without increasing the computational load.



Figure 3.13: Defects and overlaps of various surfaces created during surface wrapping in StarCCM+

Final Numerical grid

Illustrated in Fig. 3.14 and 3.15 is an overview of the volume mesh. The strategy regarding the volume mesh was in-line with standard procedures for road vehicle aerodynamics where emphasis is put into wake refinement regions.



Figure 3.14: Volume mesh of complete domain



Figure 3.15: Close up of volume mesh to show different refinement regions

In order to resolve the wall bounded flow, thus capturing the near-wall effects, the implementations of prism layers are essential. To achieve appropriate y+ values, thus a suitable first cell height, prism layers were used throughout the vehicle. As for external surfaces, a low y+ value, less or equal to unity, was targeted to properly resolve the boundary layer. However, a common problem in CFD is the question of how many prism layers are necessary to use to fulfil said criterion's. Using too many will dramatically increase the computational cost and the complexity of the volume mesh, using too few and the boundary layer might not be resolved properly, thus losing overall accuracy of the simulation. However, as seen in Fig. 2, a y+ value in the interval of 5 to 30 is unwanted due to the problematic modeling necessary. For all external surfaces, 12 prism layers were used. For lesser significant surfaces, such as surface in the engine bay and wheel houses, one prism layer was used, hence a high y+ value. Total grid count (including surface mesh) was approximately 300M, and a volume meshing time of 6 h using a parallel configuration of 60 cores at Volvo Car dedicated meshing server.



Figure 3.16: Illustration to show different prism layer settings. The right is a picture of the engine bay and the left is a picture of the sill moulding region

3.3.4 Boundary Conditions

The inlet domain was and outlet of the domain was set to velocity and pressure, respectively. As for the inlet, a velocity of 38.88 m/s (140 km/h) was used following standard procedures for external aerodynamic simulations at Volvo Car. Additionally, turbulent intensity of 0.1 % was added to the inlet of the domain as it corresponds

to the magnitude found in the test section of the wind tunnel. Furthermore, the viscosity of the fluid used in the simulations was based on the temperature of the air in the wind tunnel. The temperature was measured to 21°C and is the normal working temperature of the wind tunnel at Volvo Car. In Tab. 3.4, used boundary conditions are summarized.

Region	Boundary Condition
Inlet	Velocity Inlet
Outlet	Pressure Outlet
Ground	Moving wall
Wheel and rims	Rotating wall
Vehicle surfaces	No-slip wall
Domain sides and top	No-slip wall
Air between rim and spokes	Moving reference frame (MRF)

Table 3.4: Boundary Conditions

3.3.5 Solver settings

Simulations were carried out using the unsteady IDDES turbulence model. A pressure-based solver was chosen due to the typical velocities of road vehicles where compressibility effects are negligible. A segregated flow solver was chosen for the pressure-velocity coupling. The segregated flow solver solves the integral conservation equations of mass and momentum in a sequential procedure. For the non-linear governing equations, field variables such as velocity components and pressure, the solutions are solved in an iterative manner. The segregated solver apply a pressure-velocity coupling algorithm to the mass conservation constraint where the velocity field is satisfied by the solution from a pressure-correction equation. A pressure-velocity coupling is achieved when the continuity equation is re-written in terms of the mass flux correction. For the thesis, the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm for the pressure-velocity coupling is used. As several inner iterations are needed for each time-step, SIMPLE is preferred due to its capabilities of quickly computing iterations compared to e.g. a coupled scheme. Furthermore, the convective flux term in Eq. 2.15 requires a value of the fluid property ϕ at the face of an element. The accuracy and stability of the computed fluid property is particularly dependent of the chosen numerical scheme. For DES, hence IDDES, a hybrid second-order upwind/bounded central scheme was recommended to be used. [24] In addition to the fluid properties, spatial gradients of said properties, e.g. pressure gradients, are required. For these computations, a hybrid Gauss/Least-Squares method was used. To keep calculated gradients within reasonable bounds, the gradient limiter Venkatakrishnan was used. [28] The limiter uses the minimum and maximum values of neighbouring cells to limit the reconstructed gradients. In Tab. 3.5, used solver and discretisation schemes are summarized.

Table 3.5: Solver and discretisation schemes

Equation	Discretisation
p-v coupling	SIMPLE
Spatial gradients	Hybrid Gauss-LSQ
Pressure	Standard
Momentum	2^{nd} Order Upwind
k, ω	2^{nd} Order Upwind

As the nature of IDDES is complex and unsteady, a steady-state initialization using RANS will be used. By applying this technique, the robustness of the unsteady simulation is increased and the problem mathematically well posed hence the consistent initial guesses provided by RANS. As a standard procedure with using a DES based turbulence model in STAR-CCM+ [24], a steady-state simulation is applied to the initial 1500 iterations. Subsequently, the simulation transitions to an unsteady approach using a ramping time-step. Illustrated in Fig. 3.17 is the physical time plotted against iteration. From the plot it is possible to observe that multiple time steps are used due to the distinctive difference in gradients throughout the simulation. In other words, a larger time step is initially used and gradually made smaller as the simulation will proceed. For the first 400 iterations, corresponding to 0.1 s of physical time of the transient simulation, a time step of 5E-3 s is used along with 20 inner iterations. Furthermore, same time step size is kept until 2.0 s of physical time, but with a reduction by half of the amount of inner iterations. As for the interval of 2.0 s to 2.4 s of physical time, the time step is reduced to 1E-3 with 8 inner iterations. Finally, a time step of 2.5E-4 s with 6 inner iterations is kept until 3.7 s of physical time is reached at 40 000 iterations.



Figure 3.17: Ramping time step to show Physical time vs number of iterations

4 Results

The first and foremost aim for this thesis was to set up a workflow that would lead to a successful investigation of the physics behind the dirt/dust intrusions in the door cavities. Thus, an initial proper aerodynamic setup that also captured the air flow in these cavities was required. Initially, the need to solve the known difficulties to resolve the cavities and have a fully functional numerical grid that resolved the cavities well were fundamental for the progress of the project.

4.1 Numerical Grid and CFD results

4.1.1 Numerical Grid

Arriving at a volume mesh (or numerical grid) that resolved the cavities in a satisfactory manner was vital for this project. This section illustrates the the surface wrap, volumetric mesh along with the cavities at the doors.



Figure 4.1: Surface wrap that captures the cavities well at the door. The right image shows the features captured on the chassis side in the cavity

The above figure illustrates the different mesh settings that were used in order to create a surface wrap that would resolve the cavities in the right manner. Once this surface mesh was achieved without any holes that would create a grid in the interior of the cabin, the volume mesh was created to eventually arrive at a satisfactory numerical grid. In fig 4.2 & 4.3. the resultant volumetric mesh is illustrated.



Figure 4.2: Final Volumetric mesh



(b) Close up of the cavity resolved by the volumetric mesh.Figure 4.3: Snapshots of Final Volumetric Mesh



(a) Top view of the volumetric mesh at A pillar.



(b) Close up of the cavity at the intersection of the front fender, front door and sill moulding resolved by the volumetric mesh.



Figures 4.4a & 4.4b depict the detailed view of the volumetric mesh at the A pillar cavity. Typically, in an aero methodology these cavities would be closed.

Part	Element Count
Air stream	2.691070e + 08 cells
Car	$3.288252e{+}07$ faces
Ground	$4.912670e{+}05$ faces
Inlet	1.200000e+03 faces
Outlet	1.200000e+03 faces
Symmetry	6.800000e+03 faces

Table 4.1: Details of resultant volumetric mesh

The above table 4.1 gives the specific number of cells and faces of each part in the final volumetric mesh. Air stream is the complete volumetric mesh which includes all the subsequent parts like the domain and the car.

4.1.2 CFD results

In order to resolve the issues faced to prepare the geometry and subsequently surface wrap, which are important steps to set up the CFD workflow, many man weeks were spent. There are certain issues that remain unresolved as mentioned in the discussion section. Hence, running the simulations and thus achieving the results from the CFD simulations was unfortunately not possible within the time frame of this work. Since a CFD workflow is now in place, future projects can look into this along with support from Siemens. Therefore, the CFD results will not be presented in this thesis report.

4.2 Experimental Study

4.2.1 Wind tunnel test results

The data of the results presented below are from a wind tunnel test conducted during a previous master's thesis study. These results were not presented before as the scope of that project and this project are different. The previous thesis project was targeted to analyse flow in the tailgate, for further information the reader can go through the thesis report cited here. [3]

Initial Test

Investigating flow properties in door cavities is not a typical case, it is rather uncharted territory, thus a proof of concept was required to check whether the existing equipment was sufficient enough to measure the pressure in the cavities. This initial test of a simplified aerodynamic wind tunnel test was done with this purpose and hence was kept simple by using the available facilities i.e. the full scale Volvo wind tunnel, physical car and pressure sensors. The set up time in the wind tunnel was kept to a minimum by not using rotating wheel and any force measurement equipment. 20 pressure sensors were placed at different locations in the cavities around the vehicle. These were placed on the right hand side of the vehicle for this initial test, under the assumption of symmetry. Fig. 4.5 illustrates the locations of these sensors on the door cavities.



Figure 4.5: The points highlight the location of sensors on the right hand side door cavities in the initial wind tunnel test. [3]

Each of these sensors were connected using a yellow cable to the wind tunnel measurement equipment. These cables were arranged with caution along the edges of the inner cavity sealing so that they do not have an effect on the air flow in the cavities. The main concern was if the sensors would be able to measure the pressure differences due to the unconventional approach, and the squeezed location for some sensors. Importance was paid to whether a general trend could be captured rather than absolute pressure values. Fig. 4.6 are test results of three identical tests not performed consecutively. These initial results seems to be reliable and consistent measurement data which deemed the method possible and gave a proof of concept. The aerodynamic test was conducted at a wind velocity of 140kph and zero degree yaw angle.

Complete test

After a successful proof of concept, a new test session in the wind tunnel was performed according to Volvo Car aerodynamic testing routines, which meant rotating wheels were used and all aerodynamic loads and moments were measured. The ride height of the vehicle was accurately adjusted based on the CFD simulation setup to maintain an accurate similarity amongst the two. Various test cases were run in the wind tunnel where a typical start point was 50 kph wind velocity and increased to a velocity of 140 kph in steps of 5 kph. In order to get stable measurements each step was maintained for 1 minute. Tests were conducted to see dependence of pressure coefficient values on rotating wheels. Fig. 4.7 shows a pictorial summary of the different test conducted.



Figure 4.6: C_p measured by 20 sensors during three identical wind tunnel tests at 140kph wind velocity. [3]



Figure 4.7: C_p measured by 20 sensors around the door cavities for tests at 140kph with or without rotating wheels. [3]

4.2.2 On road driving test results



Figure 4.8: The dust pattern observed after the driving test, the vehicle was completely clean before the test. The area above the red line is dry dust and the area below the red line is wet due to the water picked up by the wheel wake from the wet roads.

This section will illustrate the different results that were obtained during the on road driving test. In spite of these tests being conducted on random roads and very low scope of repeatability, some interesting trends were observed during the road tests. In an ideal scenario the vehicle would be tested in a desert region that would assure a particular particle size distribution.

Test1

The results shown below are a combination of both tests conducted on Road1 [3.1]. As mentioned earlier this test was conducted during rainy weather conditions. Hence a combination of dust and water droplets can be observed in the images below. A common trend was observed that the contaminants would intrude at the intersection of the front fender, front door and the sill moulding. In Fig. 4.9 this can be visualised. This region is the front wheel wake region.

Another interesting region is the cavity at B pillar. One would expect the split line sealing to be sufficient to avoid dust intrusion. However dust particles were observed which could be due to the intrusion from wheel wake region. The dust particles may have been carried along by the air flow. This is depicted in Fig. 4.10 where dust particles are also observed at the top region of the door cavity. This similar trend does not continue towards the C pillar cavity.

$\mathbf{Test2}$

This section is to illustrate the dust patterns observed when driving on Road2 [3.2]. The weather conditions was dry but the road was wet. A similar trend of dust intrusion pattern was observed during this test as well. The front wheel wake carried minute dust particles into the A pillar cavity or the cavity region at the intersection of the front fender, front door and the lower sill moulding, as observed in 4.11

The trend at the B pillar cavity can also be observed in this test, in Fig. 4.12. The C pillar cavity had no deposits of dust on it.



Figure 4.9: Intersection region of Front fender, front door and sill moulding

Figure 4.13 gives an external visual description of the various split lines cavities described in this section.



Figure 4.10: Dust pattern observed at the B pillar cavity between the front door and back door



Figure 4.11: Dust pattern observed at the split line at the A pillar



Figure 4.12: Similar dust pattern observed at the B pillar cavity during test on Road2.



Figure 4.13: Different split lines where dust intrusions were observed

5 Discussion

To analyse and weigh experimental data and simulation results keeping in mind the objectives, it is important to provide unbiased conclusions. Following is a discussion of the different methods and strategies used throughout the thesis.

5.1 Virtual assessments

In this section the CFD workflow is discussed as well as the geometry preparation, numerical grid and the computational domain required for the CFD simulations.

5.1.1 Geometry, numerical grid and Computational domain

In order to get reliable results from the CFD simulations, the virtual and physical models need to be as close as possible. However, this can be computationally expensive, hence a trade off is made to maintain the closeness between the models and the computational cost. Some simplifications are possible based on the application, for e.g. in this case the inlets were closed off to reduce computational cost. In an ideal world, the CAD surface geometry would be of very high quality and perfectly resolved using extremely small surface elements to capture all details. Nevertheless, for industrial applications where time and computational resources are most often limited, a perfect geometry is not always achieved. For this project, in which, small volumes with a narrow inlet and outlet (as wide as the split line) when compared to the complete vehicle had to be resolved, the actual size of the surface mesh elements were going to be very small. This was tricky as making the mesh too fine at the cavities of interest. During the initial phases of the thesis, this trade off was hard to achieve as the cavities were being closed off in spite of having a fine mesh. The geometry had to be manually prepared in a way to control the mesh sizes at specific regions (edges of the cavities). Multiple man weeks were spent to prepare the geometry as the in built surface wrapper could not differentiate between unwanted holes in the geometry that would typically be closed off and the cavities that were required to be resolved.

One of the major setbacks during this project was this issue of resolving the cavities while closing of unwanted holes in the geometry, such that the wrapper does not leak in the cabin and create a large number of cells, which were not required to capture the external flow dynamics. Since the objective was to resolve all the cavities around the complete vehicle and with the limitation of the surface wrapper technique being able to use only one CPU to compute the mesh, i.e. the meshing would be performed in serial. Moreover, after closing off many holes manually, the wrapping technique would still mesh the inside of the cabin and thus create a large grid size that the software and/or the computer could not handle. Additionally, on getting support from Siemens, it was pointed out that a bug existed in the software that was causing the failure in the surface wrapping technique. Unfortunately, this was not resolved within the time of the thesis. Furthermore, other methods of meshing could be explored for e.g. using a different software, using only half of the geometry by creating a symmetrical plane along the z-axis. These methods were not fully explored due to limitations of the thesis and the complexity of the complete vehicle.

In the final phase of the thesis, a wrapped surface that could resolve the cavities in an appropriate manner was achieved along with the support of Siemens. However, an issue was faced when re-meshing the surface and the subsequent steps of volume meshing. The software was not able to handle the grid size and would eventually run out of memory. Thus, a trade off was not feasible between the computational size and resolving the cavities.

The size of the computational domain is a typical size used during an external aerodynamic simulation used at Volvo Car to evaluate drag and lift forces. Mesh refinement regions used are targeted for the wake regions downstream of the wheels, side mirrors and the complete vehicle to achieve a smooth transition of the volume mesh to ease the complexity of the computation. An argument can be made that this size of the domain is larger than required as the key objective of this thesis is not to estimate the drag. On one hand this could be a valid argument. On the other hand, the domain size was maintained due to the amount of turbulence created downstream of the vehicle and the cavity of interest was affected by this as there would be an interaction of the wheel wake with the tail wake. A strong standpoint to do so was the available computational power, and the computational cost related to the scale-resolved simulations. The cluster had an availability of 1920 cores and similar prior tests were conducted that amounted up to approximately 30 to 35 hours, equaling to a computational cost of 70, 000 CPU hours. [3] This method was maintained to resolve all the cavities in an appropriate way and evaluate the flow dynamics in the cavities. Additionally, an existing workflow is in place for external aerodynamics. Once the numerical grid is resolved one can evaluate the mesh dependency and reduction in computational cost should be considered.

5.1.2 CFD workflow

Evaluation of flows in the door cavities is distant from standard procedure at Volvo Car, and little to no previous studies or literature is available, the industrial supervisor for this project recommended to follow the existing procedures for aerodynamic CFD simulations within the company. Hence, the IDDES turbulence model was chosen for this study. This model is extensively used by Volvo Car and the in house method is constantly updated and developed based on the current software developments and turbulence models. The objective of this thesis was not to evaluate the appropriateness of the chosen turbulence model. However, as the cavities of interest are located close to wake regions (wheel wake to be precise) and are affected by the chaotic and irregular behavior of the turbulence from the downstream wake, one can recognize the importance of utilizing a time resolving model. In a real world scenario the flow is unsteady, hence using a steady turbulence model would not be sufficient to accurately predict the flow around the cavities.

A key note observed during the span of the thesis is the lack of connectivity when the surface is wrapped externally and an attempt is made to accommodate this is in the existing aerodynamic method to create a new method for evaluation of contaminants. On finally achieving an appropriate surface wrap in the final phase of the thesis, the inclusion of this wrap into the existing setup was leading to new issues, unfortunately due to the time constraints of the study it was not possible to evaluate the cause of this. For further studies, it is recommended to explore the idea to deviate from the existing standard procedure or even create a whole new procedure to evaluate the flow in cavities. The current numerical method is developed for road vehicle external aerodynamics simulations based on the general recommendations of StarCCM+ and can accurately predict the aerodynamic forces, it can be argued that the approach of using the similar method to evaluate the internal flow in cavities was not the best decision. Nevertheless, the upside of sticking to this method was to have a basis for comparison and validation since evaluation of flow in cavities is not a conventional procedure.

5.2 Experimental Method

Experimental data is used as validation for the CFD simulations to capture the flow in cavities. Hence it is worthy to discuss the reliability of the experimental methods used during the thesis.

5.2.1 Wind tunnel tests

The method adopted was to measure pressure in the cavities using pressure sensors. The measurement equipment used during the tests to measure the pressure have previously been used in various other projects mostly to measure pressure on external surfaces and provided reliable, consistent data. The uncertainty of achieving reliable data for the internal flow in cavities was cleared during the initial wind tunnel tests.

The location of the pressure sensors were purely based on intuition and suggestions from experienced employees at the department. It was observed that the sensors were placed only on one side of the vehicle. It could be interesting to evaluate whether a difference existed in the data if pressure was measured on both sides of the vehicle. Additionally, precise locations of the sensors should be achieved in order to have a reliable comparison with CFD results. Various methods for e.g. digital position measuring devices can be used to provide precise global xyz-coordinates in order to increase reliability n the wind tunnel measurements. Additionally, different experimental methods can be used for future studies in order to visualise the flow in cavities. Firstly, flow-vis fluid (commonly known as surface oil) could be used in treating split lines and in-cavity surfaces to visualise on surface flow and gain information regarding the entry and exit of air in the cavities. Secondly, smoke and tuft visualization can be used to detect the vortices around the wheels and rear wake to provide diagnostic information about air flow around the cavities. Furthermore, one can use other flow visualization techniques based on the constrains of the project.

5.2.2 On road driving tests

One of the objectives for this thesis was to be able to pin point certain trends followed by dust particles in the

door cavities. The on road tests are ideally conducted in a confined environment, where the conditions can be easily recreated. However, due to constrains of the thesis this was not possible. The on road tests were instead conducted on a road that was selected based on sheer intuition and recommendations from experienced employees of Volvo Car Contamination Strategy and Implementation (CSI) team. It can be argued that hence due to the randomness of the roads chosen and little to no scope of repeatibility of the tests, the data can not be relied upon. However, since the aim was to observe a base trend for the accumulation of dust the results from this method can be used as an initial guess of the pattern. In spite of the randomness of the roads a common trend was observed between two different roads. Hence, this can be used as a basis to design future tests and experiments.

It is definitely recommended to conduct future similar tests on a confined test track. Additionally, during these tests no particular sensors were mounted on the vehicle, the reliability of the tests can be further strengthened by mounting of various measurement devices, for e.g. the pressure sensors used during the wind tunnel tests. Standard procedures or expeditions (in Arizona for dust) are conducted in Volvo Car to test the contamination in various regions of the vehicle. Future experiments can be set up in a similar manner to evaluate different types of contaminants (for e.g. snow and water) in the cavities of interest.

6 Conclusion

The objectives of this study were divided into three categories. Firstly, evaluating the possibilities of observing trends for accumulation of dirt/dust particles in door cavities. Secondly, preparing the geometry in order to capture and resolve the door cavities from raw CAD models. Finally, create a CFD workflow to virtually assess the flow field and pressure distribution in the door cavities.

Based on the information presented in the method and result sections, and further discussed in the discussion section, definite indications have been presented on the possibility to observe a common trend of the accumulation of dirt/dust particles by conducting on road tests and gathering supporting pressure measurement data in the wind tunnel tests. This method is not confined only to the selected door cavities but can be used to evaluate any other kind of cavity.

To virtually resolve and capture the cavities of the door from the raw CAD models has been proved to be possible but comes with a substantial amount of manual effort. In-spite of being time consuming, this provides a basis for resolving different cavities around the complete vehicle while maintaining the computational cost.

A virtual assessment is possible based on the CFD workflow set up during this thesis. Analysing the flow properties in the door cavities is a complex problem that requires some improvements in the software used for the analysis before it can be deemed robust enough. Nevertheless, with these improvements in the future it should work out well.

7 Future Work

Evaluation of in-cavity flows using CFD simulations is an unconventional and unexplored area of application, the future possibilities are significant. A standardised method can be set up to evaluate all the cavities in the vehicle while maintaining computational costs. This can be used to analyze different types and models of vehicle, as the flow around each model would be different due to different shapes of the current fleet of Volvo Car vehicles. Improvisation of the current wrapper techniques and fixing of various bugs would drastically reduce the amount of manual work required to prepare the geometry. Furthermore, this could lead to setting up the multiphase simulations, which would include the dust particles, can help evaluate and support physical experiments. These simulations can be extended to include different types of contaminants too. An aerodynamic simulation can definitely provide the basis for these simulations, but would not provide information to the full extent. Improvements in the existing seals can be evaluated in order to reduce the accumulation of dirt/dust particles in order to improve the user experience and brand value of the current fleet. Additionally, different cavities can be evaluated all around the vehicle based on the accumulation of contaminant to determine optimal positions

of various functional devices and sensors such as cameras, LIDARs or RADARs.

References

- [1] B. Andersson et al. *Computational Fluid Dynamics for Engineers*. Cambridge University Press, 2012. ISBN: 978-1-107-01895-2.
- [2] H. Braun. Neue Erkenntnisse über Radabdeckungen. Düsseldorf: VDI Verlag, 1972.
- [3] A. Brohmé. "Assessment of Flow Properties in Door Cavities by Means of Experiments and CFD". Master's Thesis. Linköpings University, 2020.
- B. Cabral and L. C. Leedom. "Imaging vector fields using line integral convolution". Proceedings of the 20th annual conference on Computer graphics and interactive techniques. SIGGRAPH '93. Anaheim, CA: ACM, 1993, pp. 263-270. ISBN: 0-89791-601-8. DOI: 10.1145/166117.166151. URL: http://doi.acm. org/10.1145/166117.166151.
- [5] R. Clarke. "Heavy Truck Splash and Spray Suppression: Near and Long Term Solutions". Aug. 1983. DOI: 10.4271/831178.
- C. Crowe et al. Multiphase Flows with Droplets and Particles. CRC Press, 2011. ISBN: 9781439840511.
 URL: https://books.google.se/books?id=MmnRBQAAQBAJ.
- [7] L. Davidson. Large Eddy Simulations: How to evaluate resolution. International Journal of Heat and Fluid Flow 30.5 (2009), 1016–1025.
- [8] K. Hanjalic et al. A robust near-wall elliptic-relaxation eddy-viscosity turbulence model for CFD. Int. J. Heat Fluid Flow 25 (2004), 1047–1051.
- [9] S. Hoyas and J. Jiménez. Reynolds number effects on the Reynolds-stress budgets in turbulent channels. *Physics of Fluids* 20.10 (2008), 101511. DOI: 10.1063/1.3005862. eprint: https://doi.org/10.1063/1.3005862.
 URL: https://doi.org/10.1063/1.3005862.
- [10] W. Hucho and G. Sovran. Aerodynamics of road vehicles. Annual review of fluid mechanics 25.1 (1993), 458–537.
- [11] W.-H. Hucho. Aerodynamics of road vehicles : from fluid mechanics to vehicle engineering. British Library Cataloguing in Publication Data, 1987. ISBN: 0-408-01422-9.
- [12] A. Iserles. A First Course in the Numerical Analysis of Differential Equations. Cambridge University Press, 2004. ISBN: 0-521-55655-4.
- [14] W. P. Jones and B. Launder. The prediction of laminarization with a two-equation model of turbulence. International Journal of Heat and Mass Transfer 15 (1972), 301–314.
- [15] P. Koeßler. Kotflügeluntersuchungen. Düsseldorf: VDI Verlag, 1965.
- [17] F. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal 32 (1994), 1598–1605.
- [18] F. Menter and M. Kuntz. "Adaptation of Eddy-Viscosity Turbulence Models to Unsteady Separated Flow Behind Vehicles". Springer, 2004, pp. 339–352.
- [19] F. Menter, M. Kuntz, and R. Langtry. "Ten Years of Industrial Experience with the SST Turbulence Model". 2003.
- [20] J. G. Olivier, K. Schure, and J. Peters. Trends in global CO₂ and total greenhouse gas emissions:2017 Report (2017).
- [21] M. Pfadenhauer, G. Wickern, and K. Zwicker. On the Influence of Wheels and Tyres on the Aerodynamic Drag of Vehicles (1996).
- [23] M. Shur et al. A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities. International Journal of Heat and Fluid Flow 29 (2008), 1638–1649.
- [24] Simcenter STAR-CCM+ Release Notes 2019.3. StarCCM+ Theory Guide. Siemens, 2020.
- [25] P. R. Spalart et al. "Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach, International conference; 1st, Advances in DNS/LES: Direct numerical simulation and large eddy simulation". Advances in DNS/LES: Direct numerical simulation and large eddy simulation, International conference; 1st, Advances in DNS/LES: Direct numerical simulation and large eddy simulation. Greyden Press; 1997, pp. 137–148. ISBN: 1570743657.
- [26] J. Sternéus, T. Walker, and T. Bender. Upgrade of the Volvo cars aerodynamic wind tunnel. SAE Transactions SP-2066 (2007), 1089–1099.
- [27] M. Strelets. "Detached Eddy Simulation of Massively Separated Flows". AIAA Paper 2001-0879. 2001, pp. 1–18.
- [28] V. Venkatakrishnan. On the convergence of limiters and convergence to steady state solutions (1994).

- [29] H. K. Versteeg and W. Malalasekera. An introduction to computational fluid dynamics: the finite volume method. Pearson education, 2007.
- [30] Wikipedia, ed. Airfoil. 2021. URL: https://en.wikipedia.org/wiki/Airfoil.
- [31] D. Wilcox. Formulation of the k-w Turbulence Model Revisited. AIAA Journal 46 (2008), 2823–2838.
- [32] D. C. Wilcox. Trbulence modeling. *DCW Industries* (1993).
- [33] D. C. Wilcox. Turbulence Modeling for CFD Third Edition. DCW Industries, Inc., 2006. ISBN: 978-1-928729-08-2.



DEPARTMENT OF MECHANICS AND MARITIME SCIENCES DIVISION OF VEHICLE ENGINEERING AND AUTONOMOUS SYSTEMS CHALMERS UNIVERSITY OF TECHNOLOGY Gothenburg, Sweden