Degree Project in Vehicle Engineering
Second Cycle, 30 Credits

Improving Water Droplet Prediction for Vehicle Exterior Water Management: Insights from Experimental and Simulation Studies

ANTON LABBÉ

MAHIM AHSAN
Master of Science in Engineering
Master programme in Vehicle Engineering
KTH Royal Institute of Technology

Supervisor at (company): David Kallin
Examiner at KTH: Mikael Nybacka

Date of presentation: 13/06/2023

TRITA-SCI-GRU 2023:061

KTH Royal Institute of Technology
School of Engineering Sciences
KTH SCI SE-100 44 Stockholm, Sweden
URL: http://www.kth.se/sci
Abstract

This thesis focuses on the study of water transportation on vehicle surfaces, which is crucial for ensuring the unobstructed operation of sensors and cameras in autonomous vehicles. The research aims to develop and validate experimental and simulation methods to enhance the understanding of water droplet behaviour and to create accurate models for computational fluid dynamics (CFD) simulations. The primary objective is to investigate the feasibility of simulating water droplets using CFD. The study examines the behaviour of water droplets on a lacquered metal sheet and a glass surface. Physical experiments and CFD simulations are conducted to analyse droplet movement under the influence of gravity and airflow. The findings provide insights into the factors influencing droplet behaviour and validate the accuracy of the simulation models through physical tests. The research also discusses the limitations of the study and the implications for Volvo Cars, aiming to improve their ability to predict water droplet movement on their vehicles.

The results indicate that water droplet behaviour can be accurately studied and simulated using a combination of experimental and CFD approaches. The findings of this study provide a good foundation for Volvo cars to reach their ultimate goal of using simulations to study water transportation exclusively.

Keywords: Computational Fluid Dynamics, Volume of Fluid, Kistler Correlation, Water Droplet, Wind Tunnel Measurement, Exterior Water Management.
Sammanfattning


Resultaten indikerar att vattendroppars beteende kan studeras och simuleras på ett noggrant sätt med hjälp av en kombination av experimentella och CFD-metoder. Studiens resultat utgör en bra grund för Volvo Cars att nå sitt slutmål att använda simuleringar för att studera vattenförflyttning exklusivt.

Acknowledgements

We would like to express our sincere gratitude to several individuals who have contributed to the successful completion of this thesis.

First and foremost, we would like to thank Mikael Nybacka, our examiner, for his valuable feedback, guidance, and support throughout the entire thesis process. His insightful comments have been instrumental in shaping this research.

We are immensely grateful to David Kallin, our supervisor at Volvo Cars, for his continuous support, encouragement, and invaluable guidance. His expertise, knowledge, and dedication have been crucial in the development of this thesis. We are truly grateful for his mentorship and the opportunities he has provided us. We would also like to extend our heartfelt appreciation to Martin Larsson for his assistance and support throughout this journey. His contributions and willingness to offer help and guidance have been greatly appreciated. Furthermore, we would like to show our gratitude to the Contamination Team at Volvo Cars for making the time of our thesis truly enjoyable with all the discussions at the weekly fika.

Special thanks go to Dr. Mathijs Van Gorcum for providing us with the software necessary for conducting the experiments. His expertise and guidance have been indispensable in the successful execution of this research. We would also like to thank Twan Bakker for his willingness to share his insights and knowledge regarding the research. His input and prompt responses to our queries have been immensely helpful.

Lastly, We would like to thank our families and friends for their constant support during our years at KTH Royal Institute of Technology.

Stockholm, June 9, 2023
Anton Labbé and Mahim Ahsan
# Contents

1 Introduction  
1.1 Background  .................................................. 1  
1.1.1 Surface contamination  .................................... 2  
1.2 Problem  ....................................................... 2  
1.3 Purpose  ......................................................... 3  
1.4 Goals  ........................................................... 3  
1.5 Delimitations  .................................................. 3  
1.6 Structure of the thesis  ........................................ 3  

2 Theory  
2.1 Fluid Mechanics  ............................................... 5  
2.1.1 Fluid Properties  ........................................... 5  
2.1.2 Governing Equations  ....................................... 6  
2.1.3 Boundary Layer  ............................................. 7  
2.1.4 Wettability and Droplet Dynamics  ......................... 9  
2.2 Computational Fluid Dynamics  ............................... 11  
2.2.1 Multiphase Flow Modelling  ............................... 11  
2.2.2 Kistler Correlation  ......................................... 13  
2.2.3 Turbulence Models  .......................................... 13  

3 Method  
3.1 Gravity  ......................................................... 15  
3.1.1 Experiments  ............................................... 15  
3.1.2 Video analysis  ............................................. 18  
3.1.3 Comparison to the Kistler Correlation  ...................... 19  
3.1.4 Simulations  ................................................. 20  
3.2 Wind Tunnel  ................................................... 22  
3.2.1 Experiments  ............................................... 22  
3.2.2 Simulations  ................................................ 23  
3.3 Mesh Convergence Study  ..................................... 24  

4 Results and Discussion  .......................... 25
4.1 Gravity ................................................................. 25
  4.1.1 Experiments .................................................... 25
  4.1.2 Simulations ..................................................... 28
4.2 Wind Tunnel ......................................................... 30
  4.2.1 Experiments .................................................... 30
  4.2.2 Simulations ..................................................... 31
4.3 Residuals ............................................................. 34

5 Conclusion and Future Work ........................................... 37

References ................................................................. 39

Appendix A ..................................................................... i

Appendix B ..................................................................... vii
## List of Figures

<table>
<thead>
<tr>
<th>Figure</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Illustration of the development of a boundary layer.</td>
<td>8</td>
</tr>
<tr>
<td>2</td>
<td>Contact line, or triple point, between three phases and corresponding equilib-</td>
<td>9</td>
</tr>
<tr>
<td></td>
<td>rium contact angle.</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Hydrophobicity of a droplet-based on the static contact angle.</td>
<td>10</td>
</tr>
<tr>
<td>4</td>
<td>Illustration of contact angle hysteresis, where intersection points of ( \theta_a ) and ( \theta_r ) with the y-axis where the Capillary number is zero, ( Ca ), highlight the start points on droplet movement.</td>
<td>10</td>
</tr>
<tr>
<td>5</td>
<td>Advancing and receding contact angles.</td>
<td>11</td>
</tr>
<tr>
<td>6</td>
<td>Contact angle measurement made with Krüss Mobile Surface Analyser.</td>
<td>16</td>
</tr>
<tr>
<td>7</td>
<td>Standard deviation of the Agilent 10 µL syringe measuring a 60 µL droplet, ten times.</td>
<td>17</td>
</tr>
<tr>
<td>8</td>
<td>Difference in illumination angle of the droplet. Notice the difference in perspective as well, with b) being completely in profile.</td>
<td>17</td>
</tr>
<tr>
<td>9</td>
<td>Comparison of one set of tests from the gravity experiment, illustrating the effect of noise reduction using the moving mean method.</td>
<td>20</td>
</tr>
<tr>
<td>10</td>
<td>Porous region, highlighted in the red circle, in the steady state simulation before running implicit unsteady.</td>
<td>23</td>
</tr>
<tr>
<td>11</td>
<td>Experimental data for the lacquered surface (49 degrees) compared to the Kistler Correlation.</td>
<td>26</td>
</tr>
<tr>
<td>12</td>
<td>Experimental data for the glass surface (32 degrees) compared to the Kistler Correlation.</td>
<td>27</td>
</tr>
<tr>
<td>13</td>
<td>Average droplet velocities for the different inclinations, both surfaces.</td>
<td>28</td>
</tr>
<tr>
<td>14</td>
<td>Volume continuity for the different inclinations, both surfaces.</td>
<td>29</td>
</tr>
<tr>
<td>15</td>
<td>Droplet appearance of the lacquered surface at 49°.</td>
<td>30</td>
</tr>
<tr>
<td>16</td>
<td>Release of smaller a droplet before the whole droplet started to move. The smaller droplet is highlighted in the red circle and the original droplet is highlighted in the blue circle.</td>
<td>31</td>
</tr>
<tr>
<td>17</td>
<td>Droplet appearance of the lacquered surface at 30°.</td>
<td>32</td>
</tr>
<tr>
<td>18</td>
<td>Change in contact line position.</td>
<td>32</td>
</tr>
<tr>
<td>19</td>
<td>Volume fluctuation for each inclination.</td>
<td>33</td>
</tr>
</tbody>
</table>
20  *Continuity and water residuals for the gravity simulations, both surfaces.* . . . 34
21  *Continuity and water residuals for the wind tunnel simulations.* . . . . . . . 35

22  Experimental data for the glass surface (39 degrees) compared to the Kistler Correlation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . i
23  Experimental data for the glass surface (44 degrees) compared to the Kistler Correlation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . ii
24  Experimental data for the glass surface (54 degrees) compared to the Kistler Correlation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . ii
25  Experimental data for the glass surface (59 degrees) compared to the Kistler Correlation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . iii
26  Experimental data for the glass surface (22 degrees) compared to the Kistler Correlation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . iii
27  Experimental data for the glass surface (27 degrees) compared to the Kistler Correlation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . iv
28  Experimental data for the glass surface (37 degrees) compared to the Kistler Correlation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . iv
29  Experimental data for the glass surface (42 degrees) compared to the Kistler Correlation. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . v
List of Tables

1  Contact angle (CA) measurements and standard deviation from Krüss Mobile Surface Analyser. .......................................................... 16
2  Inclination increments of the motor actuated table for the different surfaces during the gravity experiments. ................................. 18
3  Advancing and receding contact angles used for the Kistler model in STAR-CCM+. .............................................................. 21
4  Inclination increments of the motor actuated table for the different surfaces in the wind tunnel experiments. ............................... 22
5  Average droplet velocity in on the lacquered metal sheet for the five inclinations. 26
6  Average droplet velocity in on the glass surface for the five inclinations. 26
7  Air velocity at droplet movement in Wind tunnel tests. .................... 29
8  Air velocity at droplet movement in Wind tunnel tests. .................... 30
9  Initial droplet movement in the simulations. .................................. 33
10 Volume Change Over the Course of the Simulation. ....................... 33
11 Results of Wind tunnel tests for the lacquered surface 35 degrees. .... vii
12 Results of Wind tunnel tests for the lacquered surface 30 degrees. .... vii
13 Results of Wind tunnel tests for the lacquered surface 25 degrees. .... vii
14 Results of Wind tunnel tests for the glass surface 18 degrees. .......... viii
15 Results of Wind tunnel tests for the glass surface 13 degrees. .......... viii
16 Results of Wind tunnel tests for the glass surface 10 degrees. .......... viii
## List of Abbreviations and Acronyms

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ADAS</td>
<td>Advanced Driver Assistance Systems</td>
</tr>
<tr>
<td>AMR</td>
<td>Adaptive Mesh Refinement</td>
</tr>
<tr>
<td>CA</td>
<td>Contact Angle</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CoM</td>
<td>Centre of Mass</td>
</tr>
<tr>
<td>DAQ</td>
<td>Data Acquisition</td>
</tr>
<tr>
<td>LiDAR</td>
<td>Light Detection And Ranging</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier Stokes</td>
</tr>
<tr>
<td>UV</td>
<td>Ultraviolet</td>
</tr>
<tr>
<td>VOF</td>
<td>Volume of Fluid</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

Autonomous vehicles are reliant on sensors such as cameras, LiDARs and radars to map their surroundings. These components can be obstructed by different types of contaminations and need purposefully designed features to guide contaminants away. To intentionally create such features the study of fluid behaviour, or fluid dynamics, is necessary. It is a broad and complex field that plays a key role in many different industries. Ranging from mechanical and chemical engineering to biological systems and astrophysics, it is used to study a fluid under static and dynamic loads [1]. Computational fluid dynamics (CFD) is a simulation technique that uses numerical methods and algorithms to analyse and predict the behaviour of fluid flow, heat transfer, and related phenomena. It involves solving complex equations that describe the motion of fluids and provides detailed insights into the fluid dynamics and performance of various systems and designs. In this study, experiments on droplet movement on a flat plate will be conducted in an attempt to make a realistic model of the same phenomena in CFD. At large, this can be seen as a (first) step in trying to simulate fully developed fluid movement on vehicle surfaces.

1.1 Background

With the rapid development of computational capabilities since the mid-1980s, simulation tools have become increasingly adept at modelling complex systems and processes. Within the automotive industry, the application of CFD simulation tools has been instrumental in studying the contamination of aforementioned sensors that are vital to the operation of autonomous vehicles. The use of CFD simulations provides numerous opportunities for sustainable practices. By simulating the performance of a component or product before physical prototyping, companies can identify potential issues and make design changes more effectively, reducing the cost of design and testing from an economic perspective. Additionally, if a component or product can be simulated instead of being built as a physical prototype, raw material, energy consumption, and transportation emissions can be saved, which is ecologically beneficial.
The sustainability benefits of CFD simulations extend beyond economic and ecological factors, encompassing critical safety aspects, particularly in the case of autonomous vehicles. Unobstructed sensors are paramount to ensuring the safe operation of these vehicles. CFD simulations can help achieve this goal by accurately modelling the flow of fluids around vehicle components and identifying areas where contamination may accumulate. By keeping sensors unobstructed, passengers are better protected, and the risk of accidents is reduced, leading to significant societal benefits. Given the importance of keeping sensors unobstructed in autonomous vehicles, it is crucial to be able to investigate how surface contamination affects their performance, either through real-world testing or CFD.

1.1.1 Surface contamination

Initially, a phenomenon only concerning the vision of the driver and the visibility of the vehicle, surface contamination has gotten increased attention in recent years as advanced driver assistance systems (ADAS) have increased in number, complexity, and popularity. In a business expected to reach USD 91 billion by 2030, ADAS involves advanced cruise control, lane-keeping assist, automatic emergency braking, parking assist, and many more [2]. Mapping the vehicle’s environment for these systems are components such as cameras, radars, LiDARs and ultrasonic sensors located on different vehicle surfaces.

While the sensitivity to contamination varies among different sensors, it is important to note that the aforementioned components are affected to some extent and perform optimally when they are free from obstructions. There are three sources of contamination, as per Hagemeier et al.’s condensed definition. Primary sources in the form of rainfall, third-party (or foreign) sources, which are dirt or splashes and sprays from other cars, and self-soiling, which is contamination brought on by the vehicle itself, such as that produced by the wheels and tires [3]. The ability to guide water and solid contaminants away is strongly affected by the interaction with wakes, vortices and shear flows. Hence, such unstable aerodynamic effects also influence water droplets, rivulets and films on vehicle surfaces [4].

1.2 Problem

The transportation of water on vehicle surfaces can have a significant impact on vehicle performance and safety. However, understanding the behaviour of water on vehicle surfaces remains a challenging problem, as it involves complex fluid dynamics interactions between the surface and the water. Therefore, this thesis will exclusively focus on the behaviour of water droplets on surfaces relevant to vehicles. The use of CFD simulations to model the flow of water droplets on vehicle surfaces has the potential to provide valuable insights into this problem. However, the accuracy of these simulations is dependent on the validity of the assumptions and the derived models, which can be verified through physical testing. This research will attempt to answer the following questions:

- How feasible is it to simulate a water droplet using CFD and leverage it as a foundation to achieve the ultimate goal of simulating water transportation on vehicle surfaces?
1.3. PURPOSE

- How do water droplets behave on the various surfaces commonly found on vehicles, and what factors influence their behaviour?
- How can verification of CFD simulations be achieved through physical tests?

1.3 Purpose

The purpose of this thesis is to develop and validate experimental and simulation methods for studying water transportation on vehicle surfaces. The objective is to create a model in the CFD software STAR-CCM+ that investigates the movements of water droplets and to validate the simulations through physical experiments on an inclined surface. Furthermore, this research endeavours to meet the academic and engineering community’s quest for new knowledge, while simultaneously contributing to the advancement of Volvo Cars. By enhancing their predictive capabilities regarding the behaviour of water droplets on vehicle surfaces, the study aims to streamline the testing procedures, minimising the reliance on wind tunnels and optimising their experimental methodologies. Moreover, the purpose is to meet the degree objectives and course requirements for the thesis course.

1.4 Goals

The goals of this thesis are to contribute to Volvo Cars’ research on water transportation on vehicle surfaces and to establish a foundation for the study of this phenomenon through rigorous experimentation and simulation. Through the delivery of dependable, precise, and replicable outcomes, this study aids Volvo Cars in progressing towards their goal of diminishing reliance on wind tunnel testing in favour of simulation-based approaches.

1.5 Delimitations

The scope of this thesis project is limited to the investigation of water transportation on vehicle surfaces, specifically focusing on lacquered metal sheet and glass surfaces with an inclination angle of up to 75 degrees, due to the properties of the actuated table. The study will only examine the behaviour of individual water droplets under the influence of gravity and airflow, with no consideration given to rivulets or fluid films. The research will be limited to macroscopic droplet properties and models, microscopic-level properties will not be studied. Moreover, the study will utilise Simcenter STAR-CCM+ as the primary CFD software to ensure consistency with modelling techniques and results. Other CFD software will not be considered in this thesis.

1.6 Structure of the thesis

The report is divided into the following chapters: Introduction, Theory, Method, Results and Discussion, and Conclusion and Future Work.
Chapter 1 offers a general overview of CFD and its significance in sensor contamination. It also discusses the advantages of utilising simulations over real-world tests and introduces the thesis problem and purpose, which outlines the structure of the report. In Chapter 2, the relevant theory required to understand the research methodology is presented. Chapter 3 presents the research methods employed to solve the problem stated in Chapter 1. Chapter 4 provides a detailed analysis of the experimental and simulation results, while the discussion of the findings is presented in the same chapter for coherence. Lastly, Chapter 5 summarises the research findings and how they relate to the thesis problem and purpose. It also suggests possible areas for future research.
Chapter 2

Theory

This chapter aims to define the theory required to comprehend the method presented in this research study and interpret the findings. The initial segment covers the fundamental equations of fluid mechanics that dictate the flow dynamics of the current problem. Subsequently, an overview of the equations essential for numerically simulating the flow of liquids over surfaces will be provided.

2.1 Fluid Mechanics

Fluid mechanics is a branch of physics that deals with the study of fluids, including liquids and gases, and their behaviour under different conditions. Fluid mechanics plays a crucial role in a wide range of applications, from aerospace engineering and bio-mechanics to chemical engineering and meteorology. Fluid mechanics explores the same basic principles found in physics and mechanics, such as Newton’s laws of motion, the conservation of mass, and the first and second laws of thermodynamics. Therefore, the fundamental laws governing fluid mechanics are similar to those that govern rigid-body and deformable-body solid mechanics, resulting in a shared approach to analysing these fields.

The current state of fluid mechanics is characterised by a growing interest in the development of advanced numerical methods for simulating fluid flow, as well as the exploration of new areas of application. With the increasing availability of high-performance computing resources, researchers are now able to perform simulations at unprecedented levels of detail, which has led to a deeper understanding of the underlying physics.

2.1.1 Fluid Properties

This section of the thesis delves into the fundamental properties that govern fluid behaviour and their relationship with fluid dynamics. The density of a fluid, denoted by the symbol $\rho$, is defined as its mass per unit volume. Liquids typically exhibit minimal changes in density
due to variations in pressure and temperature, while the density of a gas is significantly af-
fected by these factors. Although incompressibility is often assumed when analysing liquid
flow behaviour, it is important to note that real liquids are not completely incompressible.
At high pressures or temperatures, the density of liquids can change significantly, neces-
sitating more complex mathematical models and simulations to accurately predict fluid
behaviour.

Viscosity, a fundamental property of fluids, is related to their flow and deformation charac-
teristics. It can be thought of as internal friction or resistance to flow within a fluid. The
strain rate of a fluid, or the rate at which it deforms, is inversely proportional to its coeffi-
cient of viscosity, represented by the symbol \( \mu \). A comprehensive understanding of viscosity
properties is vital in designing fluid systems.

When a liquid droplet is suspended in air, it assumes a spherical shape due to the phe-
nomenon of surface tension. The attractive (cohesion) force between water molecules causes
a force asymmetry at the surface of the liquid which results in surface tension. Minimising
surface area is energetically favourable for liquids since it reduces the energy required to
maintain the attractive forces between molecules. Consequently, droplets and other liquid
bodies tend to take on spherical shapes to minimise their surface area. The ability to control
surface tension is crucial in various fields of science and engineering.

By highlighting these fundamental properties and their significance in fluid dynamics, this
section provides a foundation for the subsequent chapters of the thesis. The concepts dis-
cussed here provide the necessary background for understanding the research problem and
the methods employed to solve it.

### 2.1.2 Governing Equations

Listed below are the mathematical equations that govern the motion of a small fluid element
or infinitesimal control volume, which can be defined using three conservation laws [5]:

- Conservation of mass;
- Conservation of linear momentum;
- Conservation of energy.

The law of conservation of mass states that mass flow per unit time through a section remains
constant in a steady flow, regardless of changes in section diameter. This is described by
the continuity equation,

\[
\frac{\partial}{\partial t} \int_{cv} \rho \, dV + \int_{cs} \rho \mathbf{V} \cdot \mathbf{n}, \, dA = 0,
\]

where the first term represents the rate of change of mass with respect to time inside the
control volume, and \( \rho \) is the fluid density. The second term represents the net mass flux
through the control surface due to fluid movement across it, and accounts for the mass
flow rate into or out of the control volume. Overall, the equation ensures that the rate of
change of mass inside the control volume is equal to the net mass flux through it. To ensure the conservation of linear momentum, the total momentum of the control volume under consideration must remain constant. This conservation law is expressed by the momentum equation,

$$\frac{\partial}{\partial t} \int_{cv} (r \times V) \rho, dV + \int_{cs} (r \times V) \rho V \cdot \hat{n}, dA = \sum_{cv} (r \times F)_{cv}$$

(2)

where the left-hand side of the equation represents the time rate of change of the total momentum within the control volume and the momentum flux through the control surface, while the right-hand side represents the net force acting on the contents of the control volume. Lastly, the equation that ensures the conservation of energy,

$$\frac{\partial}{\partial t} \int_{cv} e \rho, dV + \int_{cs} e \rho V \cdot \hat{n}, dA = (\dot{Q}_{Netc, in} + \dot{W}_{Netc, in})_{cv},$$

(3)

where $e$ is the total energy per unit mass of the fluid, which includes both the internal energy and the kinetic and potential energies. The left-hand side of the equation represents the time rate of change of the total energy within the control volume and the energy flux through the control surface, while the right-hand side represents the net rate of energy transfer into the control volume through a heat transfer and work done by external forces. The terms $\dot{Q}_{Netc, in}$ and $\dot{W}_{Netc, in}$ represent the net heat transfer and work done on the system by external forces, respectively, both evaluated at the control volume. The subscript “in” indicates that these terms are evaluated only for the energy entering the control volume.

The conservation laws are used to derive the Navier-Stokes equations, which are the fundamental equations of fluid dynamics [6]. These equations describe the motion of fluids and are used extensively in CFD simulations, as they define how the velocity, pressure, temperature, and density of a moving fluid are related. The equations were derived independently by G.G. Stokes, in England, and M. Navier, in France, in the early 1800s. The equations are extensions of the Euler Equations and include the effects of viscosity on the flow.

The equations are a set of coupled differential equations and could, in theory, be solved for a given flow problem by using methods from calculus. But, in practice, these equations are too difficult to solve analytically. In the past, engineers made further approximations and simplifications to the equation set until they had a group of equations that they could solve. Recently, high-speed computers have been used to solve approximations to equations using a variety of techniques like finite difference, finite volume, finite element, and spectral methods.

### 2.1.3 Boundary Layer

In the field of aerodynamics, the interaction between an object and a fluid can generate aerodynamic forces, which depend on various factors such as the shape of the object, the speed of the object, the mass of the fluid, and the properties of the fluid, including viscosity
and compressibility. To properly model these effects, similarity parameters are utilised, which are ratios of these effects to other forces in the problem. If two experiments have the same similarity parameters, then the relative importance of the forces is being correctly modelled.

When a fluid flows past an object, the molecules near the object are disturbed, resulting in a thin layer of fluid near the surface of the object in which the velocity changes from zero at the surface to the free stream value away from the surface. This layer is known as the boundary layer, visualised in Figure 1 below, and is critical for understanding many problems in aerodynamics such as wing stall, skin friction drag, and heat transfer during high-speed flight [7].

![Diagram of a boundary layer](image)

**Figure 1:** Illustration of the development of a boundary layer.

The boundary layer can be either laminar or turbulent depending on the Reynolds number, $\text{Re}$, which is a dimensionless number defined as the ratio of inertial to viscous forces,

$$
\text{Re} = \frac{V L}{\nu},
$$

where $V$ is the velocity, $L$ is a length scale, and $\nu$ is the kinematic viscosity and is equal to the dynamic viscosity divided by the density, $\nu = \mu/\rho$. When the $\text{Re}$ number is lower, the boundary layer is laminar, and the streamwise velocity changes uniformly as one moves away from the wall. This is illustrated in the left-hand side of Figure 1. However, when the $\text{Re}$ number is higher, the boundary layer is turbulent, and the streamwise velocity is characterised by unsteady swirling flows inside the boundary layer, which is illustrated in the right-hand side of Figure 1.

The difference between laminar and turbulent flow lies in the behaviour of the fluid particles. In laminar flow, the fluid particles move in parallel layers without any disruptions or mixing, resulting in a smooth and predictable flow. However, in turbulent flow, the fluid particles move chaotically and irregularly, with eddies and vortices mixing the fluid layers. This results in an unpredictable flow pattern with fluctuations in velocity, pressure and
direction. Simulating turbulent flow is generally more challenging than simulating laminar flow. The primary reason is that the turbulent flow contains smaller-scale eddies that must be accounted for in the simulation, which requires a higher level of detail and computational power. Additionally, since turbulent flow is inherently unpredictable, simulations may require complex mathematical models that involve solving a system of differential equations, making it even more challenging.

One important aspect of turbulent flow is the formation of wakes. In fluid dynamics, a wake is a region of disturbed flow downstream of an object that moves through a fluid. The wake is formed as the fluid particles flow around the object, causing a drop in pressure and an increase in turbulence. In turbulent air flows, wakes can have a significant impact on the surrounding flow field, leading to drag, increased turbulence and mixing. The characteristics of the wake depend on the Reynolds number and the shape and size of the object.

### 2.1.4 Wettability and Droplet Dynamics

The process of wetting takes place when a droplet collides with a solid surface and then spreads across it [8]. The wetting phenomenon involves the interaction of three phases: liquid, gas, and solid. The extent of liquid spreading varies depending on the surface properties. If the surface is completely wettable, the liquid will spread uniformly and form a thin layer over the surface. However, if the surface is partially wettable, the liquid will form a thicker layer. This behaviour is characterised by the contact line, which is the point where the three phases meet, and is depicted in Figure 2 below.

![Figure 2: Contact line, or triple point, between three phases and corresponding equilibrium contact angle.](image)

In the context of wetting phenomena at rest, there is an equilibrium contact angle, which represents the angle formed between the surface and the liquid-gas surface tension force. Young’s equation provides a way to define this equilibrium contact angle by relating it to the surface tension forces of the three phases involved [9],

\[
\sigma_{lg} \cos(\theta_E) = \sigma_{sg} - \sigma_{sl},
\]

where \(\theta_E\) refers to the contact angle and \(\sigma_{sl}\), \(\sigma_{sg}\), and \(\sigma_{lg}\) the surface tensions between the liquid-gas, liquid-solid and solid-gas phases, respectively. Depending on if the droplet and
its corresponding contact line are at rest, the angle between the surface and the contact line may vary and are referred to as static and dynamic contact angles. The static angle is measured when a droplet is resting on a surface with no inclination. When the droplet is at rest, it can be classified by evaluating $\theta_s$, as depicted in Figure 3. If $\theta_s$ is less than 90 degrees the surface would be classified as hydrophilic and if $\theta_s$ is larger than 90 degrees as hydrophobic.

![Figure 3: Hydrophobicity of a droplet-based on the static contact angle.](image)

The dynamic contact angles are observed once the surface starts to tilt, resulting in the contour of the droplet changing. Note that the droplet will not immediately start to move, there exists a range of contact angles, which is referred to as contact angle hysteresis, where the droplet will stay stagnant due to adhesive forces preventing its movement. The contact angle hysteresis is enclosed by a maximum and minimum value called advancing ($\theta_a$) and receding ($\theta_r$) angle, respectively, as depicted in Figure 4.

![Figure 4: Illustration of contact angle hysteresis, where intersection points of $\theta_a$ and $\theta_r$ with the y-axis where the Capillary number is zero, Ca, highlight the start points on droplet movement.](image)
2.2. COMPUTATIONAL FLUID DYNAMICS

The capillary number \((C_a)\) is a dimensionless parameter that expresses the relative magnitude of viscous forces compared to interfacial forces and can be mathematically expressed as,

\[
C_a = \frac{\mu V_{cl}}{\sigma},
\]

where \(\mu\) is the kinematic viscosity, \(V_{cl}\) is the velocity, and \(\sigma\) is the interfacial force. Figure 5 illustrates the contour of a droplet on an inclined plane and visualises the dynamic contact angles, \(\theta_a\) and \(\theta_r\). As the surface inclination increases, there will be a point where the droplet overcomes the adhesive forces and starts to move and the contact angle at the leading edge will be referred to as advancing and receding at the trailing edge.

![Figure 5: Advancing and receding contact angles.](image)

2.2 Computational Fluid Dynamics

CFD is a rapidly growing field that has revolutionised the way engineers approach fluid dynamics problems. CFD uses numerical techniques and algorithms to simulate fluid flows and heat transfer processes in complex geometries. Residuals, which represent the difference between the calculated and actual values in the computational model, play a crucial role in assessing the accuracy and convergence of CFD simulations. With the increasing availability of high-performance computing resources and development, CFD has become an indispensable tool for the design and optimisation of a wide range of engineering systems, including aircraft, automobiles, wind turbines, and biomedical devices. By providing detailed insights into flow behaviour, CFD enables engineers to make informed decisions and design more efficient and reliable systems.

2.2.1 Multiphase Flow Modelling

This thesis deals with multiphase flows and their numerical modelling. The interface between phases is a critical factor that causes a sudden change in fluid properties and continuous exchange of mass, momentum, and heat transfer. STAR-CCM+ categorises such flows based on the interface’s increasing spatial scales, i.e., dispersed, intermittent, and stratified flows.
Two main approaches are taken when modelling multiphase flows based on the reference framework: the Euler-Euler approach that treats all phases as a continuum medium [11] and the Euler-Lagrange approach, where one phase is a continuum and the second phase is tracked from a Lagrangian framework [12]. STAR-CCM+ offers seven multiphase models for modelling all types of multiphase flows, ranging from dispersed to stratified, typically broken down based on the modelling frameworks of Euler-Euler or Euler-Lagrange.

The present study will focus on the Volume of Fluid (VOF) method [13], which is a widely used method for simulating free surface flows. Volvo Cars recommended this method as it aligns with their specific interests. The VOF model tracks the volume fraction of each fluid in a computational cell while solving a single set of momentum equations [14]. Each computational cell takes a value $0 \leq a_i \leq 1$, which represents the volume fraction of phase $i$ in the cell. $a_i = 0$ means that the cell is completely void of phase $i$, and $a_i = 1$ means that the cell is filled with phase $i$. Values between the two limits indicate the presence of an interface between phases. The volume fraction of phase $i$ is defined as,

$$a_i = \frac{V_i}{V},$$

where $V_i$ is the volume of phase $i$ in the cell and $V$ is the volume of the cell. Moreover, it should be noted that the material properties computed within the cells that include an interface are contingent upon the material properties of the individual fluids involved. These fluids that exist within the same cell as an interface are dealt with as a composite mixture and their properties are used to calculate the overall properties of the cell:

$$\rho = \sum_i \rho_i a_i,$$

$$\mu = \sum_i \mu_i a_i,$$

where, $\rho_i$ is the density and $\mu_i$ is the dynamic viscosity of phase $i$. The phase mass conservation equation governs the distribution of phase $i$,

$$\frac{\partial}{\partial t} \int_V a_i dV + \int_A a_i \mathbf{v} \cdot da = \int_V \left( S_{a_i} - \frac{a_i}{\rho_i} \frac{D\rho_i}{Dt} \right) dV - \int_V \frac{1}{\rho_i} \nabla \cdot (a_i \rho_i \mathbf{v}_{d,i}) dV,$$

where $\mathbf{a}$ is the surface area vector, $\mathbf{v}$ is the mixture (mass-averaged) velocity, $\mathbf{v}_{d,i}$ is the diffusion velocity, $S_{a_i}$ is a user-defined source term of phase $i$, and $\frac{D\rho_i}{Dt}$ is the material or Lagrangian derivative of the phase densities $\rho_i$. STAR-CCM+ calculates the volume fractions of phases as follows [14]:

- When two VOF phases are present, only the volume fraction transport for the first phase is solved. To ensure the conservation of mass, the volume fraction of the second phase is adjusted in each cell such that the sum of the volume fractions of all phases in each cell remains constant.
• When there are three more VOF phases present, the volume fraction transport is solved for all phases. The volume fraction of each phase is then normalised based on the sum of the volume fractions of all phases in each cell.

2.2.2 Kistler Correlation

To ensure accurate simulation of dynamic contact angles, it is essential to select a suitable model that can capture its complexity effectively. Several models based on empirical studies have been proposed, one of which is the Kistler Correlation introduced by Kistler in 1993 [15]. This model is an empirical dynamic contact angle model that utilises the capillary number, $C_a$, and the Hoffman function. The Kistler contact angle is defined as,

$$\theta_k = f_{Hoff}(C_a + f_{Hoff}^{-1}(\theta_s)),$$

where $f_{Hoff}$ is the Hoffman function,

$$f_{Hoff}(x) = \cos^{-1}\left(1 - \tanh\left(5.16\left(\frac{x}{1 + 1.31x^{0.99}}\right)^{0.706}\right)\right)$$

with $f_{Hoff}^{-1}$ being its inverse. STAR-CCM+ uses a modified version of the Hoffman function, which enables a closed form for its inverse function [16]. In this modification, the term $x^{0.99}$ in the original function is replaced with a simple $x$, where $x$ is the contact angle. This closed form of the inverse function is valid for contact angle values ranging from 0 to 176 degrees, with larger angles limited to 176 degrees. The Kistler Correlation was mainly chosen due to its consideration of contact angle hysteresis, a phenomenon found to be highly present in reality, depicted in Figure 4.

2.2.3 Turbulence Models

Reynolds-Averaged Navier-Stokes (RANS) turbulence models are widely employed in CFD simulations to analyse turbulent flow phenomena [17]. These models aim to solve the time-averaged Navier-Stokes equations, allowing for efficient computation of mean flow characteristics and turbulence quantities. One prominent RANS turbulence model is the K-Omega model, which incorporates the concept of eddy viscosity to capture the effects of turbulence [18]. The K-Omega turbulence model encompasses two transport equations: one for the turbulent kinetic energy ($k$) and another for the specific dissipation rate of turbulence ($\omega$). These equations describe the evolution of turbulence quantities and their interaction with the mean flow field. By solving these equations, the model provides insights into turbulence intensity, turbulent eddies, and their impact on flow behaviour.

One specific variant of the K-Omega model that has gained considerable popularity is the Shear Stress Transport (SST) model developed by Menter. The SST K-Omega model combines the advantages of both the K-Omega and K-Epsilon turbulence models, making it well-suited for a wide range of flow scenarios. It incorporates two distinct regions: the near-
wall region, where the K-Epsilon model is more accurate, and the outer region, where the K-Omega model performs better. In the near-wall region, the SST K-Omega model utilises the K-Epsilon model to accurately capture the turbulence characteristics close to the solid boundaries. This region is vital for accurately predicting wall-bounded flows, where the velocity gradients are high. Conversely, in the outer region, the model transitions to the K-Omega formulation, which offers superior performance in capturing the turbulence features away from the boundaries. The SST K-Omega model accounts for the transition between these two regions through a blending function that smoothly combines the K-Epsilon and K-Omega equations based on the local flow conditions. This transition ensures an accurate representation of both near-wall and far-field turbulence effects, making the SST K-Omega model a reliable choice for a wide range of turbulent flow simulations.
Chapter 3

Method

This chapter outlines the preparations and methodologies used for this thesis. The work was divided into two main parts: gravity experiments and wind tunnel experiments, with corresponding simulations. The gravity experiments involved testing droplet behaviour on two different surfaces: a lacquered metal sheet and a glass sheet. The experiments were conducted by suspending a droplet of water from a needle and measuring its movement using a smartphone camera. The wind tunnel experiments were carried out to study the effect of airflow on droplet behaviour. The droplet was placed on the surface of the test specimen, and the wind tunnel was used to control the airflow. The experiments were conducted at various inclinations, and the droplet’s movement was recorded using a smartphone camera. The following sections will provide a detailed description of the equipment, procedures and data analysis techniques used in each experiment, and simulations made in STAR-CCM+.

3.1 Gravity

The gravity experiments were conducted to investigate the behaviour of droplets on two different surfaces: a lacquered metal sheet and an untreated glass sheet. The choice of these materials was based on their relevance to real-world applications, both materials are commonly found on vehicles, and their different surface properties, such as surface tension and contact angle. Worth noting is that the glass sheet is not from an actual car but is rather of a more generic type, hence it does not possess the same characteristics, especially in the form of surface tension.

3.1.1 Experiments

Before starting experimenting, some preparations were necessary. To allow for a comparative analysis in STAR-CCM+ static contact angles of the lacquered metal and glass sheet are principal. These values were obtained by using a Krüss Mobile Surface Analyser (MSA), where three measurements per surface were recorded and then averaged [19]. The results
from the measurements are presented in Table 1 below and Figure 6 visualises the measurements made with the Krüss Mobile Surface Analyser.

Table 1: Contact angle (CA) measurements and standard deviation from Krüss Mobile Surface Analyser.

<table>
<thead>
<tr>
<th></th>
<th>Left CA (°)</th>
<th>Right CA (°)</th>
<th>CA (°)</th>
<th>Standard Deviation (°)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lacquered metal</td>
<td>72.9</td>
<td>73.2</td>
<td>73.0</td>
<td>5.5</td>
</tr>
<tr>
<td>Glass sheet</td>
<td>33.1</td>
<td>33.1</td>
<td>33.1</td>
<td>6.1</td>
</tr>
</tbody>
</table>

Furthermore, an approach for measuring equally sized droplets was required to assure the repeatability of the experiments. Different methods were evaluated by measuring ten, 60 µL droplets and reviewing their consistency and volume deviation. Ultimately, an Agilent 10 µL glass syringe was deemed sufficiently accurate as it had a standard deviation of 0.9989 µL, showcased in Figure 7.

Figure 6: Contact angle measurement made with Krüss Mobile Surface Analyser.
Moreover, some composing of ultraviolet (UV) light positioning and UV-liquid concentration was required. UV-liquid was used to illuminate the droplet in the dark, allowing for greater contrast when converting the recording to grey scale. Fair results were attained when the UV light illuminated the droplet straight from above, normal to the tested surface. This was mainly due to shadows being absent which was a problem from other angles. Shadows could in turn disrupt the image processing software, see Figure 8 for an illustration of the difference. Another approach for minimising the risk of shadows was increasing the UV-liquid concentration. In the end, a concentration of 0.2 ml of UV-tracer to 100 ml of water resulted in a satisfying appearance of the droplet.

The last preparation was finding an appropriate cleaning solution for the surfaces. In the end, a solution of 96 per cent ethanol and four per cent water resulted in a surface without residues that could affect the behaviour of the droplet. Its quick evaporation added further to its suitability as it granted less waiting and quicker testing.
CHAPTER 3. METHOD

For this part, the surface was securely taped to a motor-actuated table, allowing for inclinations between 0° and 75°. Also attached to the table were two cameras, recording the droplet from two angles; top and side view. More specifically an iPhone 13 Pro recorded the side view, while an iPhone 14 Pro recorded the top view. Both recorded 1080p in 60fps, utilising the iPhone’s built-in macro mode with a fixed exposure of -2.0. The mounting of the phones allowed them to rotate with the table, resulting in a constant perspective. Smartphones were chosen as recording devices due to their high camera quality in combination with their accessibility.

For the droplet, a volume of 60 \( \mu L \) was applied to the surface, which meant fully draining the syringe six times. The selection of this droplet size was based on its adherence to the typical range of water droplet sizes \[20\]. At this size, it was discovered that the droplet began to move at an inclination of 39° and 22° for the lacquered metal and glass sheet, respectively. This was used as the initial angle at which droplet movement was analysed. As seen in Table 2 the inclination was increased at increments of five degrees, five times. Each angle was tested with ten different droplets. The rotational speed of the motor-actuated table was also carefully chosen to ensure that the droplet did not start to flow before reaching the desired inclination. A speed of 6.98 \( \text{deg/s} \) was used as it provided sufficient force to rotate the table while also ensuring that the droplet remained in-frame until the desired inclination was reached.

To verify the droplet velocity, some scale must be present in the videos. This was attained by taking a photograph of a ruler on the surface, from the same angle and distance. This photograph could then be superimposed onto the video and used as a reference.

<table>
<thead>
<tr>
<th>Inclination (deg)</th>
<th>Lacquered metal</th>
<th>Glass sheet</th>
</tr>
</thead>
<tbody>
<tr>
<td>39</td>
<td>44</td>
<td>22</td>
</tr>
<tr>
<td>49</td>
<td>49</td>
<td>27</td>
</tr>
<tr>
<td>54</td>
<td>54</td>
<td>32</td>
</tr>
<tr>
<td>59</td>
<td>59</td>
<td>37</td>
</tr>
</tbody>
</table>

3.1.2 Video analysis

The contact angles and velocity of physical droplets were analysed using a video processing software called Sessile Drop Analysis, which was developed by Dr Mathijs van Gorcum. To input the video into the software, it was necessary to first process it into greyscale with the droplet appearing in black. The software then identified various properties of the droplet, including the contact angles, drop volume, contact line, and centre of mass (CoM). The contact angles were determined through sub-pixel edge detection, using either linear interpolation between two pixels around the edge or by fitting an error function around the edge. \[21\]

It was critical to ensure that the contact angle measurements obtained from the software were accurate. To validate the software’s results, the static contact angles were compared to those obtained by the Kruess Mobile Surface Analyser. It was found that the static contact
angle measured by the software were within the standard deviation of the static contact angles from the Krüss Mobile Surface Analyser, indicating that the software was suitable for this study. The contact line position difference between each frame was then utilised to determine the droplet velocity. Moreover, the software includes a feature that enables the user to use a crop box to regulate the area of the video being analysed, thereby minimising measurement noise.

Although choosing a relatively high angular velocity for the table was necessary, it simultaneously resulted in disturbing the software once it stopped rotating, as depicted in Figure 9. When the rotation ceased both the advancing and receding angles spike, especially the receding angle, something that might have consequences when deriving the results later. This was, however, an adequate compromise as the droplet would not have been in frame with a lower angular velocity, as stated earlier.

\subsection{Comparison to the Kistler Correlation}

The data gathered from the gravity experiments contained a significant amount of noise, which could affect the accuracy of calculations made with the data. To mitigate this, data cleaning was performed using the moving mean method. This method effectively removes high-frequency noise from the data and provides a smooth version of the original data, which is essential for accurate calculations. Figure 9 illustrate the noise reduction process for one of the tests, with (a) showing the original data before noise reduction and (b) showing the data after noise reduction using the moving mean method. As seen in (b), the noise reduction process results in a smoother curve, which is more amenable to accurate analysis.

After reducing the noise, the velocity of the droplet was calculated using various parameters. The velocity was calculated for each frame by dividing the distance travelled per frame by the time it took to travel that distance in that specific frame. This calculation was looped through for all tests, and the results were then appended. To further analyse the data, linear regression was implemented on the appended results. Linear regression was selected due to the linear relationship exhibited by the Kistler Correlation within the examined interval. The resulting values were plotted along with the original data. Lastly, the results were compared to the Kistler correlation to determine whether the model used to simulate the droplet corresponds well with real-world experiments.
3.1.4 Simulations

First, a description of the general settings applicable to both gravity and wind tunnel simulations will be provided. These simulations aim to study the motion of droplets under the effects of gravity and airflow, respectively. The software used for these simulations was STAR-CCM+, version 23.02. A hexahedron mesh type was employed in the setup, with adaptive mesh refinement (AMR) applied to the free surface of the droplet. The refinement level was set to three, the transition width was set to five and a limited cell size of $5 \times 10^{-5} m$.

The AMR aims to further refine the mesh resolution, in this case on the interface between the droplet and the air. The droplet itself was defined using a specific VOF field function, which injected the droplet at predetermined coordinates with a specified volume matching that of the physical experiments. Surfaces and boundaries were defined to simulate the
3.1. GRAVITY

inclined table and the environment surrounding the droplet. Multiphase modelling was realised using VOF, while dynamic contact angle modelling employed the Kistler correlation, where Table 3 displays the contact angles used for the surfaces. Note that the table displays the contact angles when the droplet starts moving, not static contact angles. Moreover, water was designated as the primary phase, and air as the secondary phase. To control the update at each physical time, an implicit unsteady solver was utilised, paired with an adaptive time step.

Table 3: Advancing and receding contact angles used for the Kistler model in STAR-CCM+.

<table>
<thead>
<tr>
<th></th>
<th>Lacquered</th>
<th>Glass</th>
</tr>
</thead>
<tbody>
<tr>
<td>Advancing Contact Angle ($deg$)</td>
<td>91.6</td>
<td>81.4</td>
</tr>
<tr>
<td>Receding Contact Angle ($deg$)</td>
<td>21.2</td>
<td>20.6</td>
</tr>
</tbody>
</table>

The gravity simulations involved the utilisation of a cuboid with well-defined surfaces. The surface on which the droplet advanced was characterised as a wall, while the remaining surfaces were designated as pressure outlets. Given the low velocities and correspondingly small Reynolds numbers, a laminar model was selected. To replicate the rotation of the actuated table, modifications were made to the gravity field function to introduce time dependence, as described by the following equation 13

\[
(Time > t) = \left[9.81 \cdot \sin\left(\theta/57.3\right), -\cos\left(\theta/57.3\right) \cdot 9.81, 0\right],
\]

\[
: \left[9.81 \cdot \sin\left((38.8 + Time \cdot 6.98)/57.3\right),
- \cos\left((38.8 + Time \cdot 6.98)/57.3\right) \cdot 9.81, 0\right].
\]

Where $Time$ is the physical time passed in the simulation. If $Time$ is greater than $t$, which is the time it takes for the table to rotate to the desired inclination $\theta$, the inclination is constant at $\theta$. However if smaller than $t$, it will instead increase until the desired value is reached. Finally, 6.98 is the angular velocity of the table in $deg/s$. Furthermore, the physical simulation time varied for each inclination, where lower inclinations were simulated for longer, all to emulate the physical experiments.

For the gravity simulations, the aim was to attain similar velocities to those achieved during the physical experiments and provide a foundation for the wind tunnel simulations. However, the velocity did not converge, rather it changed depending on the size of the time step, mesh size or other factors determining how well the solution was resolved. In the end, when criteria such as appearance, volume continuity and interface were fulfilled, only the time step was manipulated. The time step had an inverse relationship to the velocity, whereas if it decreased, the velocity increased. To approach this, the time step was decided by focusing on one inclination, for one of the surfaces and using it for all simulations. This was primarily done to make the results comparable. The performance of the simulations was monitored by analysing the mass conservation of the droplet, the contact line velocity and the residuals. Each plot was obtained through appurtenant field functions.
CHAPTER 3. METHOD

3.2 Wind Tunnel

This section provides a comprehensive description of the physical experiments in the wind tunnel, as well as the simulations. The experiments were carried out by placing the motor-actuated table, along with the surface(s) and droplet, inside a closed model wind tunnel of scale 1:5. The primary objective of the experiments was to determine the critical airflow velocity required to initiate droplet motion at various inclination angles and to use the results to validate the simulations in STAR-CCM+.

3.2.1 Experiments

To ensure the motor-actuated table did not damage the wind tunnel, move/vibrate and affect the droplet movement and have repeatable placement, it was fastened to the wind tunnel by using a ratchet strap. In addition to this, some features from the gravity experiments were carried over as well, such as the cleaning procedures, droplet application and video recording from the side view.

To attain the airflow velocity of the wind tunnel two pitot tubes were attached to the inside of the wind tunnel, one on the roof and one on the side wall. The pitot tubes were connected to Sensor Technics HCLA0025DB pressure sensors, one each, which in turn was coupled to a DEWESoft Sirius Data Acquisition (DAQ) system. Through calibration of the pressure sensors, the DEWESoft X software could translate the voltage readings into pressure values, which were monitored in real-time. Subsequently, the pressure values were recalculated to obtain velocity information.

To prevent the droplet from ending up in the wake of the table, all tests were done at an inclination, with a total of three inclinations per surface with five-degree increments, as per Table 4. Further reassuring the droplet being hit by the airflow was attaching a small cylindrical tube in front of the table to break up the flow, allowing it to reattach to the surface earlier.

Moving on, the methodology was similar to the gravity experiments. The surface was cleaned with ethanol and applied with a 60 µL droplet when dried. The sliding table was put back into the wind tunnel and sealed off and the airflow could be engaged to a set velocity. Each inclination was tested with five droplets. Facilitating observing droplet movement was a video camera recording the side view. The recording also allowed matching the DAQ to the video, making them easily comparable in post-analysis.

Table 4: Inclination increments of the motor actuated table for the different surfaces in the wind tunnel experiments.

<table>
<thead>
<tr>
<th>Surface</th>
<th>Inclination (deg)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lacquered metal</td>
<td>25 30 35</td>
</tr>
<tr>
<td>Glass sheet</td>
<td>10 13 18</td>
</tr>
</tbody>
</table>
3.2. WIND TUNNEL

3.2.2 Simulations

To investigate the droplet motion on surfaces under the influence of airflow, the wind tunnel experiments were replicated in STAR-CCM+. Due to time limitations, only the lacquered surface was simulated. To ensure accurate replication, the set-up used in the gravity simulations was used as a starting point and necessary changes were made to accommodate the wind tunnel experiments. The geometry of the motor-actuated table was created using CAD. Only the table surface, the lacquered sheet, and the cylinder to reduce the wake were included in the simulation to reduce computational time. The fluid domain was defined as a large cuboid with appropriate boundary conditions since modelling the wind tunnel was deemed too complex. Different physics models were used compared to the gravity simulations, such as Reynold-Averaged Navier-Stokes (RANS) and the K-Omega turbulence model family in combination with SST (Menter) K-Omega, to accommodate the changes required for the wind tunnel experiments. Lastly, the Y+ wall treatment physics model was desirable for turbulence resolution near walls.

Before simulating the turbulent flow, a steady state flow was simulated with only airflow to get a good starting point for the turbulent flow. In conjunction with the steady-state simulation, a porous region, depicted in Figure 10, was defined where the droplet was injected. A porous region refers to a specific area or volume within a computational domain that is characterised by its ability to allow fluid flow to pass through it while offering resistance to that flow. In this particular case, the designated region was configured to prevent any fluid from passing through. This setup ensured that the airflow in that area was effectively zero during droplet insertion, preventing the droplet from being dispersed or disturbed when running the implicit unsteady simulation.

![Figure 10: Porous region, highlighted in the red circle, in the steady state simulation before running implicit unsteady.](image)

Once the steady state flow had converged, the porous region was removed and the turbulent simulation was started by setting the airflow velocity to match the experimental conditions, ramping up the velocity between 8 m/s to 18 m/s, and placing the droplet on the surface at the same location and orientation as in the physical experiments. The simulation was then run for a defined period to allow for droplet motion and interaction with the airflow. Equation 14 shows the field function used in STAR-CCM+ to linearly vary the velocity,

\[ (\text{Time} > 2) ? 18 : (8 + \text{Time} \times 5), \quad (14) \]
where $Time$, again, is the physical time passed in the simulation. If $Time$ is larger than two then the velocity is constant at 18 m/s otherwise the velocity increases linearly from 8 m/s up to 18 m/s.

To validate the simulation results, the simulation output was compared to the experimental data obtained from the wind tunnel experiments. This comparison allowed for the identification of any discrepancies between the simulated and experimental droplet motion, as well as an assessment of the accuracy of the simulations.

### 3.3 Mesh Convergence Study

To ensure the reliability and accuracy of the simulation results, a mesh convergence study was performed to investigate the droplet motion simulations. The study encompassed the examination of the mesh size, AMR parameters, and adaptive time step to assess their influence on the droplet’s appearance, contact line velocity, and volume. The convergence analysis specifically focused on a gravity simulation for the lacquered surface, at an inclination of 49 degrees. It should be noted that the convergence study was limited to this specific inclination due to challenges encountered in achieving consistent contact line velocity values across various inclinations when altering the time step. Consequently, a compromise was made wherein a suitable time step was determined for the selected inclination, and this value was subsequently employed in all simulations to ensure uniformity and comparability.

The study commenced by establishing a baseline mesh size of $5 \times 10^{-4} m$, which was then refined by generating a series of finer meshes with decreasing sizes down to $3 \times 10^{-5} m$. Simulations were performed for each mesh size, and the results were compared to assess mesh convergence. It was determined that a mesh size of $2.5 \times 10^{-4} m$ offered the optimal combination of resolution and accuracy for capturing the behaviour of the droplet.

To further enhance simulation accuracy while minimising computational costs, AMR was employed on the free surface of the droplet. The transition width parameter was systematically tested across a range from one to eight, the minimum adaption size was set at $1 \times 10^{-4} m$, and the maximum refinement level was explored from one to five. Following an in-depth analysis, it was concluded that the most favourable settings for the adaptive mesh parameters were a transition width of five, a minimum adaption size of $5 \times 10^{-5} m$, and a maximum refinement level of three. These parameters facilitated efficient adaptivity and ensured that the mesh appropriately captured the intricate dynamics of the droplet.
Chapter 4

Results and Discussion

This chapter presents and discusses the results from the experiments and simulations described in Chapter 3.

4.1 Gravity

This section presents the obtained results for the gravity experiments and the corresponding simulations.

4.1.1 Experiments

The gravity experiments aimed to determine the velocity of the droplet and to compare the results obtained from the Sessile Drop Analysis with the Kistler Correlation, a dynamic contact angle model implemented in STAR-CCM+. The lacquered surface yielded consistent outcomes, presented in Table 5, showing an increase in velocity with higher inclinations. The glass surface demonstrated a similar trend, except for the highest inclination of 42° per Table 6, where deviations were observed. The deviations can be attributed to challenges faced by the software in accurately capturing the lower contact angles exhibited on the glass surface. Additionally, maintaining the droplet within the crop box at higher inclinations proved to be complex, which likely further contributed to the deviation.

It is intriguing to observe that the onset of water droplet movement occurs earlier on the glass surface, despite its higher hydrophilicity compared to the lacquered surface. Intuitively, one would expect a hydrophilic surface to exhibit greater droplet adhesion in comparison to a hydrophobic surface. However, several factors may contribute to this unexpected behaviour.

One possible cause for the droplet to start running earlier on a hydrophilic surface could be the presence of surface defects or irregularities. While the surface may have an overall hydrophilic nature, localised variations such as roughness or imperfections can create pref-
CHAPTER 4. RESULTS AND DISCUSSION

Differential paths for the droplet to overcome its adhesive forces and initiate movement. These irregularities can disrupt the cohesive forces within the droplet, leading to an earlier onset of running. Moreover, it is of interest to explore and ascertain whether the initiation of water droplet movement on regular glass surfaces occurs at an earlier or later stage compared to treated glass surfaces.

Table 5: Average droplet velocity in on the lacquered metal sheet for the five inclinations.

<table>
<thead>
<tr>
<th>Inclination (deg)</th>
<th>39</th>
<th>44</th>
<th>49</th>
<th>54</th>
<th>59</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean Velocity (m/s)</td>
<td>0.0026</td>
<td>0.0090</td>
<td>0.0112</td>
<td>0.0114</td>
<td>0.0116</td>
</tr>
</tbody>
</table>

Table 6: Average droplet velocity in on the glass surface for the five inclinations.

<table>
<thead>
<tr>
<th>Inclination (deg)</th>
<th>22</th>
<th>27</th>
<th>32</th>
<th>37</th>
<th>42</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean Velocity (m/s)</td>
<td>0.0010</td>
<td>0.0020</td>
<td>0.0049</td>
<td>0.0106</td>
<td>0.0068</td>
</tr>
</tbody>
</table>

The accuracy of the model is crucial to ensure a close representation of reality. Figure 11 and Figure 12 depict the outcomes of two gravity experiments conducted on the lacquered and glass surfaces, respectively, and their comparison with the Kistler Correlation. Further results can be found in Appendix 5. The scattered data points represent the raw experimental data, which were processed according to the methodology described in the preceding chapter. Linear regression analysis was performed on the raw data, yielding the blue and red lines, while the black and dark red lines correspond to the advancing and receding angles predicted by the Kistler Correlation, respectively.

Figure 11: Experimental data for the lacquered surface (49 degrees) compared to the Kistler Correlation.
The comparison of the obtained results with the Kistler Correlation reveals a favourable agreement between the model and the real-world data, as depicted in the aforementioned figures. Particularly noteworthy is the close alignment of the advancing angle measurements for both the lacquered and glass surfaces with the predictions of the Kistler Correlation, indicating the validity of the model within the scope of this project. It is noteworthy to emphasise that the contact angles exhibit a nearly constant behaviour for the considered low velocities. The observed alterations in contact angles in relation to velocity are substantially smaller compared to the inherent variability and fluctuations observed in the measurement values.

A less satisfactory match is observed for the receding angle measurements. This discrepancy can be attributed to limitations in the accuracy of the software used to measure contact angles, especially in cases where the receding angle was relatively small and left residual traces that were detected by the software. Consequently, the quality of the collected data for the receding angle was comparatively lower than that for the advancing angle. Nevertheless, considering these limitations, the data obtained for the receding angle remain acceptable for achieving the objectives of this project.

Despite the challenges faced, the advancing angle measurements exhibited robustness and were deemed reliable for analysis. It is acknowledged that further advancements in measurement techniques or alternative software solutions may enhance the accuracy of receding angle measurements in future investigations. However, it is important to note that even more precise measurements of the contact angles may not yield significant differences. The current method employed in this study proved to be highly comparable to the Kistler Correlation for lower velocities, suggesting that the accuracy achieved was already quite close to the established benchmark. Nonetheless, exploring higher velocities in future studies would
be of interest to determine if any notable distinctions emerge.

4.1.2 Simulations

Figure 13 illustrates that the simulated droplet velocities do not exhibit the same behaviour as observed in the physical experiments, where higher inclinations corresponded to higher droplet velocities. Instead, the simulated velocities appear random without a discernible pattern. For instance, at an inclination of 22°, the glass surface exhibits the highest velocity, contrary to the experimental findings. However, this outcome was anticipated, as achieving velocity convergence has been a major challenge throughout the thesis. After extensive discussions with the STAR-CCM+ support team, it was determined that the lack of convergence is attributed to the semi-implicit treatment of the surface-tension force in the momentum equation, which is proportional to the time-step. This treatment diffuses any force-pressure imbalances across the interface that may arise due to insufficient resolution of the interface. To address this issue, the resolution of the interface was enhanced by increasing the refinement level to four and decreasing the limited cell size to 10µm, as suggested by the support team. Unfortunately, despite these efforts, no improvements were achieved, and due to time constraints, further investigation into the issue was not possible. Issues related to velocity would be problematic when e.g. simulating a sensor to see if water is dispersed or not, and if so at what speed. Judging by these results, that would not be possible.

Also, worth noting is that the initial spike is due to the initialisation of the adaptive mesh and the droplet beginning to tilt because of the gravity and overall initial conditions. As seen from Figure 13 this phenomenon was present for all inclinations, for both surfaces and was difficult to mitigate.

By analysing Figure 14 and Table 7 it is noticeable that both surfaces did not exhibit ideal mass conservation, with the glass surface being less satisfactory. Generally though, achieving perfect mass conservation can be challenging due to the discrete nature of computational
4.1. GRAVITY

grids and the complexities of fluid flow phenomena. The numerical algorithms used to track and update the interface can introduce errors and inconsistencies, accumulating over time. It is also worth noting that while attaining mass continuity is desirable, some variations in mass can be expected in practical simulations. The focus can rather be put on minimising these variations, something that is achieved here, at least for the lacquered surface. Finally, measures to mitigate and reduce continuity issues have been employed through e.g. adaptive mesh refinement and adaptive time step.

Other than what is mentioned above, the higher volume change of the glass surface might be a consequence of the lower contact angles. This correlation was discovered when experimenting with different contact angles, where especially a low receding contact angle made the droplet elongated and the volume increase significantly. This would most likely be less of a problem for an actual car window since they’re hydrophobically treated.

Lastly, the appearance of the droplet, which is depicted for the lacquered surface at 49° in Figure 15. As seen, the droplet has a physical appearance to it, much like the experiments, along with a sharp interface due to the AMR. The colouring of the figure represents the volume fraction of water, where red equals water and blue equals air.

Table 7: Air velocity at droplet movement in Wind tunnel tests.

<table>
<thead>
<tr>
<th>Inclination (deg)</th>
<th>39</th>
<th>44</th>
<th>49</th>
<th>54</th>
<th>59</th>
<th>22</th>
<th>27</th>
<th>32</th>
<th>37</th>
<th>42</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volume Change (%)</td>
<td>2.8</td>
<td>4.4</td>
<td>2.7</td>
<td>2.3</td>
<td>3.2</td>
<td>11.0</td>
<td>12.6</td>
<td>11.0</td>
<td>10.1</td>
<td>9.5</td>
</tr>
</tbody>
</table>

Figure 14: Volume continuity for the different inclinations, both surfaces.

Lastly, the appearance of the droplet, which is depicted for the lacquered surface at 49° in Figure 15. As seen, the droplet has a physical appearance to it, much like the experiments, along with a sharp interface due to the AMR. The colouring of the figure represents the volume fraction of water, where red equals water and blue equals air.
4.2 Wind Tunnel

This section presents the obtained results for the wind tunnel experiments and the corresponding simulations.

4.2.1 Experiments

The results obtained from the wind tunnel experiments are summarised in Table 8 below. Comprehensive results from each test can be found in Appendix 5 for further reference and analysis. Consistent with expectations, it is observed that a greater inclination necessitates an increased air velocity to initiate droplet movement, for both the lacquered and glass surfaces.

<table>
<thead>
<tr>
<th>Inclination (deg)</th>
<th>Lacquered</th>
<th>Glass</th>
</tr>
</thead>
<tbody>
<tr>
<td>25</td>
<td>13.7</td>
<td>11.0</td>
</tr>
<tr>
<td>30</td>
<td>15.9</td>
<td>11.1</td>
</tr>
<tr>
<td>35</td>
<td>16.1</td>
<td>11.6</td>
</tr>
</tbody>
</table>

In the experiments conducted for the lacquered surface, the occurrence of droplet wobbling and the release of smaller droplets were observed. The release of a smaller droplet only occurred for the two higher inclinations, 30 and 35 degrees, and is illustrated in Figure 16. Neither of these phenomena was observed for the glass surface. The droplet wobbling and the release of smaller droplets can be attributed to several factors. One possible cause is the interaction between the droplet and the airflow in the wind tunnel. As the air flows around the droplet, it creates fluctuations and disturbances in the vicinity of the droplet, leading to its wobbling motion. These fluctuations may be amplified at certain inclinations and higher velocities, causing more pronounced wobbling effects.

Furthermore, the release of smaller droplets before the main droplet movement can be attributed to the presence of instabilities in the droplet shape or surface tension effects. As the airflow interacts with the droplet, it can induce deformations or instabilities in the droplet’s shape, causing it to break into smaller droplets before the entire droplet starts to move. Surface tension, which governs the cohesive forces within the droplet, can also
contribute to the formation of smaller droplets when the droplet is subjected to external forces.

It is important to note that the specific mechanisms causing the observed phenomena may require further investigation and analysis. Factors such as droplet size, surface properties, and airflow characteristics can all play a role in the observed behaviour.

4.2.2 Simulations

The results presented in this section pertain exclusively to the lacquered surface, as simulations were solely conducted for this particular surface. The simulations conducted in this study did not exhibit any observable droplet wobbling or the release of smaller droplets, which were distinct phenomena observed in real-world experiments. Replicating these behaviours accurately within the simulations poses a notable challenge due to the complexity involved in capturing such intricacies. Several factors contribute to this complexity, including the interaction between the droplet and the surrounding airflow, surface characteristics, and the inherent limitations of the chosen CFD model. Additionally, the interaction between the droplet and the surface could be further investigated by considering the effects of surface roughness, wetting properties, and the influence of boundary layer flows. These factors play a significant role in the behaviour of droplets on surfaces and can greatly impact their motion and ultimate detachment. Figure 17 illustrates the droplet appearance in the simulations.
CHAPTER 4. RESULTS AND DISCUSSION

To ascertain the initial movement of the droplet in the simulations, monitoring of the droplet’s contact line position was conducted. Figure 18 visually represents the variations in the contact line position observed during each of the three simulations. Through examination of this figure, the critical air velocities necessary to initiate droplet movement can be deduced. The determination of droplet movements was based on the point at which the change in the contact line position reached a relatively constant state. Specifically, this occurred at approximately 1.2 seconds for an inclination of 25 degrees, 1.5 seconds for an inclination of 30 degrees, and 1.7 seconds for an inclination of 35 degrees.

![Figure 17: Droplet appearance of the lacquered surface at 30°.](image)

The resultant air velocities required to initiate the movement of the droplet are summarised in Table 9 below. The simulations exhibit a consistent trend with the physical experiments, wherein a higher inclination necessitates a correspondingly higher air velocity to induce droplet movement. Upon comparing the simulated velocities with the results obtained from the physical experiments, it is evident that there is a discrepancy. The physical experiments

![Figure 18: Change in contact line position.](image)
4.2. WIND TUNNEL

recorded velocities of 13.7 m/s, 15.9 m/s, and 16.1 m/s, which deviate from the simulated values. This disparity suggests that the current simulation setup and modelling techniques require further refinement to better align with real-world observations.

The observed differences between the simulated and experimental velocities may be attributed to several factors. Firstly, the accuracy of the physical measurements plays a crucial role. Although efforts were made to ensure precise velocity measurements, inherent limitations in the measurement devices or environmental conditions could have introduced some level of error.

Table 9: Initial droplet movement in the simulations.

<table>
<thead>
<tr>
<th>Inclination (deg)</th>
<th>25</th>
<th>30</th>
<th>35</th>
</tr>
</thead>
<tbody>
<tr>
<td>Free Stream Velocity (m/s)</td>
<td>14.0</td>
<td>15.5</td>
<td>16.5</td>
</tr>
</tbody>
</table>

Analysing the results presented in Figure 19 and Table 10, it is evident that the wind tunnel simulations also exhibit deviations from ideal mass conservation, mirroring the observations made in the gravity simulations. As mentioned earlier, attaining perfect mass conservation in computational fluid dynamics simulations is a complex task. Furthermore, it is important to consider the limitations and trade-offs associated with achieving perfect mass conservation. The discrete nature of computational grids inherently introduces some level of numerical error, and complete elimination of such errors may not be feasible. Instead, the emphasis should be on quantifying and minimising these errors to ensure that the simulations provide meaningful and reliable results.

Figure 19: Volume fluctuation for each inclination.

Table 10: Volume Change Over the Course of the Simulation.

<table>
<thead>
<tr>
<th>Inclination (deg)</th>
<th>25</th>
<th>30</th>
<th>35</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volume Change (%)</td>
<td>3.0</td>
<td>9.6</td>
<td>2.5</td>
</tr>
</tbody>
</table>
4.3 Residuals

Upon analysing the residuals depicted in Figure 20, it is evident that both surfaces demonstrate similar performance, with the water residuals displaying smaller and more consistent values compared to the continuity residuals. The oscillatory nature of the continuity residuals indicates instability, which aligns with the observed increase in volume. Additionally, it is worth noting that the magnitude of the continuity residuals is relatively high, particularly in the later stages of the simulations. However, when considering the progress achieved earlier in the thesis, these levels can be deemed acceptable. Furthermore, it should be mentioned that the continuity residuals do not exhibit strong convergence behaviour.

(a) Residuals for the lacquered surface.

(b) Residuals for the glass surface.

Figure 20: Continuity and water residuals for the gravity simulations, both surfaces.

When comparing the residuals obtained from the gravity simulations to the wind tunnel experiments, notable differences emerge. In particular, the continuity and water residuals in the wind tunnel experiments, depicted in Figure 21, were consistently lower compared to the gravity experiments. This indicates that the simulated droplet behaviour is more closely
4.3. RESIDUALS

aligned with the real-world observations in the wind tunnel setup. Notably, the continuity residuals in the wind tunnel experiments exhibited improved convergence behaviour, with smaller oscillations and closer proximity to zero. This suggests a higher level of stability and accuracy in the numerical simulations within the wind tunnel context. Conversely, the simulations conducted under gravity conditions displayed larger and more fluctuating continuity residuals, indicating a greater degree of uncertainty and instability.

![Figure 21: Continuity and water residuals for the wind tunnel simulations.](image-url)
Chapter 5

Conclusion and Future Work

In summary, this study has demonstrated the feasibility of water simulation in CFD, particularly using STAR-CCM+. The simulations have shown that droplets respond realistically to gravity and airflow, capturing their behaviour and visual appearance. The inclusion of surface tension and the ability to manipulate contact angles have proven crucial in accurately modelling droplet motion. However, there are limitations to consider, such as the instability and ease of manipulation of droplet velocity, which may impact the usefulness of the simulations in certain scenarios.

Moving forward, it is recommended to explore alternative CFD tools to further improve the accuracy and reliability of water simulations. Additionally, future research should focus on simulating fluid films and rivulets, as these aspects are of interest and relevance in understanding water transportation on vehicle surfaces. Another significant limitation is the simplifications made in the experimental setup and methodology. The experiments were conducted under controlled laboratory conditions, which may not fully replicate the complex dynamics observed in real-world scenarios. This consideration is essential to bear in mind for future investigations, as the simplifications introduce inherent uncertainties and potentially limit the generalizability of the obtained results. By addressing these areas, Volvo Cars can enhance their understanding and predictive capabilities of water behaviour in CFD simulations, bringing Volvo Cars closer to their ultimate goal of using simulations to study water transportation exclusively.
CHAPTER 5. CONCLUSION AND FUTURE WORK
References


Appendix A

Additional Results from Gravity Experiments

Figure 1: Experimental data for the glass surface (39 degrees) compared to the Kistler Correlation.
Figure 2: Experimental data for the glass surface (44 degrees) compared to the Kistler Correlation.

Figure 3: Experimental data for the glass surface (54 degrees) compared to the Kistler Correlation.
Figure 4: Experimental data for the glass surface (59 degrees) compared to the Kistler Correlation.

Figure 5: Experimental data for the glass surface (22 degrees) compared to the Kistler Correlation.
Figure 6: Experimental data for the glass surface (27 degrees) compared to the Kistler Correlation.

Figure 7: Experimental data for the glass surface (37 degrees) compared to the Kistler Correlation.
Figure 8: Experimental data for the glass surface (42 degrees) compared to the Kistler Correlation.
Appendix B

Individual Results from Wind Tunnel Experiment

Table 1: Results of Wind tunnel tests for the lacquered surface 35 degrees.

<table>
<thead>
<tr>
<th>Test Number</th>
<th>Velocity Up ((m/s))</th>
<th>Velocity Side ((m/s))</th>
<th>Mean Velocity ((m/s))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>16.8</td>
<td>16.4</td>
<td>16.6</td>
</tr>
<tr>
<td>2</td>
<td>16.3</td>
<td>15.9</td>
<td>16.1</td>
</tr>
<tr>
<td>3</td>
<td>16.2</td>
<td>15.8</td>
<td>16.0</td>
</tr>
<tr>
<td>4</td>
<td>15.6</td>
<td>16.0</td>
<td>15.8</td>
</tr>
<tr>
<td>5</td>
<td>16.3</td>
<td>15.8</td>
<td>16.05</td>
</tr>
<tr>
<td>Overall Mean Velocity ((m/s))</td>
<td></td>
<td></td>
<td>16.1</td>
</tr>
</tbody>
</table>

Table 2: Results of Wind tunnel tests for the lacquered surface 30 degrees.

<table>
<thead>
<tr>
<th>Test Number</th>
<th>Velocity Up ((m/s))</th>
<th>Velocity Side ((m/s))</th>
<th>Mean Velocity ((m/s))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>15.8</td>
<td>15.5</td>
<td>15.65</td>
</tr>
<tr>
<td>2</td>
<td>16.6</td>
<td>16.2</td>
<td>16.4</td>
</tr>
<tr>
<td>3</td>
<td>16.0</td>
<td>15.6</td>
<td>15.8</td>
</tr>
<tr>
<td>4</td>
<td>16.1</td>
<td>15.8</td>
<td>15.95</td>
</tr>
<tr>
<td>5</td>
<td>16.2</td>
<td>15.9</td>
<td>16.05</td>
</tr>
<tr>
<td>Overall Mean Velocity ((m/s))</td>
<td></td>
<td></td>
<td>15.9</td>
</tr>
</tbody>
</table>

Table 3: Results of Wind tunnel tests for the lacquered surface 25 degrees.

<table>
<thead>
<tr>
<th>Test Number</th>
<th>Velocity Up ((m/s))</th>
<th>Velocity Side ((m/s))</th>
<th>Mean Velocity ((m/s))</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>13.7</td>
<td>13.3</td>
<td>13.5</td>
</tr>
<tr>
<td>2</td>
<td>13.9</td>
<td>13.6</td>
<td>13.75</td>
</tr>
<tr>
<td>3</td>
<td>13.9</td>
<td>13.6</td>
<td>13.75</td>
</tr>
<tr>
<td>4</td>
<td>13.9</td>
<td>13.6</td>
<td>13.75</td>
</tr>
<tr>
<td>5</td>
<td>13.9</td>
<td>13.6</td>
<td>13.75</td>
</tr>
<tr>
<td>Overall Mean Velocity ((m/s))</td>
<td></td>
<td></td>
<td>13.7</td>
</tr>
</tbody>
</table>
Table 4: Results of Wind tunnel tests for the glass surface 18 degrees.

<table>
<thead>
<tr>
<th>Test Number</th>
<th>Velocity Up (m/s)</th>
<th>Velocity Side (m/s)</th>
<th>Mean Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11.7</td>
<td>11.2</td>
<td>11.45</td>
</tr>
<tr>
<td>2</td>
<td>11.8</td>
<td>11.2</td>
<td>11.5</td>
</tr>
<tr>
<td>3</td>
<td>12.1</td>
<td>11.4</td>
<td>11.75</td>
</tr>
<tr>
<td>4</td>
<td>11.8</td>
<td>11.4</td>
<td>11.6</td>
</tr>
<tr>
<td>5</td>
<td>11.9</td>
<td>11.4</td>
<td>11.65</td>
</tr>
<tr>
<td>Overall Mean Velocity (m/s)</td>
<td></td>
<td></td>
<td>11.6</td>
</tr>
</tbody>
</table>

Table 5: Results of Wind tunnel tests for the glass surface 13 degrees.

<table>
<thead>
<tr>
<th>Test Number</th>
<th>Velocity Up (m/s)</th>
<th>Velocity Side (m/s)</th>
<th>Mean Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11.1</td>
<td>10.7</td>
<td>10.9</td>
</tr>
<tr>
<td>2</td>
<td>11.1</td>
<td>10.6</td>
<td>10.85</td>
</tr>
<tr>
<td>3</td>
<td>11.1</td>
<td>10.5</td>
<td>11.8</td>
</tr>
<tr>
<td>4</td>
<td>10.9</td>
<td>10.5</td>
<td>10.7</td>
</tr>
<tr>
<td>5</td>
<td>11.5</td>
<td>10.9</td>
<td>11.2</td>
</tr>
<tr>
<td>Overall Mean Velocity (m/s)</td>
<td></td>
<td></td>
<td>11.1</td>
</tr>
</tbody>
</table>

Table 6: Results of Wind tunnel tests for the glass surface 10 degrees.

<table>
<thead>
<tr>
<th>Test Number</th>
<th>Velocity Up (m/s)</th>
<th>Velocity Side (m/s)</th>
<th>Mean Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>11.1</td>
<td>11.6</td>
<td>10.85</td>
</tr>
<tr>
<td>2</td>
<td>11.4</td>
<td>10.9</td>
<td>11.15</td>
</tr>
<tr>
<td>3</td>
<td>11.1</td>
<td>10.6</td>
<td>10.85</td>
</tr>
<tr>
<td>4</td>
<td>11.4</td>
<td>10.8</td>
<td>11.1</td>
</tr>
<tr>
<td>5</td>
<td>11.6</td>
<td>10.9</td>
<td>11.25</td>
</tr>
<tr>
<td>Overall Mean Velocity (m/s)</td>
<td></td>
<td></td>
<td>11.0</td>
</tr>
</tbody>
</table>