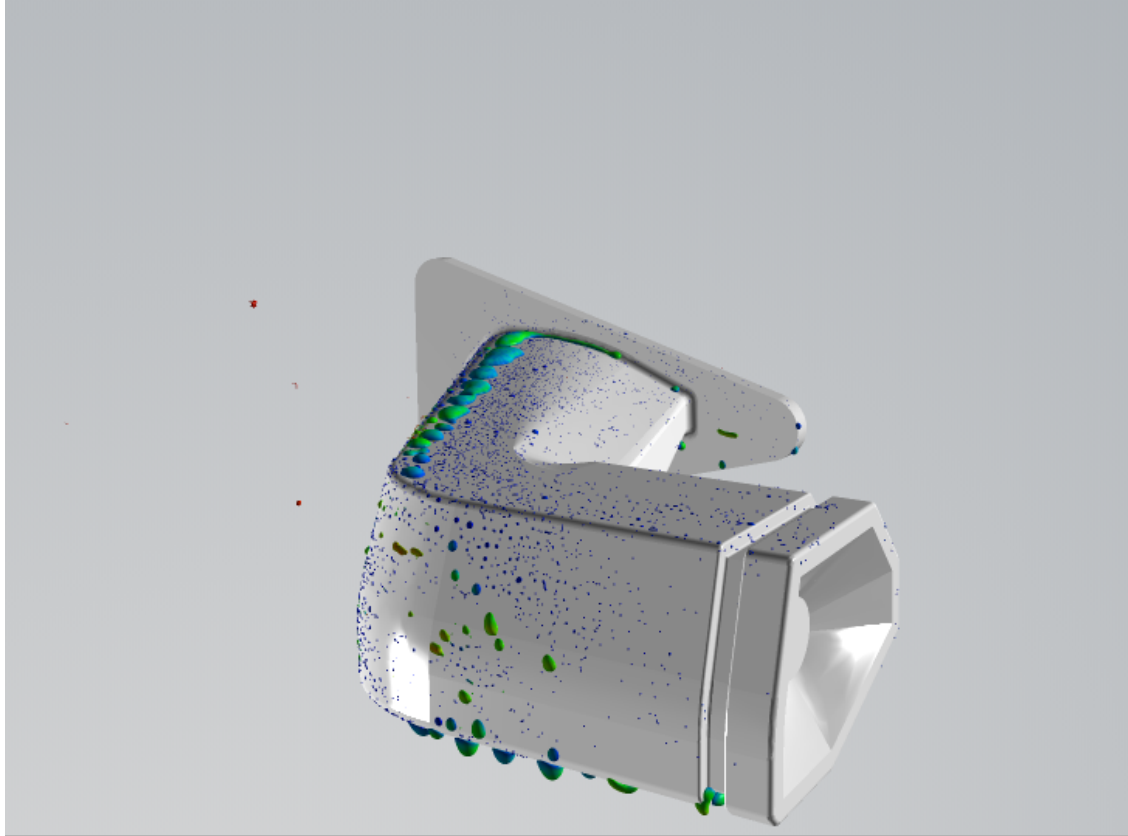




CHALMERS
UNIVERSITY OF TECHNOLOGY



External Water Management and Wetting Phenomenon Using CFD

Investigating the wetting phenomenon and droplet transport on the external surfaces of interest on a car using CFD solvers

Master's thesis in Applied Mechanics

Rutuj Mukund Adhau

DEPARTMENT OF MECHANICS AND MARITIME SCIENCES

CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden 2026

www.chalmers.se

MASTER'S THESIS 2026

External Water Management and Wetting Phenomenon Using CFD

Investigating the wetting phenomenon and droplet transport on the external surfaces of interest on a car using CFD solvers

Rutuj Mukund Adhau



CHALMERS
UNIVERSITY OF TECHNOLOGY

Department of Mechanics and Maritime Sciences

Division of Fluid Dynamics

CHALMERS UNIVERSITY OF TECHNOLOGY

Gothenburg, Sweden 2026

External Water Management and Wetting Phenomenon Using CFD
Investigating the wetting phenomenon and droplet transport on the external surfaces
of interest on a car using CFD solvers
Rutuj Mukund Adhau

© Rutuj Mukund Adhau, 2026.

Supervisor:

Tobias Eidevåg
Technical Expert Sensor Contamination
Volvo Cars

Examiner:

Henrik Ström
Professor
Department of Mechanics and Maritime Sciences
Chalmers University of Technology

Master's Thesis 2026
Department of Mechanics and Maritime Sciences
Division of Fluid Mechanics
Chalmers University of Technology
SE-412 96 Gothenburg
Telephone +46 31 772 1000

Cover: Rendered Frame from AVL PreonLab Simulations for droplet transfer on a
prototype CMS Body

Typeset in L^AT_EX
Printed by Department of Mechanics and Maritime Sciences
Gothenburg, Sweden 2026

Abstract

Advanced Driver Assistance Systems (ADAS) rely heavily on exterior mounted sensors such as cameras, radar and LiDAR to ensure safe and reliable vehicle operation. However, the performance of these sensing systems can be significantly affected by environmental conditions, particularly rain and water droplet accumulation on sensor surfaces. This thesis focuses on developing a methodology to quantitatively evaluate water droplet behavior on a prototype Camera Monitoring System (CMS) using both experimental and numerical approaches. The study combines wind tunnel experiments with computational water droplet transfer simulations to investigate the interaction between airflow and water droplets around the CMS geometry. High speed experimental videos obtained from controlled wind tunnel testing were systematically analyzed to extract quantitative metrics. These metrics served as a validation framework for the numerical simulations and enable a direct comparison between experimental observations and computational predictions.

The numerical methodology employs a hybrid Eulerian Lagrangian approach using Computational Fluid Dynamics (CFD) approach. The external airflow around the CMS is modeled using the Finite Volume Method (FVM), which solves the governing Navier Stokes equations to accurately capture the aerodynamic flow field and pressure distribution surrounding the sensor housing. Based on the computed airflow solution, water droplet transport and surface interaction are simulated using Smoothed Particle Hydrodynamics (SPH), a mesh free Lagrangian particle method well suited for modeling free surface flows, droplet breakup, coalescence, adhesion and detachment phenomena. The outcomes of this work provide a foundation for the future development of predictive design tools aimed at enhancing the reliability, robustness and safety of ADAS sensor systems operating under adverse weather conditions.

Keywords: Advanced Driver Assistance Systems (ADAS), Camera Monitoring System (CMS), Computational Fluid Dynamics (CFD), Finite Volume Method (FVM), Smoothed Particle Hydrodynamics (SPH), Water Droplet Dynamics, Wind Tunnel Testing, Experimental Validation, Multiphase Flow, Sensor Reliability.

Acknowledgements

I would like to sincerely thank Volvo Cars for providing me with the opportunity to undertake this master's thesis project. My special thanks go to Jacob Vikström for entrusting me with this opportunity and supporting the initiation of the work. I am deeply grateful to my supervisor at Volvo Cars, Tobias Eidevåg, for his continuous guidance, encouragement and constructive feedback throughout the project. His technical expertise and support, particularly in relation to cluster computing and HPC workflows, were invaluable to the successful completion of this research. I would also like to extend my appreciation to my examiner, Henrik Ström, at Chalmers University of Technology for his guidance, insights and feedback during the course of the thesis. Furthermore, I would like to thank Wolfgang Schwarz and the AVL PREONLAB team for their continuous assistance, technical support and collaboration throughout the project. Their contributions have been greatly appreciated and have played an important role in the completion of this work.

Rutuj Mukund Adhau, Gothenburg, June 2026

Contents

List of Figures	xiii
List of Tables	1
1 Introduction	2
1.1 Background	2
1.1.1 Camera Monitoring System (CMS)	3
1.2 Objectives	4
1.3 Scope and Limitations	4
1.3.1 The scope of the work includes:	4
1.3.2 Limitations	4
1.4 Societal, Ethical and Ecological Considerations	5
1.4.1 Societal Aspects	5
1.4.2 Ethical Aspects	5
1.4.3 Ecological Aspects	5
2 Theory	6
2.1 Computational Fluid Dynamics (CFD)	6
2.1.1 Finite Volume Method (FVM)	7
2.1.1.1 Boundary Layers	7
2.1.1.2 Wall Models	8
2.1.2 Smoothed Particle Hydrodynamics	8
2.1.2.1 SPH based fluid solvers	9
2.2 SPH vs FVM	10
2.3 Droplet Dynamics	10
2.3.1 Dimensionless Numbers	11
2.3.2 Reynolds number (Re_p):	11
2.3.3 Weber number (We):	11
2.3.4 Ohnesorge number (Oh):	11
2.3.5 Forces Acting on a Droplet	12
2.3.5.1 Aerodynamic Drag Force	12
2.3.5.2 Gravitational and Buoyancy Forces	12
2.3.5.3 Surface Tension Force	12
2.3.5.4 Adhesion Force from a Solid Surface	12
2.3.5.5 Lift Force	13
2.3.6 Drag Coefficient Models	13

2.3.6.1	Stokes Drag (Creeping Flow)	13
2.3.6.2	Schiller Naumann Correlation	13
2.3.6.3	Liu Dynamic Drag Correction for Deformed Droplets	13
2.3.6.4	Summary of Drag Correlations	14
2.3.7	Droplet Inertia and Particle Response Time	14
2.3.7.1	Stokes Response Time	14
2.3.7.2	Corrected Response Time (Finite Re_p)	14
2.3.7.3	Response Time with Liu Dynamic Drag	15
2.3.7.4	Physical Interpretation	15
2.3.8	Stokes Number	15
2.3.9	Droplet Deformation: Taylor Analogy Breakup (TAB) Model	15
2.3.9.0.1	Physical interpretation of terms:	16
2.3.9.0.2	Breakup criterion:	16
2.3.9.0.3	Equilibrium deformation:	16
2.3.10	Droplet Deformation Prior to Breakup	16
2.3.11	Droplet Breakup Regimes	16
2.3.12	Force Balance for Droplet Detachment from a Surface	17
3	Methods	18
3.1	Geometry	18
3.1.1	Wedgeoid	18
3.1.2	CMS	19
3.2	Wind Tunnel Setup	20
3.3	Extraction of Measurable Metrics	20
3.3.1	Velocity Data	21
3.4	Aerodynamic Simulations of the Wind Tunnel	22
3.4.1	Computational Model	22
3.4.2	Physics Models	23
3.4.3	Boundaries	23
3.5	Droplet Transport Model	24
3.5.1	Particle Size	24
3.5.2	Sessile Droplet on a Plate	25
3.5.3	Volume Flow Rate	26
3.5.3.1	Preon Solver	26
3.5.3.2	Scene Setup	27
3.5.3.3	Droplet Injection Upstream of the CMS	27
4	Results	29
4.1	Quantitative Metrics for Validation	29
4.1.1	Contour Plots	29
4.1.2	Box Plots	30
4.1.3	Trajectory Analysis	32
4.1.4	Comparison between 70 km/h and 120 km/h Cases	32
4.2	Droplet Transport Simulations	34
4.2.1	High y^+ and Low y^+ setup	34
4.2.1.1	Tuning of Lateral Adhesion and Roughness Factors	36
4.2.2	Validation	38

4.2.3	Steady State Versus Transient Velocity Fields	39
5	Conclusion	42
	Bibliography	45
A	Appendix	I
A.1	Detailed Procedure for Droplet Detection, Tracking and Metric Ex- traction	I
A.1.1	Video Preparation	I
A.1.2	Droplet Detection Using the Hessian Detector	I
A.1.3	Droplet Tracking	I
A.1.4	Export of Tracking Data	II
A.1.5	Velocity Calculation	II
A.1.6	Extraction of Additional Metrics	III
A.2	Particle Response Time Estimation	III
A.2.1	Problem Setup and Parameters	III
A.2.2	Governing Equations	III
A.2.3	Schiller Naumann correlation [27]:	IV
A.2.4	Liu et al. (1993) deformed droplet model [19]:	IV
A.2.5	Particle Response Time	IV
A.2.6	Results	IV
A.3	Adhesion Factor vs Contact Angle	V

List of Figures

1.1	Typical functional ranges of the sensors present in an ADAS [5]	2
1.2	CMS prototype, from CAD used in this thesis	3
3.1	Experimental Setup in Volvo Cars wind tunnel	18
3.2	CAD Model in STARCCM+	19
3.3	CMS prototype with the basic groove, Baseline design	19
3.4	Tracks for droplets on the side face for wind speed of 70 kmph	20
3.5	Tracks for droplets on the side face for wind speed of 120 kmph . . .	21
3.6	Volume Mesh for the low y+ setup, 70 kmph case	22
3.7	Computational domain setup in STARCCM+	24
3.8	Generated volume source in AVL PREONLAB scene at t=0	25
3.9	Droplet at rest on a flat plate as seen on AVL PREONLAB scene in- spector. Particle arrangement in the droplet (L) and rendered droplet (R)	26
3.10	Estimating contact angles in AVL PREONLAB scene. Average con- tact angle here is 83.107 deg	26
3.11	Scene Setup in AVL PREONLAB	27
3.12	Droplets injected upstream of the CMS at $t = 20$ ms. Droplet de- formation increases the drag coefficient and enables the droplets to rapidly align with the airflow, resulting in transport towards the CMS. 28	
4.1	Contour plots for some tracks 70 kmph experiment video	30
4.2	Box plots for all tracks identified in FIJI for 70 kmph case	31
4.3	Box plots for all tracks identified in FIJI for 120 kmph case	31
4.4	Median of all velocities accross the tracks for 70 kmph and 120 kmph case	32
4.5	Trajectories of droplets for 70 kmph case	33
4.6	Trajectories of droplets for 120 kmph case	33
4.7	High y+(top) vs low y+(bottom) aero simulation	34
4.8	Low y+ simulation results for 70 kmph. the droplets seem to fall under the influence of gravity rather than following the expected tra- jectories.Behaviouur can be clearly perceived from the first and sixth pic	35
4.9	High y+ simulation results for 70 kmph. the droplets follow the expected trajectories, Behaviouur can be clearly perceived from the first and sixth pic	36

4.10	High y^+ simulation results for 70 kmph. the droplets follow the expected trajectories	37
4.11	Trajectory of droplets for 70 kmph case, high y^+ simulations.	38
4.12	Velocity field comparison for transient and steady state aerodynamic simulations. The transient SST $k-\omega$ DES solution is shown above and the steady state $k-\varepsilon$ solution below.	39
4.13	Distribution of instantaneous droplet velocities obtained from PRE-ONLAB simulations using steady-state and transient CFD velocity fields, together with experimental measurements.	40
4.14	Mean velocity field obtained from the transient SST $k-\omega$ DES simulation.	40
A.1	Plot Showing relation between Contact angles and Adhesion factors obtained from Sessile Droplet on flat plate simulations (3.5.2), for different particle sizes	V

List of Tables

2.1	Summary of drag coefficient correlations for droplets.	14
2.2	Classification of droplet breakup regimes.	17
3.1	Mesh Settings for the low y^+ setup, 70 kmph case	22
3.2	Experimental Setup Parameters in AVL PREONLAB	25
A.1	Physical parameters used for response time estimation.	III
A.2	Comparison of particle response time τ_p for different drag models at $v_f = 70$ km/h.	IV

1

Introduction

1.1 Background

In recent years, the challenge of operating under non ideal driving conditions has become more crucial due to climate change. Rising global temperatures have led to increased and accelerating precipitation levels worldwide, with severe and erratic rainfall and snowfall observed all over the world [1]. Many Nordic countries now experience more frequent extreme weather events, posing substantial challenges for vehicle sensor reliability [2]

Advanced Driving Assistance Systems (ADAS) and autonomous driving (AD) technologies have significantly improved vehicle safety, comfort and driving efficiency. Features such as lane keeping assistance, adaptive cruise control, automatic emergency braking and object detection rely heavily on accurate perception of the vehicle's surroundings. These capabilities are primarily enabled by exterior mounted sensors, including cameras, radar and LiDAR systems, which together form the foundation of modern intelligent vehicles [3, 4]. The integration of multiple sensing modalities allows vehicles to detect, classify and track surrounding objects while supporting increasingly automated driving functions [3].

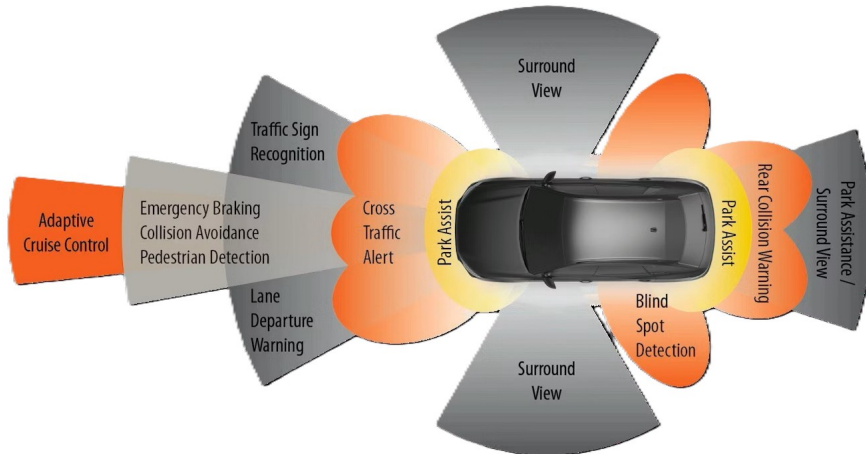


Figure 1.1: Typical functional ranges of the sensors present in an ADAS [5]

While ADAS and autonomous systems perform reliably under ideal operating con-

ditions, their effectiveness is strongly influenced by environmental factors. In real world driving scenarios, vehicles are frequently exposed to adverse weather conditions such as rain, snow and slush, which can contaminate sensor surfaces and degrade perception performance. A camera with a water covered lens or a radar obstructed by snow can significantly reduce the system’s ability to detect objects, thereby affecting both driver assistance and automated decision making. Thus there is a growing need to study and develop methods for accurately modeling water transport on external vehicle surfaces. Such models enable the simulation of rain, snow and water behavior under realistic driving conditions.

To ensure the best design these sensors are extensively tested over a large range of conditions that mimics the actual environment. Traditionally, the assessment of exterior wetting behavior relies on physical wind tunnel rain experiments. While these tests provide high fidelity data, they are costly, time consuming and difficult to apply repeatedly during early design phases. Consequently, the automotive industry is increasingly adopting a *Shift Left* approach, where predictive simulation tools are used earlier in the development process to enable faster design iterations and early verification of performance. This thesis uses two sets of wind tunnel tests to study and validate the simulations for transport of water droplets on **Camera Monitoring System (CMS)** surface.

1.1.1 Camera Monitoring System (CMS)

A side view Camera Monitoring System (CMS) is a monitoring equipment which enables both wider side and rear view vision, revealing the driver’s blind spot. The CMS consist of an interior monitor which displays the exterior camera’s surveillance, retrieved from the camera head attached through an arm to the lateral of the vehicle. The head has a rear looking face from which monitors the road through a lens. The CMS can also increase visibility at glaring conditions such as dusk and dawn conditions, and has a generally higher image clarity than a traditional side view mirror [[12], p.3, 15–26]. To reduce glare from light pollution, some CMS designs use a glareshield that acts as a wall between the lens and light. These benefits indicate that complementing or replacing side view mirrors with a CMS may lead to a reduction in traffic collisions and contribute to road safety. This however requires that the camera function reliably, which is not guaranteed under all weather conditions as its vision may be obstructed when soiled.

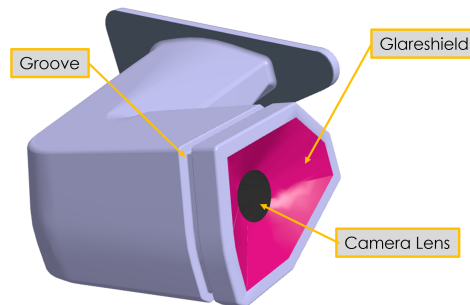


Figure 1.2: CMS prototype, from CAD used in this thesis

1.2 Objectives

The objectives of the thesis are to:

- Conduct a literature review on exterior wetting, water transport, SPH methods and automotive contamination modeling.
- Analysis of available wind tunnel experimental datasets and define relevant key performance indicators (KPIs) for validation.
- Reproduce the aerodynamic conditions of the wind tunnel experiments using CFD simulations.
- Develop and apply SPH based wetting simulations using the **AVL PreonLab**.
- Compare numerical predictions with experimental results using the defined KPIs.
- Assess the strengths and limitations of SPH based wetting simulations for early design phases.
- Provide recommendations for future virtual wetting simulations at Volvo Cars.

1.3 Scope and Limitations

This thesis focuses on the simulation and analysis of water transport phenomena on selected exterior vehicle components, specifically the Camera Monitoring System (CMS). The study is limited to conditions representative of rain exposure as defined by existing wind tunnel experiments.

1.3.1 The scope of the work includes:

- Numerical modeling of water behavior on predefined vehicle geometries.
- Comparison of simulation results with experimental validation data obtained from wind tunnel tests.
- Analysis of flow patterns and identification of regions prone to water accumulation affecting sensor performance.

1.3.2 Limitations

- The study is restricted to rain conditions and does not include snow, ice formation or mixed phase precipitation.
- The simulations rely on predefined boundary conditions and may not fully capture all real world environmental complexities.
- Surface properties such as roughness, coatings and hydrophobic effects are simplified.
- The work does not include real time implementation or integration into vehicle systems.
- Computational constraints limit the resolution, domain size and simulation time.

These limitations define the boundaries of the study while still enabling meaningful insights into water transport behavior and its impact on sensor performance.

1.4 Societal, Ethical and Ecological Considerations

1.4.1 Societal Aspects

The project has been evaluated with regard to its potential societal impact. At its current stage, the work is primarily technical and research oriented and does not directly affect specific social groups or public structures. However, the broader application of the developed concepts may influence users, stakeholders or organizational processes. Care has therefore been taken to ensure transparency in methodology, clear documentation of assumptions and reproducibility of results. Any future implementation should consider accessibility, inclusivity and potential unintended consequences for different user groups.

1.4.2 Ethical Aspects

Ethical considerations have been reviewed for the planning and execution of this work. The handling of the sensitive data generated in the experiments and subsequent simulations is thoroughly reviewed. Nevertheless, the work follows general principles of academic integrity, including proper citation of sources, honest reporting of results and avoidance of data manipulation or misrepresentation. If the project were to be extended to include user data or automated decision making, data protection regulations and fairness considerations would need to be addressed.

1.4.3 Ecological Aspects

The ecological impact of the project stems from digital resources as it is mainly conducted in a digital environment. No physical materials, chemical substances or energy intensive laboratory processes are involved. This project can enable efficient methodologies to simulate the experiments on computers which can further reduce the energy consumptions by those systems. Indirect environmental impacts, such as energy consumption due to computing resources have been considered. The computational workload has been kept efficient and proportionate to the project scope. If scaled to larger systems, energy efficiency and sustainable IT infrastructure should be taken into account.

2

Theory

2.1 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a numerical framework used to approximate solutions to the governing equations of fluid flow. These equations are derived from the fundamental conservation laws of mass, momentum and energy. The most general form of these equations for a Newtonian fluid is given by the Navier Stokes equations.

The continuity equation expresses conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (2.1)$$

The momentum equation (Navier Stokes equation) is given by:

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{g} \quad (2.2)$$

where $\boldsymbol{\tau}$ is the viscous stress tensor, defined for a Newtonian fluid as:

$$\boldsymbol{\tau} = \mu \left(\nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \quad (2.3)$$

In many engineering applications, turbulence plays a crucial role. Direct Numerical Simulation (DNS) is often computationally prohibitive, and therefore turbulence models such as Reynolds-Averaged Navier Stokes (RANS) or Large Eddy Simulation (LES) are employed.

For RANS, the velocity field is decomposed into mean and fluctuating components:

$$\mathbf{u} = \bar{\mathbf{u}} + \mathbf{u}' \quad (2.4)$$

which introduces additional unknowns known as Reynolds stresses:

$$-\rho \overline{u'_i u'_j} \quad (2.5)$$

These stresses require modeling, commonly using approaches such as the k - ϵ or k - ω models. CFD simulations involve three main steps: pre processing (geometry and mesh generation), solution (numerical discretization and solving) and post-processing (analysis of results).

2.1.1 Finite Volume Method (FVM)

The Finite Volume Method is the most widely used discretization technique in CFD due to its strict conservation properties. The computational domain is divided into a finite number of non overlapping control volumes and the governing equations are integrated over each control volume.

For a general transport equation of a scalar quantity ϕ , the integral form is:

$$\frac{\partial}{\partial t} \int_V \rho \phi dV + \int_S \rho \phi \mathbf{u} \cdot d\mathbf{S} = \int_S \Gamma \nabla \phi \cdot d\mathbf{S} + \int_V S_\phi dV \quad (2.6)$$

Applying Gauss's divergence theorem converts volume integrals of divergence terms into surface integrals:

$$\int_V \nabla \cdot (\rho \phi \mathbf{u}) dV = \int_S \rho \phi \mathbf{u} \cdot d\mathbf{S} \quad (2.7)$$

The discretized form for a control volume P can be written as:

$$a_P \phi_P = \sum_N a_N \phi_N + b \quad (2.8)$$

where a_P and a_N are coefficients arising from convection and diffusion fluxes, and b is a source term.

Interpolation schemes are required to evaluate variables at control volume faces. Common schemes include:

- Upwind differencing (first order accurate, stable)
- Central differencing (second order accurate, less stable)
- Higher order schemes (e.g., QUICK)

Pressure velocity coupling is another important aspect of FVM. Algorithms such as SIMPLE (Semi Implicit Method for Pressure linked Equations) are commonly used to enforce mass conservation in incompressible flows.

The key advantage of FVM is that it guarantees local and global conservation of mass, momentum, and energy, making it highly suitable for complex engineering flows.

2.1.1.1 Boundary Layers

Boundary layers are thin regions adjacent to solid surfaces where viscous effects dominate and velocity gradients are steep. Within this region, the fluid velocity transitions from zero at the wall (due to the no slip condition) to the free stream value. Accurate resolution of boundary layers is essential in CFD as they strongly influence drag, heat transfer, and flow separation.

In the Finite Volume Method, resolving the boundary layer requires refined meshes near walls, often achieved using structured layers of anisotropic cells (inflation layers). The mesh resolution is commonly characterized using the non dimensional wall distance y^+ , defined as:

$$y^+ = \frac{\rho u_\tau y}{\mu} \quad (2.9)$$

where u_τ is the friction velocity, y is the distance from the wall, ρ is the density, and μ is the dynamic viscosity. Depending on the turbulence modeling approach, different y^+ ranges are targeted. Low Reynolds number models require $y^+ \approx 1$, while wall function approaches typically operate in the logarithmic region ($30 < y^+ < 300$). Proper boundary layer resolution is crucial for accurately predicting near wall phenomena such as shear stress and flow separation.

2.1.1.2 Wall Models

Wall models are used in CFD to approximate the behavior of the flow near solid boundaries without fully resolving the boundary layer. This approach significantly reduces computational cost, especially in high Reynolds number flows where resolving the viscous sublayer would require extremely fine meshes.

In the context of FVM, wall models are typically implemented through wall functions, which bridge the near wall region using empirical or semi empirical relations. These functions relate the velocity at the first computational cell center to the wall shear stress using logarithmic velocity profiles.

Wall models are widely used in industrial CFD applications due to their balance between accuracy and computational efficiency. However, their accuracy depends on the validity of underlying assumptions, such as equilibrium boundary layers and attached flow conditions. In cases involving strong pressure gradients, separation, or complex geometries, fully resolving the boundary layer may be necessary for reliable predictions.

2.1.2 Smoothed Particle Hydrodynamics

Smoothed Particle Hydrodynamics (SPH) is a mesh free, Lagrangian particle method in which the fluid is represented by discrete particles. Each particle carries properties such as mass, velocity, density, pressure and moves according to the governing equations.

The interpolation theory is at the foundation of SPH. The Dirac Shifting property states

$$f(x) = \int_{\Omega} f(x') \delta(x - x') d^n x' \quad (2.10)$$

where $f(x)$ is some smooth function defined in the domain $\Omega \in \mathbb{R}$ and evaluated at the generic point $x \in \Omega$, $\delta(x - x')$ is the Dirac delta distribution and n denotes the spatial dimension ($n = 1, 2$, or 3) [6], a fundamental concept in SPH is kernel interpolation. Any field variable $A(\mathbf{r})$ is approximated as:

$$A(\mathbf{r}) = \int A(\mathbf{r}') W(\mathbf{r} - \mathbf{r}', h) d\mathbf{r}' \quad (2.11)$$

In discrete form, this becomes:

$$A(\mathbf{r}_a) = \sum_b m_b \frac{A_b}{\rho_b} W_{ab} \quad (2.12)$$

where $W_{ab} = W(\mathbf{r}_a - \mathbf{r}_b, h)$ is the smoothing kernel.

The Navier Stokes equation in SPH form is, continuity equation:

$$\frac{d\rho_a}{dt} = \sum_b m_b (\mathbf{u}_a - \mathbf{u}_b) \cdot \nabla W_{ab} \quad (2.13)$$

The momentum equation is written as:

$$\frac{d\mathbf{u}_a}{dt} = - \sum_b m_b \left(\frac{p_a}{\rho_a^2} + \frac{p_b}{\rho_b^2} \right) \nabla W_{ab} + \mathbf{g} + \mathbf{F}_{\text{visc}} \quad (2.14)$$

SPH naturally handles large deformations, moving interfaces and free surface flows without the need for mesh regeneration. This makes it particularly advantageous in applications such as:

- Multiphase flows with strong interface dynamics
- Fluid structure interaction
- Fragmentation and breakup processes

However, SPH also has limitations, including lower accuracy near boundaries, sensitivity to kernel choice and relatively high computational cost compared to grid based methods [30].

2.1.2.1 SPH based fluid solvers

Generally, the SPH based fluid solvers compute the acceleration for each fluid particle 'i' by solving the Navier Stokes equation [9]. In SPH based solvers, the fluid is discretized into particles and field quantities are approximated using kernel interpolation over neighboring particles. Each particle interacts only with particles within a finite support radius, which is determined through an efficient neighborhood search. The governing equations are thus converted into particle summations, where spatial derivatives such as gradients and Laplacians are evaluated using smoothing kernels. A typical SPH solver consists of three main steps: (i) neighbor search, (ii) computation of density, pressure, forces and (iii) time integration to update particle positions and velocities. [16] These steps are repeated for each time step to evolve the fluid in time.

$$\underbrace{\frac{Dv_i}{Dt}}_{\text{I}} = \underbrace{-\frac{1}{\rho_i} \nabla p_i}_{\text{II}} + \underbrace{\nu \nabla^2 v_i}_{\text{III}} + \underbrace{\frac{\mathbf{F}_i^{\text{external}}}{m_i}}_{\text{IV}} \quad (2.15)$$

Equation 2.15 has 4 terms.

Term **I** represents the acceleration of the particle. Term **II** corresponds to the pressure gradient force, which drives fluid motion. The pressure is computed from density using an Equation of State (EOS), typically in the context of weakly compressible SPH (WCSPH) [23].

Term **III** represents the viscous forces, which model momentum diffusion within the fluid. In SPH, this term is commonly discretized using formulations such as artificial viscosity or physically based viscosity models to ensure numerical stability and realistic dissipation of kinetic energy. These formulations are designed to handle shear flows and prevent unphysical particle interpenetration while maintaining consistency with the Navier Stokes equations [23].

Term **IV** includes external forces acting on the particles. These typically include body forces such as gravity as well as additional effects such as surface tension, adhesion and interaction forces with boundaries. In SPH, surface tension is often modeled using inter particle force formulations or continuum based approaches, enabling accurate simulation of droplet formation, deformation and breakup [23].

SPH methods are particularly well suited for problems involving free surfaces, large deformations and complex boundary interactions, as they avoid the need for mesh generation and naturally handle interface tracking. However, they are computationally expensive due to the large number of particle interactions and require careful treatment of numerical stability, boundary conditions and time stepping.

2.2 SPH vs FVM

Droplet dynamics can be simulated using mesh based Finite Volume Methods (FVM) and mesh free Smoothed Particle Hydrodynamics (SPH) approaches.

FVM discretizes the governing Navier Stokes equations over a fixed computational grid and employs interface capturing techniques such as the Volume of Fluid (VOF) or Level Set methods. These approaches provide high accuracy in resolving smooth interfaces and benefit from well established pressure velocity coupling algorithms, but require sophisticated numerical treatment to handle topological changes such as breakup and coalescence [21].

In contrast, SPH is a Lagrangian particle based method that naturally handles free surfaces, large deformations and fragmentation. It is particularly well suited for simulating droplet breakup, splashing, and multiphase interactions [22]. Surface tension effects are incorporated through inter particle forces or curvature based formulations. The choice between FVM and SPH depends on the specific requirements of accuracy, computational efficiency, and the complexity of the physical phenomena being modeled.

2.3 Droplet Dynamics

Droplet dynamics is governed by the balance of inertial, viscous, surface tension, gravitational and adhesion forces acting on the fluid interface. The motion and deformation of a droplet in an incompressible carrier phase is described by the

Navier–Stokes equations:

$$\rho \frac{D\mathbf{u}}{Dt} = -\nabla p + \mu \nabla^2 \mathbf{u} + \mathbf{f}_\sigma + \rho \mathbf{g} \quad (2.16)$$

where the surface tension force is modeled using the Continuum Surface Force (CSF) formulation [17]:

$$\mathbf{f}_\sigma = \sigma \kappa \mathbf{n} \delta_s \quad (2.17)$$

with σ the surface tension coefficient, κ the local interface curvature, $\mathbf{n}$ the unit normal to the interface, and δ_s a surface delta function confining the force to the interface.

2.3.1 Dimensionless Numbers

The behavior of droplets in a flow is characterized by key dimensionless numbers that quantify the relative importance of competing forces:

2.3.2 Reynolds number (Re_p):

Ratio of inertial to viscous forces,

$$Re_p = \frac{\rho_c |\mathbf{u}_c - \mathbf{u}_p| d_p}{\mu_c} \quad (2.18)$$

where ρ_c is the carrier-phase density, $\mathbf{u}_c - \mathbf{u}_p$ is the relative velocity between the fluid and the droplet, d_p is the droplet diameter, and μ_c is the dynamic viscosity of the carrier phase.

2.3.3 Weber number (We):

Ratio of inertial (aerodynamic) forces to surface tension forces,

$$We = \frac{\rho_c |\mathbf{u}_c - \mathbf{u}_p|^2 d_p}{\sigma} \quad (2.19)$$

2.3.4 Ohnesorge number (Oh):

Ratio of viscous forces to the combined inertial and surface tension forces,

$$Oh = \frac{\mu_d}{\sqrt{\rho_d \sigma d_p}} \quad (2.20)$$

where μ_d and ρ_d are the dynamic viscosity and density of the droplet liquid, respectively.

These dimensionless groups govern droplet deformation, breakup and transport behavior and are widely used to classify breakup regimes and predict droplet dynamics in spray and external aerodynamic flows.

2.3.5 Forces Acting on a Droplet

A droplet in an external aerodynamic flow and in contact with (or near) a solid surface is subject to the following forces:

2.3.5.1 Aerodynamic Drag Force

The drag force is the dominant mechanism transporting droplets in air. It acts opposite to the relative motion between the droplet and the surrounding gas:

$$\mathbf{F}_D = \frac{1}{2} C_D \rho_c A_p |\mathbf{u}_c - \mathbf{u}_p| (\mathbf{u}_c - \mathbf{u}_p) \quad (2.21)$$

where C_D is the drag coefficient, $A_p = \pi d_p^2/4$ is the projected frontal area of the droplet, and the relative velocity $\mathbf{u}_c - \mathbf{u}_p$ defines both the magnitude and direction of the force. The drag force accelerates the droplet toward the local gas velocity, and its magnitude depends on the droplet Reynolds number and degree of deformation (see Section 2.3.6).

2.3.5.2 Gravitational and Buoyancy Forces

The net gravitational force (gravity minus buoyancy) acting on a droplet of volume $V_p = \pi d_p^3/6$ is:

$$\mathbf{F}_g = (\rho_d - \rho_c) V_p \mathbf{g} \quad (2.22)$$

For water droplets in air ($\rho_d/\rho_c \approx 815$), buoyancy is negligible and the gravitational force simplifies to $\mathbf{F}_g \approx \rho_d V_p \mathbf{g}$. Gravity is primarily responsible for drainage of droplets on vertical and inclined surfaces and for the settling of airborne droplets.

2.3.5.3 Surface Tension Force

Surface tension acts at the liquid gas interface to minimize the droplet surface area, resisting deformation and breakup. For a sessile droplet on a surface, the surface tension also provides a capillary restoring force that opposes displacement from equilibrium.

2.3.5.4 Adhesion Force from a Solid Surface

When a droplet is in contact with a solid surface, adhesion forces resist removal by aerodynamic drag and gravity. The adhesion force depends on the wettability of the surface (quantified by the contact angle θ) and the contact line geometry. For a droplet with a circular contact line of diameter d_{cl} on a surface with advancing contact angle θ_A and receding contact angle θ_R , the maximum adhesion force in the direction of the applied shear is [37, 38]:

$$F_{adh} = k \sigma d_{cl} (\cos \theta_R - \cos \theta_A) \quad (2.23)$$

where k is a geometric constant that depends on the contact line shape (typically $k = \pi/4$ for a circular footprint [38]), and $\theta_A - \theta_R$ is the contact angle hysteresis. A large contact angle hysteresis implies strong pinning and high resistance to droplet

removal. On hydrophobic surfaces ($\theta > 90^\circ$), the contact area is small and adhesion is weak, facilitating droplet shedding; on hydrophilic surfaces ($\theta < 90^\circ$), the large wetted area increases adhesion. The critical condition for incipient motion (onset of droplet sliding or detachment) is achieved when the applied aerodynamic force exceeds the adhesion force:

$$F_D > F_{\text{adh}} + F_{g,\parallel} \quad (2.24)$$

where $F_{g,\parallel}$ is the component of gravity opposing motion.

2.3.5.5 Lift Force

The aerodynamic lift force arises from velocity gradients (shear) in the carrier flow and can cause lateral migration of droplets away from or toward a surface. For a droplet near a wall in a shear flow, the Saffman lift [39] is:

$$F_L = 1.615 \mu_c d_p (\mathbf{u}_c - \mathbf{u}_p) \sqrt{\frac{\rho_c}{\mu_c} \left| \frac{\partial u_c}{\partial n} \right|} \quad (2.25)$$

where $\partial u_c / \partial n$ is the velocity gradient normal to the wall.

2.3.6 Drag Coefficient Models

The drag coefficient C_D depends on the droplet Reynolds number and shape. Three progressively more realistic models are employed:

2.3.6.1 Stokes Drag (Creeping Flow)

For $Re_p \ll 1$:

$$C_{D,\text{Stokes}} = \frac{24}{Re_p} \quad (2.26)$$

2.3.6.2 Schiller Naumann Correlation

For the intermediate regime ($Re_p < 1000$), the Schiller Naumann correlation [27] extends the Stokes drag:

$$C_{D,\text{SN}} = \frac{24}{Re_p} \left(1 + 0.15 Re_p^{0.687} \right) \quad (2.27)$$

This correlation treats the droplet as a rigid sphere and does not account for internal circulation or deformation.

2.3.6.3 Liu Dynamic Drag Correction for Deformed Droplets

Unlike rigid spheres, droplets deform under aerodynamic loading. As a droplet distorts from a sphere toward a disk-like shape, its projected frontal area increases and the drag coefficient rises. Liu et al. [19] proposed a dynamic drag model that

interpolates between the drag of a sphere ($C_{D,\text{sphere}}$) and that of a disk ($C_{D,\text{disk}} = 1.54$) based on the droplet distortion parameter y :

$$C_D = C_{D,\text{sphere}} (1 + 2.632 y) \quad (2.28)$$

Equivalently, this can be written as a linear interpolation:

$$C_D = C_{D,\text{sphere}} (1 - y) + C_{D,\text{disk}} y \quad (2.29)$$

where y is the normalized distortion parameter obtained from the TAB model (Section 2.3.9). When $y = 0$, the droplet is a sphere; when $y = 1$, breakup occurs.

At equilibrium (negligible oscillation), the distortion parameter can be estimated from the Weber number:

$$y_{\text{eq}} = \min\left(\frac{We}{12}, 1\right) \quad (2.30)$$

This expression follows from the static balance of the aerodynamic forcing and surface tension restoring terms in the TAB equation.

2.3.6.4 Summary of Drag Correlations

Table 2.1: Summary of drag coefficient correlations for droplets.

Model	Expression	Regime
Stokes	$C_D = 24/Re_p$	$Re_p \ll 1$
Schiller Naumann	$C_D = (24/Re_p)(1 + 0.15 Re_p^{0.687})$	$Re_p < 1000$
Liu (dynamic drag)	$C_D = C_{D,\text{sphere}}(1 + 2.632 y)$	Deforming droplets

2.3.7 Droplet Inertia and Particle Response Time

Droplet inertia refers to the resistance of a droplet to changes in its velocity. Owing to its mass, a droplet does not instantaneously follow variations in the surrounding flow field, instead, it exhibits a lag characterized by the *particle response time* (or relaxation time).

2.3.7.1 Stokes Response Time

In the Stokes regime ($Re_p \ll 1$), the particle response time is [40, 25]:

$$\tau_{p,\text{Stokes}} = \frac{\rho_d d_p^2}{18 \mu_c} \quad (2.31)$$

2.3.7.2 Corrected Response Time (Finite Re_p)

For finite Reynolds numbers, the Schiller Naumann correction modifies the response time:

$$\tau_p = \frac{\rho_d d_p^2}{18 \mu_c} \cdot \frac{24}{C_D Re_p} = \frac{\rho_d d_p^2}{18 \mu_c} \cdot \frac{1}{1 + 0.15 Re_p^{0.687}} \quad (2.32)$$

2.3.7.3 Response Time with Liu Dynamic Drag

Incorporating the Liu deformation correction (Eq. A.3):

$$\tau_p = \frac{\rho_d d_p^2}{18 \mu_c} \cdot \frac{1}{\left(1 + 0.15 Re_p^{0.687}\right) (1 + 2.632 y)} \quad (2.33)$$

Deformation ($y > 0$) reduces the response time, meaning the droplet accelerates more quickly toward the gas velocity due to the increased drag.

2.3.7.4 Physical Interpretation

- A **small** τ_p implies the droplet responds quickly to flow changes and behaves as a tracer, faithfully following the carrier phase.
- A **large** τ_p implies significant inertia: the droplet deviates from streamlines, crosses them, and exhibits quasi ballistic motion.

The transient velocity response of a droplet released from rest into a uniform flow is:

$$u_p(t) = u_c \left(1 - e^{-t/\tau_p}\right) \quad (2.34)$$

At $t = \tau_p$, the droplet attains approximately 63.2% of the carrier phase velocity.

2.3.8 Stokes Number

The Stokes number St is defined as the ratio of the particle response time τ_p to a characteristic flow time scale τ_f [40, 41]:

$$St = \frac{\tau_p}{\tau_f}, \quad \tau_f = \frac{L}{U} \quad (2.35)$$

where L is a characteristic length (e.g., obstacle dimension, groove width, or boundary layer thickness) and U is the free stream velocity. The Stokes number governs droplet impaction efficiency and deposition:

- $St \ll 1$: Droplets follow the flow faithfully around obstacles and deposit minimally.
- $St \sim 1$: Maximum preferential concentration and impaction efficiency.
- $St \gg 1$: Droplets are largely unaffected by the flow and impact surfaces ballistically.

2.3.9 Droplet Deformation: Taylor Analogy Breakup (TAB) Model

The Taylor Analogy Breakup (TAB) model [35] describes droplet deformation and breakup by analogy to a spring mass damper system:

- **Forcing:** External aerodynamic pressure (analogous to external force).
- **Restoring force:** Surface tension (analogous to spring stiffness).
- **Damping:** Liquid viscosity (analogous to dashpot).

The governing equation for the dimensionless distortion parameter $y = 2x/r$ (where x is the displacement of the droplet equator from its spherical equilibrium and r is the undisturbed radius) is:

$$\ddot{y} = \frac{C_F}{C_b} \frac{\rho_c}{\rho_d} \frac{u_{\text{rel}}^2}{r^2} - \frac{C_k \sigma}{\rho_d r^3} y - \frac{C_d \mu_d}{\rho_d r^2} \dot{y} \quad (2.36)$$

where the model constants are $C_F = 1/3$, $C_b = 1/2$, $C_k = 8$, and $C_d = 5$.

2.3.9.0.1 Physical interpretation of terms:

1. $\frac{C_F}{C_b} \frac{\rho_c}{\rho_d} \frac{u_{\text{rel}}^2}{r^2}$ — aerodynamic forcing that drives deformation.
2. $\frac{C_k \sigma}{\rho_d r^3} y$ — surface tension restoring force that opposes deformation.
3. $\frac{C_d \mu_d}{\rho_d r^2} \dot{y}$ — viscous damping that dissipates oscillation energy.

2.3.9.0.2 Breakup criterion: Breakup occurs when $y \geq 1$, at which point the droplet is assumed to fragment into smaller child droplets whose sizes are determined from energy conservation arguments [35].

2.3.9.0.3 Equilibrium deformation: Setting $\ddot{y} = \dot{y} = 0$ in Eq. (2.36) yields the equilibrium (steady-state) deformation:

$$y_{\text{eq}} = \frac{C_F}{C_b C_k} \frac{\rho_c u_{\text{rel}}^2 r}{\sigma} = \frac{We}{12} \quad (2.37)$$

Breakup ($y_{\text{eq}} = 1$) thus corresponds to $We = 12$, consistent with the experimentally observed critical Weber number for bag breakup [?].

2.3.10 Droplet Deformation Prior to Breakup

For moderate Weber numbers ($We < 100$) and low Ohnesorge numbers ($Oh < 0.1$), the maximum deformation of a droplet prior to breakup is correlated by Hsiang and Faeth [?]:

$$\left(\frac{d_{\text{def}}}{d_p} \right)_{\text{max}} = 1 + 0.19 We^{1/2} \quad (2.38)$$

where d_{def} is the cross-stream dimension of the deformed droplet.

2.3.11 Droplet Breakup Regimes

Droplet breakup is primarily governed by the Weber number and modified by the Ohnesorge number [?, 20]. The principal regimes are:

Increasing the Ohnesorge number shifts the regime boundaries to higher Weber numbers, as viscous forces suppress deformation and delay breakup [20].

Table 2.2: Classification of droplet breakup regimes.

Regime	Weber number range	Description
Vibrational	$We < 12$	Oscillation without breakup
Bag breakup	$12 < We < 50$	Thin bag inflates and ruptures
Bag-and-stamen	$50 < We < 100$	Bag + central ligament breakup
Shear/stripping	$100 < We < 350$	Surface stripping of small droplets
Catastrophic	$We > 350$	Violent multi mode fragmentation

2.3.12 Force Balance for Droplet Detachment from a Surface

For a sessile droplet resting on a vehicle surface exposed to an external airflow, the condition for incipient motion (sliding or detachment) is determined by the balance of aerodynamic drag, gravity, and adhesion. The force balance in the flow direction is:

$$F_D = F_{\text{adh}} + F_{g,\parallel} \quad (2.39)$$

The drag force on a sessile droplet protruding into the boundary layer can be estimated as:

$$F_D \approx \frac{1}{2} C_D \rho_c U_{\text{eff}}^2 A_{\text{proj}} \quad (2.40)$$

where U_{eff} is an effective velocity evaluated at the droplet height within the boundary layer, and A_{proj} is the projected frontal area of the sessile droplet.

3

Methods

This chapter presents information on the methods used to obtain the results. The method of extraction of relevant quantitative data from the available experiment videos is discussed. It also includes a description of the geometry used , the motivation on choices for the simulation parameters and the models used.

3.1 Geometry

To simplify the study, aerodynamic bluff bodies were used instead of a full vehicle in both experiments and simulations. This approach reduced computational cost and complexity, allowing for faster simulations and easier analysis of flow behavior and CMS design features.

3.1.1 Wedgeoid

A body with a malgation of a wedge and a cuboid (Wedgeoid) was used in wind tunnel experiment. The wedge shape faces the wind and the cuboid section has the CMS mounted (Figure 3.1). This setup retains the key aerodynamic features of a car. The same geometry was available for further analysis in SIMCENTER STARCCM+ (Figure 3.2)



Figure 3.1: Experimental Setup in Volvo Cars wind tunnel

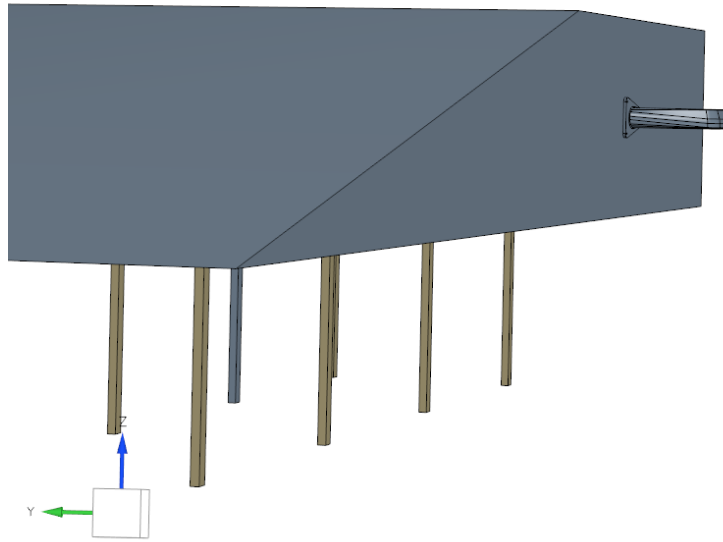


Figure 3.2: CAD Model in STARCCM+

3.1.2 CMS

The CMS prototype has an intentionally simplified geometry and it has been designed by removing complex geometric features such as protrusions and recesses. The connection between the camera head and arm was smoothed to improve aerodynamic performance and material differences (e.g. lens vs. housing) were neglected to ensure uniform surface properties. A baseline design was established and used for most analyzes. The baseline CMS contains a running groove parallel to the transverse section of the CMS prototype (Figure 3.3). This groove design would be referred to as basic group throughout this thesis.

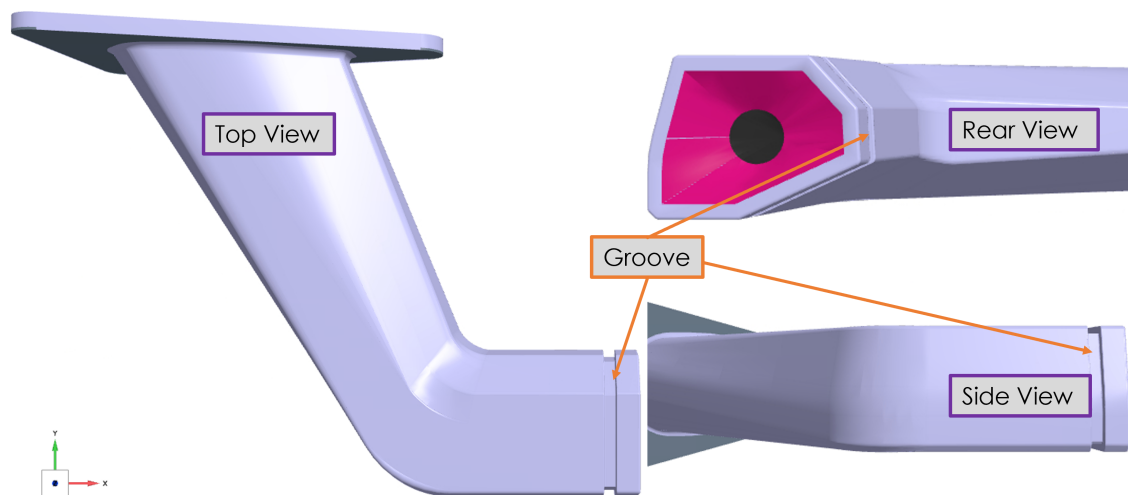


Figure 3.3: CMS prototype with the basic groove, Baseline design

3.2 Wind Tunnel Setup

The wind tunnel simulates driving conditions by generating controlled airflow over a stationary test object. Key components include a main fan to generate airflow, a settling chamber to reduce turbulence, a nozzle to accelerate the flow, and a test section where the model is placed. For soiling studies, a spray system introduces water mixed with UV sensitive liquid, allowing droplet behavior to be visualized under UV lighting. The Wedgeoid was centrally positioned in the test section and all CMS designs were evaluated. Two cameras were used to record each test, one capturing the front of the CMS and the other the rear providing detailed visualization of droplet deposition and flow behavior. Different wind speeds mimicking the driving speeds were used to collect the data.

3.3 Extraction of Measurable Metrics

The experimental video recordings at a free stream velocity of 70 km/h were post processed to extract quantitative information on droplet dynamics over the CMS surface as a baseline case. For this purpose, the open source image analysis platform FIJI (Fiji Is Just ImageJ), an extended distribution of ImageJ widely used in scientific image processing, was employed. FIJI provides a robust environment for particle detection, tracking and analysis, supported by a large collection of plugins and algorithms tailored for fluid and multiphase flow visualization studies.

Droplet identification was performed using a Hessian based feature detection approach. The Hessian detector, commonly used in image analysis for identifying blob like and curvilinear structures, relies on the computation of second order intensity derivatives to detect regions with strong local curvature. This method has been shown to be particularly effective in identifying small scale features such as droplets in noisy backgrounds, as it enhances structures based on their local intensity variations while suppressing noise [11, 13]. In the context of the present study, the detector was tuned to capture the characteristic size and contrast of droplets illuminated under UV lighting conditions in the wind tunnel experiments.

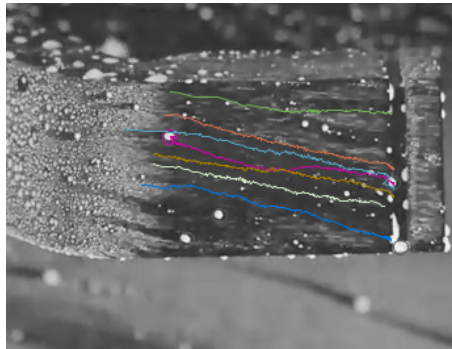


Figure 3.4: Tracks for droplets on the side face for wind speed of 70 kmph

Figure 3.4 and Figure 3.5 illustrates representative droplet trajectories extracted

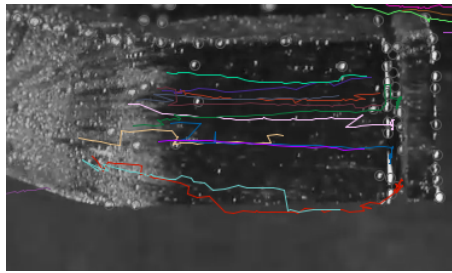


Figure 3.5: Tracks for droplets on the side face for wind speed of 120 kmph

from both the 70 km/h and 120 km/h experimental cases respectively. The detected droplets were tracked across consecutive frames, enabling the reconstruction of their motion paths over the CMS surface. The tracking process yielded time resolved positional data for each identified droplet in image coordinates. Notable differences that can be visually seen in both the figures is that the droplets in 120 kmph test videos are much smaller than that in the 70 kmph test. The tracks are also much more horizontal compared to the 70 kmph test.

Subsequently, the extracted coordinate data were processed using Python to compute kinematic quantities. By evaluating the frame by frame displacement of individual droplets and accounting for the known frame rate of the video, instantaneous and average droplet velocities were calculated. The detailed steps involved in metrics extraction is discussed in appendix A.1

3.3.1 Velocity Data

The extracted droplet trajectories were further analysed to quantify their kinematic behaviour on the CMS surface. From the frame by frame positional data, instantaneous velocities were computed and subsequently used to determine statistical measures such as the mean and median velocities for each experimental case. These metrics provide a robust representation of droplet motion, with the mean capturing the overall transport behaviour and the median reducing sensitivity to outliers arising from tracking noise or intermittent detection errors.

In addition, curve fitting techniques were applied to the droplet trajectories to extract quantitative information about their directional behaviour. By fitting appropriate functions to the spatial evolution of droplet positions, the slope of each trajectory was determined. This slope represents the dominant direction of droplet motion and provides insight into the influence of local flow structures and surface interactions on droplet transport. The combination of statistical velocity measures and trajectory based slope analysis offers a comprehensive framework for comparing experimental cases and evaluating the impact of design variations on droplet dynamics. These results are extensively discussed in section 4.1

3.4 Aerodynamic Simulations of the Wind Tunnel

The Wedgeoid geometry was imported into SIMCENTER STARCCM+ to perform aerodynamic simulations representative of the wind tunnel conditions. The computational setup was designed to closely replicate the experimental configuration, including inlet velocity corresponding to the test cases and appropriate boundary conditions such as velocity inlet, pressure outlet and no slip wall conditions on all solid surfaces. A standardized Volvo Cars simulation workflow was used to ensure consistency, numerical stability and comparability with existing aerodynamic studies.

3.4.1 Computational Model

The computational domain was discretized using a trimmed cell mesher with prism layer refinement near solid boundaries to accurately resolve the boundary layer development. Special attention was given to mesh refinement in regions of high flow gradients, particularly around the CMS and in the wake region behind the Wedgeoid using the surface and volume controls. The mesh included multiple prism layers to capture near wall effects, with a target dimensionless wall distance (y^+) suitable for resolving the viscous sublayer. Figure 3.6 Shows a low y^+ where the cell height of the first layer is $30\text{ }\mu\text{m}$ at the CMS. For the high y^+ setup that height is 2 mm.

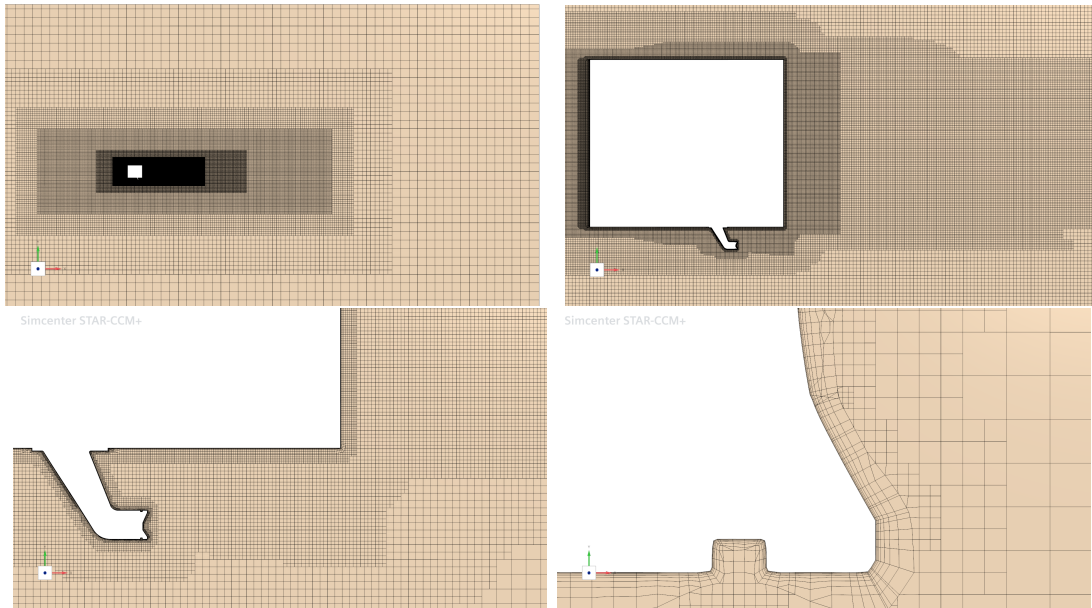


Figure 3.6: Volume Mesh for the low y^+ setup, 70 kmph case

Meshing Parameters	Values
No. of prism layers (domain)	2
No. of prism layers (CMS)	10

Table 3.1: Mesh Settings for the low y^+ setup, 70 kmph case

3.4.2 Physics Models

The airflow simulations were performed using two turbulence modeling approaches: a steady state realizable $k-\varepsilon$ Reynolds-Averaged Navier Stokes (RANS) model and the SST $k-\omega$ Detached Eddy Simulation (DES) model. The steady $k-\varepsilon$ model was employed to provide a computationally efficient prediction of the mean flow field, while the SST $k-\omega$ DES model combines a RANS treatment in the near wall region with Large Eddy Simulation (LES) in separated and free shear flow regions. This hybrid approach is well suited for capturing large scale unsteady flow structures and flow separation typical of bluff body aerodynamics. Simulations were conducted on identical computational meshes to enable a direct comparison between the turbulence models. Second-order discretization schemes were employed to improve numerical accuracy, and convergence was assessed through residual reduction together with the stabilization of key flow quantities. The comparison between the steady RANS and DES solutions provided insight into the influence of turbulence modeling on the predicted aerodynamic flow field around the CMS geometry.

3.4.3 Boundaries

Appropriate boundary conditions were applied to accurately replicate the wind tunnel environment. A uniform velocity inlet was prescribed at the domain entrance, corresponding to the experimental test conditions. At the outlet, a pressure outlet boundary condition was used, allowing the flow to exit the domain with minimal artificial reflections.

All solid surfaces of the Wedgeoid geometry, including the CMS housing, were treated with no slip wall boundary conditions to capture the development of boundary layers and surface shear effects. The wind tunnel walls were modeled either as slip walls or moving walls, depending on the specific test configuration, to minimize artificial boundary layer growth and better represent the experimental setup. Symmetry boundary conditions were not employed (Figure 3.7), as the full geometry was simulated to capture asymmetric flow features, particularly in the wake region. This ensured that complex flow structures influencing droplet transport were accurately resolved.

The resulting solution provided detailed velocity fields, capturing essential aerodynamic features such as boundary layer development, flow separation and wake formation. These flow structures are critical in determining droplet transport and deposition behaviour on the CMS.

For further analysis, velocity fields were extracted from selected cell regions of interest surrounding the CMS and wake zone. These data were exported and used as tensor field inputs for AVL PREONLAB, where they served as the basis for droplet transport simulations. This coupling approach enabled the integration of high fidelity aerodynamic flow information into multiphase modelling, allowing for more realistic prediction of droplet trajectories and surface interactions.

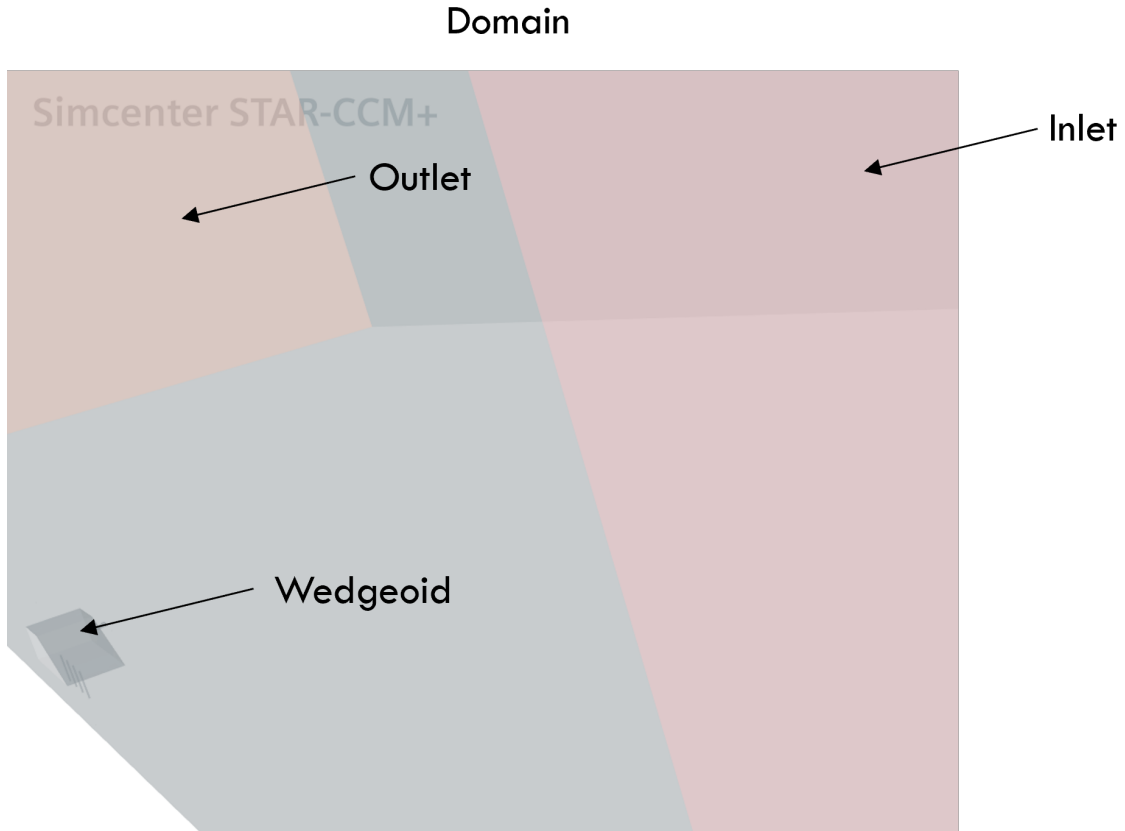


Figure 3.7: Computational domain setup in STARCCM+

3.5 Droplet Transport Model

The droplet transport simulations were performed using AVL PREONLAB, a particle based solver suitable for modelling complex multiphase flows involving droplet interaction, breakup and surface wetting phenomena.

3.5.1 Particle Size

The selection of particle size is a critical aspect in SPH based simulations, as it directly influences both computational cost and solution accuracy. A recommended guideline in AVL PREONLAB is that, within the near wall prism layer, the droplet particle diameter should be at least twice the local cell height. This ensures adequate resolution of velocity gradients near the wall and improves the accuracy of drag force calculations.

At the same time, the particle size must be sufficiently small to properly discretize the droplet and capture deformation and surface interactions. Therefore, a compromise between computational efficiency and physical accuracy is required. Based on these considerations, a particle size of 0.25 mm was selected for the present study.

3.5.2 Sessile Droplet on a Plate

Sessile droplet experiments were conducted on a flat plate to characterize the wetting behaviour of the surface. By analysing droplet shape and contact angle, important surface properties such as adhesion and wettability were determined. These parameters were subsequently used to define the interaction models in AVL PREONLAB.

Mesurements of Contact angle of a sessile droplet using an industrial experimental setup were made available. This setup was replicated with the particle size selected in 3.5.1 to further tune the parameters like Solid Adhesion factors and roughness [Table 3.2 and Figure 3.8]. The results show that only the Solid adhesion factor is determined using this method. The lateral adhesion factor and roughness factor had to be determined iteratively.

Parameter	Value
Particle Size	0.25 mm
Volume Source	2.017 μL
Volume Dimensions	$1.26348 \times 1.26348 \times 1.26348 \text{ m}^3$
Volume Source Position	(0, 0, 2) mm

Table 3.2: Experimental Setup Parameters in AVL PREONLAB

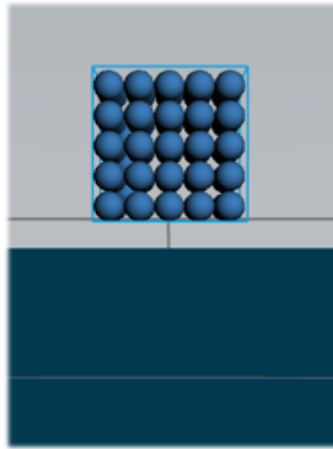


Figure 3.8: Generated volume source in AVL PREONLAB scene at $t=0$

Figure 3.9 shows a droplet on the flat plate after it has been allowed to rest on a flat plate. Here it is evident that visually it is very difficult to measure the contact angles of the rendered droplet since the curvature due to the particle at bottommost layer is present much more above the ground. To remedy this the angles were measured using a common tangent for extreme particles in lowermost layer with the adjacent layer on top of it as illustrated in Figure 3.10. Average is taken for the angles measured in 4 side views as left and right contact angles are not symmetrical.

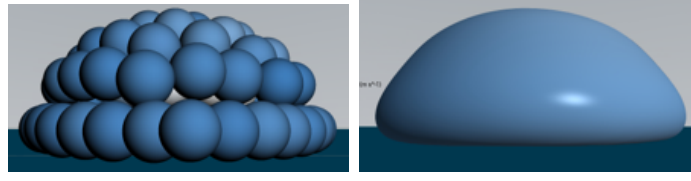


Figure 3.9: Droplet at rest on a flat plate as seen on AVL PREONLAB scene inspector. Particle arrangement in the droplet (L) and rendered droplet (R)

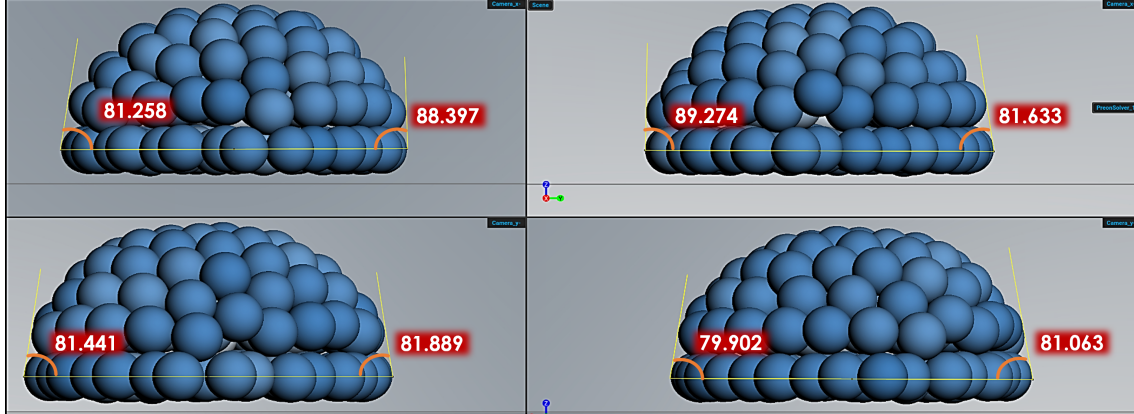


Figure 3.10: Estimating contact angles in AVL PREONLAB scene. Average contact angle here is 83.107 deg

3.5.3 Volume Flow Rate

To reduce computational cost while maintaining representative droplet surface interactions, the simulation domain was shortened compared to the full experimental setup. Additionally, larger droplet sizes were introduced in the simulations to ensure consistent impingement on the CMS surface within the reduced domain length. This approach allowed for faster simulations while preserving the essential physics governing droplet transport and deposition.

3.5.3.1 Preon Solver

The droplet transport simulations in AVL PREONLAB were performed using a Weakly Compressible Smoothed Particle Hydrodynamics (WCSPH) solver. In this approach, the fluid is represented by discrete particles, and pressure is computed as a function of density using an equation of state, allowing for efficient simulation of incompressible like flows. The solver operates in a fully Lagrangian framework, where particles carry physical properties such as mass, velocity, and density, and move according to the governing equations of motion. Time integration is performed explicitly, with stability controlled by a Courant Friedrichs Lewy (CFL) condition and viscous constraints.

3.5.3.2 Scene Setup

The CMS geometry was imported into the AVL PREONLAB environment and the PREON solver was initialized. Velocity fields obtained from aerodynamic simulations were imported as tensor fields and mapped onto the computational domain. A computational domain was defined and an area source representing rainfall was introduced (Figure 3.11). The imported geometry and velocity fields were kept unchanged to preserve consistency with the aerodynamic simulation setup.

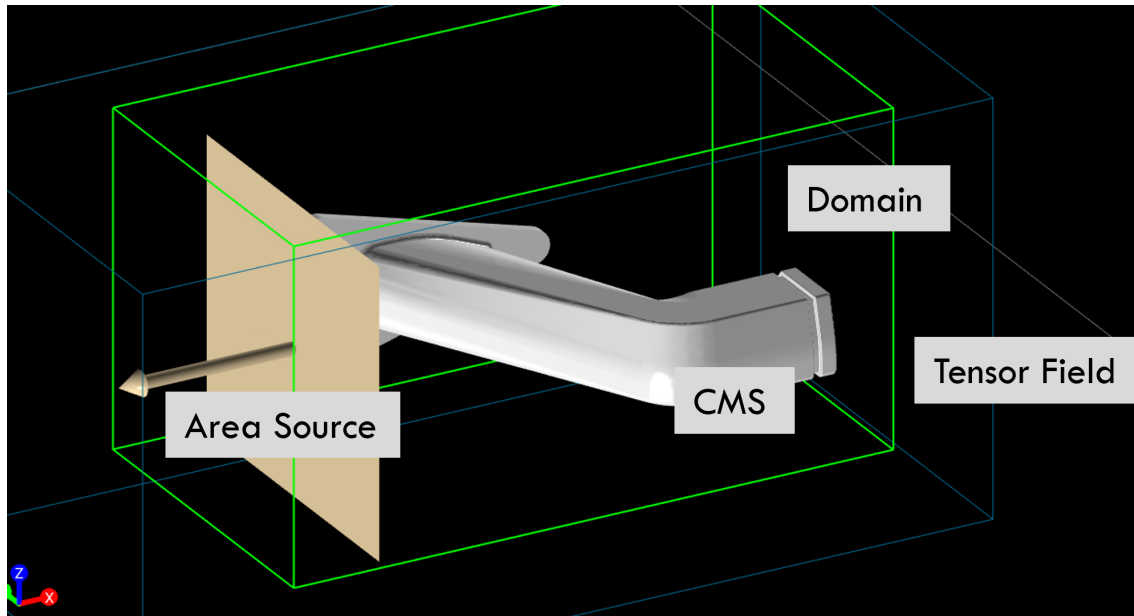


Figure 3.11: Scene Setup in AVL PREONLAB

3.5.3.3 Droplet Injection Upstream of the CMS

The response time of the droplets are calculated in A.2. To keep the simulation domain small and to address the issue of droplets not following the airflow towards the Camera Monitoring System (CMS), droplets were injected upstream of the sensor location. Figure 3.12 illustrates the initial stage of the simulation, corresponding to approximately 20 ms after droplet release.

At this stage, the droplets experience significant aerodynamic loading, causing them to deform from their initial spherical shape into flattened disc like structures. This deformation increases the effective projected area of the droplets and consequently increases the drag coefficient. As a result, the droplet response time is reduced, allowing the droplets to adapt more rapidly to changes in the surrounding airflow.

The increased aerodynamic coupling causes the droplets to reverse their initial momentum and align more closely with the local flow field. Consequently, the airflow transports the droplets toward the CMS surface, resolving the previously observed issue where droplets failed to follow the flow and did not reach the sensor region. These observations indicate that upstream injection provides a more realistic repre-

sensation of droplet transport and improves the interaction between the liquid phase and the aerodynamic flow field.

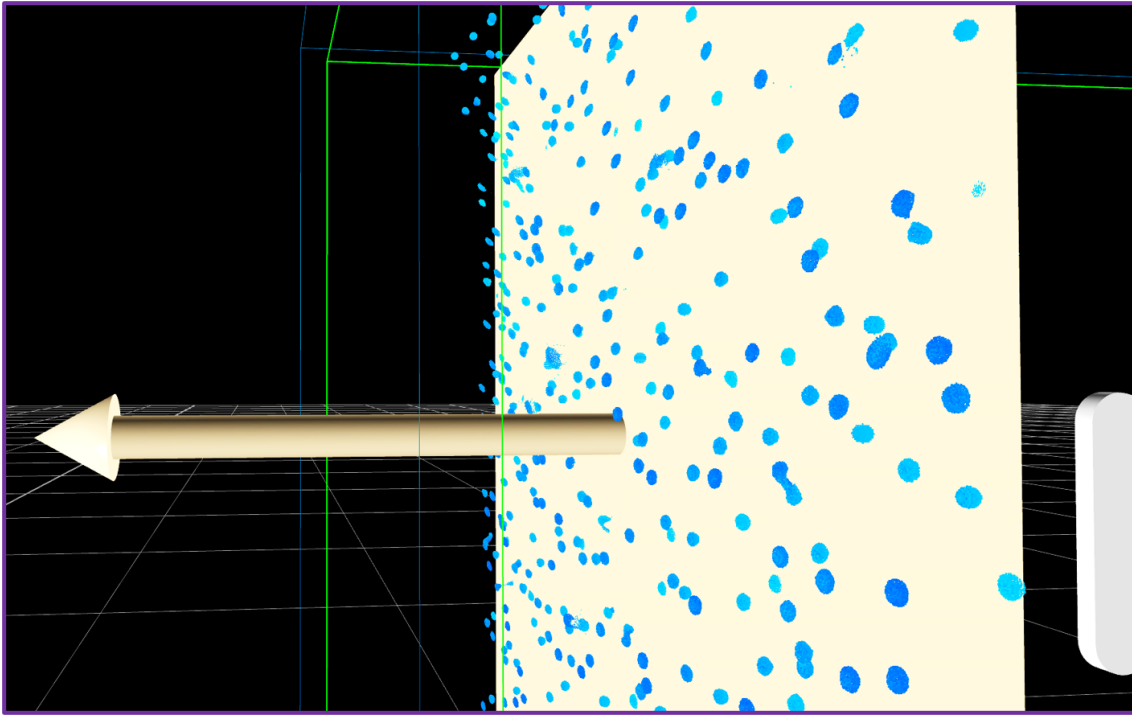


Figure 3.12: Droplets injected upstream of the CMS at $t = 20$ ms. Droplet deformation increases the drag coefficient and enables the droplets to rapidly align with the airflow, resulting in transport towards the CMS.

4

Results

This chapter presents and analyses the results obtained from the methodologies described in the previous chapter. The validation data extracted from experimental recordings using FIJI are first discussed. Finally, the results from droplet transport simulations performed in AVL PREONLAB are presented and compared against the experimental metrics to assess the predictive capability of the numerical approach.

4.1 Quantitative Metrics for Validation

Quantitative metrics were extracted from experimental image sequences to enable validation of the numerical simulations. Droplet motion was tracked frame by frame, allowing the computation of instantaneous velocities and trajectories.

Instantaneous velocities were calculated between consecutive frames and visualized using contour plots, where each point represents the droplet position colored by velocity magnitude. These plots provide a spatial overview of droplet dynamics and highlight regions of acceleration and deceleration. To enable robust comparison, statistical representations of the velocity data were also generated. In particular, median values were used as a representative measure due to their reduced sensitivity to outliers.

4.1.1 Contour Plots

Contour plots were generated to visualize the instantaneous velocity magnitude of droplets across different experimental cases (Fig. 4.1). Each point represents the droplet position at a given time step, colored according to velocity magnitude. A consistent trend can be observed across all cases, where the droplet velocity decreases progressively along the trajectory. This indicates the influence of aerodynamic drag and energy dissipation as the droplet moves downstream. In some cases, localized deviations from the smooth gradient are visible, suggesting the presence of flow disturbances or measurement noise.

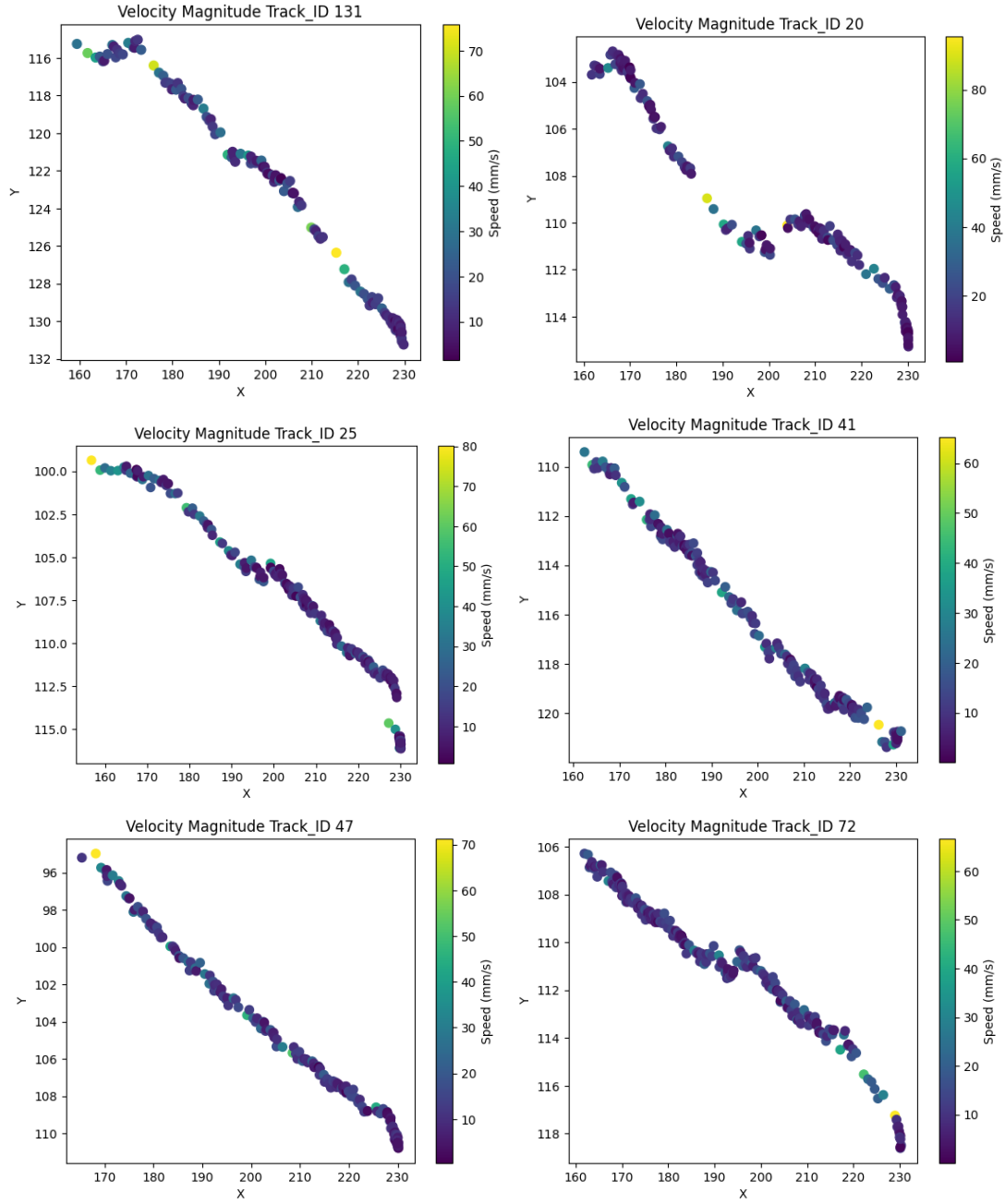


Figure 4.1: Contour plots for some tracks 70 kmph experiment video

4.1.2 Box Plots

Box plots (Figure 4.2, Figure 4.3 and Figure 4.4) were used to statistically analyze the velocity distributions. The box plots indicate a relatively narrow interquartile range, suggesting that the majority of velocity values are clustered around a central tendency. The presence of outliers highlights occasional deviations, likely caused by transient flow effects or tracking uncertainties. The median velocity is observed to be a more stable and reliable metric compared to the mean, as it is less influenced by extreme values. This makes it particularly suitable for validation purposes when

comparing experimental and simulation data.

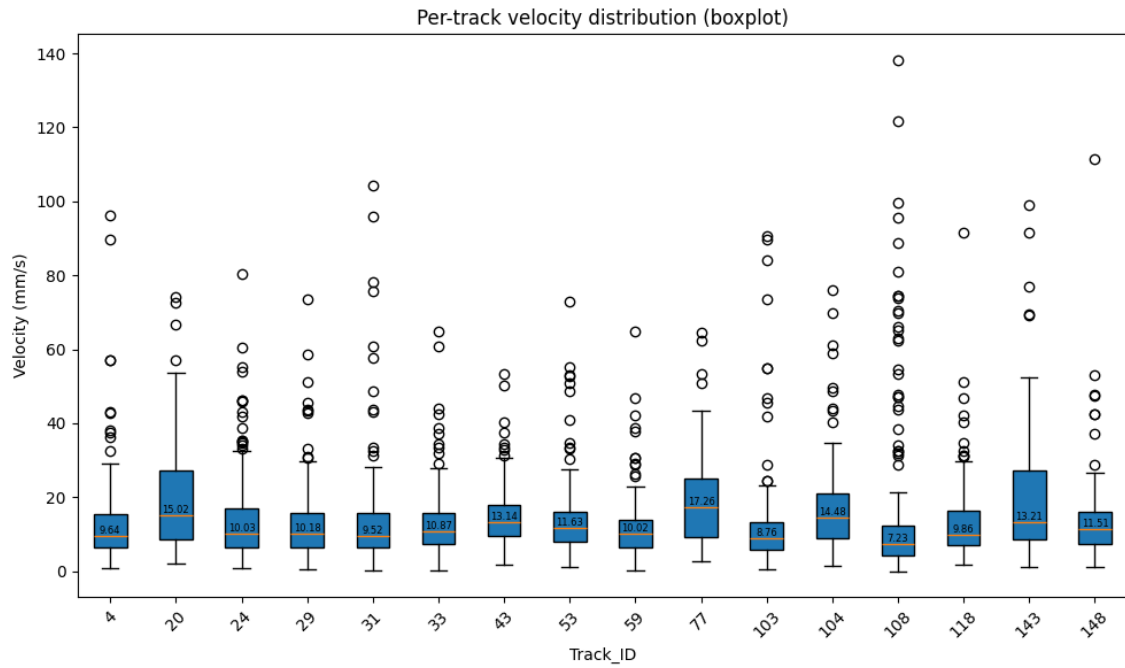


Figure 4.2: Box plots for all tracks identified in FIJI for 70 kmph case

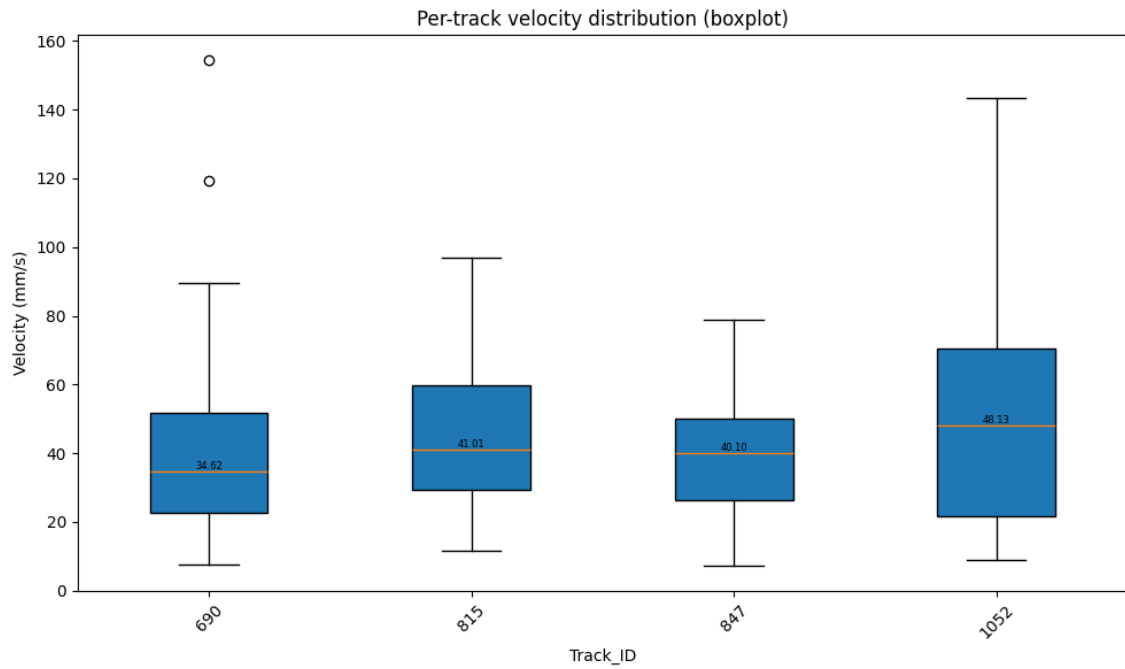


Figure 4.3: Box plots for all tracks identified in FIJI for 120 kmph case

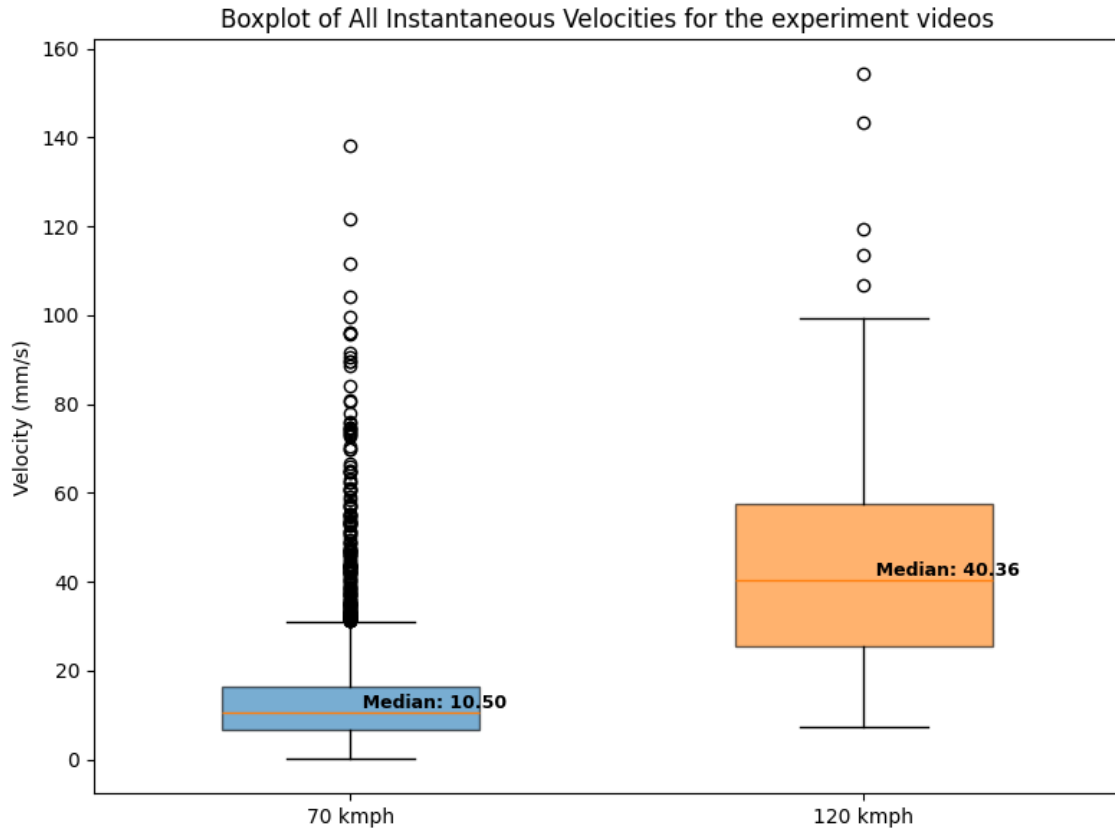


Figure 4.4: Median of all velocities across the tracks for 70 kmph and 120 kmph case

4.1.3 Trajectory Analysis

The droplet trajectories (Figure 4.5 and Figure 4.6) provide a time resolved representation of particle motion and allow for the identification of dominant transport paths. The fitted trajectories indicate that droplets generally follow a smooth, downward curving path, reflecting the combined influence of aerodynamic drag and gravitational forces. The slope analysis shows that the overall direction of motion remains consistent across cases, suggesting that the large scale flow field dominates droplet transport.

These observations confirm that droplet transport is governed by a combination of deterministic aerodynamic forces and localized flow disturbances, which must be considered in predictive modeling.

4.1.4 Comparison between 70 km/h and 120 km/h Cases

A comparative analysis was performed between the 70 kmph and 120 kmph cases to evaluate the influence of freestream velocity on droplet dynamics and aerodynamic behavior. Trajectory analysis highlights these differences. At 120 kmph, droplet trajectories are smoother and more streamlined, with reduced curvature. This indicates that aerodynamic drag forces are significantly stronger, allowing droplets

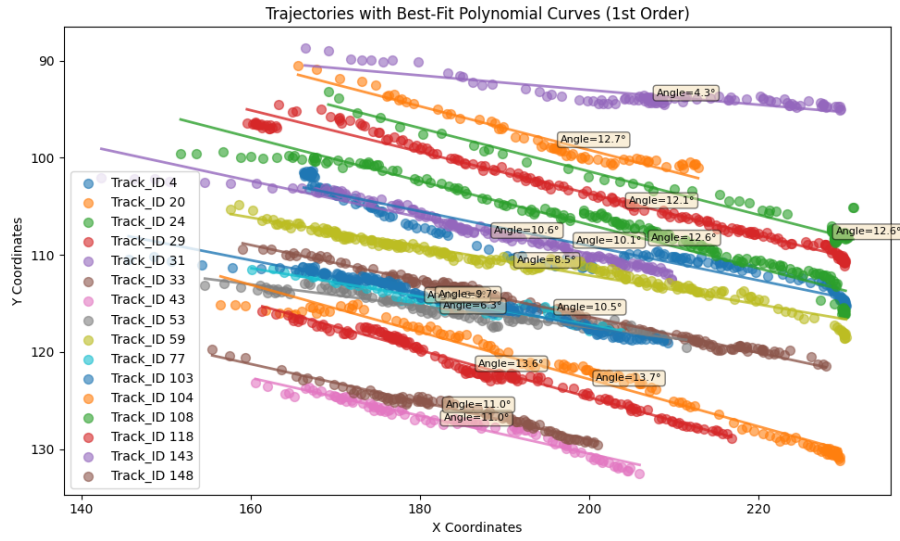


Figure 4.5: Trajectories of droplets for 70 kmph case

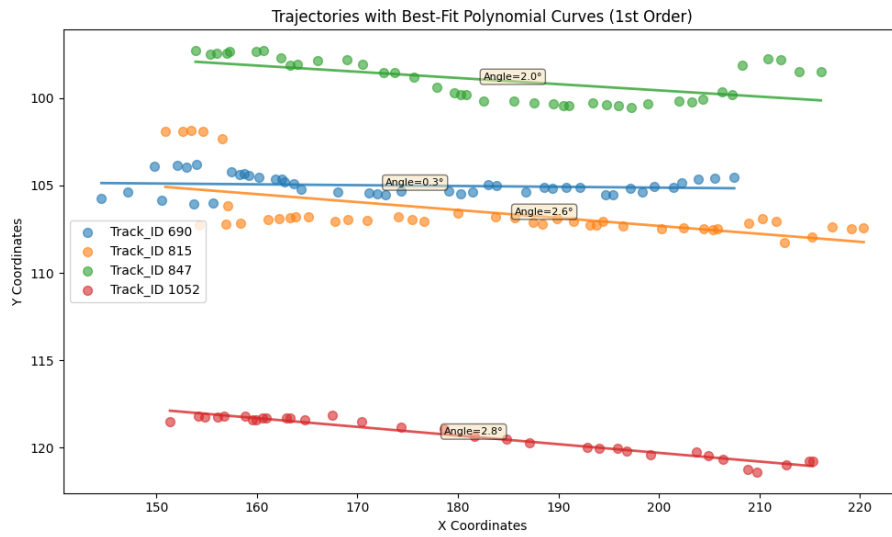


Figure 4.6: Trajectories of droplets for 120 kmph case

to closely follow the airflow. Conversely, in the 70 kmph case, trajectories exhibit more pronounced deviations and curvature, reflecting a stronger relative influence of gravity and localized flow disturbances.

An important observation is that droplet accumulation behavior differs between the two cases. At lower velocity (70 kmph), reduced aerodynamic forces allow droplets to remain longer on the surface, increasing the likelihood of adhesion and accumulation. At higher velocity (120 kmph), increased drag and shear forces enhance droplet transport and removal, although localized recirculation zones may still lead to accumulation in specific regions.

4.2 Droplet Transport Simulations

The setup focused on establishing key simulation parameters before conducting the analysis. The first step involved selecting an appropriate particle size based on recommendations from AVL PreonLab, which suggest that the particle size should be at least twice the height of the first mesh cell. Considering a low y^+ mesh cell height of $30\text{ }\mu\text{m}$ and a high y^+ mesh cell height of 2 mm , a particle size of 0.25 mm was chosen to reduce computational cost and enable faster simulations. The setup also included estimating the volume flow rate for the fluid source. A water flow rate of 1.3 L/min was used as the reference condition, and the corresponding volume flow rate was scaled to the area source, resulting in a value of $2 \times 10^{-6}\text{ m}^3/\text{s}$. For improved computational efficiency and reduced simulation time, a higher volume flow rate of $1 \times 10^{-5}\text{ m}^3/\text{s}$ was ultimately adopted. In addition, the process involved determining the adhesion factor values and tuning the lateral adhesion and surface roughness parameters to ensure realistic particle surface interaction behavior within the computational domain. From subsequent analysis it was found that lateral adhesion factor is more influential over the droplet speeds on the surface whereas roughness factor affects the shape of the droplets.

4.2.1 High y^+ and Low y^+ setup

The first phase of the simulation study focused on comparing the performance of low y^+ and high y^+ mesh configurations. Both setups employed identical liquid properties and surface interaction parameters, with a lateral adhesion coefficient of 5 and a roughness value of 3. The objective was to determine which wall treatment strategy better reproduces the experimentally observed droplet trajectories.

Figure 4.7 shows the mesh distribution and velocity field for both configurations.

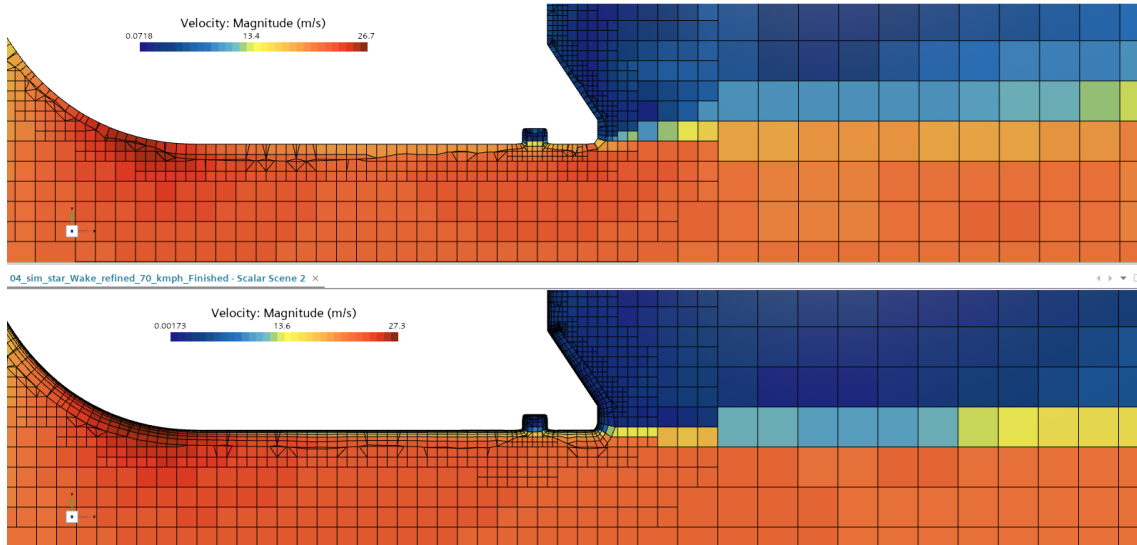


Figure 4.7: High y^+ (top) vs low y^+ (bottom) aero simulation

The low y^+ setup was expected to provide a more accurate representation of the near

wall flow structures due to the finer wall resolution. However, the resulting droplet behaviour shown in Figure 4.8 did not agree well with the experimental observations. The droplets tended to move predominantly under the influence of gravity, exhibiting excessive downward motion and limited transport along the surface. In contrast, the high y^+ setup shown in Figure 4.10 produced trajectories that more closely resembled those observed in the experimental recordings (Figure 4.5). The droplets followed the expected transport paths and exhibited more realistic interaction with the surrounding airflow. Although this result appears counterintuitive, it is likely related to the chosen particle resolution and wall treatment approach. The selected SPH particle size may not adequately resolve the near wall flow structures present in the low y^+ simulation, resulting in an overprediction of gravitational effects and reduced aerodynamic transport.

Overall, the high y^+ configuration demonstrated better qualitative agreement with the experiments and was therefore selected for further parameter tuning and validation studies.

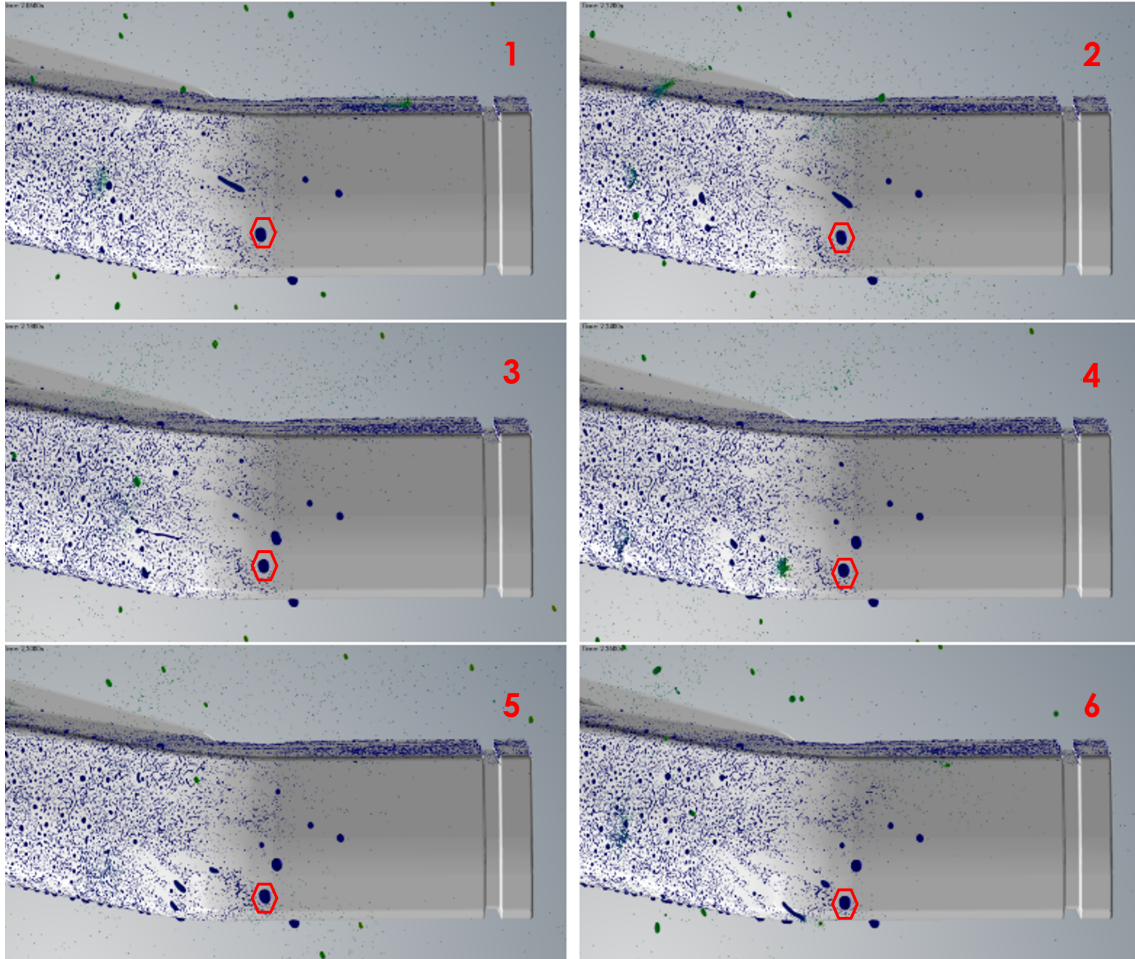


Figure 4.8: Low y^+ simulation results for 70 kmph. the droplets seem to fall under the influence of gravity rather than following the expected trajectories. Behaviour can be clearly perceived from the first and sixth pic

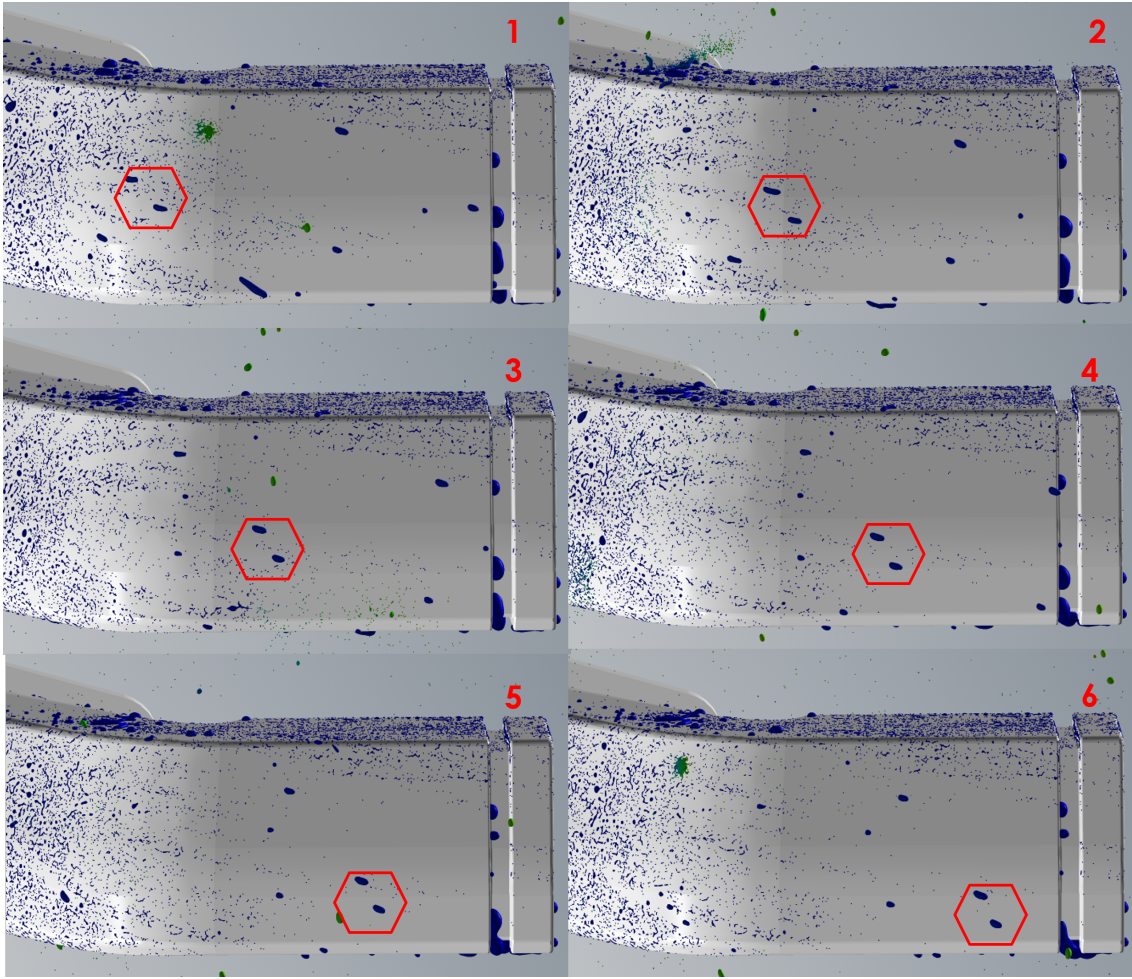


Figure 4.9: High y^+ simulation results for 70 kmph. the droplets follow the expected trajectories, Behaviour can be clearly perceived from the first and sixth pic

4.2.1.1 Tuning of Lateral Adhesion and Roughness Factors

Following the observations discussed in Section 4.2.1, an iterative parameter study was performed to improve the agreement between the numerical predictions and the experimental measurements. The lateral adhesion coefficient and surface roughness factor were systematically varied while monitoring their influence on droplet velocity, trajectory shape and overall transport behaviour.

The results confirmed that the lateral adhesion coefficient is the dominant parameter governing droplet motion along the surface. Increasing the adhesion coefficient reduced droplet velocity and promoted stronger interaction with the wall. Conversely, lower values resulted in excessive droplet transport and unrealistic trajectories. The roughness parameter primarily affected droplet morphology and spreading behaviour.

Through successive simulations, a lateral adhesion coefficient of 6.2 combined with a roughness value of 3 was found to provide the closest agreement with the experimen-

tally observed droplet trajectories and velocity distributions. The resulting droplet motion is shown in Figure 4.10, where the simulated transport behaviour closely follows the trends identified in the wind tunnel recordings.

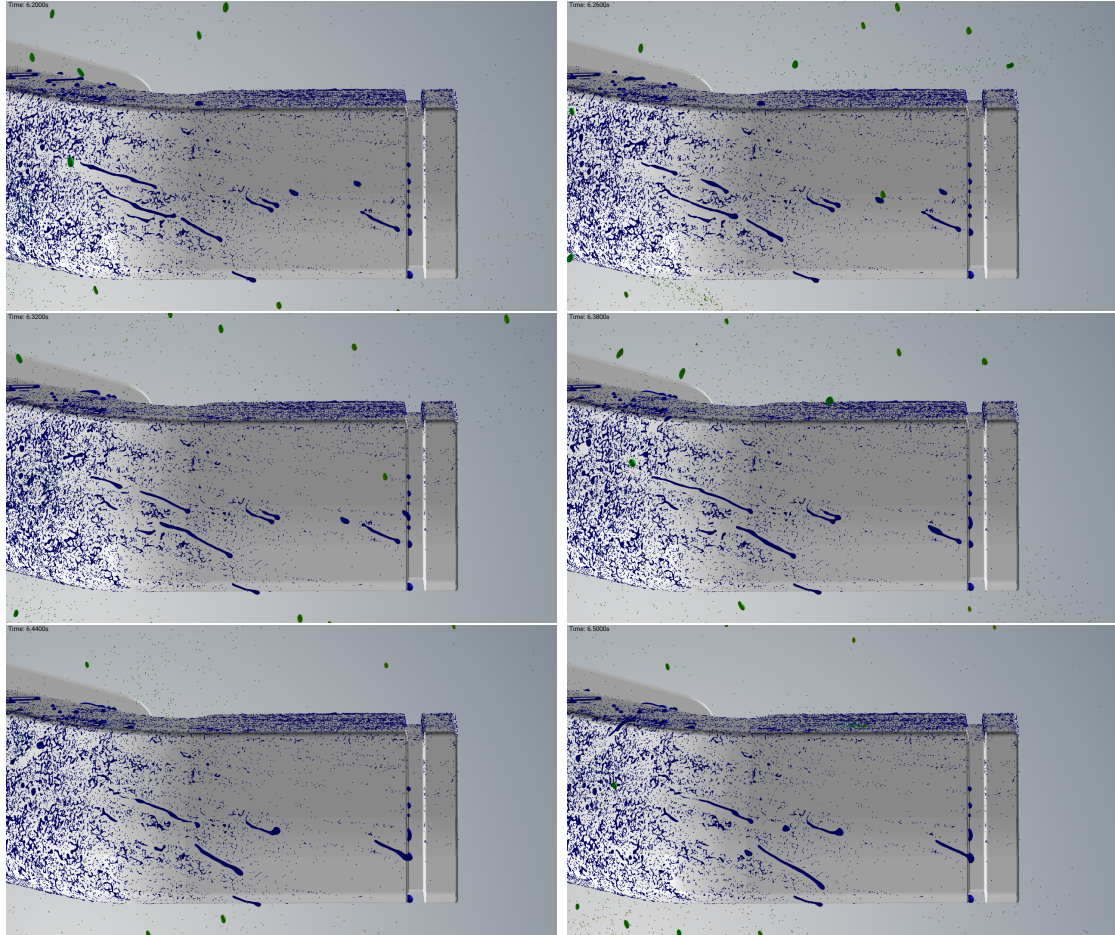


Figure 4.10: High y^+ simulation results for 70 kmph. the droplets follow the expected trajectories

4.2.2 Validation

To validate the numerical model, droplet trajectories obtained from the simulations were compared directly with the experimentally tracked droplets. Similar to the experimental analysis, particle tracking techniques were applied to the simulation output, allowing droplet positions and velocities to be extracted on a frame by frame basis. Figure 4.11 presents a qualitative comparison between the experimental and numerical trajectories for the 70 kmph case. The high y^+ simulation setup successfully reproduced the overall direction and curvature of the experimentally observed droplet paths. The agreement indicates that the dominant aerodynamic transport mechanisms are captured by the numerical model and that the selected adhesion and roughness parameters provide a realistic representation of droplet surface interaction. For the 120 kmph case, however, additional challenges were encountered.

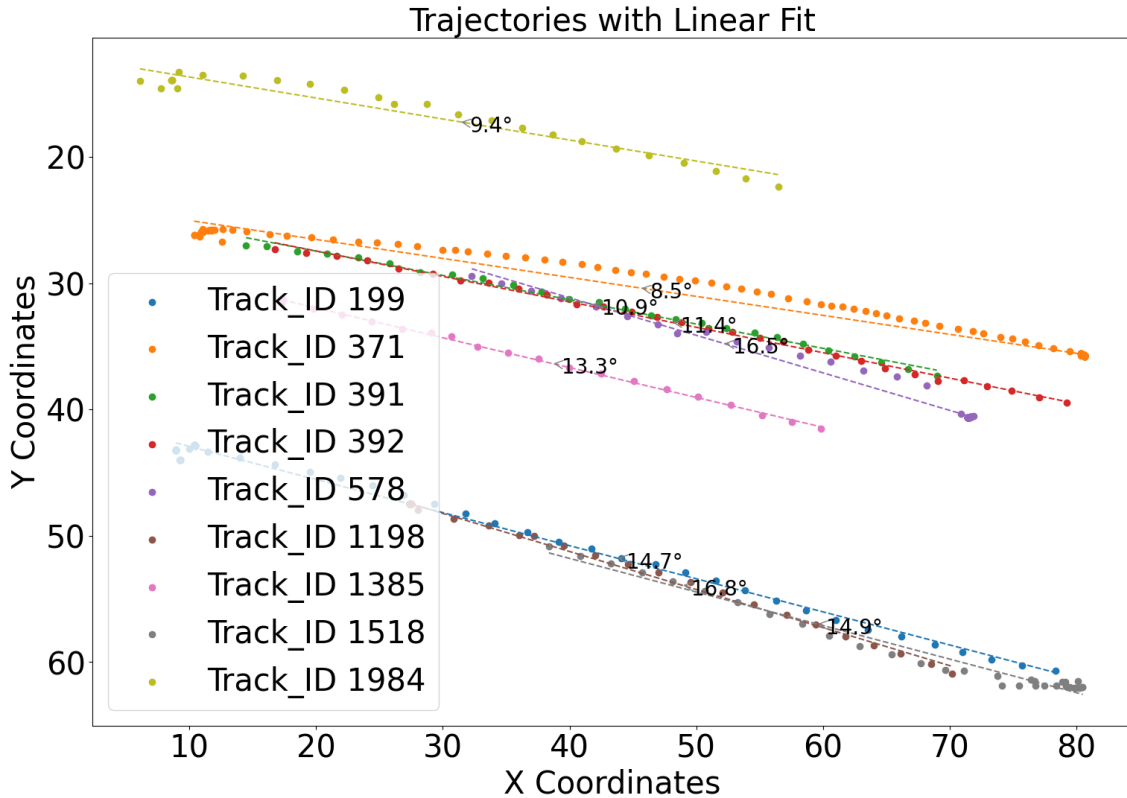


Figure 4.11: Trajectory of droplets for 70 kmph case, high y^+ simulations.

When the same lateral adhesion and roughness parameters calibrated for the 70 kmph case were applied, the droplets exhibited significant breakup during transport. Individual SPH particles detached from the main droplet and became trapped on the surface, resulting in behaviour that was not observed experimentally. This phenomenon is likely caused by insufficient droplet discretization, where the number of particles representing each droplet is too low to maintain droplet integrity under increased aerodynamic loading.

These findings suggest that particle resolution becomes increasingly important at higher velocities. A finer SPH discretization, corresponding to a smaller particle

size, would likely improve droplet cohesion and provide a more accurate representation of breakup and transport phenomena. Future work should therefore investigate the influence of particle size and droplet discretization on simulation accuracy, particularly for high speed operating conditions.

Although the results indicate that the high y^+ configuration provides better agreement with the experiments than the low y^+ setup, this outcome remains somewhat counterintuitive. Additional studies involving finer particle resolution, alternative wall treatments and further experimental validation are required to fully understand the underlying causes and establish robust modelling guidelines for droplet transport simulations on ADAS sensor systems.

4.2.3 Steady State Versus Transient Velocity Fields

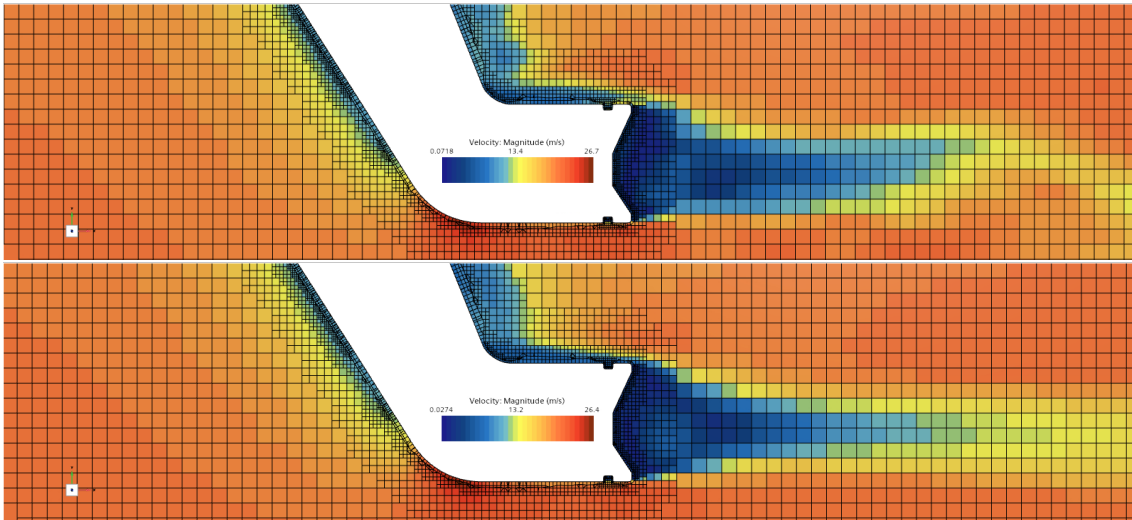


Figure 4.12: Velocity field comparison for transient and steady state aerodynamic simulations. The transient SST $k-\omega$ DES solution is shown above and the steady state $k-\epsilon$ solution below.

Figure 4.12 presents the velocity contours obtained from the transient SST $k-\omega$ DES simulation and the steady state $k-\epsilon$ simulation. Both approaches predict a similar global flow topology around the CMS geometry, including the acceleration of the flow around the sensor housing and the formation of a wake region downstream of the body. The extent and location of the separated flow region are also comparable between the two simulations. Despite these similarities, differences can be observed in the wake structure. The transient simulation predicts a more diffuse wake with stronger spatial variations in velocity, indicating the presence of unsteady flow structures that are averaged out in the steady-state solution. In contrast, the steady state simulation produces a smoother velocity distribution due to the time averaged nature of the RANS formulation. Nevertheless, both simulations predict comparable near body flow characteristics, suggesting that the primary aerodynamic features governing droplet transport are captured by both turbulence models.

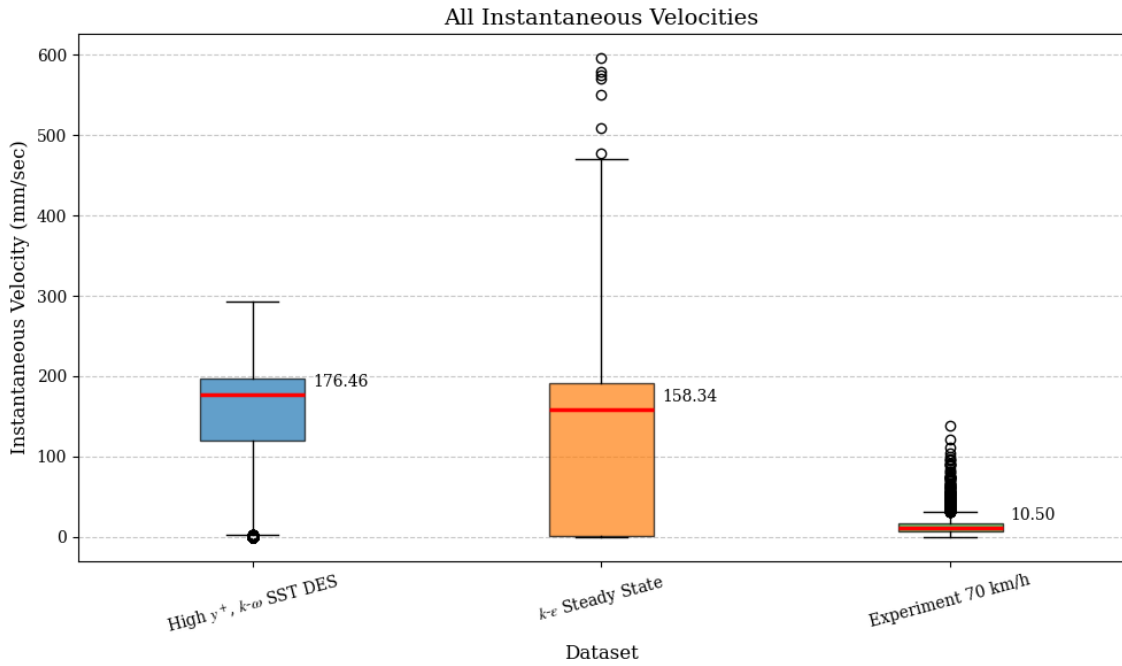


Figure 4.13: Distribution of instantaneous droplet velocities obtained from PREONLAB simulations using steady-state and transient CFD velocity fields, together with experimental measurements.

The resulting droplet velocity statistics obtained from the AVL PREONLAB simulations are shown in Figure 4.13. These simulations were performed using a lateral adhesion factor of 3.5 and a surface roughness value of 3. The comparison was conducted using the high y^+ CFD setup, as this configuration produced droplet trajectories that closely matched the experimentally observed behaviour obtained from video analysis.

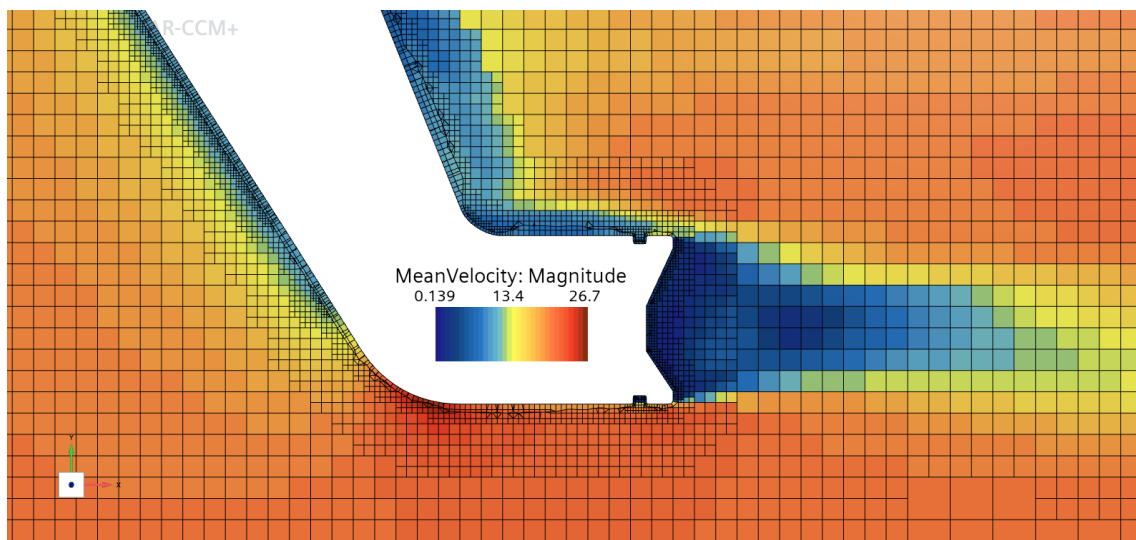


Figure 4.14: Mean velocity field obtained from the transient SST $k-\omega$ DES simulation.

To investigate this behaviour, the mean velocity field obtained from the transient CFD simulation was analysed and used as input for PREONLAB simulations. Figure 4.14 presents the temporal mean velocity field calculated from the transient simulation. Visually, the mean field closely resembles the steady state CFD solution, with similar wake dimensions and velocity gradients around the CMS housing. This observation suggests that the differences observed in the droplet transport results are not solely caused by differences in the mean flow structure.

The PREONLAB results indicate that averaging the transient velocity field reduces the influence of local velocity fluctuations on droplet motion. Consequently, droplet transport speeds become more stable and exhibit a narrower statistical distribution. In contrast, the steady state solution generates a broader range of droplet velocities despite predicting a similar mean flow field. This demonstrates that the manner in which the aerodynamic velocity field is represented can significantly influence predicted droplet transport behaviour.

5

Conclusion

This thesis presented an experimental and numerical methodology for investigating water droplet transport on a prototype Camera Monitoring System (CMS) used in Advanced Driver Assistance Systems (ADAS). By combining wind tunnel experiments, image based droplet tracking, and SPH based droplet transport simulations, a framework was established for evaluating sensor contamination under representative operating conditions.

The experimental analysis successfully quantified droplet trajectories and velocity distributions for both 70 kmph and 120 kmph operating conditions. The results demonstrated that increasing freestream velocity significantly alters droplet behaviour. At 120 kmph, droplets followed smoother and more streamlined trajectories due to stronger aerodynamic forces, whereas at 70 kmph greater trajectory curvature was observed because gravitational effects and localized flow disturbances played a more significant role. Furthermore, lower velocities promoted droplet retention and accumulation on the surface, while higher velocities enhanced droplet transport and removal.

A numerical simulation workflow was developed using AVL PREONLAB, combining CFD generated airflow fields with SPH based liquid modelling. Sensitivity studies revealed that the lateral adhesion coefficient is the most influential parameter affecting droplet velocity and transport behaviour, while surface roughness primarily influences droplet morphology and spreading characteristics. Through iterative calibration, a lateral adhesion value of 6.2 and a roughness value of 3 were identified as providing the closest agreement with the experimental observations.

Comparison between numerical and experimental results showed that the high y^+ simulation setup produced trajectories that more closely matched the measured droplet motion than the low y^+ configuration. Although unexpected, this result suggests that the chosen SPH particle resolution and wall treatment are better suited to the high y^+ approach for the investigated conditions. The simulations were able to capture the dominant transport mechanisms observed experimentally, demonstrating the potential of the methodology for predicting water contamination behaviour during the design phase.

The 120 kmph simulations highlighted that the drag and the lateral forces are high which maybe a limitations associated with droplet discretization. Droplet breakup and particle separation indicated that the selected particle resolution was insufficient to accurately represent liquid cohesion under increased aerodynamic loading. Future

work should therefore focus on reducing particle size, improving droplet discretization, and extending the validation to additional operating conditions and sensor geometries.

Overall, the developed methodology supports the ***Shift Left*** design philosophy by enabling contamination performance to be assessed during the early stages of product development. The integration of experimental validation with FVM SPH simulations provides a promising foundation for the development of predictive design tools aimed at improving the reliability, robustness and safety of ADAS sensor systems operating in adverse weather conditions.

Bibliography

- [1] Tabari H. Climate change impact on flood and extreme precipitation increases with water availability. *Scientific Reports*, 10, 13768, 2020. <https://doi.org/10.1038/s41598-020-70816-2>
- [2] Benestad, R. E., Parding, K. M., and Dobler. Downscaling the probability of heavy rainfall over the Nordic countries. *Hydrology and Earth System Sciences*, 29, 45–63, 2025. <https://hess.copernicus.org/articles/29/45/2025/>
- [3] Kukkala, V. K., Tunnell, J., Pasricha, S., and Bradley, T. H. Advanced Driver-Assistance Systems: A Path Toward Autonomous Vehicles. *IEEE Consumer Electronics Magazine*, 7(5):18–25, 2018.
- [4] Li, X., Lin, K.-Y., Meng, M., Li, X., Li, L., Hong, Y., and Chen, J. Composition and Application of Current Advanced Driving Assistance System: A Review. *arXiv preprint arXiv:2105.12348*, 2021.
- [5] Types of ADAS Sensors in Use Today. [Online]. Available: <https://dewesoft.com/blog/types-of-adassensors>
- [6] Sigalotti, L. D. G., Klapp, J., and Gesteira, M. G. The Mathematics of Smoothed Particle Hydrodynamics (SPH) Consistency. *Frontiers in Applied Mathematics and Statistics*, 7:797455, 2021. doi:10.3389/fams.2021.797455
- [7] Driving with a Camera Monitor System or Mirrors. [Online]. Available: <https://www.volvotrucks.com/en-en/news-stories/insights/articles/2024/jan/driving-with-a-camera-monitor-system-or-mirrors.html>
- [8] Becker, M. and Teschner, M. Weakly compressible SPH for free surface flows. In *Proceedings of the ACM SIGGRAPH/Eurographics Symposium on Computer Animation*, pp. 209–217, 2007.
- [9] Gissler, C., Band, S., Peer, A., Ihmsen, M., and Teschner, M. Generalized drag force for particle-based simulations. *Computers and Graphics*, 69:1–11, 2017. <https://doi.org/10.1016/j.cag.2017.09.002>
- [10] Gaylard, A. and Weatherhead, D. Dynamic Tailgate Water Management Simulation Using Smoothed Particle Hydrodynamics. SAE Technical Paper 2025-01-8625, 2025. doi:10.4271/2025-01-8625
- [11] Schneider, C. A., Rasband, W. S., and Eliceiri, K. W. NIH Image to ImageJ: 25 years of image analysis. *Nature Methods*, 2012.
- [12] Mazzae, E. N., Andrella, A. T., et al. Examination of a Prototype Camera Monitor System for Light Vehicle Outside Mirror Replacement. U.S. Department of Transportation, 2016.
- [13] Frangi, A. F., et al. Multiscale vessel enhancement filtering. MICCAI, 1998.

- [14] Pilch, M. and Erdman, C. A. Use of breakup time data and velocity history data to predict the maximum size of stable fragments for acceleration-induced breakup of a liquid drop. *International Journal of Multiphase Flow*, 13(6):741–757, 1987. [https://doi.org/10.1016/0301-9322\(87\)90063-2](https://doi.org/10.1016/0301-9322(87)90063-2)
- [15] Reitz, R. D. Modeling atomization processes in high-pressure vaporizing sprays. *Atomisation and Spray Technology*, 3(4):309–337, 1987.
- [16] Ihmsen, M., Cornelis, J., Solenthaler, B., Horvath, C., and Teschner, M. Implicit incompressible SPH. *IEEE Transactions on Visualization and Computer Graphics*, 20, 2014.
- [17] Brackbill, J. U., Kothe, D. B., and Zemach, C. A continuum method for modeling surface tension. *Journal of Computational Physics*, 100(2):335–354, 1992.
- [18] Hsiang, L.-P. and Faeth, G. M. Near-limit drop deformation and secondary breakup. *International Journal of Multiphase Flow*, 21(4):545–560, 1995. [https://doi.org/10.1016/0301-9322\(95\)00009-G](https://doi.org/10.1016/0301-9322(95)00009-G)
- [19] Liu, A. B., Mather, D., and Reitz, R. D. Modeling the effects of drop drag and breakup on fuel sprays. SAE Technical Paper 930072, 1993. doi:10.4271/930072
- [20] Guildenbecher, D. R., López-Rivera, C., and Sojka, P. E. Secondary atomization. *Experiments in Fluids*, 46:371–402, 2009. <https://doi.org/10.1007/s00348-008-0593-2>
- [21] Tryggvason, G., Scardovelli, R., and Zaleski, S. *Direct Numerical Simulations of Gas-Liquid Multiphase Flows*. Cambridge University Press, 2011.
- [22] Monaghan, J. J. Smoothed particle hydrodynamics. *Reports on Progress in Physics*, 68(8):1703–1759, 2005. <https://doi.org/10.1088/0034-4885/68/8/R01>
- [23] Koschier, D., Bender, J., Solenthaler, B., and Teschner, M. Smoothed particle hydrodynamics techniques for the physics-based simulation of fluids and solids. *Computer Graphics Forum*, 2017. <https://doi.org/10.1016/j.cag.2017.09.002>
- [24] Crowe, C. T., Schwarzkopf, J. D., Sommerfeld, M., and Tsuji, Y. *Multiphase Flows with Droplets and Particles*, 2nd Edition. CRC Press, 2012.
- [25] Maxey, M. R. and Riley, J. J. Equation of motion for a small rigid sphere in a nonuniform flow. *Physics of Fluids*, 26(4):883–889, 1983. doi:10.1063/1.864230
- [26] Clift, R., Grace, J. R., and Weber, M. E. *Bubbles, Drops, and Particles*. Academic Press, New York, 1978.
- [27] Schiller, L. and Naumann, A. Über die grundlegenden Berechnungen bei der Schwerkraftaufbereitung. *Zeitschrift des Vereines Deutscher Ingenieure*, 77:318–320, 1933.
- [28] Elghobashi, S. On predicting particle-laden turbulent flows. *Applied Scientific Research*, 52(4):309–329, 1994. doi:10.1007/BF00936835
- [29] Brennen, C. E. *Fundamentals of Multiphase Flow*. Cambridge University Press, 2005.
- [30] Liu, M. B. and Liu, G. R. Smoothed Particle Hydrodynamics (SPH): An Overview and Recent Developments. *Archives of Computational Methods in Engineering*, 17(1):25–76, 2010.
- [31] Soldati, A. and Marchioli, C. Physics and modelling of turbulent particle deposition and entrainment. *International Journal of Multiphase Flow*, 35(9):827–839, 2009. doi:10.1016/j.ijmultiphaseflow.2009.02.016

-
- [32] Balachandar, S. and Eaton, J. K. Turbulent dispersed multiphase flow. *Annual Review of Fluid Mechanics*, 42:111–133, 2010. doi:10.1146/annurev.fluid.010908.165243
 - [33] Hinze, J. O. *Turbulence*, 2nd Edition. McGraw-Hill, 1975.
 - [34] Toschi, F. and Bodenschatz, E. Lagrangian properties of particles in turbulence. *Annual Review of Fluid Mechanics*, 41:375–404, 2009. doi:10.1146/annurev.fluid.010908.165210
 - [35] O’Rourke, P. J. and Amsden, A. A. The TAB method for numerical calculation of spray droplet breakup. SAE Technical Paper 872089, 1987. doi:10.4271/872089
 - [36] Liu, L., Litchfield, J., and Belmonte, A. Drag coefficient correction for evaporating droplets. *International Journal of Multiphase Flow*, 38(1):12–19, 2012.
 - [37] Extrand, C. W. and Kumagai, Y. Liquid drops on an inclined plane: the relation between contact angles, drop shape, and retentive force. *Journal of Colloid and Interface Science*, 170:515–521, 1995.
 - [38] ElSherbini, A. I. and Jacobi, A. M. Retention forces and contact angles for critical liquid drops on non-horizontal surfaces. *Journal of Colloid and Interface Science*, 299:841–849, 2006.
 - [39] Saffman, P. G. The lift on a small sphere in a slow shear flow. *Journal of Fluid Mechanics*, 22(2):385–400, 1965.
 - [40] Crowe, C. T., Schwarzkopf, J. D., Sommerfeld, M., and Tsuji, Y. *Multiphase Flows with Droplets and Particles*, 2nd Edition. CRC Press, 2011.
 - [41] Israel, R. and Rosner, D. E. Use of a generalized Stokes number to determine the aerodynamic capture efficiency of non-Stokesian particles from a compressible gas flow. *Aerosol Science and Technology*, 2:45–51, 1983.
 - [42] Ranz, W. E. and Marshall, W. R. Evaporation from drops, Parts I and II. *Chemical Engineering Progress*, 48(3):141–146; 48(4):173–180, 1952.
 - [43] Haider, A. and Levenspiel, O. Drag coefficient and terminal velocity of spherical and nonspherical particles. *Powder Technology*, 58(1):63–70, 1989. doi:10.1016/0032-5910(89)80008-7
 - [44] Sirignano, W. A. *Fluid Dynamics and Transport of Droplets and Sprays*, 2nd Edition. Cambridge University Press, 2010.
 - [45] Lefebvre, A. H. and McDonell, V. G. *Atomization and Sprays*, 2nd Edition. CRC Press, 2017.

A

Appendix

A.1 Detailed Procedure for Droplet Detection, Tracking and Metric Extraction

A.1.1 Video Preparation

The experimental recordings obtained during the wind tunnel tests were exported as image sequences and imported into FIJI for analysis. Prior to droplet detection, the videos were inspected to ensure consistent illumination and image quality. The ultraviolet (UV) illumination used during testing produced bright fluorescent droplets against a comparatively dark background, enabling reliable feature identification. The preprocessing parameters were maintained consistently for all videos belonging to the same experimental condition to ensure comparability of the extracted results. A scale was calibrated according to the known distance.

A.1.2 Droplet Detection Using the Hessian Detector

Droplet identification was performed using the Hessian based feature detector available in FIJI. The detector evaluates second order image intensity derivatives and identifies regions exhibiting strong local curvature associated with blob like structures such as droplets.

The detection workflow consisted of the following steps:

1. Selection of an appropriate scale range corresponding to the expected droplet size.
2. Computation of the Hessian matrix at each pixel location.
3. Evaluation of the Hessian eigenvalues to identify blob like features.
4. Detection of local maxima corresponding to droplet centres.
5. Application of size and intensity thresholds to eliminate false detections caused by image noise.

The detector parameters were tuned using representative frames from the experimental recordings to ensure reliable detection of droplets throughout the image sequence.

A.1.3 Droplet Tracking

Following detection, droplets were tracked across consecutive frames using the particle tracking tools available within FIJI. The tracking algorithm associates droplets

between successive frames based on their spatial proximity and expected displacement. The tracking procedure generated a set of trajectories, each represented by a unique track identifier and a sequence of image coordinates (x_i, y_i) corresponding to frame i . Tracks that could not be reliably linked across multiple frames were discarded to minimise tracking errors. Examples of the extracted trajectories for the 70 kmph and 120 kmph test conditions are shown in Figures 4.5 and 4.6, respectively.

Only tracks located within the side face region of the CMS were retained for further analysis. Tracks occurring outside the side face region were excluded, as they did not contribute to the assessment of water transport behaviour relevant to the study. This filtering step ensured that all calculated metrics were representative of droplet motion on the surface of interest.

A.1.4 Export of Tracking Data

The validated trajectories were exported from FIJI in comma separated value (CSV) format. For each detected droplet, the exported dataset contained:

- Track identification number,
- Frame number,
- Horizontal image coordinate (x),
- Vertical image coordinate (y),
- Time stamp derived from the video frame rate.

The exported coordinate data formed the basis for all subsequent quantitative analyses performed in Python.

A.1.5 Velocity Calculation

Droplet velocities were calculated using the frame to frame displacement of each tracked droplet. Let the position of a droplet at frame i be given by (x_i, y_i) and the time interval between successive frames be Δt .

The displacement between two consecutive frames was computed as

$$\Delta s_i = \sqrt{(x_{i+1} - x_i)^2 + (y_{i+1} - y_i)^2}.$$

The instantaneous velocity was then calculated as

$$v_i = \frac{\Delta s_i}{\Delta t}.$$

The average velocity for an individual droplet track containing N velocity samples was obtained as

$$\bar{v} = \frac{1}{N} \sum_{i=1}^N v_i.$$

These calculations were implemented using Python scripts, allowing automated processing of all tracked droplets.

A.1.6 Extraction of Additional Metrics

In addition to velocity, several trajectory based quantities were extracted from the tracking data, including:

- Total track length,
- Track duration,
- Average trajectory angle,
- Horizontal and vertical displacement components,
- Number of droplets detected within the ROI.

These metrics were used to characterise droplet transport behaviour under different wind speed conditions and to support comparisons with CFD predictions.

The combined FIJI Python workflow provides a flexible and reproducible methodology for extracting quantitative droplet transport information from experimental flow visualisation recordings.

A.2 Particle Response Time Estimation

The particle response time τ_p characterizes how quickly a water droplet adjusts its velocity to match the surrounding air flow. This is a critical parameter for understanding droplet transport, deposition, and breakup on vehicle surfaces at highway speeds.

A.2.1 Problem Setup and Parameters

A spherical water droplet of diameter $d_p = 2$ mm is considered in air at a free-stream velocity of $v_f = 70$ km/h (≈ 19.44 m/s). The initial particle velocity is $v_p = 1$ m/s. The relevant fluid and particle properties are summarized in Table A.1.

Table A.1: Physical parameters used for response time estimation.

Parameter	Symbol	Value
Particle diameter	d_p	2 mm
Particle density (water)	ρ_p	998.2 kg/m ³
Fluid density (air)	ρ_f	1.225 kg/m ³
Dynamic viscosity (air)	μ	1.81×10^{-5} Pa·s
Surface tension (water)	σ	0.072 N/m
Free stream velocity	v_f	19.44 m/s (70 km/h)

A.2.2 Governing Equations

The particle Reynolds number is defined as:

$$Re_p = \frac{\rho_f d_p |v_f - v_p|}{\mu} \quad (\text{A.1})$$

For the given conditions, $Re_p \approx 2497$, placing the flow in the intermediate regime where inertial effects are significant but the flow remains sub critical.

A.2.3 Schiller Naumann correlation [27]:

The standard rigid sphere drag coefficient valid for $Re_p < 800$:

$$C_D = \frac{24}{Re_p} \left(1 + 0.15 Re_p^{0.687} \right) \quad (\text{A.2})$$

A.2.4 Liu et al. (1993) deformed droplet model [19]:

Incorporates the droplet deformation parameter y from the Taylor Analogy Breakup (TAB) model:

$$C_D = C_{D,\text{sphere}} (1 + 2.632 y) \quad (\text{A.3})$$

where the equilibrium deformation is related to the Weber number through:

$$y_{\text{eq}} = \min \left(\frac{We}{12}, 1 \right), \quad We = \frac{\rho_f (\Delta v)^2 d_p}{\sigma} \quad (\text{A.4})$$

A.2.5 Particle Response Time

The particle relaxation (response) time is defined as:

$$\tau_p = \frac{4}{3} \frac{\rho_p d_p}{C_D \rho_f |\Delta v|} \quad (\text{A.5})$$

The transient velocity response of a particle released into a uniform flow is given by:

$$v_p(t) = v_f \left(1 - e^{-t/\tau_p} \right) \quad (\text{A.6})$$

At $t = \tau_p$, the droplet reaches approximately 63.2% of the free-stream velocity

A.2.6 Results

Table A.2 summarizes the computed response times for the different drag models at the experimental conditions ($Re_p \approx 2497$, $We \approx 11.58$).

Table A.2: Comparison of particle response time τ_p for different drag models at $v_f = 70$ km/h.

Model	C_D	τ_p (ms)	Reduction vs. sphere
Schiller Naumann (sphere)	—	367.71	baseline
Liu et al. (deformed)	—	103.79	71.1 %

A.3 Adhesion Factor vs Contact Angle

Figure A.1 presents the relationship between adhesion factor and contact angle for three particle sizes (0.25 mm, 0.15 mm and 0.1 mm), each fitted with a third-order polynomial curve. The adhesion factor ranges from 0.001 (near zero adhesion, representing superhydrophobic conditions) to 1.0 (full adhesion, corresponding to hydrophilic surfaces). A clear inverse relationship is observed: as the contact angle increases, the adhesion factor decreases monotonically across all particle sizes. For the smallest particles (0.1 mm), higher contact angles are required to achieve the same reduction in adhesion compared to the larger 0.25 mm particles, indicating that smaller particles adhere more strongly at equivalent surface wettability conditions. At a contact angle of approximately 87.5, the adhesion factor lies near 0.47 for the

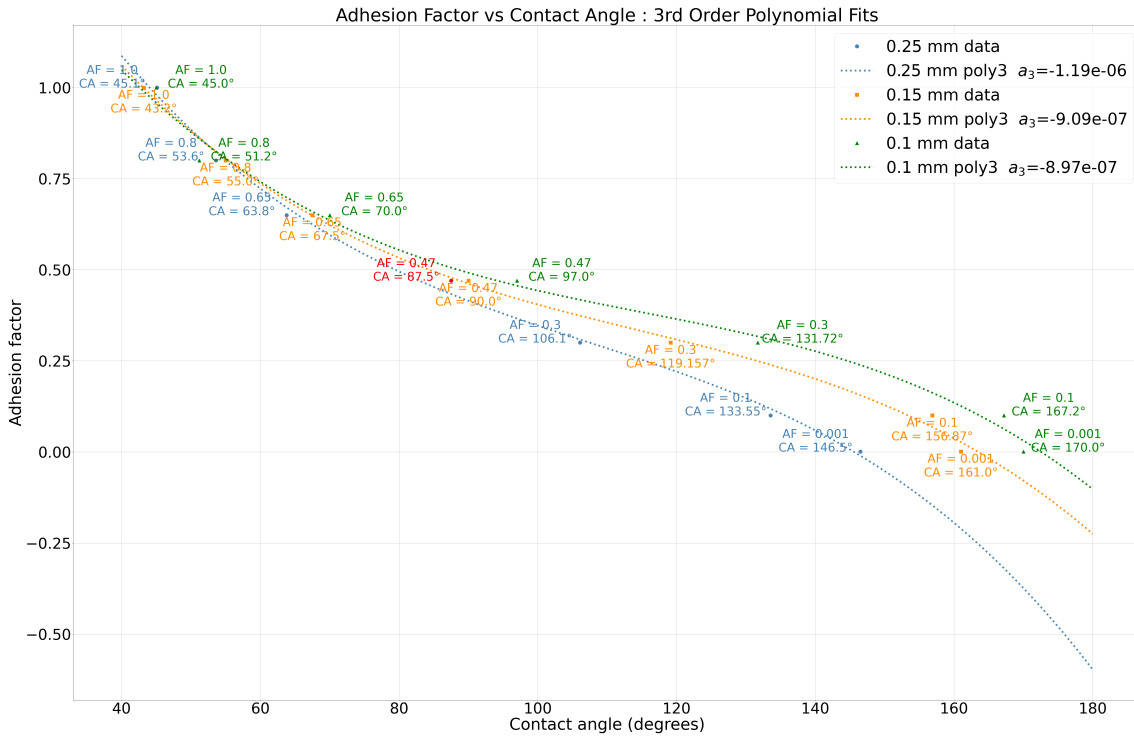


Figure A.1: Plot Showing relation between Contact angles and Adhesion factors obtained from Sessile Droplet on flat plate simulations (3.5.2), for different particle sizes

0.25 mm particles. The third order polynomial fits capture the nonlinear nature of the adhesion wettability relationship, particularly the steeper transition in the mid range contact angles (60°-130°) and the plateau behaviour at the extremes. These fitted curves provide a continuous mapping between contact angle and adhesion factor that can be used as a boundary condition input for CFD simulations of particle surface interactions at 70 km/h.

DEPARTMENT OF SOME SUBJECT OR TECHNOLOGY
CHALMERS UNIVERSITY OF TECHNOLOGY
Gothenburg, Sweden
www.chalmers.se



CHALMERS
UNIVERSITY OF TECHNOLOGY