A CFD Method for Simulation of Gas-Liquid Flow in Cooling Systems

An Eulerian-Eulerian Approach

Karl Johan Josefsson
Malin Lind

Academic Supervisor: Petter Ekman
Industrial Supervisor: Fredrik Öhrby
Examiner: Johan Renner
Abstract

When designing modern engines it is important to construct a cooling system that cools the engine structure efficiently. Within the cooling system there is always a certain amount of air which can accumulate and form air pockets in critical areas, such as the water jacket, which can lead to wall degradation. A Computational Fluid Dynamics (CFD) method in STAR-CCM+ from CD-adapco, was derived at Volvo Cars in order to study the accumulation of air bubbles in the water jacket. The method was derived by investigating and evaluating already existing methods. The method initially considered as the best suited was the Eulerian-Eulerian approach. The method was validated against three simpler geometries where experimental data was available. The Eulerian-Eulerian approach treats both phases, liquid and gas, as continuous phases. The idea with the method is to solve the Navier-Stokes equation, the continuity equation and the energy equation for both phases using the Eulerian approach, therefore called Eulerian-Eulerian. The interaction between the two phases was important to model properly which was done by including several interaction models within STAR-CCM+. By tuning different coefficients, which were investigated by a thorough parameter study, the method resembled the experimental data in a satisfying way. The best suited mesh for these simpler geometries was a directed mesh. However, the mesh in the water jacket was automatically generated by STAR-CCM+ and the simpler cases were therefore validated with an automated mesh as well. To capture the experimental data the convection scheme for volume fraction had to be of second order when simulating with automated mesh. This resulted in convergence issues when implementing the method on the water jacket. Instead first order convection scheme, which did not present as satisfying results as second order, had to be implemented. Simulations of the water jacket were performed with two different velocities, that were 10 m/s and 19 m/s, and different flow split ratios for the three outlets. Air with volume fraction 0.1 was injected at the inlet during the first 0.5 s followed by 0.5-1.1 s of further simulation without injecting air. Increased velocity resulted in increased flow through of gas, whereas no big difference could be seen between the different outlet flow split ratios. At two different zones lower pressure was found which resulted in gas holdup. To be able to validate the results from the water jacket, experiments would be necessary to perform in order to provide experimental data for comparison. Velocity profiles from the derived two-phase method resemble the velocity profiles from the one-phase simulation from Volvo, which indicated that the two-phase method did not affect the solution in a remarkable way. Granted that the zones of lower pressure and gas holdup normally coincides, the pressure field from the one-phase simulation could be directly studied, which would lower the computational costs significantly.
Acknowledgements

We would like to begin thanking Volvo Cars for the opportunity to perform an interesting master thesis on the engine cooling system. Volvo provided us with work space and computational power in order to perform our thesis, for which we are thankful. We would also like to thank our industrial supervisor Fredrik Öhrby for his guidelines on practical matters and his valuable inputs regarding the implementation of the method on the water jacket.

We would like to thank our academic supervisor Petter Ekman for his inputs throughout the work. In addition we would like to thank our examiner Johan Renner.

Finally we would like to thank Roman Thiele, the CD-adapco support engineer supporting Volvo Cars, for his advice on how to set up a proper method in STAR-CCM+ by CD-adapco.

Karl Johan Joseffson and Malin Lind

2016, May
## Nomenclature

### Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>EOC</td>
<td>Engine Oil Cooler</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier-Stokes</td>
</tr>
<tr>
<td>SRS</td>
<td>Scale Resolving Simulation</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>CPU</td>
<td>Central Processing Unit</td>
</tr>
<tr>
<td>VOF</td>
<td>Volume Of Fluid</td>
</tr>
<tr>
<td>DEM</td>
<td>Discrete Element Method</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrichs-Lewy</td>
</tr>
</tbody>
</table>

### Symbols and Mathematical Notation

<table>
<thead>
<tr>
<th>Notation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$F$</td>
<td>Force</td>
</tr>
<tr>
<td>$m$</td>
<td>Mass</td>
</tr>
<tr>
<td>$a$</td>
<td>Acceleration</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Density</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Volume fraction</td>
</tr>
<tr>
<td>$u$</td>
<td>Velocity vector</td>
</tr>
<tr>
<td>$x$</td>
<td>Directional vector</td>
</tr>
<tr>
<td>$S$</td>
<td>Source term</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic viscosity</td>
</tr>
<tr>
<td>$g$</td>
<td>Gravity</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Transport variable</td>
</tr>
<tr>
<td>$C$</td>
<td>Coefficient</td>
</tr>
<tr>
<td>$d$</td>
<td>Diameter</td>
</tr>
<tr>
<td>$\nabla$</td>
<td>Divergent</td>
</tr>
<tr>
<td>$n$</td>
<td>Number of particles</td>
</tr>
<tr>
<td>$P$</td>
<td>Probability density function</td>
</tr>
<tr>
<td>$A$</td>
<td>Area</td>
</tr>
<tr>
<td>$y$</td>
<td>Distance</td>
</tr>
<tr>
<td>$n$</td>
<td>Outward facing unit normal</td>
</tr>
<tr>
<td>$u$</td>
<td>Velocity vector</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>$L$</td>
<td>Characteristic length</td>
</tr>
</tbody>
</table>
## Subscripts and Superscripts

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>l</td>
<td>Liquid</td>
</tr>
<tr>
<td>g</td>
<td>Gas</td>
</tr>
<tr>
<td>b</td>
<td>Bubble</td>
</tr>
<tr>
<td>i</td>
<td>Tensor notation</td>
</tr>
<tr>
<td>j</td>
<td>Tensor notation</td>
</tr>
<tr>
<td>D</td>
<td>Drag force</td>
</tr>
<tr>
<td>L</td>
<td>Lift force</td>
</tr>
<tr>
<td>WL</td>
<td>Wall lubrication force</td>
</tr>
<tr>
<td>TD</td>
<td>Turbulent dispersion force</td>
</tr>
<tr>
<td>P</td>
<td>Pressure</td>
</tr>
<tr>
<td>G</td>
<td>Gravity</td>
</tr>
<tr>
<td>VM</td>
<td>Virtual mass</td>
</tr>
<tr>
<td>WD</td>
<td>Wall deformation</td>
</tr>
<tr>
<td>γ</td>
<td>Order of moment</td>
</tr>
<tr>
<td>w</td>
<td>Wall</td>
</tr>
<tr>
<td>r</td>
<td>Relative</td>
</tr>
<tr>
<td>t</td>
<td>Turbulence</td>
</tr>
</tbody>
</table>
## Contents

1 Introduction 1
  1.1 Background .......................................................... 1
    1.1.1 Cooling System ............................................... 1
    1.1.2 Multiphase Problem ........................................... 2
  1.2 Previous Work ...................................................... 3
  1.3 Objectives .......................................................... 4
  1.4 Limitations .......................................................... 4
  1.5 Outline ............................................................. 5

2 Theory 7
  2.1 Two-Phase Flows .................................................... 7
  2.2 Lagrangian versus Eulerian Approaches ......................... 7
  2.3 Eulerian-Eulerian Method ......................................... 8
  2.4 Eulerian-Lagrangian Method ....................................... 8
  2.5 Governing Equations ................................................ 8
    2.5.1 Eulerian-Eulerian ............................................. 8
    2.5.2 Eulerian-Lagrangian .......................................... 9
    2.5.3 Reynolds Averaged Navier-Stokes Equations .............. 9
  2.6 Turbulence .......................................................... 10
    2.6.1 Turbulence Modelling ....................................... 10
    2.6.2 Near Wall Treatment ........................................ 11
  2.7 Gas-Liquid Flow .................................................... 11
    2.7.1 Phase Interaction .......................................... 12

3 Method 17
  3.1 Ekambara ............................................................ 18
    3.1.1 Mesh Independence Study .................................. 21
    3.1.2 Force and Parameter Study ................................ 22
    3.1.3 Simulation Time and Convergence ......................... 22
  3.2 Hibiki ............................................................... 23
  3.3 Bottin ............................................................... 24
  3.4 Water Jacket ........................................................ 25
    3.4.1 Solver Settings .............................................. 27
    3.4.2 Time Step Independence Study ......................... 28

4 Results 29
  4.1 Validation of Methods .............................................. 29
    4.1.1 Ekambara ..................................................... 29
    4.1.2 Hibiki ....................................................... 35
    4.1.3 Bottin ...................................................... 36
  4.2 Water Jacket ........................................................ 38
    4.2.1 Time Step Independence Study ............................ 38
    4.2.2 One Cylinder Model ....................................... 40
    4.2.3 Complete Water Jacket .................................... 41

5 Discussion 47

6 Conclusions 53

7 Outlook 55
List of Figures

1.1 Overview of the cooling system ............................................. 1
1.2 Water jacket in engine structure ......................................... 2
1.3 Water jacket ................................................................. 2
2.1 Lagrangian measurements .................................................... 7
2.2 Eulerian measurements ....................................................... 7
2.3 Flow patterns relative to superficial velocity ......................... 12
2.4 Flow patterns ............................................................... 12
2.5 Breakup and coalescence ................................................... 14
3.1 Superficial velocities for test cases ....................................... 17
3.2 Geometry of the Ekambara case .......................................... 18
3.3 Cross sectional and axial mesh for sim. 2 ............................... 21
3.4 Axial and radial overviews of Mesh 9 ..................................... 22
3.5 Geometry in the Hibiki case ............................................... 23
3.6 Geometry in the Bottin case ............................................... 24
3.7 Water jacket and one cylinder model geometries ..................... 25
3.8 Lines used for velocity comparison ...................................... 27
3.9 Lines used for time step independence study ......................... 28
4.1 Axial mesh independence study .......................................... 29
4.2 Cross sectional mesh independence study ............................. 30
4.3 Prism layer independence study ......................................... 30
4.4 Polyhedral mesh independence study .................................. 31
4.5 Velocity and volume fraction for Ekambara parameter study ...... 32
4.6 Velocity and volume fraction for Ekambara drag coefficient study 32
4.7 Velocity and volume fraction for Ekambara lift coefficient study 33
4.8 Velocity and volume fraction for Ekambara turbulent dispersion study 33
4.9 Velocity and volume fraction for case 2 ................................ 34
4.10 Different inner iterations ............................................... 35
4.11 Velocity and volume fraction for Hibiki ............................... 36
4.12 Velocity and volume fraction for Bottin @5D ......................... 37
4.13 Velocity and volume fraction for Bottin @20D ....................... 37
4.14 Velocity and volume fraction for Bottin @40D ....................... 37
4.15 Time step independence study at total time 0.05 s ................. 39
4.16 Time step independence study at total time 0.1 s ................... 40
4.17 Volume fraction for one cylinder model .............................. 41
4.18 Volume fraction for the water jacket .................................. 42
4.19 Distribution of volume fraction for the water jacket in z-direction 43
4.20 Pressure field for the one-phase model and the two-phase method 44
4.21 Comparison of velocity profiles between one-phase and two-phase simulations 46
## List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1</td>
<td>Models</td>
<td>18</td>
</tr>
<tr>
<td>3.2</td>
<td>Models for the phases</td>
<td>19</td>
</tr>
<tr>
<td>3.3</td>
<td>Properties for density and dynamic viscosity</td>
<td>19</td>
</tr>
<tr>
<td>3.4</td>
<td>Interaction models</td>
<td>20</td>
</tr>
<tr>
<td>3.5</td>
<td>Operating settings for the Ekambara cases</td>
<td>20</td>
</tr>
<tr>
<td>3.6</td>
<td>Mesh settings for the Ekambara cases</td>
<td>21</td>
</tr>
<tr>
<td>3.7</td>
<td>Mesh and simulation settings for the Hibiki cases</td>
<td>23</td>
</tr>
<tr>
<td>3.8</td>
<td>Operating settings for the Hibiki case</td>
<td>24</td>
</tr>
<tr>
<td>3.9</td>
<td>Operating settings for the Bottin case</td>
<td>24</td>
</tr>
<tr>
<td>3.10</td>
<td>Mesh and simulation settings for the Bottin cases</td>
<td>25</td>
</tr>
<tr>
<td>3.11</td>
<td>Simulation settings for the water jacket and the one cylinder model</td>
<td>26</td>
</tr>
<tr>
<td>3.12</td>
<td>Solver settings</td>
<td>27</td>
</tr>
<tr>
<td>4.1</td>
<td>$y^+$ interval for Ekambara case</td>
<td>31</td>
</tr>
<tr>
<td>4.2</td>
<td>$y^+$ interval for Hibiki case</td>
<td>36</td>
</tr>
<tr>
<td>4.3</td>
<td>$y^+$ interval for Bottin case</td>
<td>38</td>
</tr>
<tr>
<td>4.4</td>
<td>Injected, released and remaining volume of gas in the water jacket</td>
<td>45</td>
</tr>
<tr>
<td>4.5</td>
<td>$y^+$ values for water jacket case</td>
<td>45</td>
</tr>
</tbody>
</table>
1 Introduction

An important aspect when designing modern engines is the construction of an efficient cooling system that cools the engine structure and makes sure that the temperature is maintained below damaging levels. Within the cooling system there is always a certain amount of air, that enters the system during filling or from leakage. Depending on the design of the cooling system, air bubbles can accumulate and form air pockets in critical areas such as the water jacket. Presence of air in the system can cause hot spots on the walls which can lead to wall degradation through thermal stresses, fatigue and in worst case cracking.

1.1 Background

A method to simulate air bubbles in the cooling system can become an important tool to be able to understand how to design water jackets to prevent accumulation of air bubbles and to enable an efficient deaeration of the system. An investigation of different methods in order to develop the best suited Computational Fluid Dynamics (CFD) method to be able to study the influence of air bubbles in the cooling system, will be performed at Volvo Cars.

1.1.1 Cooling System

In order to understand the cooling system in the engine, a simplified system overview is presented in Fig. 1.1. In the system a coolant mixture consisting of water, glycol and corrosion inhibitors circulates. Cooled coolant enters the water jacket in the engine structure through a pump. The coolant transport heat out of the engine structure. The thermostat positioned outside the engine structure will lead the cooled coolant back to the engine structure and the heated coolant to the radiator, which will cool down the coolant again. Heated coolant from the engine structure will also be directed to the climate circuit (coupe) whereas cooled coolant will be directed to the engine oil cooler (EOC). An overview of the engine structure can be seen in Fig. 1.2 and the water jacket inside the structure can be seen in Fig. 1.3 where the mentioned flow direction is represented by the inlet and the outlets.

![Diagram of the cooling system](image)

**Fig. 1.1:** Overview of the cooling system. Cooled coolant enters the water jacket in the engine structure through a pump. The thermostat positioned outside the engine block will lead the cooled coolant back to the engine block and the heated coolant to the radiator. Heated coolant from the engine structure can also be directed to the climate heater whereas cooled coolant can be directed to the engine oil cooler.
1.1.2 Multiphase Problem

As mentioned previously, the problem concerns a flow consisting of two phases, coolant and air, further referred to as liquid and gas. Multiphase flows are common and can be found in several natural phenomena as well as in technical processes. The physics being present is complex and CFD simulations are an important tool in order to understand such flows. The two phases, liquid and gas, which are not chemically related to each other can be modelled in different ways. A two-phase model is one approach which will be considered in this project.
1.2 Previous Work

In this section previous work and its literature is presented. If further explanations are needed see the theory section.

As mentioned, multiphase problems are common and have been widely studied in different setups, where one recurring setup is called the “bubble column”. A bubble column is a simple setup where gas in a liquid can be studied. Hibiki, Ishii and Xiao [1], who are widely cited, performed an experiment on a vertical bubble column with water and air. Air was introduced into a chamber by a compressor. In the chamber, air and water were mixed and the mixture travel upward in the column.

Fraga, Stoesser, Lai and Socolofsky [2] stated that three main methods exist for modelling gas and liquid. The methods are called Eulerian-Eulerian which is a volume fraction method, Eulerian-Lagrangian which is a particle tracking method and Volume of Fluid interface tracking. Interface tracking is more computationally heavy and involves resolving the surfaces between the bubbles and the liquid, which for ideal tracking means that a fine mesh is required. This method seems to only have been applied for smaller amount of bubbles.

Several comparisons between Eulerian-Eulerian and Eulerian-Lagrangian have been performed. Idelsohn, Onate, Nigro, Becker and Gimenez [3] compared the numerical errors between Eulerian-Eulerian and Eulerian-Lagrangian and concluded that the errors in Eulerian-Eulerian are higher in general. They also stated that the Eulerian-Eulerian approach performs better for lower Reynolds numbers whereas the Eulerian-Lagrangian approach performs better for higher Reynolds numbers. The drawbacks of Eulerian-Eulerian are convergence issue and not being able to represent the interaction between particles directly whereas the drawbacks of Eulerian-Lagrangian are that it is more computational heavy and that the method only can be applied on cases with smaller concentration of particles, which in CD-adapco STAR-CCM+ is up to 0.4 volume fraction of gas. [4].

Xiao, Jang and Li [5] modelled a bubble column using Eulerian-Eulerian and denoted one phase containing liquid and small bubbles and one phase containing large bubbles. This method showed advantages within different aspects, for example improved prediction of overall gas holdup. Xiao et al. also stated that small bubbles tend to stay longer in the column since they follow the motion of the liquid whereas the larger bubbles leaves the column during the mixing of liquid and small bubbles. According to Simonnet, Gentic, Olmos and Midoux [6] bubble coalescence is negligible during low superficial gas velocity, thus smaller bubbles occurs, whereas coalescence occurs during increased velocity.

Dhotre, Deen, Niceno, Khan and Joshi [7] found that bubbles induce turbulence even in laminar flows. This turbulence is of anisotropic nature in contrast to the assumption of most Reynolds Averaged Navier-Stokes (RANS) turbulence models where the turbulence is assumed to be isotropic. This is also stated by Mattson and Mahesh [8] who further concluded that due to the anisotropic nature, Scale Resolving Simulations (SRS) such as Large Eddy Simulation (LES) is preferred over RANS modeling.

Horizontal flow have not received as much attention in literature as vertical flow which is also stated by Ekambara, Sanders, Nandakumar and Masliyah [9]. Ekambara et al. performed a CFD simulation, using the Eulerian-Eulerian method, with liquid and gas in a horizontal pipe and included experimental data from other researchers. Another experiment with liquid and gas in a horizontal pipe was performed by Bottin, Berlandis, Hervieu, Lance, Marchand, Öztürk and Serre [10]. Liquid and gas entered an injection section via two independent pipes. The injection section consisted of 320 tubes for the liquid and 37 tubes for the gas and the test section that followed after the injection section contained bubbly flow.

A lot of research has been performed concerning forces acting on a single bubble. However, D. Lucas, E. Krepper and H.-M. Prasser [11] investigated the validity of these single bubble correlations with multiple bubbles and found that the correlations are valid for a group or cluster of bubbles as well.
Further on, concerning the mesh, Peric and Ferguson [12] presented a discussion of the benefits of using polyhedral mesh over tetrahedral mesh. Tetrahedral cells are easy to generate automatically, however, the tetrahedral cells have only four neighbours which results in problems when computing gradients using standard approximations. To obtain an accurate solution special discretization schemes and a large amount of cells are required which leads to higher computational costs. The polyhedral cells however, have more neighbours, about ten, which makes the approximation of gradients easier. On the other hand, more neighbours results in more computational operations. This is however, more than compensated by the higher accuracy. Peric and Ferguson used a water jacket of an engine as geometry and performed several meshes of both tetrahedral and polyhedral cells in order to validate their statement. They used the same discretization scheme and solution method for all simulations and showed that a simulation with polyhedral mesh was slightly more accurate than a simulation with tetrahedral cells which consisted of six times as many cells. This meant remarkably lower computational time for the polyhedral mesh, less than one tenth of the time used for the tetrahedral mesh.

1.3 Objectives

- The aim of the study is to derive a method for simulating gas bubbles in the water jacket. The method will be used to study the accumulation of bubbles in critical areas.
- An evaluation of existing two-phase methods will be performed, on which the derived method will be based.
- A validation of the derived method will be performed on published experiments before implementing it on the water jacket.
- The method will be used to study the accumulation of bubbles in the water jacket with different operating conditions.

1.4 Limitations

The project was limited to the available computer power, that is the Volvo Cars Central Processing Unit (CPU) cluster. The first part of the project was limited to the use of one computer. In addition, the beginning of the project was directed towards learning the commercial CFD software STAR-CCM+ from CD-adapco, since no previous knowledge existed. The physics and the turbulence modelling within the software were limiting factors and could contribute to errors. The research was limited to the models, forces and physics included in the software since there was not enough time nor knowledge to implement new models.

Further on, the project was delimited to only consider the water jacket and the rest of the cooling system was not investigated. The validation of the derived method was limited to three test cases with simpler geometries, that are circular pipes with varying dimensions, and the derived method was assumed to be valid on the water jacket. In addition the derived method was, as stated in the objectives, limited to existing methods and no development of a new method has been performed.
1.5 Outline

The section that follows is the theory section where the theoretical background for the work will be presented. The method section will thoroughly describe the derived method using test cases as well as the implementation on the water jacket. The result section that follows, presents all results obtained in this work. The section naturally follows the same order as in the method section. After the result section a discussion is presented where all results as well as the method is thoroughly discussed. The conclusion section shortly and concisely presents the conclusions of the work. Finally outlook and perspective sections follows, where future work and society as well as commercial perspectives are presented.
2 Theory

2.1 Two-Phase Flows

Multiphase flows can either be dispersed or separated. Dispersed flow is flow containing finite particles such as bubbles, that are distributed in a continuous phase whereas separated flow is flow consisting of phases separated by larger interfaces \[13\]. For dispersed flow, as in this project, the Eulerian-Lagrangian or the Eulerian-Eulerian are recommended \[14\]. The Eulerian-Lagrangian is recommended when it is necessary to track each particle in detail and Eulerian-Eulerian is recommended if no details of the particles are needed, instead averaged values are enough \[14\]. In addition, as mentioned in section \[1.2\] previous work there exist a method called interface tracking. However, this method will not be considered since it only has been applied for smaller amount of bubbles which indicates a high computational cost.

2.2 Lagrangian versus Eulerian Approaches

The focus in the theory section will be on the Eulerian-Eulerian method and the Eulerian-Lagrangian method. The main differences between the methods is explained here. The Lagrangian approach tracks the properties of each particle, relative to its starting position which means that the properties of each particle are known independently off its location \[15\]. In addition, the location and the path of each individual particle are tracked. Hence, simulating with enough amount of particles should give a good understanding of the flow field. A principle sketch of the Lagrangian approach can be seen in Fig. 2.1

![Fig. 2.1: Shows the principles of a Lagrangian approach. The vectors symbolizes the velocity of each particle at its current position. Each particle can be seen as a measuring gauge.](image)

In the Eulerian approach the particles are not tracked but the properties of the particle are measured at certain positions which in STAR-CCM+ is the centre of each cell \[15\]. By dividing the fluid domain in to multiple cells a map of the flow field can be obtained and in each cell the properties of the flow are obtained. The cells can be viewed upon as measuring gauges. Hence, a more detailed overview of the flow can be obtained by increasing the amount of cells. A principle sketch of the Eulerian approach can be seen in Fig. 2.2

![Fig. 2.2: Shows the principles of an Eulerian approach. The vectors symbolizes the velocity field for the flow. The particles are passing by the measuring points (cells) and provide information about the velocity in that certain point.](image)
2.3 Eulerian-Eulerian Method

The Eulerian-Eulerian method is a two-phase method where the dispersed phase is treated as a second continuous phase [13]. The idea with the method is to solve the Navier-Stokes equation, the continuity equation and the energy equation for both phases using the Eulerian approach, thus called Eulerian-Eulerian [4]. The method can model turbulence for each phase and can be applied on cases with volume fraction of the dispersed phase ranging from zero to one [4]. As mentioned, this method does not give information of each particle path, instead the properties of the dispersed phase are averaged [13]. The method is an efficient way of visualizing the volume fraction of each phase in the domain.

2.4 Eulerian-Lagrangian Method

Eulerian-Lagrangian solves Navier-Stokes equations for the continuous phase whereas the particles in the dispersed phase are treated by solving the equation of motion, in a Lagrangian way [4], thus called Eulerian-Lagrangian. As mentioned, this provides knowledge about each particle which makes it possible to study particle size distribution, interaction between particles in terms of collision, coalescence and agglomeration and heat and mass transfer between particles [4]. The properties of the dispersed phase can be in form of the actual particle or by larger representative particles, which in term lower the computational cost [13].

2.5 Governing Equations

Fluid dynamics is based on three physical principles, which are:

- The conservation of mass
- Newtons second law, \( F = ma \)
- The conservation of energy (First law of thermodynamics) [16]

These principles results in three mathematical statements, which are the fundamental governing equations of fluid dynamics:

- Continuity equation
- Navier-Stokes momentum equations
- Energy equation [16]

2.5.1 Eulerian-Eulerian

Under incompressible assumption, i.e. constant density, and using the Einstein summation convention [17], the equations for the Eulerian-Eulerian method are as follows:

The continuity equation for the continuous phase:

\[
\frac{\partial \rho \alpha \mathbf{u}_j}{\partial x_j} = 0
\]

(1)

The continuity equation for the dispersed phase:

\[
\frac{\partial \rho_d \alpha_d \mathbf{u}_j}{\partial x_j} = S_i
\]

(2)
where \( \rho \) is the density, \( \alpha \) is the volume fraction, \( u \) is the velocity vector and \( x \) is the spatial vector \([9]\). The indexes \( g \) and \( l \) stands for gas and liquid. \( S \) is a source term that brings the effects of coalescence and break-up into the equation \([9]\).

The Navier-Stokes momentum equation for the continuous phase:

\[
\frac{\partial \alpha_l u_{i,l}}{\partial t} + u_{j,l} \frac{\partial \alpha_l u_{i,l}}{\partial x_j} = -\frac{\alpha_l \rho}{\partial x_i} + \alpha_l \nu \frac{\partial^2 u_{i,l}}{\partial x_j^2} + \alpha_l g_i + F_{lg} \tag{3}
\]

The Navier-Stokes momentum equation for the dispersed phase:

\[
\frac{\partial \alpha_g u_{i,g}}{\partial t} + u_{j,g} \frac{\partial \alpha_g u_{i,g}}{\partial x_j} = -\frac{\alpha_g \rho}{\partial x_i} + \alpha_g \nu \frac{\partial^2 u_{i,g}}{\partial x_j^2} + \alpha_g g_i + F_{gl} \tag{4}
\]

where \( p \) is the pressure, \( \nu \) is the kinematic viscosity and \( g \) is the gravity \([18]\). \( F \) is the total interfacial force which includes different forces affecting the interface between the two phases according to:

\[
F_{i,lg} = -F_{i,gl} = F_{PD} + F_{PL} + F_{VL} + F_{TD} + F_{WD} \tag{5}
\]

which represents the drag force, lift force, wall lubrication force and the turbulent dispersion force \([18]\).

The energy equation will not be used and is therefore not included in this report. The principle of the equation is although mentioned, that is: \textit{Rate of change of energy of the particle} = \textit{Net rate of heat added to the particle} + \textit{Net rate of work done on the particle} \([16]\).

### 2.5.2 Eulerian-Lagrangian

The continuous phase is solved in the same way as in the Eulerian-Eulerian method, thus the continuity equation and the Navier-Stokes momentum equation are solved as in equations \([1] \) and \([3]\). For the dispersed phase the motion of a bubble can be expressed according to Newton’s second law \([7]\). Under the assumption of constant mass the equation is as follows:

\[
\sum F_i = m_b \frac{du_{i,b}}{dt} \tag{6}
\]

where \( m \) is the mass. Index \( b \) stands for bubbles and \( \sum F_i \) is the sum of all forces acting on a bubble:

\[
\sum F_i = F_{PD} + F_{CG} + F_{VD} + F_{VL} + F_{VM} + F_{TD} + F_{WD} + F_{WD} \tag{7}
\]

which represents forces concerning pressure, gravity, drag, lift, virtual mass, turbulent dispersion, wall lubrication and wall deformation \([7]\).

### 2.5.3 Reynolds Averaged Navier-Stokes Equations

The Navier-Stokes equations includes instantaneous quantities which can be solved by using Direct Numerical Simulations (DNS). However, this demands unreasonably high computational power which can be lowered by instead using RANS equations. RANS equations are obtained by introducing Reynolds decomposition followed by a time averaging of each term. Reynolds decomposition is obtained by dividing the instantaneous quantities into a time-averaged part and a fluctuating part as follow:

\[
\phi(x,t) = \bar{\phi}(x) + \phi'(x,t) \tag{8}
\]

where \( \phi \) is any transport variable. The decomposed and averaged RANS equations for Eulerian-Eulerian are then obtained as:
For the continuous phase:

\[
\frac{\partial \alpha_l \bar{u}_{i,l}}{\partial t} + \bar{u}_{j,l} \frac{\partial \alpha_l \bar{u}_{i,l}}{\partial x_j} = -\frac{\alpha_l}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \alpha_l \nu_l \frac{\partial^2 \bar{u}_{i,l}}{\partial x_j^2} - \frac{\partial \bar{u}_{i,l} \bar{u}_{j,l}}{\partial x_j} + \alpha_l g_l + F_l^{\text{rg}} \tag{9}
\]

For the dispersed phase:

\[
\frac{\partial \alpha_g \bar{u}_{i,g}}{\partial t} + \bar{u}_{j,g} \frac{\partial \alpha_g \bar{u}_{i,g}}{\partial x_j} = -\frac{\alpha_g}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \alpha_g \nu_g \frac{\partial^2 \bar{u}_{i,g}}{\partial x_j^2} - \frac{\partial \bar{u}_{i,g} \bar{u}_{j,g}}{\partial x_j} + \alpha_g g_l + F_l^{\text{rg}} \tag{10}
\]

The continuity equation is, in contrast to the Navier-Stokes equation, linear, which results in an expression obtained on the same form after decomposition:

For the continuous phase:

\[
\frac{\partial \rho \alpha_l \bar{u}_{i,l}}{\partial x_j} = 0 \tag{11}
\]

For the dispersed phase:

\[
\frac{\partial \rho \alpha_g \bar{u}_{i,g}}{\partial x_j} = S_i \tag{12}
\]

The decomposed and averaged equation for Eulerian-Lagrangian is obtained as:

\[
\sum F_i = m_b \frac{d\bar{u}_{b,i}}{dt} \tag{13}
\]

### 2.6 Turbulence

Turbulence is characterized by random and chaotic three-dimensional vorticity. When present, turbulence dominates all other flow phenomena, resulting in increased energy dissipation, mixing, heat transfer and drag [19]. Turbulence is therefore a complex phenomenon even for one-phase flow in a simple geometry. When multi-phase flow is considered, it is obvious that the turbulence will be further complex, for examples due to particles being present which influence the turbulence by strengthening or weakening it [13].

#### 2.6.1 Turbulence Modelling

By introducing the Reynolds decomposition in the Navier-Stokes equations six additional terms are introduced, which are called the Reynolds stresses [16]. The introduction of the Reynolds stresses is referred to as the closure problem. The closure problem can be dealt with by introducing, for example the Boussinesq assumption, which is relating the Reynolds stresses to be proportional to the mean deformation rate of the continuum. Two variables are introduced, the turbulent viscosity \( \mu_t \) and the term \( k \), turbulent kinetic energy, which occurs from modifying the pressure. The six unknown terms are replaced with the more convenient number. Introducing a two-equation turbulence model is one way of solving the two unknown where \( \mu_t \) is related to \( k \) and either \( \varepsilon \) or \( \omega \), and thus closing the closure problem. \( k \) is as mentioned the turbulent kinetic energy, \( \varepsilon \) is the rate of turbulent dissipation and \( \omega \) is the specific turbulence dissipation rate. The \( k - \varepsilon \) model will be used in this work and will be further explained.
$k - \varepsilon$ Model

$k - \varepsilon$ is a two-equation turbulence model that solves transport equations for the turbulent kinetic energy, $k$, and the rate of dissipation of turbulent kinetic energy, $\varepsilon$ [16]. The $k - \varepsilon$ model is one of the most widely used and validated models since the model can handle a wide range of flows and works well in the free stream. The drawbacks of the model are that it presents poor results near walls, in anisotropic turbulence and in flows with large strain rate. The model exist in different forms due to different attempts to improve it. The form that is used at Volvo Cars as well as in this project is realizable $k - \varepsilon$ two-layer model. Realizable $k - \varepsilon$ two-layer combines the realizable $k - \varepsilon$ model with the two-layer approach [20]. The realizable $k - \varepsilon$ includes a new transport equation for the dissipation rate and a coefficient that were assumed to be constant in the standard model is here a function of mean flow and turbulence properties. The model is thus an improvement for many applications compared to the standard model. The two-layer approach enables the model to be used with fine mesh in the viscous sublayer, i.e. near wall. In standard $k - \varepsilon$ the normal stress $\overline{u^2}$ is by definition positive, but can obtain negative values and thus becoming "non-realizable" [21]. In contrary the formulation of realizable $k - \varepsilon$ prevents the stresses from obtaining negative values by satisfying certain mathematical constraints, thus being "realizable".

2.6.2 Near Wall Treatment

Throughout the boundary layer there exists wall bounded flow with large gradients. In order to resolve the boundary layer, i.e. resolve the near wall flow it is important to achieve an appropriate $y^+$ value. The non-dimensional wall distance, $y^+$ is defined as

$$y^+ = \frac{u_t y}{\nu}$$  \hspace{1cm} (14)

where $u_t$ is the friction velocity which depends on the wall shear stress and the density of the fluid [22]. $y$ is the distance to the wall from the first cell center and $\nu$ is the kinematic viscosity. According to [20] the $y^+$ value should be around 1 or above 30 when simulating with realizable $k - \varepsilon$ two-layer. Further, the $y^+$ value should not exceed 100.

2.7 Gas-Liquid Flow

Gas-Liquid flows can be divided in to multiple types such as stratified flow, bubbly flow, slug flow and annular flow [23]. Bubbly flow occurs when the flow rate of the gas is low in relation to the liquid flow rate. In this flow the gas forms bubbles of various sizes. Stratified flow has a distinct surface that separates the liquid and the gas phases. In a pipe, for example, the liquid phase flow in the lower region due do its higher density whereas the gas tends to flow in the upper region. This flow type occurs during low flow rates. When the flow rate increases a slug flow is present which contain slugs of large asymmetric bubbles combined with small bubbles. During very large gas flow rates the gas creates a film of liquid along the circumference of the pipe, with the gas flowing in the centre of the pipe, a phenomena called annular flow. The conclusion is that the flow type depends on the superficial velocities of both the gas and the liquid. The different flow types are shown in Fig. 2.3 and Fig. 2.4. In order to capture the behaviour of the gas phase the interaction forces between the liquid and the gas phase have to be studied.
Fig. 2.3: Shows the flow pattern found in (a) horizontal flow and (b) vertical flow. The figure describes what flow patterns can be expected depending on the superficial velocity of each phase. The letters a-f corresponds to the letters in Fig. 2.4. The figures are principal sketches of how the flow behaves in relation to the superficial velocity, and should not be viewed as scientifically accurate values.

Fig. 2.4: Shows the flow patterns found in horizontal flows (a-d) and vertical flows (e-f). a) and e) represents bubbly flow, b) and f) represents slug flow. c) represents stratified flow whereas g) represents churn flow. d) and h) represents annular flow.

2.7.1 Phase Interaction

For gas-liquid flows the CFD code has to contain constitutive laws for the interaction between the gas and liquid phase, i.e forces acting on the bubbles. With complicated three-dimensional geometries the situation becomes complex and these forces not only depend on the flow structures but also the bubble sizes. The forces, phenomenon and models that hereinafter will be explained are the drag force, lift force, breakup and coalescence, turbulent dispersion force, particle induced mixing and wall lubrication force.
**Drag Force**

Drag force is a resistive force that acts in the opposite direction of the velocity. There are two types of drag which are called skin drag and form drag [9]. A gas bubble moving in a liquid phase will experience skin drag due to viscous stress and form drag due to the pressure distribution around the moving bubble. This force is calculated as:

\[ F_D = \frac{3}{4} C_D \alpha_g \rho_l d_b |u_g - u_l| (u_g - u_l) \]  

where \( C_D \) is the drag coefficient [9]. The drag coefficient is a science on its own where multiple researchers has contributed with models for capturing the drag force of a single bubble. The force on a cluster of bubbles is therefore harder to predict.

**Lift Force**

If the bubble flows in a liquid where velocity gradients are present the relative velocity will not be the same on the whole bubble surface [23]. This will result in an unequal pressure distribution and thus a force called lift force is created. In upward or vertical flow the lift force will push the bubbles towards the wall of the pipe. However, in horizontal pipes the lift force will force the bubbles towards the centre of the pipe. The sign of the lift coefficient is positive for large bubbles but decrease to negative values as the diameter of the bubbles decreases [11]. The lift force acting on a bubble is calculated as:

\[ F_{Lg} = C_L \alpha_g \rho_l \cdot [(u_{i,g} - u_{i,l}) \times (\nabla \times u_{i,l})] \]  

Where \( C_L \) is the lift coefficient [9]. There are several available method derived to model the lift coefficient. However, as recommended in Ekambara et al. [7] the lift coefficient was to be constant, and therefore, no further explanation of these methods will be provided.

**Breakup and Coalescence**

Bubbles flowing through a liquid will experience phenomena called breakup and coalescence. Breakup describes the phenomena when two or more bubbles are created from one existing bubble [23]. The mechanisms behind are breakup due to impact of the liquid eddies against the bubbles caused by turbulence, breakup due to smaller bubbles shearing off from larger bubbles and breakup due to bubbles falling apart due to surface instabilities. Coalescence on the other hand describes the phenomena when new bubbles are created from existing bubbles. There are two main mechanism for coalescence in gas-liquid flows, random collision due to turbulence and collision due to different velocities of the bubbles [23]. A principle sketch of breakup and coalescence can be seen in Fig. 2.5.
Due to breakup and coalescence the particle size i.e. the surface area can change continuously in gas-liquid flows [20]. Since interfacial terms depends on the surface area of the gas phase, it is important to take the particle size into account when simulating multi-phase flows. The $S_\gamma$ model in STAR-CCM+ has therefore been implemented to take the particle size and its distribution into account. The particle size distribution is assumed to be log-normal which includes a mean diameter and its variance. When the $S_\gamma$ model is activated the mean diameter is updated during the simulation but to account for the diameter variance, the breakup and coalescence models are required as well. The breakup model describes the balance between disruptive and restoring forces on the particle [20]. Different effects dominates in laminar and turbulent flow which have resulted in two different types of breakup. These are called viscous breakup and inertial breakup since viscous effects dominates in laminar flow and interactions with turbulence eddies dominates in turbulent flow. The coalescence model describes the probability of collision between bubbles as well as their contact time and the time for the liquid film between the bubbles to disappear [20]. Like the breakup model the coalescence model contains a viscous coalescence and an inertial coalescence. The $S_\gamma$ model in STAR-CCM+ which predicts the transport of the moments of the particle size distribution is defined as:

\[
S_\gamma = n \int_0^\infty d^\gamma P(d)d(d)
\]  

(17)

where $\gamma$ is the order of moment, $n$ is the number of particles per unit volume and $P(d)$ is the probability density function of particle diameter [20].

**Turbulent Dispersion Force**

Turbulent dispersion force strongly affects the gas concentration in a bubbly flow and determines, together with the wall lubrication force and lift force, the peak of volume fraction close to the walls [23]. The force is a result of the interaction between the phases in terms of drag force and the interaction between particles in the gas phase and the eddies of the liquid phase [23] [24]. The force per volume off the liquid phase due to the gas phase is defined in STAR-CCM+ as:

\[
F_{tg}^{TD} = A_{tg}^D \cdot \left[C_{tg}^{TD} \cdot \left(\frac{\nabla \alpha_g}{\alpha_g} - \frac{\nabla \alpha_l}{\alpha_l}\right)\right]
\]  

(18)

where $C_{tg}^{TD}$ is the tensor diffusivity coefficient [20].
Particle Induced Mixing

When simulating the gas phase as laminar the turbulence induced by the bubbles have to be added to the liquid phase. A model called Sato is available for particle induced mixing and it enhances the effective viscosity of the liquid phase by adding a term in the turbulence formulation for the liquid phase. In this way turbulent effects from the gas phase are accounted for [20].

Wall Lubrication Force

In the region close to the walls the flow on the bubble surface differs from the bulk flow. A force on the bubbles is generated due to the generated velocity gradients caused by the no-slip condition on the wall [23]. This force pushes the bubbles away from the wall and enables the prediction of the slight offset peak of volume fraction close to the walls [23]. The force is called wall lubrication force and Antal et al. developed a model which is implemented in STAR-CCM+ [20]. The force is defined as:

\[ F_{\text{WL}}^{\text{lg}} = C_{\text{WL}}(y_w) \alpha_g \rho_l \frac{|u_t||u_t|}{d} \cdot n \]  

(19)

\[ u_t = (u_l - u_g) - [(u_l - u_g) \cdot n] \cdot n \]  

(20)

where \( n \) is the outward facing unit normal at the closest point on the wall which means that the force prevents contact between the bubbles and the wall [20]. The model is defined such that if the distance from the wall \( y_w \) equals five bubble diameters there is no wall lubrication force acting on the bubble.
3 Method

Both the Eulerian-Eulerian and the Eulerian-Lagrangian approach were suited for this project. However, the Eulerian-Eulerian method was considered as the most advantageous method based on the fact that most of the researchers reviewed in section 1.2 previous work used this method. In addition, in this project there was no need to track each particle as in the Eulerian-Lagrangian approach, instead the focus was towards finding areas with accumulated gas. This was possible with the Eulerian-Eulerian approach which at the same time was less computational heavy compared to the Eulerian-Lagrangian as stated in section 1.2 previous work. This further motivated the use of the Eulerian-Eulerian approach. This method was therefore further investigated by applying it on three previous works that were cited in section 1.2 previous work. A validation of the method was possible since experimental data existed. The different cases will be referred to as Ekambara, Hibiki and Bottin, named after the first author. Most effort was put into the Ekambara case since it best resembled the water jacket in terms of diameter and Reynolds number. The knowledge gained from Ekambara was then used when working with the two remaining cases. The test cases were all mainly simulated under steady state conditions although some simulations in the Ekambara case were performed under transient conditions. This was done in order to obtain knowledge before applying the method on the water jacket, which was simulated only under transient conditions. The test cases have a relatively structured flow compared to the water jacket, which due to the complicated geometry has a varying flow structure. This motivates the use of steady and transient conditions. Further on, since experimental data for the test cases existed, the steady state approach could be validated. As stated in section 1.2 previous work LES is preferred over RANS modelling. However, RANS modelling was although used for all simulations since the computational cost when using LES was considered unreasonably high due to the amount of simulations that were to be performed.

According to the flow types mentioned in section 2.7 gas-liquid flow, the current flows in the test cases were determined. For all test cases slug flow could be expected. However, the points representing the test cases seen in Fig. 3.1 are located close to bubbly flow which means that a mixture of slug flow and bubbly flow could occur.

![Horizontal Flow Patterns](image1)

![Vertical Flow Patterns](image2)

Fig. 3.1: Shows the horizontal flow patterns in (a) and the vertical flow patterns in (b). The black points are representing the superficial velocities in the test cases. E1 stands for Ekambara case 1 and E2 for Ekambara case 2. B stands for Bottin and H stands for Hibiki. For all cases slug flow could be expected.
3.1 Ekambara

Ekambara et al. [9] performed CFD simulations and evaluated the results against experimental data. The geometry was a 9 m long horizontal circular pipe with inner diameter 0.05 m and can be seen Fig. 3.2. Ekambara et al. used constant bubble size of 0.002 m in diameter at the inlet as well as groups with different bubble diameters to resemble the experiments. Two different cases, with different velocities and volume fractions, from the Ekambara research were studied in this work. The Reynolds number in these cases were 270 000 - 300 000.

Fig. 3.2: Shows the geometry of the Ekambara case as well as the line where simulation data was extracted. The length of the pipe and the position of the extraction line is calculated with the diameter, \( D \). The gravity is represented by \( g \).

The setup that was used in this work was derived by iteratively testing and evaluating different settings as well as using knowledge from previous work and tutorials within STAR-CCM+.

Tab. 3.1 presents the models that were selected in STAR-CCM+.

<table>
<thead>
<tr>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Eulerian Multiphase</td>
</tr>
<tr>
<td>Gradients</td>
</tr>
<tr>
<td>Gravity</td>
</tr>
<tr>
<td>Multiphase Equation of State</td>
</tr>
<tr>
<td>Multiphase Interaction</td>
</tr>
<tr>
<td>Multiphase Segregated Flow</td>
</tr>
<tr>
<td>Steady</td>
</tr>
<tr>
<td>Three Dimensional</td>
</tr>
<tr>
<td>Turbulent</td>
</tr>
</tbody>
</table>

Eulerian multiphase was chosen since this approach was to be used instead of Langrangian. Gravity was specified in the negative y-direction, seen in Fig. 3.2 due to the horizontal orientation of the pipe and the model was selected to be able to account for the gravitational effects. Multiphase interaction was needed in order to model the interaction between the phases. Steady state was used as previously motivated. Three dimensional and turbulent flow were selected in order to represent the dimensions and the Reynolds number found in Ekambara. The models gradients and multiphase equation of state were automatically selected when other models were chosen. The remaining model, multiphase segregated flow, was selected based on knowledge gained from tutorials within STAR-CCM+.

Two Eulerian phases were created under the Eulerian multiphase model, air and water. The models used for the liquid and gas phase are stated in Tab. 3.2.
Tab. 3.2: Models for the liquid and gas phase within STAR-CCM+ that were selected.

<table>
<thead>
<tr>
<th>Liquid phase</th>
<th>Gas phase</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant Density</td>
<td>Constant Density</td>
</tr>
<tr>
<td>Exact Wall Distance</td>
<td>Exact Wall Distance</td>
</tr>
<tr>
<td>$k - \varepsilon$ Turbulence</td>
<td>Gas</td>
</tr>
<tr>
<td>Liquid</td>
<td>Laminar</td>
</tr>
<tr>
<td>Realizable $k - \varepsilon$ Two Layer</td>
<td>$S_{\gamma}$</td>
</tr>
<tr>
<td>Liquid-averaged Navier-Stokes</td>
<td>Two-Layer All $y^+$ Wall Treatment</td>
</tr>
</tbody>
</table>

Constant density was selected since incompressible flow was assumed. The models Reynolds-Averaged Navier-Stokes, $k - \varepsilon$ turbulence, realizable $k - \varepsilon$ two layer and two-layer all $y^+$ wall treatment were selected since RANS modelling and realizable $k - \varepsilon$ were to be used as motivated both previously and in section 2.6.1 $k - \varepsilon$ model. Laminar was selected for the gas phase as done in Ekambara et al. [9]. In addition a model called particle induced mixing was selected for the interaction model as described in section 2.7.1 particle induced mixing. $S_{\gamma}$ was selected in order to account for bubble breakup and coalescence as described in section 2.7.1 breakup and coalescence. Exact wall distance was automatically selected when other models were chosen.

The properties for density and dynamic viscosity at the assumed temperature, $20^\circ$ C, can be seen in Tab. 3.3. The liquid phase was initialized with volume fraction equal to one and horizontal velocity according to the superficial velocity in Tab. 3.5 whereas the gas phase was initialized with zero both for volume fraction and velocity. This approach was used after recommendations from STAR-CCM+. Further on concerning the initial conditions, a turbulent length scale of 0.005 m was set for the liquid phase according to Hibiki et al. [11]. The sauter mean diameter for the gas phase was set to 0.002 m, i.e. the bubble size.

Tab. 3.3: Properties for density and dynamic viscosity used in the simulations for Ekambara.

<table>
<thead>
<tr>
<th>Temperature $[^\circ C]$</th>
<th>20</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density $[\text{kgm}^{-3}]$</td>
<td>1.189 998.2</td>
</tr>
<tr>
<td>Dynamic viscosity $[\text{kgm}^{-1}\text{s}^{-1}]$</td>
<td>1.81e-5 1.005e-3</td>
</tr>
</tbody>
</table>

As mentioned, the model multiphase interaction was needed in order to model the interaction between the phases. The model contains a lot of "sub-models" in order to model different types of interaction. Tab. 3.4 shows the models that were selected.
Continuous-dispersed phase interaction was selected since the gas phase was dispersed in the liquid phase. As stated in section 2 theory, drag force, lift force, turbulent dispersion force, wall lubrication force and particle induced mixing are important interaction models and were therefore selected. The drag coefficient was set as Schiller-Naumann after recommendations from Ekambara et al. [9]. The lift coefficient and the turbulent dispersion Prandtl number were adjusted in order to resemble the experimental data and the chosen settings can be seen in Tab. 3.5. Interaction length scale was chosen and set to the bubble diameter as in tutorials within STAR-CCM+. Interaction area density and virtual mass coefficient were selected automatically when other models were chosen. The interaction area density was set to spherical particles. Multiphase material was selected in order to specify the surface tension between the phases. The surface tension was set to 0.0726 N/m according to Bottin et al. for distilled water [10]. Finally, $S_\gamma$ breakup and coalescence were selected in order to account for these phenomenon which was important as stated in the section 2 theory. Short collision time was chosen as the coalescence probability according to tutorials within STAR-CCM+.

Concerning the boundary conditions, a velocity inlet, a flow-split outlet and a wall condition were used. The inlet velocities, which were calculated from the superficial velocities, and the volume fractions can be seen in Tab. 3.5 for the two different cases, case 1 and case 2. For the walls a no-slip condition was used for both phases.

Tab. 3.4: Models for the multiphase interaction within STAR-CCM+ that were selected.

<table>
<thead>
<tr>
<th>Continuous-Dispersed Phase Interaction</th>
<th>Drag Force</th>
<th>Interaction Area Density</th>
<th>Interaction Length Scale</th>
<th>Lift Force</th>
<th>Multiphase Material</th>
<th>Particle Induced Mixing</th>
<th>$S_\gamma$ Breakup</th>
<th>$S_\gamma$ Coalescence</th>
<th>Turbulent Dispersion Force</th>
<th>Virtual Mass Coefficient</th>
<th>Wall Lubrication Force</th>
</tr>
</thead>
</table>

Tab. 3.5: Operating setting for the Ekambara cases. The abbreviation vel. stands for velocity and TD stands for turbulent dispersion Prandtl number. The abbreviation coeff. stands for coefficient.

<table>
<thead>
<tr>
<th></th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Superficial vel. [ms$^{-1}$]</td>
<td>0.8</td>
<td>0.25</td>
</tr>
<tr>
<td></td>
<td>Liquid</td>
<td>Liquid</td>
</tr>
<tr>
<td>Volume fraction [-]</td>
<td>0.139</td>
<td>0.043</td>
</tr>
<tr>
<td>TD - Directed mesh</td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>TD - Automated mesh</td>
<td>0.5</td>
<td>0.5</td>
</tr>
<tr>
<td>Lift coeff. - Directed mesh</td>
<td>-0.2</td>
<td>-0.2</td>
</tr>
<tr>
<td>Lift coeff. - Automated mesh</td>
<td>-0.2</td>
<td>-0.2</td>
</tr>
</tbody>
</table>
3.1.1 Mesh Independence Study

A thorough study regarding the mesh was conducted for Ekambara case 1 in Tab. 3.5. The purpose of the study was to gain knowledge about how to create an independent and proper mesh for the test cases as well as for the water jacket. The study was performed by creating a directed mesh with polygonal cells and investigate the influence of axial mesh, cross section mesh and prism layers. Despite the fact that poly mesh was stated as the preferred mesh in section 1.2, the directed mesh type was proved to be more suited on the simpler cases since the flow overall was aligned with the length direction, thus aligned with the mesh. Further on, when a suitable directed mesh was selected a corresponding automated mesh with polyhedral cells was created in order to validate both mesh types. This mesh type was in advance determined as the best suited mesh for the water jacket, due to the complex geometry and the fact that Volvo Cars applies this mesh type when simulating the water jacket. Both first and second order convection scheme was used for volume fraction since large differences occurred for the different mesh types. The different meshes that were used in the mesh independence study can be seen in Tab. 3.6.

Tab. 3.6: Mesh settings for the Ekambara cases. Simulation 1-7 are directed meshes consisting of polygonal cells stretched in axial direction. Simulation 8-9 are automated meshes consisting of polyhedral cells. The abbreviation g.r stands for growth rate, h. stands for height, dir. stands for direction, k stands for 1000 and vf conv. stands for volume fraction convection.

<table>
<thead>
<tr>
<th>Simulation</th>
<th>Directed</th>
<th>Automated</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>Base size [m]</td>
<td>0.003</td>
<td>0.003</td>
</tr>
<tr>
<td>Surface g.r</td>
<td>1.1</td>
<td>1.1</td>
</tr>
<tr>
<td>Prism layers</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>Prism g.r</td>
<td>1.3</td>
<td>1.3</td>
</tr>
<tr>
<td>Prism h. [m]</td>
<td>0.003</td>
<td>0.003</td>
</tr>
<tr>
<td>Cells axial dir.</td>
<td>1000</td>
<td>500</td>
</tr>
<tr>
<td>k cells</td>
<td>400</td>
<td>200</td>
</tr>
<tr>
<td>vf conv.</td>
<td>1st</td>
<td>1st</td>
</tr>
</tbody>
</table>

The meshes in simulation 2, a directed mesh, and simulation 9, an automated mesh, were selected as suitable meshes and were further used. The meshes can be seen in Fig. 3.6.

(a) Cross sectional mesh  
(b) Axial mesh  

Fig. 3.3: Shows (a) the cross sectional and (b) the axial mesh distribution for sim. 2 in Tab. 3.6. 
Total amount of cells with these settings was 200 000.
3.1.2 Force and Parameter Study

Several forces and parameters that were widely discussed in reviewed works were investigated in order to further understand their influence in this specific case. The study concerned neglecting forces as well as studying the influence of parameter settings. Forces that were neglected were wall lubrication force, $S_\gamma$ and lift force. Wall lubrication force was investigated since this force later had to be neglected in the water jacket. The influence of $S_\gamma$ was investigated since it was one of the heaviest models in terms of computational cost, thus neglecting such model would lower the computational cost. Parameters were studied for turbulent dispersion Prandtl number, lift coefficient and drag coefficient. These parameters varied widely in previous works and was therefore investigated further to gain deeper knowledge of their influences.

3.1.3 Simulation Time and Convergence

As stated previously Ekambara was mainly simulated under steady state condition although some simulations were performed under transient conditions. All simulations, both steady and transient, were continued until the flow had reached through the whole pipe, which was confirmed by studying scalar scenes with volume fraction and velocities. Convergence was determined by converging monitor points of velocity and volume fraction at several locations as well as stable residuals. No specific upper limit for the residuals was determined due to convergence issues being present during the work. Each case was therefore treated independently.

For the steady state simulations, 5000 iterations were needed for the directed mesh to reach steady state, which was approximately 16 core hours of simulation time. The automated mesh, on the other hand, needed approximately 11000 iterations which was 160 core hours of simulation time. The number of cores used for the simulations was typically 96.

The purpose of the transient simulations was to study the amount of iterations needed within each time step. This was done in order to gain knowledge about the amount needed in the water jacket.

---

Fig. 3.4: Shows the (a) radial and (b) axial mesh distribution for sim. 9 in Tab. 3.6. The total amount of cells with these settings was 1 150 000.
3.2 Hibiki

Hibiki et al. \cite{Hibiki} performed experiments on a vertical bubble column with bubbles of size 0.003 m in diameter. The geometry was a 3.06 m long tube with inner diameter 0.05 m and can be seen in Fig. 3.5. One case from the Hibiki experiments was studied in this work. The Reynolds number in this case was 60 000.

![Fig. 3.5: Shows the geometry of the Hibiki case as well as the line where simulation data was extracted. The gravity, g, is in the negative x-direction.](image)

As stated previously, most effort was directed towards Ekambara since that case best resembled the water jacket. Therefore, the same setup as for Ekambara was used except for the gravity that was in the axial direction due to the vertical orientation of the pipe. The velocities and volume fractions at the inlet was as in Tab. 3.8. The mesh settings, which also were based on the mesh knowledge gained from Ekambara, can be seen in Tab. 3.7. However, in order to fulfill the $y^+$ criteria mentioned in section 2.6.2 near wall treatment some differences concerning the prism layer had to be made. The initial sauter mean diameter was set 0.003 m to resemble the bubbles in the experiments. As in Ekambara, lift coefficient and turbulent dispersion Prandtl number were adjusted in order to resemble the experimental data. The chosen settings can be seen in Tab. 3.7.

Tab. 3.7: Mesh and simulation settings for the Hibiki cases. Simulation 1 is a directed mesh consisting of polygonal cells stretched in axial direction. Simulation 2 is an automated mesh consisting of polyhedral cells. The abbreviation g.r stands for growth rate, h. stands for height, dir. stands for direction, k stands for 1000, vf conv. stands for volume fraction convection, TD stands for turbulent dispersion Prandtl number and coeff. stands for coefficient.

<table>
<thead>
<tr>
<th></th>
<th>Directed</th>
<th>Automated</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simulation</td>
<td>1</td>
<td>2</td>
</tr>
<tr>
<td>Base size [m]</td>
<td>0.003</td>
<td>0.003</td>
</tr>
<tr>
<td>Surface g.r</td>
<td>1.1</td>
<td>1.1</td>
</tr>
<tr>
<td>Prism layers</td>
<td>3</td>
<td>3</td>
</tr>
<tr>
<td>Prism g.r</td>
<td>1.1</td>
<td>1.1</td>
</tr>
<tr>
<td>Prism h. [m]</td>
<td>0.004</td>
<td>0.004</td>
</tr>
<tr>
<td>Cells axial dir.</td>
<td>170</td>
<td>-</td>
</tr>
<tr>
<td>k cells</td>
<td>60</td>
<td>335</td>
</tr>
<tr>
<td>vf conv.</td>
<td>1st</td>
<td>1st</td>
</tr>
<tr>
<td>TD</td>
<td>0.6</td>
<td>0.6</td>
</tr>
<tr>
<td>Lift coeff.</td>
<td>-0.05</td>
<td>-0.13</td>
</tr>
</tbody>
</table>
Tab. 3.8: Operating settings for Hibiki case. The abbreviation vel. stands for velocity.

<table>
<thead>
<tr>
<th></th>
<th>Gas</th>
<th>Liquid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Superficial vel.</td>
<td>0.3220</td>
<td>0.984</td>
</tr>
<tr>
<td>Volume fraction</td>
<td>0.2</td>
<td>0.8</td>
</tr>
</tbody>
</table>

3.3 Bottin

Bottin et al. [10] performed experiments on a horizontal pipe with liquid and gas. The test section was a 5.4 m long pipe with inner diameter 0.1 m and can be seen in Fig. 3.6. The bubble size at the inlet was 0.0015 m according to experimental data. One case from the Bottin experiments was studied in this work. The Reynolds number in this case was 448 000.

Once again, the same setup as described for the Ekambara case was used. Inlet velocities and volume fractions can be seen in Tab. 3.9. The initial sauter mean diameter was set to 0.0015 m and the surface tension was 0.0074 N/m according to the experiments. Two meshes were created, see Tab. 3.10, which resembles the meshes used in Ekambara. Both first and second order convection scheme for volume fraction were used. The adjusted lift coefficient and the turbulent dispersion Prandtl number can be seen in 3.10.

Tab. 3.9: Operating setting for the Bottin case. The abbreviation vel. stands for velocity.

<table>
<thead>
<tr>
<th></th>
<th>Gas</th>
<th>Liquid</th>
</tr>
</thead>
<tbody>
<tr>
<td>Superficial vel.</td>
<td>0.0637</td>
<td>4.42</td>
</tr>
<tr>
<td>Volume fraction</td>
<td>0.0142</td>
<td>0.9858</td>
</tr>
</tbody>
</table>
Tab. 3.10: Mesh and simulation settings for the Bottin cases. Simulation 1 is a directed mesh consisting of polygonal cells stretched in axial direction. Simulation 2 and 3 is an automated mesh consisting of polyhedral cells. The abbreviation g.r stands for growth rate, h. stands for height, dir. stands for direction, k stands for 1000, vf conv. stands for volume fraction convection, TD stands for turbulent dispersion Prandtl number and coeff. stands for coefficient.

<table>
<thead>
<tr>
<th>Simulation</th>
<th>Directed</th>
<th>Automated</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base size [m]</td>
<td>0.003</td>
<td>0.003</td>
</tr>
<tr>
<td>Surface g.r</td>
<td>1.1</td>
<td>1.1</td>
</tr>
<tr>
<td>Prism layers</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>Prism g.r</td>
<td>1.3</td>
<td>1.3</td>
</tr>
<tr>
<td>Prism h. [m]</td>
<td>0.003</td>
<td>0.003</td>
</tr>
<tr>
<td>Cells axial dir.</td>
<td>300</td>
<td>-</td>
</tr>
<tr>
<td>k cells</td>
<td>295</td>
<td>1945</td>
</tr>
<tr>
<td>vf conv.</td>
<td>1st</td>
<td>1st</td>
</tr>
<tr>
<td>TD</td>
<td>0.55</td>
<td>0.7</td>
</tr>
<tr>
<td>Lift coeff.</td>
<td>-0.25</td>
<td>-0.25</td>
</tr>
</tbody>
</table>

3.4 Water Jacket

The developed method was implemented on the water jacket which was previously described in section 1.1.1. However, due to the large number of cells in the model from Volvo Cars the method was initially implemented on one cylinder of the original model. The water jacket and the one cylinder model can be seen in Fig. 3.7. Three outlets are present in the original model, called thermostat outlet, EOC outlet and climate outlet whereas only the thermostat outlet was maintained in the one cylinder model. However, the outlet had to be moved in order to be able to minimize the model.

![Fig. 3.7](image)

(a) Water jacket  
(b) One cylinder model

The same boundary conditions were used as described for Ekambara, that is inlet condition where the velocities and the volume fractions were specified, flow split outlets and walls with no-slip conditions. Split ratios could be specified for the flow split outlets and two different settings were simulated which were settings used by Volvo Cars. The first setting, referred to case 1 and 3 in Tab. 3.11, concerned only including the thermostat outlet whereas the second setting, referred to case 2 and 4, included all outlets. For the one cylinder model the
split ratio was set to one for the maintained outlet. The different simulations settings can be seen in Tab. 3.11 where the velocities and the volume fractions were assumed to be possible conditions that the water jacket may experience. The total solution time for all simulations was 1-1.6 s where the gas was injected at the inlet during the first 0.5 s. This approach was used since the focus was towards finding areas with accumulated gas which was possible since there was sufficient time for the gas to disappear. The reason for the different total solution times was due to some simulations suffered from convergence issues. Further on, the total solution time was determined as reasonable in order to be able to compare the cases and to minimize the computational cost. The injection time of 0.5 s for the gas was found to be suitable since a certain amount of gas was needed in the system in order to maintain stable residuals and to obtain a physically possible solution.

Tab. 3.11: Simulation settings for the water jacket and the one cylinder model. The abbreviation vel. stands for velocity.

<table>
<thead>
<tr>
<th></th>
<th>One cylinder model</th>
<th>Water jacket</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Case 1</td>
<td>Case 2</td>
</tr>
<tr>
<td>Inlet gas vel. [ms⁻¹]</td>
<td>19</td>
<td>10</td>
</tr>
<tr>
<td>Inlet liquid vel. [ms⁻¹]</td>
<td>19</td>
<td>10</td>
</tr>
<tr>
<td>Gas volume fraction [-]</td>
<td>0.1</td>
<td>0.1</td>
</tr>
<tr>
<td>Liquid volume fraction [-]</td>
<td>0.9</td>
<td>0.9</td>
</tr>
<tr>
<td>Split ratio thermostat</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>Split ratio EOC</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Split ratio climate</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

The mesh was, as for Hibiki and Bottin, based on the mesh generated in the Ekambara case since this case best resemble the water jacket in terms of diameter and Reynolds number. However, since smaller passages were present in the water jacket the base size and the total prism layer height was lowered to 0.001 m, which also was the settings used at Volvo Cars. Further on, the mesh had to be changed in critical areas, such as these smaller passages since they otherwise generated unrealistically increased flow velocities. Some geometry simplifications were also necessary in order to reach stable residuals. The simplifications concerned removing three small channels that connected the back and front side of the water jacket. In addition, the surface mesh size was increased in order to smooth sharp edges and thus obtain increased mesh quality in these regions. The initial suter mean diameter was set to 0.001 m which corresponded to the base size in the water jacket. The initial suter mean diameter in Ekambara, Hibiki and Bottin did not exceeded the base sizes that were used in those cases, which motivates the suter mean diameter of 0.001 m in the water jacket. The total number of cells for the water jacket was 2.3 million and for the one cylinder model 1.2 million. The water jacket has a complex geometry which means that visualizing the mesh is hard. All mesh settings are presented in this report and therefore no figures of the mesh are included.

Due to convergence issues when directly implementing the method on the water jacket the wall lubrication force within the interaction models had to be neglected. In addition, the solver settings concerning relaxation factors and AMG linear solver had to be changed, which is described further in section 3.4.1 below. The wall lubrication force was studied on the Ekambara case in order to prove or disprove its importance. In addition, a model called cell quality remediation was added according to recommendations from [25]. The model neglects bad cells surrounded by better cells and models the mesh in those bad cells with respect to the better cells. The turbulent dispersion Prandtl number was set to 0.32 and the lift coefficient was set to -0.2 according to the Ekambara case, without the wall lubrication force seen in the result section.

Ekambara and Hibiki were the best resembling cases compared to the water jacket due
to their smaller diameters. As mentioned, these cases had a Reynolds number of 270 000 - 300 000 and 60 000 respectively. However, the Reynolds number in the water jacket was difficult to calculate due to its complicated geometry. Three lines were therefore distributed where different flow characteristics were expected, see Fig. 3.8. The mean velocity \( v \) and the length of these lines as characteristic length, \( L \), in Eq. 21 gave approximated Reynolds numbers for these locations. The lines were also used in order to compare the velocities in the derived two-phase method and the one-phase model from Volvo Cars in order to investigate if differences occurred between the two approaches.

\[
Re = \frac{vL}{\nu} \tag{21}
\]

![Fig. 3.8: The lines used to investigate the Reynolds number as well as for velocity comparison between the one-phase simulation from Volvo Cars and the derived two-phase method.](image)

### 3.4.1 Solver Settings

The solver settings that were used in the water jacket simulations are stated in Tab. 3.12. This settings were used after recommendations from [25].

Tab. 3.12: Solver settings used for the water jacket simulations. The abbreviation vel. stands for velocity.

<table>
<thead>
<tr>
<th></th>
<th>Under-relaxation factor</th>
<th>AMG linear solver</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Implicit</td>
<td>Explicit</td>
</tr>
<tr>
<td>Phase coupled vel.</td>
<td>0.7</td>
<td>0.5</td>
</tr>
<tr>
<td>Pressure</td>
<td>0.2</td>
<td></td>
</tr>
<tr>
<td>Volume fraction</td>
<td>0.7</td>
<td>0.3</td>
</tr>
<tr>
<td>( S_v )</td>
<td>0.7</td>
<td>0.3</td>
</tr>
<tr>
<td>( k-\varepsilon )</td>
<td>0.5</td>
<td></td>
</tr>
<tr>
<td>( k-\varepsilon )</td>
<td>0.5</td>
<td></td>
</tr>
</tbody>
</table>
3.4.2 Time Step Independence Study

As mentioned, the water jacket was, in contrast to almost all simulations in Ekambara, Hibiki and Bottin, simulated under transient condition. As stated previously, this was due to the more unstructured flow but also due to the transient inlet condition that was used. According to [25], the Courant-Friedrichs-Lewy (CFL) number should be maintained low, around one, for Eulerian multi-phase problems. A simulation with time step 0.00001 s was initially simulated which resulted in a sufficient CFL number of 0.19. However, this time step was unreasonable low for this project since the time needed to perform such simulation of the one cylinder model with converging monitor points would be in order of months. A time step independence study was therefore performed on the one cylinder model in order to find a more suited time step. Since the simulation with sufficient CFL number only had reached through the inlet pipe, the time step study was performed in this region. Three lines were randomly distributed on the inlet pipe as seen in Fig. 3.9. Volume fraction profiles from simulations with different time steps were extracted from these lines at two different solution times. Except for time step 0.00001 s, which was unreasonable, other more reasonable time steps were used for the independence study. These time steps were 0.001, 0.01 and 0.05 s which resulted in CFL numbers of 19, 190 and 950 respectively.

![Fig. 3.9: The lines used for the time step independence study.](image-url)
4 Results

4.1 Validation of Methods

The results generated from the validation of the method will be presented here. The three cases, Ekambara, Hibiki and Bottin will follow in order.

4.1.1 Ekambara

This section includes results from the Ekambara case. All profiles were extracted from a line close to the outlet as seen in Fig. 3.2.

Mesh Independence Study

According to Tab. 3.6 the mesh in sim. 1 had three times more cells than the mesh in sim. 3 which in turn resulted in roughly three times the computational costs. As seen in Fig. 4.1 the prediction of velocity and volume fraction remains accurate regardless of the mesh density in axial direction which indicated that the mesh in sim. 3 was sufficient. On the other hand, when further decreasing the amount of cells in axial direction, i.e. increasing the aspect ratio, unstable simulations occurred. The mesh in sim. 2 was therefore determined as the best mesh for the remaining investigation.

Concerning the cross sectional mesh, Fig. 4.2 proves that no significant differences can be seen for different densities. The mesh in sim. 4, in Tab. 3.6 was the coarsest mesh needed to still capture the characteristics of the volume fraction profile. This mesh was therefore determined as the most proper mesh in terms of computational costs yet still predicting the velocity and volume fraction profiles. However, when converting the mesh used in sim. 4 to
automated polyhedral mesh the solution did not converge. The reason for the divergence was not further investigated.

As seen for sim. 4 and 6 in Fig. 4.3 no major differences can be seen when increasing the number as well as the height of the prism layers. However, removing the prism layers from the mesh, as for sim. 7, results in poor prediction of volume fraction and velocity which proves the importance of using prism layers and thus capturing the wall bounded flow. The mesh in sim. 7 was determined as insufficient and since no differences could be seen between sim. 4 and 6, the previous settings was used for further work.

The mesh settings in sim. 2 were used for the automated polyhedral mesh study since sim. 4 did not result in a converged solution when converting to automated polyhedral mesh, as stated previously. Sim. 8 in Fig. 4.4 where first order convection scheme for volume fraction was used, shows large deviations from experimental data. When instead using
second order convection scheme for volume fraction, as for sim. 9, sufficient agreement can be seen. However, sim. 2 was the overall best performing mesh which proves that a directed mesh would be favourable for this application.

![Velocity](image)

**Fig. 4.4:** Shows a comparison between different a directed mesh and a polyhedral mesh using both first and second order convection scheme for volume fraction. (a) Shows the difference in velocity and (b) shows the difference volume fraction. The legends presents which curve that corresponds to experimental data and which curves that corresponds to the simulations presented in Tab. 3.6 as well as the order of convection scheme. Sim. 8 with first order convection scheme shows poor prediction of volume fraction whereas the prediction by sim. 9 with second order convection scheme overall is good. However, the directed mesh, sim. 2 presents best agreement with experimental data.

![Gas Volume Fraction](image)

Tab. 4.1 shows the $y^+$ interval for the Ekambara sim. 2 and 9 in Tab. 3.5. Both simulations obtained $y^+$ value within the recommendations mentioned in section 2.6.2 near wall treatment.

<table>
<thead>
<tr>
<th></th>
<th>sim. 2</th>
<th>sim. 9</th>
</tr>
</thead>
<tbody>
<tr>
<td>$y^+$</td>
<td>57-75</td>
<td>30-75</td>
</tr>
</tbody>
</table>

**Tab. 4.1:** Shows the $y^+$ interval for the Ekambara sim. 2 and 9 according to Tab. 3.5. The $y^+$ values was within the recommendations mentioned in section 2.6.2.

**Force and Parameter Study**

Fig. 4.5 shows volume fraction and velocity profiles with and without the wall lubrication force as well as with and without the $S_9$ model. The results proves that the wall lubrication force can be neglected, which had to be done for the water jacket, without affecting the solution significantly. However, the turbulent dispersion Prandtl number had to be lowered, from 0.5 to 0.32 when neglecting the wall lubrication force. Concerning the $S_9$ model, the volume fraction profile does not at all capture the characteristics of the experimental data when the model is neglected, whereas the velocity profile is not that affected.
Fig. 4.5: Shows the velocity and volume fraction for sim. 2 in Tab. 3.6 with and without the wall lubrication force as well as with and without the $S_\gamma$ model. The results with and without wall lubrication force was unchanged for the velocity profile. Some differences could be seen in the upper region of the profile for volume fraction. Without the $S_\gamma$ model the volume fraction profile presents very low values.

Fig. 4.6 shows volume fraction and velocity profiles with different settings of the drag coefficient. Large differences from experimental data can be seen for volume fraction when choosing a constant drag coefficient. Concerning the velocity profile, the constant drag coefficient generates simulation data closer to the experimental data in the lower part of the pipe compared to the Schiller-Naumann drag coefficient. Schiller-Naumann drag coefficient is however undoubtedly closest to experimental data for volume fraction.

Fig. 4.7 shows volume fraction and velocity profiles with different settings of the lift coefficient. When the lift force is neglected, i.e. lift coefficient equals zero, the characteristics of both profiles are not detected at all which proves the need for including this force. Decreasing the coefficient slightly, i.e. lift coefficient of -0.05, results in a profile for volume fraction that captures the characteristics, though presenting an inaccurate magnitude. Concerning the velocity profile the profile is similar to the one where the lift force is neglected. Decreasing the coefficient further, i.e. lift coefficient of -0.2, results in a profile that resembles the
experimental volume fraction profile. Decreasing the coefficient further lowers the accuracy in capturing the magnitude of volume fraction. An over prediction of velocity is seen for the simulated velocities close to the upper wall when the lift coefficient increases.

Fig. 4.7: Shows the velocity and volume fraction for sim. 2 in Tab. 3.6 with different settings of the lift coefficient. Values towards zero gives bad prediction of volume fraction and velocity. $C_L = -0.2$ has the best prediction of volume fraction whereas $C_L = -0.35$ has a slight under prediction close to the upper wall. The velocity is almost the same for $C_L = -0.2$ and $C_L = -0.35$.

Fig. 4.8: Shows volume fraction and velocity profiles with different settings of the turbulent dispersion Prandtl number. A high value of the turbulent dispersion Prandtl number gives less turbulent dispersion, resulting in an over prediction of volume fraction. The opposite results is seen for a low value. $T_d=0.5$ gives the best agreement towards experimental data for volume fraction. The velocity profiles are unchanged for all simulations.
Case 2

As mentioned in section 3 method, Ekambara was investigated for two different operating conditions as seen in 3.5. Case 2 had lower superficial velocity and volume fraction of gas compared to case 1. The most suited directed and automated mesh from the mesh independence study, i.e. the mesh in sim. 2 and the mesh in sim. 9, was simulated for this case. With minor changes on turbulent dispersion Prandtl number and lift coefficient overall good agreement with experimental data, for lower amount of volume fraction at the inlet, can be seen. However, both the automated and directed mesh in Fig. 4.9 shows poor agreement concerning volume fraction compared to case 1. None of the meshes resembles the profile at the upper wall of the pipe. At the same time the agreement is slightly improved for the velocity profile. Fig. 4.4 and Fig. 4.9 gives an indication that the developed method, when using second order convection scheme for volume fraction, predicts the profile better for cases with higher amount of volume fraction as for case 1. Unfortunately there was no data with further increased amount of gas volume fraction provided.

![Graphs showing velocity and volume fraction](image-url)

**Fig. 4.9**: Shows the velocity and volume fraction for sim. 2 and 9 in Tab. 3.6 for case 2, i.e. different operating conditions as seen in Tab. 3.5. The automated mesh in sim. 9 captured the characteristics of the volume fraction profile seen for the experimental data in the bulk flow but gave did not predict the profile close to the upper wall. The directed mesh in sim. 2 predicted the volume fraction profile even worse.
Iteration Analysis

Under relaxation of the solver is important in order to control the advancement of the solution. Under relaxation below one means that parts of the solution are used in the next iteration. This can improve convergence as it creates a more stable simulation. The issue with under relaxation factors when simulating under transient conditions was to let the solution converge in each time step. As mentioned, some simulations in the Ekambara case were performed under transient conditions. Fig. 4.10 shows the importance of letting the solution converge before changing time step. As can be seen, a small number of inner iterations per time step resulted in a solution significantly different from a solution where each time step had reached convergence. This indicated that a sufficient number of iterations per time step was essential in order to generate a reliable solution.

![Solution Time: 0.055 s](image)

**Fig. 4.10:** Shows the difference in accuracy between a simulation where the solution converge in each time step and a simulation with very few inner iterations. (a) shows a simulations with large values on the under relaxation factors where the solution converged in each time step. (b) shows a simulations with small values on the under relaxation factors where the solution converged in each time step. (c) shows a simulations with small values on the under relaxation factors where the solution did not converge in each time step.

4.1.2 Hibiki

Fig. 4.11 shows velocity and volume fraction profiles for the meshes in Tab. 3.7. As can be seen the simulation data, both the automated and directed mesh, agrees to the experimental data in the bulk flow. However, a sudden change of characteristics can be seen for sim. 2 in volume fraction close to the walls which happens at the transition between the prism layers and the automated polyhedral mesh. The experimental data shows a peak close to the walls.
which can not be seen for the simulation data. Overall, no substantial differences can be seen between the directed and the automated mesh, they are both overall well performing.

Tab. 4.2 shows the $y^+$ interval for the Hibiki sim. 1 and 2 in Tab. 3.8. Both simulations obtained $y^+$ values within the recommendations mentioned in section 2.6.2 near wall treatment.

**Tab. 4.2:** Shows the $y^+$ interval for the Hibiki sim. 1 and 2 according to Tab. 3.8. The $y^+$ values was within the recommendations mentioned in section 2.6.2.

<table>
<thead>
<tr>
<th></th>
<th>sim. 1</th>
<th>sim. 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$y^+$</td>
<td>31-41</td>
<td>31-41</td>
</tr>
</tbody>
</table>

### 4.1.3 Bottin

Fig. 4.12, 4.13 and 4.14 shows the volume fraction and velocity profiles for the Bottin case. Concerning the velocity profiles for all lines, all simulation data overall captures the profiles from the experiment. The directed mesh presents slightly sharper edges compared to the automated meshes which agrees better with experimental data, especially at the line farthest away from the inlet. Concerning the volume fraction, best agreement with experimental data can be seen for the directed mesh and the automated mesh with second order convection scheme for volume fraction. The automated mesh with first order convection scheme for volume fraction follows, in some way, the characteristics presented by the experimental data. This contradicts the results seen in Ekambara, Fig. 4.4 where large deviations from experimental data can be seen for first order convection scheme. However, the automated mesh with first order scheme does not present as good results as either the directed mesh nor the automated mesh with second order convection scheme. The simulated volume fraction profiles farthest away from the inlet presents poor agreement with experimental data compared to the lines closer to the inlet.
Fig. 4.12: Shows the velocity and volume fraction for sim. 1-3 @5D in Tab. 3.10 The legend presents simulation number and which order of convection scheme that was used for volume fraction. Sim. 3 presents best agreement towards experimental data.

Fig. 4.13: Shows the velocity and volume fraction for sim. 1-3 @20D in Tab. 3.10 The legend presents simulation number and which order of convection scheme that was used for volume fraction. Sim. 3 presents best agreement towards experimental data.

Fig. 4.14: Shows the velocity and volume fraction for sim. 1-3 @40D in Tab. 3.10 The legend presents simulation number and which order of convection scheme that was used for volume fraction. Sim. 3 presents best agreement towards experimental data.
Tab. 4.3 shows the $y^+$ interval for the Bottin sim. 1 and 2 in Tab. 3.9. As can be seen both simulations presents $y^+$ values within the recommendations mentioned in section 2.6.2 near wall treatment.

<table>
<thead>
<tr>
<th></th>
<th>sim. 1</th>
<th>sim. 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>$y^+$</td>
<td>40-60</td>
<td>30-60</td>
</tr>
</tbody>
</table>

4.2 Water Jacket

This section includes results from the water jacket simulations.

4.2.1 Time Step Independence Study

Fig. 4.15 and 4.16 shows volume fraction profiles for different time steps extracted from lines located as in Fig. 3.9 at total solution time 0.05 s and 0.1 s respectively. As seen in all figures the time step 0.05 s differs from the other time steps and was therefore determined as too high. The remaining time steps generates approximately the same solution, although the profile from the simulation with time step 0.01 s differs slightly in some figures. The time step 0.01 s was although determined as reasonable for the simulations both concerning solution accuracy and especially due to simulation time.
Fig. 4.15: Shows the volume fraction profiles after 0.05 s of simulation time for three different lines which are located as shown in Fig. 3.9. Time step 0.05 s presents deviating results compared to the other time steps. At total simulation time 0.05 s a time step size of 0.01 s was sufficient for all lines.
Fig. 4.16: Shows the volume fraction profiles after 0.1 s of simulation time for three different lines which are located as shown in Fig. 3.9. Time step 0.05 s presents deviating results compared to the other time steps. At total simulation time 0.1 s a time step size of 0.01 s was sufficient for all lines.

4.2.2 One Cylinder Model

Fig. 4.17 shows volume fraction of air for the one cylinder model at two different solution times, 0.8 and 1 s. The velocities differs as stated in Tab. 3.11 which clearly affected the amount of air present in the domain. Higher velocity, as for case 1, resulted in increased gas flow throughout the geometry and almost no gas can be seen after 1 s of simulation time. Case 1 after 0.8 s and case 2 after 1 s corresponds with each other, which again proved the increased gas flow for higher velocity.
Fig. 4.17: Shows volume fraction for the one cylinder model at different solution times for case 1 and case 2 as stated in Tab 3.11. Higher velocity as in case 1, resulted in increased gas flow throughout. Almost no gas can be seen after 1 s of simulation time. The remaining amount of air in case 1 after 0.8 s and case 2 after 1 s corresponds with each other.

Since the simulations did not diverge and showed results as expected in terms of decreasing volume fraction for increasing solution time, the complete water jacket was investigated further.

4.2.3 Complete Water Jacket

Fig. 4.18 shows volume fraction of air for the water jacket at two different solution times, 1 and 1.6 s. The velocities and the flow split outlets differs as stated in Tab 3.11 which resulted in different outcomes. Case 1-2 with higher velocity presented, as for the one cylinder model, increased gas flow throughout the geometry compared to case 3-4. After 1.6 s of simulation time, case 1-2 presents small amount of gas at the same locations. Case 3-4 suffered from convergence issues at around 1.2 s and no results could therefore be generated at 1.6 s. It is hard to visually judge the difference in gas holdup between the cases with one outlet and the cases with three outlets although some differences are visible.
Fig. 4.18: Shows volume fraction of gas for the water jacket at different solution times for case 1-4 as stated in Tab. 3.1. Case 1-2 with higher velocity presented, as for the one cylinder model, increased gas flow throughout the geometry compared to case 3-4. Small amount of gas can be seen after 1.6 seconds of simulation time.
Fig. 4.19 where the distribution of cells with volume fraction above 0.01 in z-direction is presented, complements the visual comparison. The direction of z can be seen in Fig. 1.3. As seen in the figures, after 1 s of simulation time the largest amount of cells with higher volume fraction are present in the water jacket cylinder head. On the other hand, after 1.6 s of simulation time few cells with higher volume fraction are present in the water jacket cylinder head. Most of the cells with higher volume fraction is after 1.6 s instead present in the water jacket cylinder block. However, the number of cells after 1.6 s is in the order of thousand times less compared to the number of cells after 1 s.

Fig. 4.19: Shows the amount of cells with gas volume fraction above 0.01. The cells are sorted with respect to z in order to get an understanding of the distribution of gas in z-direction in the water jacket. The direction of z can be seen in Fig. 1.3. Case 1-4 is shown in (a) and was extracted after 1 second. Case 1 and 2 is shown in (b) and was extracted after 1.6 second. Case 1-4 is shown in Tab. 3.1. Note that (a) is presented in thousands of cells whereas (b) presents significantly lower numbers. The highest number of cells with volume fraction above 0.01 are clustered in the upper regions of the water jacket at 1 s of simulation time. However, at 1.6 s of simulation time highest number of cells can be found in the lower regions.

Fig. 4.20 (a) and (b) shows the pressure field in the one-phase model from Volvo Cars whereas (c) shows the pressure field in the derived two-phase method. Fig 4.20 (d) shows volume fraction from the derived two-phase method. Fig. 4.20 (a) and (b) shows two black zones encapsulated and surrounded by white, which represents local low pressure zones. The pressure magnitude in these zones differs from each other and can only be visualized by using different pressure intervals on the legends. On the contrary, the local pressure zones appear with same pressure magnitude in Fig. 4.20 (c). Despite the magnitude differences same zones can be found in both the one-phase model from Volvo Cars and the derived two-phase method. Fig. 4.20 (d) shows zones of gas holdup which corresponds to these pressure zones.
Fig. 4.20: Shows the pressure distribution around the outlet leading to the thermostat for the one-phase model from Volvo Cars as well as the derived two-phase method. The highlighted areas in (a), (b) shows the pressure zones in the one-phase model from Volvo Cars. These two zones appear at different pressure intervals. (c) shows local low pressure zones obtained from the derived two-phase method which, in contrary to (a) and (b) appears in the same pressure interval. These pressure zones corresponds to where gas is encapsulated, which can be seen in (d).
Tab. 4.4 presents the volume injected in the water jacket, as well as the volume released and the remaining volume of air after 1.6 s for case 1-2 and after 1 s for case 3-4. The main outcome from the table is that the remaining air in the water jacket not at all corresponds to the air injected minus the air monitored at the outlets. This indicates that most of the air has disappeared not through the outlets but instead by other phenomenon or errors. The table also shows that three outlets resulted in more released air compared to one outlet. The flow split ratios 0.76, 0.15 and 0.07 for the thermostat, EOC and climate mentioned in section 3.4 do not correspond to the ratios of released air. The ratio of released air for case 2 is 0.65, 0.3 and 0.05 and for case 4 0.57, 0.42 and 0.01 for the different outlets respectively. The EOC outlet is thus the outlet that released most air compared to its split ratio.

Tab. 4.4: Shows the injected, released and remaining volume of gas in the water jacket after 1.6 s of simulation time for case 1-2 and after 1 s of simulation time for case 3-4. Case 1-4 are stated in Tab. 3.11. The values of $V_{\text{waterjacket}}$ seen for all cases does not correspond to the measured values seen for the $V_{\text{in}} - V_{\text{out,total}}$. Note that all values are presented in $10^{-5}$.  

<table>
<thead>
<tr>
<th></th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{\text{in}}$</td>
<td>196.0</td>
<td>196.0</td>
<td>103.0</td>
<td>103.0</td>
</tr>
<tr>
<td>$V_{\text{out,thermostat}}$</td>
<td>6.591</td>
<td>5.123</td>
<td>1.718</td>
<td>1.256</td>
</tr>
<tr>
<td>$V_{\text{out,EOC}}$</td>
<td>-</td>
<td>2.337</td>
<td>-</td>
<td>0.930</td>
</tr>
<tr>
<td>$V_{\text{out,climate}}$</td>
<td>-</td>
<td>0.431</td>
<td>-</td>
<td>0.034</td>
</tr>
<tr>
<td>$V_{\text{out,total}}$</td>
<td>6.591</td>
<td>7.891</td>
<td>1.718</td>
<td>2.220</td>
</tr>
<tr>
<td>$V_{\text{in}} - V_{\text{out,total}}$</td>
<td>189.4</td>
<td>188.1</td>
<td>101.3</td>
<td>100.8</td>
</tr>
<tr>
<td>$V_{\text{waterjacket}}$</td>
<td>0.024</td>
<td>0.017</td>
<td>0.988</td>
<td>1.182</td>
</tr>
</tbody>
</table>

All values are presented in $10^{-5} m^3$

Tab. 4.5 shows $y^+$ values for the water jacket case 1-2 and 3-4 according to Tab. 3.11. The maximum and minimum values varied a lot for all cases. If the average values are assumed to represent the whole domain, the recommendation mentioned in section 2.6.2 near wall treatment is not fulfilled.

Tab. 4.5: Shows $y^+$ values for the water jacket case 1-2 and 3-4 according to Tab. 3.11. If the average values are assumed to represent the whole domain, the recommendation mentioned in section 2.6.2 near wall treatment is not fulfilled.

<table>
<thead>
<tr>
<th></th>
<th>Case 1-2</th>
<th>Case 3-4</th>
</tr>
</thead>
<tbody>
<tr>
<td>maximum $y^+$</td>
<td>129</td>
<td>83</td>
</tr>
<tr>
<td>minimum $y^+$</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>average $y^+$</td>
<td>19</td>
<td>8</td>
</tr>
</tbody>
</table>

Fig. 4.21 shows velocity profiles from the one-phase simulation from Volvo Cars compared with the derived two-phase method at lines located as in Fig. 3.8. Overall agreement can be seen for all lines, although some differences occurs, which means that the two-phase method did not affect the liquid flow in a significant way. Fig. 4.21 (b) shows that the velocity profile from the one-phase model from Volvo Cars is not fully developed. As stated in the figures, the local Reynolds numbers differ between 10 000 to 100 000 which overall agrees with the Reynolds numbers in Ekambara and Hibiki, although Hibiki resembles the water jacket best.
Fig. 4.21: Shows velocity profiles from the one-phase simulation from Volvo Cars compared with the derived two-phase method. The location of the lines are presented in Fig. 3.8. Same characteristics are seen in (a) and (c) whereas (b) shows a fully developed profile for the two-phase simulation and a undeveloped profile for the one-phase simulation.
5 Discussion

The objective of the study is highly motivated since accumulation of air bubbles can cause hot spots on the walls which could lead to wall degradation. The use of CFD in order to predict such accumulations can be a useful tool in the water jacket development process. In the absence of experimental data the method, where previous works have been studied in order to create a suitable method, must be seen as appropriate. As stated, there are lots of investigations on different two-phase methods which facilitated the theoretical background. The developed method was optimized and tested towards much simpler geometries. These geometries have a relatively structured flow compared to the water jacket, which due to the complicated geometry has a varying flow structure. The assumption that the method is directly applicable on the water jacket is therefore not well supported. Due to the turbulent nature of the water jacket experiments would be advisable to perform in order to provide experimental data for validation. The method overall performs well on simpler geometries and was tested on cases with varying boundary and physics condition which although gives a good indication of how to set up the method for the water jacket.

Implementing the method on a vertical pipe proved to be a lot easier than on a horizontal pipe. This is believed to be due to the simpler flow characteristics caused by the gravity direction being aligned with the pipe, i.e in stream wise direction. The parameter study was therefore performed on a horizontal pipe in order to include the more complicated flow. The Ekambura case was used since the Reynolds number and the diameter best resembled the water jacket. If all forces were constant, the turbulent dispersion Prandtl number and the lift coefficient are by far the most important parameters to adjust. However, Fig. 4.6 indicates that the drag coefficient also has a large impact on the characteristics. The drag force is the most researched phenomena which means that varying the drag coefficient was unjustified, which motivates the use of Schiller-Naumann drag coefficient model. There are models available for the turbulent dispersion Prandtl number and lift coefficient but the recommendations from previous work was to tune them to resemble experimental data. However, the parameters are tuned after a specific flow which could make them very flow depending. This means that they could be performing worse for other flow characteristics like the flow in the water jacket, but this could not be proved or disproved due to the lack of experimental data.

The main conclusion from the mesh study was that the directed mesh was preferred due to the lower number of cells, lower simulation time as well as more stable solutions. However, the automated mesh was the most suitable mesh strategy in the water jacket due to its simple implementation. On the other hand, if the directed mesh was to be used, the density in both axial direction and the cross sectional plane did not affect the solution as shown in Fig. 4.1 and 4.2. The most important was instead to properly resolve the wall bounded flow, as can be seen in Fig. 4.3 where the simulation without prism layers do not capture the volume fraction profile at the upper wall. In the water jacket small passages are present which limited the number of layers. According to the mesh study four prism layers was determined as enough since no significant difference could be seen when using a thicker total layer consisting of 15 prism layers. In addition, the total number of cells increased when increasing the prism layers which further motivated the use of fewer prism layers. These facts motivates the use of four layers in the water jacket. Concerning the growth rate, 1.3 was assumed to be sufficient since no significant difference could be seen in Fig. 4.3 close to the walls when comparing growth rate 1.3 and 1.05. Another upside of the directed mesh was that it was less diffusive and therefore only required the less complicated and more stable first order volume fraction convection scheme. On the other hand, the automated mesh required second order convection scheme for volume fraction in order to capture the volume fraction profile which is clearly proved in Fig. 4.4.
On the contrary, when simulating with first order convection scheme for volume fraction in the Hibiki case good agreement could be found. In addition the Bottin case with first order convection scheme did capture some of the characteristics, significantly better than in the Ekambara case. However, first order convection scheme did not present as good results as second order convection scheme. The main differences between Ekambara and Bottin was greater pipe diameter as well as lower amount of volume fraction for the latter case which could be the reason for the better accuracy when using first order convection scheme. The different flow type in Hibiki, i.e. vertical flow also seems to be less diffusive, thus presenting an accurate solution with first order convection scheme. However, the flow in Ekambara resembles the water jacket best which therefore could be an indication of the need for second order convection scheme in the water jacket.

As stated in section 1.2 previous work the drawback of Eulerian-Eulerian is convergence issues. Despite the arguments mentioned before, first order convection scheme had to be used in the water jacket due to such convergence issues when using second order convection scheme. Assuming that first order convection scheme provided accurate simulations, the results provided an interesting pattern to further investigate. By studying Fig. 4.18 a pattern can be seen in all figures, the bubbles will cluster in zones left and right of the thermostat and Fig. 4.20 shows zones of locally low pressure in which the bubbles seem to be captured. This is believed to happens due to the higher pressure surrounding the low pressure zones and thus prevent the bubbles from escaping. Further on, Fig. 4.19 (a) shows that the gas has risen to the top part at the early stages of the simulations which naturally occurs due to the lighter density of the gas phase. At the later stage of the simulations, Fig. 4.19 (b), different characteristics can be seen where most of the gas is located in the engine block which could mean that the water jacket has good deaeration in the water jacket cylinder head, although, this was initially believed to be a problematic area. The good deaeration in the upper parts further proves that the remaining gas is trapped by the low pressure zones. A simpler and much more cost efficient way of studying critical areas in the water jacket could therefore be to investigate such low pressure zones. Even though there exist big difference in pressure magnitude between the one-phase simulation from Volvo Cars and the derived two-phase method, the velocity profiles overall corresponded well as seen in 4.21 which means that the flow through of gas should be approximately the same granted that the locations of the lines represents the whole domain. Henceforth, studying the pressure field in the one-phase model is further motivated.

Another way of lowering the computational costs would be to use the fully developed velocity field from a one-phase simulation and solve the two-phase method on a frozen liquid velocity field. This was not performed in this project due to time limitations, but would be an interesting approach. However, if a frozen approach is used, some of the turbulence induced by the bubbles are lost. Another approach could instead be to initialize the two-phase method with the liquid velocity field from a one-phase simulation. This would probably not result in lowered computational costs but a more stable simulation in terms of convergence.

As stated in section 4 results, one of the velocity profiles from the one-phase model from Volvo Cars was not fully developed. Wall functions are developed for fully developed flow and since this was not the case the wall function might not work in a proper way. There are most certainly several regions with velocities that are not fully develop and the performance of the wall functions can therefore be questioned.

The big differences in pressure seen in Fig. 4.20 between the one-phase simulation from Volvo Cars and the derived two-phase method could be due to the geometrical simplifications mentioned. Three small channels that connected the back and front side of the water jacket in the engine block were removed which might be the reason for the differences in pressure field. The differences can also be due to the lower mesh resolution in the derived method compared to the one Volvo Cars used. In order to generate converging solutions coarser surface mesh had to be used in order to "mesh over" very narrow regions. In these regions the velocity of the gas phase otherwise reached unphysical values and caused the solution
to diverge. This was believed to be due to bad mesh quality in these specific regions.

In reality the engine does not operate under constant conditions as simulated in this project. The engine speed instead varies a lot when the car is in use. In addition the engine could experience vibrations and forces generated from operating the vehicle. This means that the cooling system in the engine operates under varying conditions as well. With varying conditions on the water jacket the flow field and the pressure field would experience changes that might cause the local low pressure zones to appear in different places or causing them to disappear. This might in turn result in improved deaeration of the water jacket. Running the engine in varying speed might therefore be a solution in preventing gas hold up in the water jacket. Further on, concerning the boundary conditions, the approach of applying a uniform distribution of gas at the inlet does not correspond to the reality where instead gas bubbles enters the domain in a random fashion. The outlet condition where split ratios were specified as constant values does not either represents the reality where instead, as mentioned previously, the split ratios varies depending on the temperature of the coolant. The walls are modeled as smooth walls whereas the walls in the real water jacket have a surface roughness. The overall conclusion is that the boundary conditions are simplifications of the reality and thus, most likely affects the accuracy of the solution. Concerning the initial conditions, all simulations were initialised as a water jacket filled with liquid, i.e no gas was present in the beginning. The initial liquid velocity was set to the inlet velocity in the test cases in order to speed up the simulations. However, in the water jacket the initial liquid velocity was set to zero since no uniform velocity direction was present. These initial conditions are assumed to represent the reality since gas could start entering the water jacket filled with liquid due to filling or from leakage.

By measuring the injected volume of gas and subtracting the released volume of gas the remaining volume of gas could be obtained. When adding the volume in each cell the amount is significantly lower which points to false diffusion within the domain. As discussed previously the second order convection scheme for volume fraction is essential in order to predict the gas hold up. Since first order convection scheme was used in the water jacket the big difference in remaining volume of gas might be due to false diffusion. In reality some diffusion happens naturally but should not cause such a big difference. This difference is the main evidence that proves the unreliability in the method when using first order convection scheme. Since large differences in accuracy was obtained when simulating with different conditions, i.e. Ekambara, Hibiki and Bottin, with first order convection scheme there is no guarantee that the method works for case comparison on the water jacket. However, if the differences obtained in Ekambara, Hibiki and Bottin only depends on the differences in geometry, the method could be used for case comparison in the water jacket. The difference could be due to the large CFL number gained from the large time step that were needed in order to perform reasonable simulations in terms of computational time. Further on, the study of remaining volume of gas should have been performed on Ekambara, Hibiki and Bottin in order to investigate if same differences could be seen for the test cases. This could prove or disprove the results found for the water jacket.

Second order convection scheme seems to be easier implemented on simpler geometries. The geometry in the water jacket could therefore be simplified by removing the sharp edges and widening the narrow channels. The simplified geometry then has to be validated against the one-phase simulation from Volvo Cars in order to ensure that the simplifications will not effect the results in a remarkable way. If agreement would be obtained and second order convection scheme would work this simplification would be motivated. Another approach could be a more refined mesh in the problem areas. However, this would increase the number of cells and therefore the computational cost which already was very high due to two-phase flow and all interaction models adding equations to the solver. Eulerian-Lagrangian which was previously described in the theory chapter could therefore be a better suited method on such complicated geometries. However, if only one method was to be studied the previous work pointed towards the Eulerian-Eulerian approach since this approach was
easier to implement when the main focus was to study gas holdup. On the other hand, a more thorough comparison of the two approaches in an early state might have resulted in Eulerian-Lagrangian as the best suited method. As mentioned in section 1.2 previous work, Idelsohn et al. [18] stated that the errors are generally higher in the Eulerian-Eulerian approach which further motivates an investigation of the Eulerian-Lagrangian approach.

As stated, the CFL number should be low, around one, in order to have a proper temporal resolution. This resulted in unreasonably small time steps which meant that 1.6 s of total solution time demanded approximately four to five months spent on one simulation, even when using 192 cores. This motivated the larger time step of 0.01 s which according to the time step study provided similar accuracy. Therefore, the CFL number is not that important granted that the inlet pipe, where the time step study was performed, represents the whole domain. The large number of inner iterations that were needed further increased the simulation time. However, if the inner iterations are lowered convergence within each time step is not fulfilled and thus resulting in an unphysical solution which was proven in Ekambara, see Fig. 4.10.

As mentioned the interaction models were computationally heavy and Fig. 4.5(b) shows that by neglecting the wall lubrication force no major differences were obtained. On the other hand, neglecting the \( S_\gamma \) model, i.e. neglecting the breakup and the coalescence phenomenon, results in major differences which therefore proves the need for this model and those phenomenon. However, the parameter study was performed with first order convection scheme and the \( S_\gamma \) model might not be that important when using second order convection scheme. The \( S_\gamma \) model is the heaviest interaction model due to its demanding breakup and coalescence models and neglecting that model resulted in lowered computational cost. The \( S_\gamma \) model on the other hand, is important if heat transfer is to be studied since it helps predicting the interaction area between two bubbles which is an essential parameter for capturing the correct heat transfer between two bubbles. When simulating the heat transfer the energy equation is important. All simulations were performed without the energy equation which means that the temperature was constant within the whole domain. This simplification can therefore have large effect on the flow due to constant densities of both phases. If heat transfer would be added to the method the density of both phases would then vary with the increasing or decreasing temperature. This could change the characteristics of the flow which means that the method has to be validated towards experiments with different densities.

As stated in section 3 method, RANS modelling was used over LES since the computational cost when using LES was considered unreasonably high for this project. The use of RANS modelling was further motivated since it was used in previous works that were studied. As turbulence model the realizable \( k-\varepsilon \) two-layer model was used. Since Volvo Cars applies this model on the water jacket, there was no thought of using another model. The fact that the model, as stated in section 2.6.1 \( k-\varepsilon \) model, is one of the most widely used and validated models since the model can handle a wide range of flows and works well in the free stream, further motivates the choice. Different turbulence models were used in previous works, which indicates that other models also could be suited. Realizable \( k-\varepsilon \) was although the most widely applied model. Modelling the turbulence using RANS over resolving it using LES may result in a less accurate solution. However, the results generated using RANS were assumed to be more valuable than performing few, more accurate simulations using LES.

There are always errors in experimental data due to experimental setup, reading errors etc. The experimental velocity profile in Ekambara has unexpected structures in the bulk flow which can not be seen for the experimental data in Bottin. These unexpected structures contradicts the theoretical profile of pipe flow. However, since the profile in Bottin do not have this unexpected structure in the bulk flow points to error in the experimental data in Ekambara. The unusual structure in experimental data in Ekambara might be different from the theoretical profiles due to gas being dispersed in the liquid phase which could explain
the slightly deformed velocity profile presented by the simulations. This is believed to occur due to the lower density of the gas phase causing an obstacle close to the upper wall which results in decreased liquid velocity which can be seen in the experimental data for Ekambara and Bottin.

Concerning the $y^+$ values for the water jacket, the average values were not in the $y^+$ criteria mentioned in section 2.6.2 near wall treatment. However, the errors caused by this was significantly lower than the error caused by simulating with first order convection scheme for volume fraction which can be seen when comparing the differences seen in Fig. 4.3 and 4.4. Further work should therefore be focused on the implementation of second order convection scheme for volume fraction. If this succeeds it is believed that the method will present accurate results based on the accuracy on the test cases with second order, which motivates a continued development of the two-phase method.

The theoretical background included in section 2 theory is assumed to be well covering and informative in order to understand the method, results and discussion in this project. A deeper elaboration on all coefficients could have been included in the theoretical background, but was not needed to obtain a better understanding of the interaction forces. Therefore the focus was instead on presenting the forces and its main effects on the flow.

Further on, concerning the objectives stated in section 1 introduction. Despite all shortcomings that are mentioned in the discussion, all objectives are determined as fulfilled. A method for simulating gas bubbles in the water jacket has been derived. The method is based on an existing approach which together with other approaches have been carefully judged. A validation of the method has been performed on different cases that were found in the literature. Finally, the water jacket has been simulated during different operating conditions. The strengths and the weaknesses of the method are presented which gives knowledge on where further work needs to be directed, in order to create a better and more accurate method for the water jacket.
6 Conclusions

• There are two suitable methods for this two-phase application, called Eulerian-Eulerian and Eulerian-Lagrangian. A method using the Eulerian-Eulerian approach has been derived which proved to be unstable. Eulerian-Lagrangian might therefore have been the better choice.

• Using test cases was a good approach in developing a method for simulating gas and liquid in the water jacket. This provided knowledge of how to set up the method, including models and setting the values of the parameters since it could be validated towards experimental data.

• The derived method should be viewed upon as a tool to investigate gas hold up not to measure the actual volume of the gas. The settings for coefficients, constants, forces and the implemented models all contribute to sources of errors. This makes the model in some way unreliable until experimental data have been used to validate the method in the water jacket. Despite the disadvantages a method for simulating gas in liquid has been developed.

• A simpler and a more cost efficient way of studying critical areas in the water jacket could be to investigate low pressure zones. These zones were found to prevent gas from exiting the domain.

• Until convergence is reached for second order convection scheme for volume fraction the method has to be further developed. This could be solved by simplifying the geometry or by improving the mesh.

• Convergence within each time step is essential in order to obtain physical solutions. Rather than focusing on a satisfying CFL number the time step is better kept larger due to the large computational costs of the method.

• The velocity profiles from the derived two-phase method and the one-phase model from Volvo Cars overall has the same characteristics, which means that the method did not affect the flow in a remarkable way.
7 Outlook

Further work is needed concerning:

- Implementing the second order convection scheme for volume fraction with maintained stable residuals. It was obviously proved that higher order convection scheme will improve the method when using automated polyhedral mesh.

- Validation of the method on the water jacket. This means that experiments have to be conducted.

- The Eulerian-Lagrangian approach which would give a deeper knowledge about this method. This will in turn facilitate a proper decision on which approach to use for further work on the water jacket.

- Applying a fully converged one-phase liquid flow field into the flow field of the liquid phase as an initialization of the two-phase method.
8 Perspectives

From an environmental perspective the derived method for the water jacket could lead to a better cooling system, which in turn means better cooling of the engine and in long terms an improved performance of the engine with higher efficiency. A more efficient engine would lower the fuel consumption and lower the emissions caused by the exhausts.

The method will support Volvo Cars in their work on predicting problematic areas in the water jacket. The air can prevent cooling of the engine and may in worst case cause cracks in the engine structure. The costs of repairing such engine failure could be high which means that the derived method could in long terms lower the costs for car owners.

To study the gas hold up in the water jacket computationally heavy and time consuming simulations have to be performed. These simulations are expensive in terms of occupying and maintaining a CPU cluster. This in turn can prevent other simulations to be performed. However, if the pressure field of a one-phase simulation is directly studied, no additional CPU time is needed.
References


