Computational and experimental study of fuel leakage through a ventilation valve during various driving conditions

Fattahi, Sadegh
Månsson, Philip
Computational and experimental study of fuel leakage through a ventilation valve during various driving conditions

Sadegh Fattahi
Philip Månsson

Academic supervisor: Magnus Andersson
Industrial supervisors: Ehsan Yasari, Robert Palm and Anders Pihl
Examiner: Matts Karlsson
Abstract

Fuel leakage through a fill limit vent valve (FLVV) inside a fuel tank is an important factor to consider during the design of a new tank. The performance of the carbon canister which absorbs the hydrocarbon can be compromised if fuel manages to escape through the valve, so called Liquid Carry Over (LCO) and thus not fulfilling the fuel emission requirements. As of today this is not thoroughly investigated using experiments nor Computational Fluid Dynamics.

The main focus of this study was to develop a method to simulate the behaviour of the FLVV during various driving conditions at an early design stage and if this gives rise to fuel escaping through the FLVV. This method was later to be validated with an experimental set-up and later used to perform some simulations to investigate LCO by varying different parameters such as fuel level and different types of driving. What happens when the canister is purging was also investigated to see if it has a pronounced effect on LCO. Purging is when hydrocarbons, absorbed by the canister, are sent to the engine and giving rise to an under pressure in the tank. The method was developed to run on a cluster utilizing 200 Central Processing Unit Cores where each simulated physical second required an average of 3 hours of simulation time. The flow inside the tank was simulated using a Volume Of Fluid (VOF) multiphase model and the dynamic behaviour of the floater inside the FLVV was simulated using an overset mesh with a Dynamic Fluid Body Interaction. The movement of the simulated dynamic floater was validated with an experimental set-up specifically developed for the overset mesh validation and the motion of the floater was captured at a fairly accurate level. A prototype for an experimental tank was also developed and produced to validate the VOF set-up used for sloshing inside the tank which was utilized on the real tank but due to time limitation the experiments were not performed.

The results from the parameter investigation showed that LCO was present in cases with high fuel level inside the tank (95 %) and that an aggressive driving gives rise to a higher level of LCO compared to normal driving. Simulations with a fuel level of 85 % and lower showed no evidence of LCO for this particular tank model. The purging of the tank induced a pumping effect giving rise to a higher level of LCO pumped through by the floater.
Acknowledgments

This thesis has been carried out at Volvo Cars Corporation (VCC) thus we would like to thank our supervisors Ehsan Yasari and Robert Palm for providing help and knowledge in the area of CFD here at the fuel system department. We would also like to express our utmost gratitude to Anders Pihl for his technical expertise of the fuel system and the surrounding components. We would also like to thank Chrithofer Karlberg and Anders Aronsson for their continuous help throughout the project and making us feel like a part of the group. Lastly we would like to show gratitude to our fellow thesis students for their supports and inputs when needed. We sincerely are grateful to be given the opportunity to conduct our thesis at VCC.

We would like to thank and express our gratitude to our academic supervisor, Magnus Andersson, for his valuable inputs and feedback during the course of this thesis.

Sadegh Fattahi & Philip Månsson
2019-05-22
Gothenburg
Nomenclature
Abbreviations and Acronyms

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>VCC</td>
<td>Volvo Cars Corporation</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>FLVV</td>
<td>Fuel Limit Vent Valve</td>
</tr>
<tr>
<td>LCO</td>
<td>Liquid Carry Over</td>
</tr>
<tr>
<td>PEM</td>
<td>Pump Electronic Module</td>
</tr>
<tr>
<td>ECU</td>
<td>Electronic Control Unit</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier-Stokes</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds Averaged Navier-Stokes</td>
</tr>
<tr>
<td>RKE</td>
<td>Realizable $k - \varepsilon$</td>
</tr>
<tr>
<td>VOF</td>
<td>Volume of Fluid</td>
</tr>
<tr>
<td>DOF</td>
<td>Degrees of Freedom</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant-Friedrichs-Lewy</td>
</tr>
<tr>
<td>NVD</td>
<td>Normalized Variable Diagram</td>
</tr>
<tr>
<td>HRIC</td>
<td>High Resolution Interface Capturing</td>
</tr>
<tr>
<td>FOU</td>
<td>First Order Upwind</td>
</tr>
</tbody>
</table>

Latin Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$t$</td>
<td>Time</td>
<td>[s]</td>
</tr>
<tr>
<td>$S_{Mx}$</td>
<td>Source term of body forces</td>
<td>[kg/m²s²]</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure</td>
<td>[kg/ms²]</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy</td>
<td>[m²/s²]</td>
</tr>
<tr>
<td>$\ell$</td>
<td>Length scale</td>
<td>[m]</td>
</tr>
<tr>
<td>$C_{\mu}$</td>
<td>Eddy viscosity coefficient</td>
<td></td>
</tr>
<tr>
<td>$P_k$</td>
<td>Generation of $k$ due to mean velocity</td>
<td></td>
</tr>
<tr>
<td>$P_b$</td>
<td>Generation of $k$ due to bouyancy</td>
<td></td>
</tr>
<tr>
<td>$Y_m$</td>
<td>Dialation dissipation</td>
<td></td>
</tr>
<tr>
<td>$S_k$</td>
<td>Source term</td>
<td></td>
</tr>
<tr>
<td>$C_1, C_2, C_{1\varepsilon}, C_{3\varepsilon}$</td>
<td>Coefficients</td>
<td></td>
</tr>
<tr>
<td>$S_\varepsilon$</td>
<td>Source term</td>
<td></td>
</tr>
<tr>
<td>$V$</td>
<td>Volume</td>
<td>[m³]</td>
</tr>
<tr>
<td>$C_\alpha$</td>
<td>Sharpening factor</td>
<td></td>
</tr>
<tr>
<td>$I$</td>
<td>Unity tensor</td>
<td></td>
</tr>
<tr>
<td>$y^+$</td>
<td>Dimensionless wall distance</td>
<td></td>
</tr>
<tr>
<td>$C$</td>
<td>Courant number</td>
<td></td>
</tr>
<tr>
<td>$\dot{M}_{i,c}$</td>
<td>Approximated evaporation rate</td>
<td></td>
</tr>
</tbody>
</table>
Greek Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>τ_{ij}</td>
<td>Viscous Stress Components</td>
<td>[N/m²]</td>
</tr>
<tr>
<td>ρ</td>
<td>Density</td>
<td>[kg/m³]</td>
</tr>
<tr>
<td>μ</td>
<td>Eddy Viscosity</td>
<td>[m²/s]</td>
</tr>
<tr>
<td>λ</td>
<td>Second Viscosity</td>
<td>[m²/s]</td>
</tr>
<tr>
<td>φ</td>
<td>Arbitrary Variable</td>
<td>[-]</td>
</tr>
<tr>
<td>δ_{ij}</td>
<td>Kronecker delta</td>
<td>[-]</td>
</tr>
<tr>
<td>ε</td>
<td>Rate of Dissipation of Turbulent Kinetic Energy</td>
<td>[m²/s³]</td>
</tr>
<tr>
<td>θ</td>
<td>Velocity Scale</td>
<td>[m/s]</td>
</tr>
<tr>
<td>α_i</td>
<td>Phase Volume Fraction</td>
<td>[-]</td>
</tr>
<tr>
<td>μ_i</td>
<td>Dynamic Viscosity of each phase</td>
<td>[kg/m·s]</td>
</tr>
<tr>
<td>ρ_i</td>
<td>Density of each phase</td>
<td>[kg/m³]</td>
</tr>
<tr>
<td>σ</td>
<td>Surface Tension</td>
<td>[J/m²]</td>
</tr>
<tr>
<td>θ</td>
<td>Contact angle</td>
<td>[deg]</td>
</tr>
<tr>
<td>σ_{sv}</td>
<td>Interfacial Tension of solid-vapor</td>
<td>[J/m²]</td>
</tr>
<tr>
<td>σ_{sl}</td>
<td>Interfacial Tension of solid-liquid</td>
<td>[J/m²]</td>
</tr>
<tr>
<td>σ_{lv}</td>
<td>Interfacial Tension of liquid-vapor</td>
<td>[J/m²]</td>
</tr>
<tr>
<td>Φ</td>
<td>Arbitrary Variable</td>
<td>[-]</td>
</tr>
<tr>
<td>Φ̃</td>
<td>Normalized Arbitrary Variable</td>
<td>[-]</td>
</tr>
<tr>
<td>Θ</td>
<td>Angle Between interface normal and cell face vector</td>
<td>[deg]</td>
</tr>
</tbody>
</table>
## Contents

1 Introduction  
1.1 Background .............................................. 1  
1.2 Problem Statement ..................................... 2  
1.3 Objectives ............................................. 2  
1.4 Delimitations .......................................... 3  

2 Theory  
2.1 Fuel System ........................................... 4  
  2.1.1 Fuel tank ........................................... 4  
  2.1.2 Fuel Distribution System ......................... 4  
  2.1.3 FLV .................................................. 4  
  2.1.4 Carbon Canister .................................. 5  
  2.1.5 Purging ............................................. 5  
  2.2 Fluid Dynamics ...................................... 7  
    2.2.1 Fluid Properties ................................... 7  
    2.2.2 Governing Equations ............................... 7  
  2.3 Turbulence Modelling ................................. 8  
    2.3.1 Unsteady Reynolds Averaged Navier-Stokes Models . ... 9  
  2.4 Multiphase Flows .................................... 11  
    2.4.1 VOF Multiphase Model ............................ 13  
  2.5 Evaporation .......................................... 16  
    2.5.1 Modelling of Evaporation .......................... 16  
    2.5.2 Antoine’s Equation ................................. 17  
  2.6 Overset Meshing ...................................... 17  
  2.7 Near-Wall-Treatment .................................. 19  
  2.8 Modelling of Solid Body Motion ..................... 20  
  2.9 Courant-Friedrichs-Lewy Condition .................. 20  
  2.10 Numerical Methods ................................... 21  
    2.10.1 Segregated solver .................................. 21  
    2.10.2 Normalized Variable Diagram (NVD) ............... 22  
    2.10.3 High Resolution Interface Capturing (HRIC) ....... 23  
    2.10.4 Temporal Discretization .......................... 24  
    2.10.5 Mesh Types and Quality .......................... 25
3 Method

3.1 Literature Review .......................................................... 28
   3.1.1 Experimental Methods ........................................... 28
   3.1.2 Numerical Methods ................................................ 29
3.2 Tank and FLVV Geometry ................................................. 30
3.3 Experimental Set-up ...................................................... 32
   3.3.1 FLVV Motion Experimental Set-up ............................. 32
3.4 Computational Domain .................................................... 34
3.5 Numerical grid .............................................................. 36
3.6 Numerical Model Set-up ................................................... 36
   3.6.1 Boundary Conditions and Physics ............................ 37
   3.6.2 Fill Limit Vent Valve Motion Modelling ..................... 39
   3.6.3 Post-processing ..................................................... 41
3.7 Spatio-Temporal Discretization Investigation ....................... 42
   3.7.1 1st vs 2nd Order Temporal Discretization ................. 43
   3.7.2Mesh Refinement .................................................... 45
   3.7.3 Adaptive Time-step Control and Computational Cost .......... 46
3.8 LCO Investigation and FLVV Performance ............................. 48
   3.8.1 Purge Resonance ................................................... 48
3.9 Evaporation Model ........................................................ 49

4 Results ................................................................. 51

4.1 FLVV Motion Validation ............................................... 51
4.2 LCO investigation and FLVV Performance ............................ 51
   4.2.1 Rigid Floater ......................................................... 51
   4.2.2 Aggressive Driving Cycle ........................................ 52
   4.2.3 Purge Resonance ................................................... 54
   4.2.4 Fuel Level ............................................................ 57
4.3 Evaporation ............................................................... 58

5 Discussion ................................................................. 59

5.1 Numerical Method ........................................................ 59
5.2 Experimental set-up and Validation .................................. 62
5.3 Model Usage and Further Model Development ....................... 63

6 Conclusions ................................................................. 64
1 Introduction

1.1 Background

At Volvo Cars Corporation (VCC) the need for simulations is ever growing especially in the field of Computational Fluid Dynamics (CFD). This is due to the higher cost of experimental testing and also the fact that CFD is a convenient way of performing simulations at a lower cost. In the fuel department the parts concerned are the tank and the surrounding sub-parts such as venting valves, pumps, fuel lines etc. They are all subjected to the fluid which in this particular case is gasoline and thus CFD is used more frequently for simulations to evaluate the performance of these parts. Similar investigations have been performed at Ford Motor Co. investigating the fuel level sensor inside tank and the effects of the sloshing using CFD [1]. Fluid sloshing inside a container in general is an interesting phenomena being investigated with different industrial applications such as fuel transportation at sea, rocket fuel etc. The sloshing induces forces giving instabilities and effecting structural integrity of the vessel thus the need of cheap and reliable methods of investigation to reduce cost of experimental testings. CFD could be a suitable candidate for such investigation if the simulation cost is kept low.

One important part of the fuel distribution assembly is the fuel limit vent valve (FLVV). Due to the evaporate nature of the fuel inside the tank the fumes created lead to a pressure build up inside the tank. The fumes have to be evacuated in order to be able to refuel the tank and not increase the pressure build up even further thus the necessity of the FLVV. This pressure build up is not just a problem during refuelling but also in a warm climate where the fumes are created at an even higher pace leading to a much higher and dangerous pressure peak. The FLVV also closes when the vehicle has rolled over and is in a upside down state. The fumes are directed from the FLVV and taken care of by a carbon canister in order to meet emission requirements set by governments around the world and most restrictively the United States of America [2]. A simplified overview of the fuel system can be seen in figure 1.

The FLVV contains two separate floaters, an upper and a lower floater. The upper floater operates depending on the buoyancy effect acting on it due the difference in density between the gas and the fluid. When the tank is refuelled and the fuel reaches a certain limit the buoyancy effect leads to the rise of the upper floater and sealing the flow to the canister (Fig. 2). This causes a pressure build up which shuts the capless unit and lets the user know that the tank is full. During normal driving conditions when the fuel is sloshing around in the tank a potential problem arises with the FLVV. If some of the fuel manages to land on top of the upper floater it could find its way to the, so called Liquid Carry Over (LCO), thus leading to higher emission which is not desired. Thus the simulation of sloshing inside the tank is also of importance to ensure that no fuel is reaching the canister. Investigation has previously been done at VCC to develop a CFD model for sloshing inside a fuel tank and the performance of the FLVV. The conducted work required further elaboration to get a fully verified and validated CFD model [3]. The performance of the FLVV might also be effected by the evaporation inside the tank which gives rise to a pressure change. Another important aspect to investigate is whether the purging of the canister induces a pumping effect on the FLVV valve which could subsequently lead to LCO.
1.2 Problem Statement

In order for a fuel tank to operate smoothly and also fulfill the emission requirements the existence of an FLVV is crucial. During various driving conditions when the vehicle is accelerated, decelerated, climbing, descending or turning the fuel inside the tank moves in a chaotic manner. The open FLVV valve inside the tank could thus lead to fuel sticking on the top floater and finding its way into the venting line and entering the canister, so called Liquid Carry Over (LCO). This will cause unwanted fuel in the system and subsequently disrupt the performance of the canister and thus increase emissions output.

1.3 Objectives

The objectives of this project are as presented below.

- Develop a reliable, accurate, time and cost efficient CFD method to investigate the performance of the FLVV during various driving conditions and investigate LCO. This method should be able to be utilized by VCC for investigating LCO.
- To conduct experimental work and use obtained data to evaluate the performance of the developed CFD method using relevant parameters, both qualitatively and quantitatively.
1.4 Delimitations

- The time span of the project is limited to 20 weeks corresponding to 30 credits per student.
- The fuel considered is gasoline only.
- The cluster at the Fuel department consist of 200 cores.
- Limited time and resources in the laboratory facilities.
- CAD clean up is performed using ANSA. Simulations are performed using STAR-CCM+ Version 13.06.011. Post-processing is done using both STAR-CCM+ and Matlab.
2 Theory

This chapter contains the relevant theory to support the used method in this report. An overview of the fuel system components are presented followed by theory of the governing equations. Some theory about multiphase flows, evaporation and overset mesh are included as well as modeling of solid body motion.

2.1 Fuel System

The fuel system comprises a wide variety of components contributing in different ways. The components of interest in this investigation are mainly the fuel tank and the venting system. The part in the venting system investigated is the FLVV. The FLVV itself contains the floaters which sole purpose is to seal the outlet hole inside the valve. The sealing will further lead to a pressure build and shuts the refuelling pistol off. Outside of the tank the most important part influencing the fuel tank is the carbon canister which function is to take care of the fumes caused by evaporation of the fuel inside the tank.

2.1.1 Fuel tank

The fuel tank in a car has the function to store the fuel and let the fuel pump, which is located inside the tank, provide the engine with the fuel. The fuel tank is usually made of metal or plastic, where most of the Volvo car models of today use a fuel tank made of plastic. The shape and size of the tank are dependent on which type of car model it is used in. For the car models that require a large fuel capacity a saddle design is used for the tank. The tank is saddle shaped to account for the drive shaft to the rear wheels for four wheel drive systems. This divides the tank into two different parts, an active side and a passive side. The active side is the one containing the fuel pump.

2.1.2 Fuel Distribution System

As stated earlier the fuel pump resides inside the active side of the tank. Fuel will continuously be distributed from the passive side to the active side if the level at the active side decreases for example when fuel sloshes from the active to the passive side during driving. This is to always make sure that the fuel pump can deliver fuel to the engine. The level of fuel inside each part of the tank is registered via level sensors, one in each side, and sent to the pump electronic module (PEM). These components together with the fuel pump provide the fuel to the high pressure system.

2.1.3 FLVV

The FLVV is a multi functional valve. The main function of this particular valve is to prevent over-filling. When the fluid inside the tank reaches a certain limit the FLVV shuts close and gives rise to an increase in pressure which subsequently leads back to the capless unit and the filler nozzle stops. The FLVV is also directly connected to the carbon canister (Fig. 1). Due to emission regulation from the United States Environmental Protection Agency the fumes inside the tank are not allowed to be lead back to the filler pipe and later released into the atmosphere. Instead they are redirected to the carbon canister.
via the FLVV. The performance of the canister is deteriorated if LCO occurs thus another important function of the valve is to prevent liquid from finding its way into the canister.

2.1.4 Carbon Canister

The carbon canister is simply put a box containing activated carbon. During refueling the vapors from the gasoline escape through the breather hose connecting the FLVV output to the canister. Inside the canister the vapor is absorbed by the carbon. The canister is also supplied with an outlet where the cleaned air exits. When the engine control unit (ECU) decides that the engine can handle the vapors the purge line is opened which induce a suction of air from the outlet. The air coming in releases the hydrocarbons in the canister and gets sucked through the purge line to the engine for combustion.

![Diagram of canister flow during refueling and purging](image)

**Figure 3: Schematic figure of the flow through the canister when refueling and when purging**

2.1.5 Purging

The purging phenomena is simply explained the cleansing of the carbon canister from the absorbed hydrocarbons. This is done by passing clean atmospheric air flow through the canister. The hydrocarbons are transported with the fresh air to the engine for combustion and the rate of that transportation is controlled by the ECU and a Pulse Width Modulated (PWM) valve (Purge valve). The purging is controlled using PWM and the flow through the purge valve is controlled by a Duty Cycle (DC) of a control signal with a frequency of 10 Hz according to design guidelines at VCC. Fig. 4 shows the purge flow depending on the DC of the purge valve. The behaviour is such that the higher the purge flow required the higher the DC.
Figure 4: The idealized DC with change in the flow through the canister during purging. Purge flow starts at 4 to 6% duty cycle depending on the conditions and tank age.
2.2 Fluid Dynamics

The behaviour of a continuum fluid flow is governed by the three conservation laws of physics. The conservation of mass, Newton’s second law and the first law of thermodynamics. The conservation of mass states that no mass can be created nor destroyed. Newton’s second law can be described as that the momentum of a fluid particle is conserved and the first law of thermodynamics relates to the conservation of energy, that no energy can be destroyed or created. The study of the motion of fluid can be done in two different ways called Lagrangian and Eulerian. The Lagrangian approach follow a fluid particle as it moves and in the Eulerian approach observe the particle pass by a fixed region in space. The Eulerian approach is the most common when studying the motion of fluids.

2.2.1 Fluid Properties

As the fluid is considered a continuum, the motion and action at a molecular level is to be neglected. This implies that the fluid behaviour can be expressed by properties like density, pressure, temperature and velocity. These fluid properties are all time and space dependent. A fluid particle can then be defined as an element where these properties mentioned are not affected by individual molecules.

2.2.2 Governing Equations

By establishing a mass balance for a fluid element the equation for mass conservation can be derived. The resulting equation is the continuity equation (Eq. 1). The three terms on the left hand side of the continuity equation for a Newtonian incompressible fluid represents the net flow of mass out of the fluid element.

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{1}
\]

The derivation of the momentum equations are done using Newton’s second law which employed on a fluid element means that the rate of change in momentum for the fluid element is equal to the sum of forces acting on the element. The fluid element can be subtracted to forces of two different kinds, body forces and surface forces. The body forces act on the volumetric mass of the element such as gravity or electromagnetic force while the surface forces act on the surfaces of the element like pressure and shear forces. The left hand side of the momentum equations represents the rate change of momentum and the right hand side represent the total force, i.e. due to surface forces and body forces (Eq. 2).

\[
\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\tau_{ij}) + S_{Mx} \tag{2}
\]

In the momentum equation the unknown viscous stress components appear, \(\tau_{ij}\). For a Newtonian fluid the viscous stresses are proportional to the time rate of strain (Eq. 3). By introducing these into the momentum equations (Eq. 2), the Navier-Stokes equations are found (Eq. 4). This is a transport equation of, in this case, the momentum. The first
The term on the left hand side is the rate of change of the velocity and the second term is the convective term. On the right hand side the first term is the pressure gradient, the second is the diffusive term and the last term is the source term. The effects of gravity is be one such source term.

\[
\tau_{xx} = 2\mu \frac{\partial u}{\partial x} + \lambda div u \quad \tau_{xy} = \tau_{yx} = \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)
\]

(3)

\[
\frac{\partial \rho u_i}{\partial t} + div (\rho u_i u_j) = - \frac{\partial p}{\partial x_i} + div (\mu \nabla u_i) + S_{Mx}
\]

(4)

The flow problem considered in this work will not take into account any effects of compressibility but the effects of heat transfer will be considered due to the evaporation of the fuel in some cases. For those cases the energy equation needs to be solved along side the other equations (Eq. 5).

\[
\rho \frac{DE}{Dt} = - div (\rho u_i) + div (k \grad T) + S_E + \frac{\partial}{\partial x_j}(u_i \tau_{ij})
\]

(5)

Where the left hand side is the rate of change of energy of the particles and the rest of the term on the right side is the sum of the rate of work done on that particle together with the net rate of heat addition to the fluid and \( S_E \) is the source term. \( E \) is the specific energy of a fluid (Eq. 6)

\[
E = i + \frac{1}{2} \left( u^2 + v^2 + w^2 \right)
\]

(6)

where \( i \) is the internal energy and the rest is kinetic energy.

## 2.3 Turbulence Modelling

Many flow related engineering problems are of turbulent nature. Turbulence can be described as chaotic and random behaviour of the flow motion with a large range of length and time scales. Because of the unsteady nature can the flow properties be decomposed into a mean value component and a fluctuating component, the so called Reynolds decomposition (Eq. 7).

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

(7)

The fluctuating component introduces additional shear stress due to the vortical motions present in turbulent flow that cause an exchange in momentum between fluid layers. These additional shear stresses are, in a time-averaged sense, called Reynolds stresses and appear in the Navier-Stokes equations when expressing them with Reynolds decomposition (Eq. 8).
\[
\frac{\partial \bar{u}}{\partial t} + \text{div}(\bar{u} \mathbf{u}) = - \frac{1}{\rho} \frac{\partial p}{\partial x} + \text{div}(\nu \text{grad} \bar{u}) + \frac{1}{\rho} \left[ \frac{\partial - \rho \bar{u}'^2}{\partial x} + \frac{\partial - \rho \bar{u}'v'}{\partial y} + \frac{\partial - \rho \bar{u}'w'}{\partial z} \right] + S_{\text{Mx}} \tag{8}
\]

2.3.1 Unsteady Reynolds Averaged Navier-Stokes Models

When writing the Navier-Stokes equations with Reynolds decomposition they are called the Reynolds-averaged Navier-Stokes (RANS) equations. The RANS equations are good when information of the mean-flow are of interest and details about the turbulent fluctuations are not needed. For many engineering problems information about the mean-flow are enough which has made models based on RANS very common because of the less computational cost compared to other types of calculation procedures for turbulent flow.

When solving unsteady flow cases with the RANS equations they are called URANS (Unsteady RANS) and the difference from solving the steady equations are that the transient term is kept. As the equations for an unsteady case are time dependent the Reynolds decomposition can be written with three terms (Eq. 9). Where \( \langle \bar{\phi} \rangle \) is the time-averaged velocity, \( \phi' \) the modelled fluctuations and \( \bar{\phi}'' \) is the resolved fluctuations. The idea with this is to have a scale separation, i.e. an assumption that the modelled turbulence should have a much smaller time scale than the resolved turbulence. Flow problems that have this large scale separation is though rather uncommon. \[5\]

\[
\phi(x,t) = \langle \bar{\phi} \rangle + \phi' + \bar{\phi}'' \tag{9}
\]

The appearance of the turbulent shear stresses in RANS equations introduce six additional unknowns into the momentum equations. To be able to solve the RANS equations these unknowns need to be predicted. This is done by a turbulence model. The most commonly used turbulence models are based on the Boussinesq hypothesis. The Boussinesq hypothesis propose that the Reynolds stresses are proportional to the mean rates of deformation (Eq. 10). \[6\]

\[
- \rho \bar{u}' \bar{w}' = \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} , \quad k = \frac{1}{2} \left( \bar{u}'^2 + \bar{v}'^2 + \bar{w}'^2 \right) \tag{10}
\]

\( k \) represents the turbulent kinetic energy and \( \mu_t \) is the introduced term called eddy viscosity, \( \mu_t \). The \( \delta_{ij} \) is the Kronecker delta that is equal to zero if \( i = j \) and equal to one otherwise. In order to solve this, the eddy viscosity needs to be estimated. The turbulence model used in this work to estimate the eddy viscosity is the relizable \( k - \varepsilon \) (RKE).

**Realizable \( k - \varepsilon \) model**

The standard \( k - \varepsilon \) turbulence model is a so called two equation model which means that it adds two additional equations beyond the mean flow momentum equations to close the system of equations. The two additional equations are transport equations for the turbulent kinetic energy, \( k \) and for the rate of dissipation of turbulent kinetic energy, \( \varepsilon \). Starting with the kinetic energy, it can be decomposed as a mean term, \( K \), and a turbulent term, \( k \) (Eq. 11). \[6\]
\[ k(t) = K + k, \quad K = \frac{1}{2}(U^2 + V^2 + W^2) \quad k = \frac{1}{2}(\bar{u}^2 + \bar{v}^2 + \bar{w}^2) \] (11)

The derivation of the turbulent kinetic energy is then done by multiplication of the respective velocity fluctuation component to the corresponding Navier-Stokes equation (Eq. 12-14).

\[ u' \left( \frac{\partial u}{\partial t} + \text{div}(u \mathbf{u}) = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \text{div}(\nu \text{ grad } u) \right) \] (12)

\[ v' \left( \frac{\partial v}{\partial t} + \text{div}(v \mathbf{u}) = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \text{div}(\nu \text{ grad } v) \right) \] (13)

\[ w' \left( \frac{\partial w}{\partial t} + \text{div}(w \mathbf{u}) = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \text{div}(\nu \text{ grad } w) \right) \] (14)

The three equations can then be added together and by doing the same thing to the RANS equations these two resulting equations can be subtracted. After subtraction of the two and some rearrangement the exact equation for the turbulent kinetic energy is obtained (Eq. 15).

\[ \frac{\partial (\rho k)}{\partial t} + \text{div}(\rho k \mathbf{u}) = \text{div}(\nabla p \mathbf{u} + 2\mu \mathbf{s} - \frac{1}{2} \mathbf{w} \cdot \mathbf{w}) - 2\mu \mathbf{s} \cdot \mathbf{S} \] (15)

By defining the rate of dissipation of turbulent kinetic energy as \( \varepsilon = 2\nu s_{ij} \cdot s_{ij} \) the large velocity and length scales can be defined by the turbulent kinetic energy, \( k \), and the dissipation of turbulent kinetic energy, \( \varepsilon \), as shown below. With this the eddy viscosity can be defined (Eq. 16).

\[ \vartheta = k^{1/2} \quad \ell = \frac{k^{3/2}}{\varepsilon} \]

\[ \mu_t = C \rho \vartheta \ell = \rho C' \frac{k^2}{\varepsilon} \] (16)

The RKE model uses another formulation for the eddy viscosity based on the constraints that the normal Reynolds stresses are by definition positive and Schwarz inequality for turbulent shear stresses. These constraints have their founding in the fact that the Reynolds Stress Tensor is positive semidefinite. In the standard formulation of the eddy viscosity the normal stresses can become negative in cases of large mean strain rate, which is not strictly positive semidefinite, and the Schwarz inequality can be violated. The realizable formulation makes the coefficient \( C_\mu \) a variable instead of constant and relates it to the mean strain rate. The modeled transport equation for \( k \) can then be written (Eq. 17 or 18).
\[
\frac{\partial (\rho k)}{\partial t} + \text{div} (\rho k \mathbf{u}) = \text{div} \left[ \frac{\mu_t}{\sigma_k} \text{grad} \varepsilon \right] + 2 \mu S_{ij} \cdot S_{ij} - \rho \varepsilon \tag{17}
\]

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial}{\partial x_j} (\rho k u_j) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu + \mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \rho \varepsilon + S_k \tag{18}
\]

The term \( P_k \) represents the generation of turbulence of kinetic energy. \( S_k \) represents the user-specified source term. The transport equation for \( \varepsilon \) in the RKE model is based on the dynamic equation of the mean-square vorticity fluctuation (Eq. 19) [7]. \( P_\varepsilon \) is the production term and \( S_\varepsilon \) the source term. The transport equations for \( k \) and \( \varepsilon \) have constants with values chosen to fit a wide range of flows. The preset constants in STAR-CCM+ are shown in Tab. 1.

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} (\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu + \mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \rho C_1 S_\varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} P_\varepsilon + S_\varepsilon \tag{19}
\]

<table>
<thead>
<tr>
<th>Constants</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_\mu )</td>
<td>0.09</td>
</tr>
<tr>
<td>( C_{1\varepsilon} )</td>
<td>1.44</td>
</tr>
<tr>
<td>( C_{2\varepsilon} )</td>
<td>1.9</td>
</tr>
<tr>
<td>( \sigma_k )</td>
<td>1.0</td>
</tr>
<tr>
<td>( \sigma_\varepsilon )</td>
<td>1.2</td>
</tr>
</tbody>
</table>

### 2.4 Multiphase Flows

The flow inside a fluid domain with more than one fluid interacting with each other is referred to as a multiphase flow. In these types of flows the region between the fluids, the interface, is well defined due to the difference in chemical and/or physical structures of the fluids. Problems involving multiphase flows are modelled with mixing at a macroscopic level with different convection velocity. These types of flows can be categorized as below, however the flow could be a mixture of the two as well [8].

- Dispersed flows, such as bubble of oil inside a water domain as visualized on top in figure 5.
- Stratified flows, such as free surface flows as seen on bottom in figure 5.
There are currently two different major ways of numerical modelling of two phase flows, the Eulerian-Eulerian approach and the Eulerian-Lagrangian approach.

**Eulerian-Eulerian**

In the Eulerian-Eulerian approach the modelled phases each have their own set of conservation equations. The phases coexist and are mixed at length scales smaller than length scales to be resolved. This concept is a so called interpenetrating continua based on the assumption that one is interested in the time-averaged flow behaviour. This is also based on volume fraction of the phases which is described more thoroughly in coming chapters. The conservation equations are solved for each phase in the domain.

The Eulerian-Eulerian has the advantages of being efficient in obtaining the mean values of velocity, pressure etc, turbulence is included and calculated at a low extra cost and can cover full range of the volume fraction [9][8]. The main disadvantage are higher cost if many different fluid particle sizes are present.

**Eulerian-Lagrangian**

This approach solves the governing equations for the continuous phase but the dispersed phase is solved by tracking each particle (droplet) in a Lagrangian approach. These two different phases, continuous and dispersed, share their momentum, mass and energy if needed. The advantages of this approach is that each scattered droplet has its own exact information of velocity, space, temperature etc. This approach is suitable for cases involving particle-particle collision, particle size distribution, transfer of different quantities between the particles and the fluid [9][8]. The disadvantages are the limitation of the coverage of the range of volume fraction and computational heavy for problems involving many particles.

**Multiphase Models**

There are several different models used to simulate the dispersed and stratified flow based on the Eulerian-Eulerian and Eulerian-Lagrangian approaches such as:
Since the problem at hand involves sloshing inside a tank and also the need for capturing droplets leading to LCO the model most suitable to use is the VOF multiphase model as stated in the STAR CCM+User Guide and previous studies conducted in this field. The Eulerian Multiphase model could also be used but since the phases never separate and also the higher computational cost it is not advantages to utilize this model. A Langrangian model is not suited for problems involving a high rate of deformation of the free boundaries in three dimensions, such as sloshing inside a tank. The VOF model allows for tracking of the free surfaces which employs additional transport equation, conservation of mass and conservation of momentum.

2.4.1 VOF Multiphase Model

The VOF multiphase model is mostly used to simulate flows involving two or more immiscible fluids. It is a simple multiphase model capable of resolving the interface between the phases of the fluids. The model can handle mixtures such as dispersed flows and also free surfaces which is the flow behaviour inside a tank during sloshing. One of the sources of discretization error using the VOF multiphase model is the assumption of shared quantities between the phases such as velocity, temperature and pressure and also that the interfaces between the phases are resolved. In order to be able to resolve the dispersed flow inside the tank each droplet needs to be covered by more than two element (Fig. 6) thus the smaller droplets inside the tank the more computational effort needed to reduce modeling error.

![Grid density for a VOF model involving dispersed flow](image)

**Figure 6: Grid density for a VOF model involving dispersed flow. Upper case is an example of not a suitable grid and lower case is an example of a suitable one**

As the method used in this model captures the interface and predicts the movement and distribution of the phases, the distribution and spatial position of the interface are described by the phase volume fraction $\alpha_i$ in a cell (Eq. 20)
Each phase is denoted \(i\) and \(V_i\) is the volume of phase \(i\) in the element and \(V\) is the volume of that element thus the sum of volume fraction of all phases present in the flow must be equal to one (Eq. 21) and \(N\) being the number of phases.

\[
\sum_{i=1}^{N} \alpha_i = 1
\]  

(21)

The volume fraction \(\alpha_i\) ranges between zero and one, zero meaning no presence of phase \(i\), one being completely filled with phase \(i\) and between these values means the existence of interface between the phases. The presence of interfaces in a cell affects the material properties in the cell and the fluid is treated as a mixture inside. Material properties density \((\rho_i)\), dynamic viscosity\((\mu_i)\) and specific heat\((\left(C_p\right)_i)\) are calculated (Eq. 22, 23 and 24) with \(n\) being the amount of fluids present in the cell.

\[
\rho = \sum_{i=1}^{n} \rho_i \alpha_i
\]  

(22)

\[
\mu = \sum_{i=1}^{n} \mu_i \alpha_i
\]  

(23)

\[
C_p = \sum_{i=1}^{n} \frac{(C_p)_i \rho_i}{\rho} \alpha_i
\]  

(24)

The driving force for distribution of phase \(i\) is the mass conservation equation (Eq. 25).

\[
\frac{\partial}{\partial t} \int_V \alpha_i dV + \oint_A \alpha_i v \cdot da = \int_V \left( S_{\alpha_i} - \frac{\alpha_i D\rho_i}{\rho_i} \right) dV - \int_V \frac{1}{\rho_i} \nabla \cdot (\alpha_i \rho_i v_{d,i}) dV
\]  

(25)

where \(v\) is the mass-averaged velocity of the mixture, \(a\) is the surface area vector, \(v_{d,i}\) is the diffusion velocity, \(S_{\alpha_i}\) is the source term and \(\frac{D\rho_i}{Dt}\) is the Lagrangian derivative of the phase densities.

The term for the diffusion velocity is present due to slip between the phases. The slip between the phases is used to model the effects due to the phases in the interface moving at different velocities. As stated earlier the VOF model assumes that the interface between phases is resolved. If this is not fulfilled, which could be due to inappropriate time or grid size, the model smears the interface between the phases.
where all phases have the same velocity. In that case the included slip between phases improves the modelling assumptions and gives a sharper interface.

To reduce numerical diffusion a sharpening factor can be added to the solver ranging between 0.0 and 1.0 where a higher number gives a lower numerical diffusion and sharper interfaces. If the sharpening factor is added an extra term is added to the transport equation above (Eq. 26).

\[
\nabla \cdot (v_{ci} \alpha_i (1 - \alpha_i))
\]

where \(v_{ci}\) is defined as:

\[
v_{ci} = C_\alpha \times |v| \frac{\nabla \alpha_i}{|\nabla \alpha_i|}
\]

and \(C_\alpha\) is the sharpening factor. The mass conservation equation for all the phases is then defined (Eq. 28).

\[
\frac{\partial}{\partial t} \left( \int_V \rho dV \right) + \oint_A \rho v \cdot da = \int_V S dV
\]

where \(S\) is the mass source term which is related to \(S_{\alpha_i}\), which can be found in the transport equation, and defined as

\[
S = \sum_i^n S_{\alpha_i} \cdot \rho_i
\]

Eq. 29 is also dependent on the volume fraction of the phases embedded in \(\rho_i\) (Eq. 22). The momentum equation is defined as

\[
\frac{\partial}{\partial t} \left( \int_V \rho v dV \right) + \oint_A \rho v \otimes v d = -\oint_A \rho I \cdot da + \oint_A T \cdot da + \int_V \rho g dV + \int_V f_B dV - \sum_i^n \oint_A \alpha_i \rho_i v_{di} \otimes v_{di} \cdot da
\]

where \(p\) is the pressure, \(I\) is the unity tensor, \(T\) is the stress tensor and \(f_B\) is the vector containing the body forces. Eq. 28 and 30 are used to resolve the shape of the interface between the phases.

**Surface Tension**

Sloshing inside a tank involves immiscible fluids as stated earlier thus the modelling of surface tension is an important feature in the VOF model. Immiscibility of fluids is due to the properties and nature of the fluids mainly affected by the molecular cohesion forces. The surface tension can be expressed with a coefficient \(\sigma\) which is the amount of work required to create a unit area of free surface.

The surface tension contact angle also needs to be considered for accurate modelling of the physics. The contact angle \(\theta\) is the angle at the triple line meaning the line where the solid surface and the two fluids are in contact (Fig. 7). The angle specifies
the wettability of a fluid where a lower angle corresponds to better wetting. The basic relation between the contact angle and surface tension is based on equation \(31\), which is the Young’s equation \[10\].

\[
\sigma_{sv} = \sigma_{sl} + \sigma_{lv} \cos \theta 
\]

Where \(\sigma_{sv}\), \(\sigma_{lv}\), and \(\sigma_{sl}\) is are the interfacial tensions of the solid-vapor, liquid-vapor and solid-liquid respectively.

Figure 7: The contact angle \(\theta\) between the solid surface, the droplet and surrounding vapor at the triple line

### 2.5 Evaporation

Inside the fuel tank the gasoline continuously transform from liquid to its gaseous state below its boiling point. This process is called evaporation and the rate at which it occurs depends on the temperature difference between the surface of the gasoline and the its surrounding atmosphere. When the thermal motion of the molecules on the surface of the gasoline exceeds the work required to break the cohesion forces between the molecules of the fluid evaporation occurs. Thus the higher rate of evaporation at higher temperature. The rate of evaporation also depends on the pressure inside the tank. Since a fluids boiling temperature depends on the ambient pressure the evaporation rate is also affected by a change in ambient pressure. When the molecules escaping from the surface of the fuel are equal to the ones returning to the fuel inside the enclosed tank the vapor inside the tank will be saturated. The pressure of the vapor inside the tank will be the saturated vapor pressure. Thus at higher temperature when more evaporation occurs the saturated vapor pressure will also be higher. The boiling temperature is identified as the temperature at which the vapor pressure is equal to the ambient pressure. The continuous evaporation will subsequently lead to a colder gasoline temperature since the energy in the surface is reduced when the molecules with high thermal motion leave the fuel.

#### 2.5.1 Modelling of Evaporation

The evaporation taking place at the interfaces between the gas and liquid phase is hydrodynamically limited as treated by the solver. Hydrodynamical limitation means that the phases are at an equilibrium state at the interface and the driving force for evaporation is the diffusion. The phase equilibrium is described by Raoult’s law which states that the partial vapor pressure of one of the components in an ideal mixture is equal to the vapor pressure of that component multiplied by its mole fraction. Eq. \[32\] describes the rate of evaporation of any given component \(i\).
\[ \dot{m}_i = - \rho_g D_{g,i} \frac{\partial Y_{g,i}}{\partial n} \bigg|_{s} \left( 1 - \sum_{j=1}^{N_v} Y_{g,j}^s \right) \]  

(32)

Where \( \rho_g \) and \( D_{g,i} \) are the density and diffusion coefficient of the gas phase. \( N_v \) is the number of components in an evaporate state, \( Y_{g,i}^s \) is the component mass fraction at the free surface and \( n \) is the surface normal coordinate. The evaporation rate for each cell is then approximated (Eq. 33).

\[ \dot{M}_{l,c} \approx - \rho_g D_{g,i} \nabla Y_{g,m} \nabla \alpha_l V_c \left( 1 - \sum_{j=1}^{N_v} Y_{g,j}^s \right) \]  

(33)

where \( \alpha_l \) is the liquid volume fraction and \( \nabla \alpha_l \) is the gradient of the volume fraction over a volume expressing the area of the interface that volume contains. \( \nabla Y_{g,m} \) expresses the effect of \( \nabla Y_{g,i} \) from Eq. 32.

The accuracy of the evaporation rate is thus dependent on how well the position of the interface is determined compared to the boundary layer above the interface. Thus for a more accurate solution the solver requires a finer mesh density to model evaporation.

### 2.5.2 Antoine’s Equation

The equilibrium of pressure, temperature and chemical potential between the vapor and its liquid can be described by the Clapeyron equation (Eq. 34) [11].

\[ \frac{d(\ln P_{vp})}{d(1/T)} = \frac{\Delta H_v}{R \Delta Z_v} \]  

(34)

\( \Delta H_v \) and \( \Delta Z_v \) are the differences in enthalpy and compressibility factors of the saturated vapor and liquid. \( P_{vp} \) is the vapor pressure and \( T \) is the phase temperature. By integrating this equation and assuming that \( \Delta H_v / \Delta Z_v \) and some further manipulation one can obtain the Antoine Equation (35)[11].

\[ \log_{10} P_{vp} = A - \frac{B}{T + C - 273.15} \]  

(35)

The constants A, B and C are obtained using extensive experimental and they are dependent on the material.

### 2.6 Overset Meshing

Overset meshes are a convenient and useful way of dealing with moving bodies in a fluid domain. Overset mesh is basically what the name suggests, multiple meshes overlapping each other used to discretize a domain. A domain with an overset mesh consists of a background region and the overset region. The background region is the entire solution domain and the overset region surrounds the moving bodies in the
domain in enclosed regions. Once these regions have been meshed there will be an overlapping between the cells in those regions. These two regions will then be used to create an overset interface between them. The overlapping cells will change based on the movement of the body inside the overset region and information transfer will occur in through the overlapping cells [8].

The coupling of the overset region and background region takes places within the overset interface. This results in the hole-cutting process where a hole is cut in the background mesh where the overset mesh lies. This hole cutting process will either follow a layered approach or a global approach [8].

- In the layered approach the donor cells (Fig. 8) reside as a layer around the overset boundary. The cells in the background region next to these donor cells become the acceptor cells (Fig. 8). The rest of the overset region that completely covers the background region results in the cells in the background to become inactive cells. In these cells no equations are being solved [8].

- In the global approach the solver recognizes if the each cell centroid in the background region resides inside or outside the overset region and if the centroid is inside the cell in the background becomes inactive [8].

The background and overset mesh are implicitly coupled meaning the solution is calculated for all the active cell simultaneously. In order for the solution to propagate every acceptor cell must have donor cells (\(N_1 - N_6\)) which are covered by the interpolation elements (green) elements, seen in Figure 8. For every active cell (C) (Fig. 8) an acceptor cell in dotted line is present nearby which provides necessary information for calculation of cell center values in the active cell and also the face fluxes between it and the acceptor cell. Multiple donor cells in the other mesh are used to calculate the contribution of the acceptor cell. The chosen donor cells depend on interpolation option, which are Distance-weighted interpolation, Linear interpolation and Least squares interpolation, and also the amount of active cells in the donor region in the vicinity of the acceptor cell [8] [12].

Figure 8: Overset mesh in blue overlapping background mesh in black. The dotted lines show acceptor cells which provide necessary information for calculation of cell center value at the active cell and the fluxes. C are the active cell ansd \(N_1 - N_6\) are the donor cells and the green cells are the interpolation elements.
The overset mesh option also includes a special feature used in vicinities where a zero gap configuration might occur meaning when a background region makes contact with the overset region called overset mesh zero gap interface. When the regions come together the algorithm automatically deactivates the cells in the gap if the distance between them is less than two cell layers\[8\].

2.7 Near-Wall-Treatment

The flow close to a solid boundary will be more affected by viscous effects in comparison to inertial forces the closer you get to the wall. The prediction of the flow in this near-wall region will be dependent on the mesh resolution near the wall. The $y^+$ parameter is a dimensionless wall distance that determines the mesh resolution in the near-wall region. The inner near-wall region is typically divided into three type of layers. The most inner layer, closest to the wall, called the viscous sub-layer are dominated by viscous effects and is defined for $y^+ < 5$. The next layer are where the viscous and turbulent effects are about the same, this layer are called the buffer layer. The last layer is called the log-law layer where the turbulent effects dominate and is defined for $30 < y^+ < 500$. [6]

The realizable $k - \varepsilon$ model has a variant in STAR-CCM+ called the Realizable Two-Layer $k - \varepsilon$ model. The Two-Layer variant allows the model to be used in the near-wall region, i.e. in the viscous affected region. The standard $k - \varepsilon$ model solves the transport equations for regions far away from the wall and applies wall functions to model the flow in the near-wall region, the high-$y^+$ wall treatment in STAR-CCM+ is then intended to be used. This implies that a mesh with $y^+ > 30$ is to be used as it assumes that the first cell lies in the log-law region (Fig. 9). For the realizable two-layer $k - \varepsilon$ model the transport equation for $k$ are solved all the way down to the wall, $\varepsilon$ and $\mu_t$ are determined near the wall by functions of wall distance. The two layer variant model works with either a fine near wall mesh, $y^+ \sim 1$, or a coarser mesh, $y^+ > 30$. This implies that the all-$y^+$ wall treatment can be used resulting in that if the mesh satisfy $y^+ \sim 1$ it resolves the viscous sub-layer, if $y^+ > 30$ it uses wall functions and if the first cell lies in the buffer layer it uses a blending function to determine the turbulence quantities. [8]

![Figure 9: An illustration of high-$y^+$ wall treatment to the left and low-$y^+$ wall treatment to the right](image_url)
2.8 Modelling of Solid Body Motion

The movement of a rigid body, due to buoyancy effect for example, inside a fluid domain can be modelled using the Dynamic Fluid Body Interaction (DFBI) module by calculating the forces and moments acting on the body. The change in position of the rigid body is calculated by solving the governing equations of motion inside the domain. This module is capable of calculating the fluid forces, moments and gravitational forces on a body with 6 Degrees of Freedom (DOF) which includes rotations about x, y and z axis and also the translation in x, y and z direction. The motion can also be limited to one-DOF translating motion which moves the body along a specified axis of direction.

The motion of the body in a one-DOF simulation can be limited using different approaches. The least computational heavy and complex is the motion limiter. The rigid body is constrained via specification of the maximum and minimum displacement. When the moving body reaches those limits the kinetic energy carried by the body is destroyed which could lead to the occurrence of nonphysical phenomena. To counteract this effect a damped motion limiter can be used which slowly retards the movement of the body. The specification of damped motion allows for use of a damping length which is based on the time step size. The damping length must be calculated and set allowing the solution to propagate several time step over the that length. This length should be set to at least allow the body to slow down in the same range as the prism layer total thickness size [8].

2.9 Courant-Friedrichs-Lewy Condition

For unsteady flow simulations the time-step size needs to be chosen wisely to ensure numerical stability and reliable results. To be able to choose a suitable time-step can the Courant-Friedrichs-Lewy (CFL) condition be used. The reasoning behind the CFL condition is that the distance of which a fluid particle moves should not be greater than the local cell size. The condition is defined as Eq. 36 for a three dimensional flow problem where the sum is over the three different planes x, y and z. $u_x$, is the velocity magnitude in each direction, $\Delta t$ is the time step and $\Delta x_i$ is the element length in each direction.

$$C = \Delta t \sum_{i=1}^{3} \frac{u_x}{\Delta x_i} \leq C_{Max} \tag{36}$$

The $C_{Max}$ is the CFL number that needs to be met. A $C_{Max}$ of unity implies that a fluid particle move the distance of one cell size each time-step. For an implicit solver the CFL number can be larger than unity in areas of low action without risking numerical stability. When dealing with multiphase flow problems the free surface needs extra attention regarding the CFL number and it is recommended that the CFL number is below 0.7 at the interface when using first order temporal discretization [8]. Temporal discretization are further explained in section 2.10.4. In URANS simulations the time scale for the resolved larger scales is “separated” from the part of the domain which is modeled turbulence which is called scale separation.
as described in section [2.3.1]. The turbulent time scales are much smaller than the
time step required for URANS simulations and are handled by the turbulence model,
and the time step is then dictated by the phenomenas occurring at higher pace.

### 2.10 Numerical Methods

In STAR-CCM+ several different numerical solver and discretization schemes exist.
In this section the schemes used in this work is explained as well as the different
mesh types and quality measurements.

#### 2.10.1 Segregated solver

The segregated solver scheme solves the equations of momentum and continuity
separately after each other in iterations until convergence. It uses a pressure-
correction equation to obtain the pressure in order to fulfill the continuity equation
for the predicted velocity field. This is done using a collocated variable arrangement
and a Rhie-and-Chow-type pressure-velocity coupling together with a Semi-Implicit
Method for Pressure Linked Equations (SIMPLE) algorithm [3]. A collocated vari-
able arrangement means that all the variables such as velocity, pressure, density etc
are stored in the same same place instead of being in either the cell center of the
control volume or at the cell faces i.e staggered grids.
2.10.2 Normalized Variable Diagram (NVD)

The Normalized Variable Diagram (NVD) is based on convective boundedness criterion. The definition of this criterion is that the distribution of the scalar variable $\Phi_f$ between the centers of the neighbouring control volumes, B and C, should be kept smooth (Fig. 10). The normalized variable are then introduced (Eq. 37 and 38).}

$$\tilde{\Phi}_f = \frac{\Phi_f - \Phi_A}{\Phi_C - \Phi_A} \quad (37)$$

$$\tilde{\Phi}_B = \frac{\Phi_B - \Phi_A}{\Phi_C - \Phi_A} \quad (38)$$

As seen in figure 10 the relation $\Phi_B < \Phi_f < \Phi_C$ holds and using equation 37 and 38 one obtains the boundedness criterion in normalized variables, $\tilde{\Phi}_B < \tilde{\Phi}_f < 1$. The boundedness criterion [13] is satisfied in the gray region (Fig. 11) by any scheme. The volume fraction of $\Phi_f$ can be calculated using Eq. 37 and 38 together with Eq. 39 and 40 below. The volume fraction $\Phi_f$ is then used to calculate the volume fraction flux in the discretized form, using Crank Nicholson method, of the equation for transport of volume fraction described earlier [13].

$$\Phi_f = \left(1 - \tilde{\beta}_f \right) \Phi_B + \tilde{\beta}_f \Phi_C \quad (39)$$

$$\tilde{\beta}_f = \frac{\Phi_f - \Phi_B}{1 - \Phi_B} \quad (40)$$
Figure 11: The region where the convective boundedness criterion is satisfied by all differencing schemes, \( C_f \) is the local courant number at the face of the control volume. UD is Upwind Differencing and DD is Downwind Differencing.

2.10.3 High Resolution Interface Capturing (HRIC)

The HRIC scheme is an interface capturing scheme used in the VOF method which implements a blending between the Bounded Downwind and Upwind Differencing scheme. This scheme is well suited for the numerical modelling of sharp interfaces which is an important criteria for simulation of immiscible fluids. HRIC scheme is based on the NVD described previously and is second order accurate. The upwind assumes that the values at the cell faces are equal to the values of the upstream node of each face when calculating the cell center values taking the flow direction into account. But the downwind uses the values at the downstream nodes. The order of accuracy refers to the truncation errors in the approximation of the gradients from the Taylor series expansion. The normalized face values are estimated according to:

\[
\tilde{\Phi}_f = \begin{cases} 
\tilde{\Phi}_B & \text{if } \tilde{\Phi}_B < 0 \\
2\tilde{\Phi}_B & \text{if } 0 \leq \tilde{\Phi}_B < 0.5 \\
1 & \text{if } 0.5 \leq \tilde{\Phi}_B < 1 \\
\tilde{\Phi}_B & \text{if } 1 \leq \tilde{\Phi}_B 
\end{cases}
\]

The value of \( \tilde{\Phi}_f \) is then corrected (Eq.41) which is the local Courant number.

\[
C_0 = \frac{a_f v_f}{V_B} \delta t 
\]

Where \( v \) is velocity, \( a_f \) is the cell-face surface vector, \( V_B \) is velocity at \( B \) and \( \delta t \) is the time step. This correction takes the availability criterion into account which states that the fluid convected through the cell face cannot be larger than the amount at hand in the donor cell. Depending on the \( C_0 \) value the correction is made according...
to following[8]:

\[
\tilde{\Phi}^* = \begin{cases} 
\tilde{\Phi}_f & \text{if } C_0 < C_{ol} \\
\tilde{\Phi}_B + (\tilde{\Phi}_f - \tilde{\Phi}_B) \frac{C_{o_u} - C_0}{C_{o_u} - C_{ol}} & \text{if } C_{ol} \leq C_0 < C_{o_u} \\
\tilde{\Phi}_B & \text{if } C_{o_u} \leq C_0
\end{cases}
\]

$C_{ol}$ and $C_{o_u}$ are the lower and upper local Courant number. The limits above show the change in schemes depending on the Courant number. If $C_0$ is lower than the lower limit the HRIC scheme is implemented, if between upper and lower limit a blending of HRIC and First Order Upwind (FOU), and if higher than the upper limit a pure FOU is used. The main reason for the shifting between the schemes is to ensure a robust and stable solver where lower specified values of $C_{ol}$ and $C_{o_u}$ promote sooner activation of FOU but this consequently leads to a smeared interface. This blending is of high importance in cases where the time step is too large to resolve an interface with large shape variation in time[8].

One last correction is introduced to $\Phi_f$ to account for cases where the flow is parallel to the interface, where the downwind differencing scheme is not suitable and leads to a non sharp interface (Eq. 42).

\[
\tilde{\Phi}^*_{ji} = \tilde{\Phi}^*_{ji} (\cos \Theta)^{C_0} + \tilde{\Phi}_B \left(1 - (\cos \Theta)^{C_0}\right)
\]

Where the angle $\Theta$ is the angle between the interface normal and the cell face surface vector $a_f$. $C_0$ is the angle factor which is solver specific used to control the blending and sharpens the free surface.

Finally the value of the cell face $\Phi_f$, introduced previously, is calculated using the corrected value from (Eq. 42) and the neighbouring control volumes (Fig. 10) according to equation 43[8].

\[
\alpha_f = \tilde{\Phi}^*_{ji} (\alpha_C - \alpha_A) + \alpha_A
\]

2.10.4 Temporal Discretization

For unsteady simulations the transient term needs to be discretized and so the physical simulation time is divided into time-steps. The temporal discretization can either be of first order or second order in STAR-CCM+ and uses a Euler backward differentiation formula[8]. The first order use the solution at the current and previous time-step to approximate the next while the second order scheme use the current and the previous two time-steps. The order of temporal discretization scheme will affect the time-step size needed. The second order scheme is known for good prediction of wave propagation compared to the first order scheme. The second order scheme however imposes a limit for the CFL number where the maximum allowed at the free surface is 0.5. If the CFL number grow to large there is a risk for the mass to not be conserved or that the solution diverge. The second order scheme can be further improved using a high-accuracy temporal discretization if the flow physics
does not suffer from stability issues. There are two different high-accuracy temporal discretization, the 4th and 5th optimized second-order. The 4th uses the solutions from 4 time-steps and 5th uses solution from 5 time-steps.

### 2.10.5 Mesh Types and Quality

In STAR-CCM+ there are three different types of volume mesh models which all have their strengths and weaknesses. The three types include the polyhedral, tetrahedral and trimmed mesher models (Fig. 12). The choice of mesh type will affect for instance the computational cost, solution accuracy and convergence rate.

![Trimmed Cell](image)

![Tetrahedral](image)

![Polyhedral](image)

**Figure 12:** The three different volume mesh types available in STAR-CCM+. The Trimmed Cell mesher trims the core mesh in the background with the grey region as the input surface.

The polyhedral mesh utilizes cells with a polyhedral shape. This type of mesh is created by generating a tetrahedral mesh and forming polygons around every node of the tetrahedral mesh element. The polyhedral mesh has the advantage of a much better approximation of gradients due to the fact that each element has around 10 neighbours. Due to this fact the discretization method is simpler than the equivalent tetrahedral mesh and also the standard approximations for calculating the gradients at the cell centres can be utilized for more efficient and faster calculations.

The quality of a polyhedral mesh is investigated using different criteria, described below, available in STAR-CCM+.

There are also some important general criteria such as volume change, aspect ratio and skewness angle that need to be fulfilled. The volume change defines the increase in element size with each element. This is important to ensure no information is
lost when a multiple of smaller elements suddenly grow into a much larger element. A value of below 0.01 is considered bad quality. The aspect ratio defines the ratio between the edges of the element. Depending on the flow behaviour the aspect ratio can differ significantly. If the flow is homogeneous in the domain then a low aspect ratio is desired. But for a flow close to a wall where the boundary layer is present with strong shear a high number of elements is needed in the direction normal to the surface but parallel to the wall the grid spacing can be larger since the flow is parallel to the wall. Skewness angle criteria shows how well the cells on each side of a face are formed to allow for a certain level of diffusion without giving rise to unboundedness. A skewness angle of above 85° indicates a bad mesh.

**Face Validity:** The Face Validity metric measures the correctness of the normals of each face relative to the cell centroid they are attach to. The quality is measured in a value between 0 and 1, where a value of one corresponds to all normal pointing away from the centroid. The lower the value below 1 the faces with normals pointing inwards towards the cell centroid (Fig. 13). A face validity below 1 is considered a bad element quality and below 0.5 show a negative cell volume.

![Figure 13: The Face Validity of two different cells. The left cell is a cell with good cell validity and right cell has a bad cell validity](image)

**Cell Quality:** Cell Quality is yet another metrics available which is based on Gauss and least-squares methods for calculation of cell gradients. Cell Quality evaluates the geometric distribution of neighbouring cell faces relative to each other and also their orientation in space. Figure 14 shows a graphical representation of such a case. The cells with the highest value between 0 and 1 have the best quality. It is recommended to keep the cell quality above 1E-5 for all cells.\[8\]
Figure 14: The Cell Quality of two different cells. The left cell has a good Cell Quality but the right cell with flat surfaces has many non orthogonal faces giving a much lower cell quality.
3 Method

In this chapter the procedure used in this work, in order to obtain the results, is presented. The experimental approach used to generate validation data will be presented followed by the numerical method. The numerical method covers the geometry pre-processing, model set-up and treatment of the valve motion as well as the generation of the computational mesh and time-steps used. First a literature review of experimental and numerical methods presented.

3.1 Literature Review

This section presents the experimental and numerical approach of previous work at VCC as well as other experimental and numerical studies that have been done within the field of tank sloshing.

3.1.1 Experimental Methods

The experimental study performed in the previous work [3] consisted of a simplified transparent plastic box equipped with pressure sensors with the intention to inspect the sloshing visually. This was preformed by recording with a high frequency camera and measure the pressure difference induced by the sloshing. A ruler were attached to the side of the box to give a measurement of the wave height. The results obtained from the experiments were not ideal as the domain was hard to get completely sealed. The pressure sensors used needed to have a relative high gauge pressure inside the domain in order to detect any pressure differences. Since the box was hard to get sealed, and because it was unsure how much pressure the box could handle, no pressure measurements were used. The ruler was deemed redundant as the domain used for the experiments did not exactly match the domain used in the simulations. The measurements used from the sloshing experiments were therefore only visual inspection of the sloshing. Experiments of the upper floater interaction with the free surface were also performed in form of a drop test. The floater was attached to a trajectory to limit its motion to one direction, then it was dropped from a certain height above the free surface. The impact with the surface was recorded with a high frequency camera to be compared with the simulation of the same case.

The results of the experiments concluded that a more rigid domain was needed in order to perform pressure measurements of the sloshing. Further experiments on the floater movement were also deemed necessary in order to validate the numerical model. Reviews of other experimental studies on tank sloshing have been done in order to further develop this experimental approach.

Yichao Chen and Mi-An Xue [15], investigated how different filling levels affect the sloshing numerically and validated with experiment. For the experiments they used a simplified box of plexiglass, 8mm thick. Pressure sensors on the tank walls were installed to measure the dynamic impact pressure. The tank was put on a hexapod to generate the motion.
Sang-Yeob Kim et al. [16], did a study on comparing different pressure sensors for sloshing experiments. They set up a rectangular tank of plexiglass, 20mm thick, with pressure sensors attached to one side. A hexapod was used to generate motion.

Minho Ha et al. [17], validated their numerical method of sloshing through experiment with a rectangular tank made of plexiglass with thickness 20 mm. They attached pressure sensors on one side and used a hexapod to generate motion.

Jiang Mei-rong et al. [18], made a study on hydro elastic effect in an elastic tank and compared it to an rigid tank. Their rigid tank experiment consisted of an rectangular plexiglass tank with wall thickness 12mm. Pressure sensors were attached to the tank walls to measure the impact pressure.

Shuo Huang et al. [19], developed and validated a boundary element method used to simulate tank sloshing. Their experimental tank was a rectangular shaped plexiglass box with pressure sensors on on side. The wall thickness was 20mm.

Conclusion of the literature findings are that the most common way of doing experimental studies on tank sloshing are with a simplified geometry made of transparent material such as plexiglass. The most common measurements are visual inspection of the sloshing and impact pressure on the walls. Experiments on sloshing using a pressurized tank were not found.

### 3.1.2 Numerical Methods

The numerical method in previous work by Eklund and Kreuger [3] was performed using a simple VOF model with an HRIC scheme and first order temporal discretization. The domain was constructed of polyhedral mesh using a two-layer approach together with the Realizable $k-\varepsilon$ turbulence model. The RKE model was concluded to be sufficient in modeling the turbulence inside the tank compared to other models with the comparable computational cost. The moving FLVV was modeled using an Overset mesh and also a dynamic mesh but it was concluded that the Overset mesh was more computationally efficient.

In the investigation conducted by Khezzar et al. [20], water sloshing inside a rectangular container was studied both experimentally and by mean of CFD. The CFD model was considered as laminar flow dominant and unsteady. The multiphase model used was VOF together with a segregated solver and also first order temporal discretization. The simulations showed good agreement with the experimental data capturing the sloshing phenomena inside the tank.

The work of Ha et al. [17] also involves a rectangular domain where the VOF method has been used together with either $k-\varepsilon$, SST or without turbulence model, i.e. with laminar assumption. The CFD model was later validated against experimental data. The $k-\varepsilon$ models showed a faster convergence but the SST showed a better prediction of pressure variation in the tank however the prediction of exerted forces inside the tank by the water is similar between the model. The visual inspection of the wave propagation inside the tank was best captured by the SST model. It was thus concluded that SST is the most suitable turbulence model. In the laminar case the results show good agreement but showed some peak values not present in the
SST model.

JIN et al. [21] also investigated sloshing inside a tank but with a perforated plate inside. The numerical model was set up with a VOF model and laminar flow model. The numerical solver was a combination of Pressure-Implicit Splitting of Operators and Semi-Implicit Method for Pressure-Link Equations called PIMPLE algorithm for a faster solver. The Van Leer scheme was used for discretization giving a second order accurate solution. The results in these investigation also showed good correlation to the experimental results obtained.

To conclude, there is a distinct trend between all these simulation being the utilization of VOF model for simulation of multiphase flows due to its efficiency in computational cost. To be noted is that these papers were only interested in the sloshing inside tank, not the importance of wetting on critical surface. In this current investigation the wetting of the FLVV is just as important as the sloshing inside the tank thus putting higher requirement on the CFD method as a whole.

3.2 Tank and FLVV Geometry

The geometrical models investigated in this work were two different geometries presented below, a 60l fuel tank and a simplified tank with a FLVV assembly inside designed and constructed to be used for experimental tests to obtain validation data.

Fuel tank geometry

The fuel tank used in this work is a standard plug-in hybrid electric vehicle (PHEV) 60l tank (Fig. 15) for the United States market. This particular tank model was used due to that the previous work had this model for the development of the method. The choice of tank geometry will not affect the development of the method as the method is to be used with any kind of tank geometry. Inside the fuel tank there are multiple components including the component of interest, the FLVV, as well as components which are not investigated but affect the flow and sloshing inside the tank. These are the baffles, the fuel pump, fuel level sensors, jet pump and different types of pipes.
Simplified tank geometry

In order to validate the sloshing behaviour for the CFD method a new sloshing rig needed to constructed. The design was based on the test rig used in the previous work[3]. The aim for the design of the new rig was to improve the rigidity to be able to pressurize the tank. The design consist of two halves, bottom and top, that is sealed with screws and window seal (Fig. 16). The two parts are designed with some connections that can be used to connect pressure sensors, air supply (for pressurization) as well as drain and filling systems. The bottom flange has holes all the way around it made to be able to fasten the tank to a hexapod platform located at VCC test facilities. Mounting holes for the FLVV is made on the top side. The tank is manufactured with a 3D-printed transparent SLA plastic to make visual observations possible.
FLVV

The FLVV (Fig. 17), which is the component of interest for this method consist of a number of highly detailed parts. The main parts of this particular valve used are the two floaters inside the casing, the upper and lower floater. These floaters have different buoyancy capacity. The lower floater is less buoyant than the upper floater i.e the upper floater shuts outlet of the valve first followed by the lower floater when the fuel level rises further. The two floaters are situated inside the casing which has three small holes located on the bottom and four rectangular shaped holes on one side of it.

![FLVV](image)

(a) Uncut view of the FLVV  
(b) Cut section of the FLVV

Figure 17: The left figure show an uncut view of the FLVV with the inlet holes seen on the left side and the outlet hole on the top to the right. The right figure show a cut section of the FLVV with the upper moving floater in turquoise, bottom floater in dark grey and casing in light grey

### 3.3 Experimental Set-up

The modeling of the moving floater inside the FLVV is one of the central parts of the developed CFD method and therefore it is of great importance to validate the accuracy of it. In this section the set-up of the experiment conducted with purpose to validate the motion modeling is explained. The setup for the sloshing experiment, using the simplified tank, is not included as due to time limitations no experiments have been performed and is suggested to be done for future work.

#### 3.3.1 FLVV Motion Experimental Set-up

In order to validate the modeling of the moving floater inside the FLVV an experiment was conducted as well as a corresponding simulation. The approach for this experiment was to place the FLVV, without the outer casing, inside a glass jar and fill it up with water and measure the movement of the floater as it rises.
The lower floater was fixed in place to the inner casing to prevent it from moving during the filling process. The casing was then glued to the bottom of the jar and a filler pipe, in form of a rubber tube, was fixed in place next to the side of the jar (Fig. 18). A slot in the casing was made to be able to see the movement of the upper floater and a ruler was fixed next to the slot in order to measure the position of the floater over time (Fig. 19). The filling process was conducted by connecting the filler pipe to a reservoir of water and then letting the water flow through the pipe (Fig. 20).

Figure 18: An illustration of the experimental setup for the validation of the motion of the floater inside the FLVV

Figure 19: Experimental setup as seen from the front with the FLVV placed inside a jar. The blue ring shows the floater inside the FLVV
Figure 20: The experimental setup prior to filling (left) and directly after starting the fill by turning the upper container upside down (right)

The mass flow rate needs to be known as it is a boundary condition for the simulation and it was determined by measuring the time it took to fill the jar, without the presence of the FLVV, with a certain volume of water. The flow rate measurement was performed three times and the average value was then used in the simulation (Table 2).

Table 2: The average mass flow rate measurements of three consecutive tests used in the simulation

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass Flow Rate Avg.</td>
<td>0.0333</td>
</tr>
<tr>
<td>Standard Deviation</td>
<td>0.0019</td>
</tr>
</tbody>
</table>

3.4 Computational Domain

The computational domain of interest was the 60 l tank (Fig. 15) and also a simplified tank as figure 16 shows but the simplified tank was not followed through due to time constraints. In both tank domains the FLVV, venting house and outlet pipe are included. This was cleaned up of smaller geometrical details as seen in figure 21 where the details not affecting the flow significantly were removed for reason related to computational efficiency and that could compromise the quality of the volume mesh. The same FLVV was used for the validation simulations of the floater motion and the complete computational domain in that case was the jar containing the FLVV.
For the 60ℓ tank the inner details were cleaned and smaller details removed. The result shows only components which is believed to have the largest impact on the flow inside the tank (Fig. 22a). The numbered components are described in table 3. The parts that are kept in the domain were given different Property ID (PID) to easier control both mesh generation and post processing.

Table 3: The components residing inside the tank numbered as seen in figure 22a and 22b

<table>
<thead>
<tr>
<th>Part Name</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1 Baffle passive side</td>
<td></td>
</tr>
<tr>
<td>2 FLVV</td>
<td></td>
</tr>
<tr>
<td>2a Upper floater</td>
<td></td>
</tr>
<tr>
<td>2b Lower floater</td>
<td></td>
</tr>
<tr>
<td>2c Outer casing</td>
<td></td>
</tr>
<tr>
<td>3 Liquid trap</td>
<td></td>
</tr>
<tr>
<td>4 FDM</td>
<td></td>
</tr>
<tr>
<td>5 Baffle active side</td>
<td></td>
</tr>
<tr>
<td>6 Outlet pipe</td>
<td></td>
</tr>
</tbody>
</table>

These parts were later surface meshed in ANSA and imported into STAR-CCM+. The surface mesh was generated fine enough to preserve all the surface details. In
3.5 Numerical grid

The volume mesh inside the domain was built of solely polyhedral mesh. The polyhedral cells were used due to the better efficiency, in terms of computational cost, than the tetrahedral cells. A trimmed cell type mesh was also considered because of its faster mesh generating capability. It was ruled out due to the complexity of the tank geometry and expected violent sloshing that will result in that the mesh not to be aligned with the flow at all times. The near-wall mesh in the tank was constructed with 3 prism layers on all walls. The size of the first prism layer thickness was set to obtain $30 < y^+ < 5$ to fulfill the criteria for the two-layer All $y^+$ wall treatment. Inside the FLVV the $y^+$ was set below 5 due to the smaller cell size necessary to capture the geometrical details whilst inside the rest of the tank the $y^+$ for the tank walls was above 30.

To make sure of reducing the error caused by bad mesh the mesh quality was checked. The mesh metrics used were the aspect ratio criteria, growth rate, skewness angle, face validity and cell quality each making sure that the mesh fulfill certain criterias for a good mesh. The threshold values and criteria definitions of each mesh metric is described thoroughly in the theory section.

The domain was divided into different regions in order to get more control over the mesh generation both for the volume and surface mesh (Fig. 23). The venting house region consist of the FLVV, liquid trap and outlet pipe volumes represented in dark grey. The overset region, represented in blue, is separated from the venting house region and contains the upper floater. The background region in light grey consists of the rest not covered by the venting house and overset regions.

![Figure 23: A section cut of the tank displaying the different regions used to control the mesh generation illustrated with the different colors. The light grey represent the background region, the dark grey represent the venting house and the blue is the overset region](image)

3.6 Numerical Model Set-up

The performed simulations for the verification, validation and general investigations of sloshing were carried out using a VOF model which is a multiphase model with two different phases involved. The solver scheme for this particular model is the
Segregated solver which solves the continuity equation and momentum equations separately which is generally more robust but has a slower rate of convergence.

The discretization of scalar transport was solved using a HRIC scheme, described thoroughly in section 2.8.2, which uses both Bounded Downwind and Upwind differencing schemes which is suited for tracking of sharp interfaces.

The temporal discretization used for the verification study was both first and second order accurate discretization method to investigate the stability of each. Due to no stability issues nor higher computational cost the second order was chosen for the rest of the simulations. Also the recommendation from STAR-CCM+ was to use a second order temporal discretization for highly unsteady flow.

The turbulence model of choice was the Realizable $k - \varepsilon$ model based on recommendations from VCC and also previous work conducted in the subject by Eklund and Kreuger [3]. The near wall treatment used was a two layer All $y^+$.  

Table 4: Solver setup used for the validation study with the most important settings included

<table>
<thead>
<tr>
<th>Solver</th>
<th>Implicit Unsteady</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulence Model</td>
<td>Realizable $k - \varepsilon$</td>
</tr>
<tr>
<td>Wall Treatment</td>
<td>Two-Layer All $y^+$</td>
</tr>
<tr>
<td>Temporal discretization</td>
<td>Second-Order</td>
</tr>
<tr>
<td>Time-Step</td>
<td>$5.2 \times 10^{-4}$ s</td>
</tr>
<tr>
<td>Convection Scheme</td>
<td>HRIC</td>
</tr>
<tr>
<td>Inner Iterations/Time-Step</td>
<td>5</td>
</tr>
</tbody>
</table>

3.6.1 Boundary Conditions and Physics

The tank domain consists of few major domains which can be initialized differently, the tank, venting house and FLVV region (Fig.15). The tank was initialized according to fuel level and the venting house with only air inside. The FLVV was initialized with no gasoline inside and filled during the first 0.5 seconds of the simulation. Inside the domain two different Eulerian phases have been specified, gasoline and air. The properties of each phase was specified according to table 5. The initialization of the fraction of each phase was defined as a function where the amount of fuel was specified. The walls of the tank and venting region were set as no slip walls and the outlet of pipe leading away from the venting house was set as a pressure outlet with atmospheric pressure during normal simulations. This pressure outlet was later modified to mimic the purging from the canister by using a transient boundary condition specifying a certain pressure drop at a certain frequency corresponding to the real driving conditions (Fig. 35). The simulation performed for validation was initialized with only air inside and the filler pipe inlet a mass flow inlet boundary condition was specified with the average mass flow rate specified as input (Table 2). The opening of the jar was set as a pressure outlet and specified atmospheric pressure.

The validation performed for the motion of the FLVV using the setup with an FLVV and a container (Fig. 19) involved air and water as the two phases whilst the simulations for the real tank sloshing involved gasoline and air.
Table 5: Specified fluid phase properties for the simulated fluids

<table>
<thead>
<tr>
<th>Phases</th>
<th>Density ([kg/m^3])</th>
<th>Dynamic Viscosity ([kg/ms])</th>
<th>Surface Tension ([N/m])</th>
<th>Contact Angle ([\cdot])</th>
</tr>
</thead>
<tbody>
<tr>
<td>Water</td>
<td>997.6</td>
<td>8.887E-4</td>
<td>0.072</td>
<td>60</td>
</tr>
<tr>
<td>Gasoline</td>
<td>720</td>
<td>3.64E-4</td>
<td>0.045</td>
<td>0</td>
</tr>
<tr>
<td>Air</td>
<td>1.18</td>
<td>1.855E-5</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

The sloshing inside the tank was induced using the obtained data from a real driving sequence on the Hällered test track in X, Y and Z (Fig. 24 and 26). The acceleration was implemented as a source term in each momentum equation respectively. The acceleration data is filtered when imported into STAR-CCM+ and when needed data is interpolated between the recorded acceleration using a B-spline. The simulated section of the obtained data was decided based on the most aggressive driving condition which was deemed to be from the 52 s mark onwards (Fig. 25).

Figure 24: Data obtained using accelerometer inside the car during a test track driving session

Figure 25: Used time interval of the Hällered test track data for the simulations. This time interval is simulated for some simulations but most of the simulations only parts of this time interval are simulated
3.6.2 Fill Limit Vent Valve Motion Modelling

Inside the FLVV there are two floater which can move depending on the conditions. Both floaters are limited in their motion in the sense that they can only move in one direction translating motion along the z axis. The upper floater moves more easily due to its lower weight which is affected easier by buoyancy. The rubber sealing attached to the top of the floater has been modelled as a non flexible and rigid body for simplification.

Dynamic Fluid Body Interaction

Due to previously stated fact and the main functionality of the FLVV being fulfilled by the upper floater movement only the upper floater has been modelled as a moving body using DFBI modeling with one degrees of freedom translating motion. When the body then hits the bottom or the top the motion limits comes into play and stops the movement of the floater. The stopping process starts when a damping length is reached. The damping length is the distance between a moving body and a wall boundary at which the body starts slowing down when the solver applies a damping force progressively. The damping length is set to the size of the prism layer total thickness in the FLVV due to the fact that the prism layer thickness is critical for a stable stopping solver. This ensures that the body moves through this length during several time steps which is a requirement according to STAR-CCM+\cite{8}.

In table \ref{tab:motion_settings} the settings used for the motion and physics of the floater are specified. The Body Mass specifies the actual weight of the moving floater. The release time is the time from simulation start when the calculation of the motion begins. This is specified in order to let the flow initialize before calculating the motion. The recommended release time for an unsteady simulation is 10-50 times the simulation time step which varies during the verification study. The higher the release time the more accurate movement of the floater thus the choice of a factor 50. The ramp time is utilized to obtain a more robust solution with a less oscillating behaviour. The forces and moments applied to the floater at an instant could cause a shock effect
thus the ramp time is used to gradually add the forces and moments and reducing the negative shock effect. A good practice is to have a factor of 10 times the release time according to STAR-CCM+[8].

The values for Initial Moment of Inertia is the specified initial moments of inertia tensor in x-, y- and z-direction. Even though the motion of the floater is specified as 1-DOF translating motion leading to no moments occurring on the body a non-zero value is not accepted by the solver thus an arbitrary low value was used.

<table>
<thead>
<tr>
<th>Properties</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Body Mass</td>
<td>0.00327 kg</td>
</tr>
<tr>
<td>Release Time</td>
<td>50*Time-step</td>
</tr>
<tr>
<td>Ramp Time</td>
<td>10*Release Time</td>
</tr>
<tr>
<td>Initial Moment of Inertia</td>
<td>[2E-4, 2E-4, 2E-4] kg/m²</td>
</tr>
</tbody>
</table>

Table 6: The setting used for the DFBI-solver

Overset Mesh

To be able to have a moving solid body inside the fluid domain an overset mesh is utilized. By subtracting the floater from the rest of the domain a background mesh could be generated. The floater was later enclosed in a cylinder domain which was then meshed as the overset region. This cylinder could allow for the overlapping of at least 4-5 cells between the background and overset mesh and the nearby walls. This cell overlapping is also a requirement for the coupling between the overset and background mesh. As stated earlier the overset and background mesh size must be of the same size in the overlapping region to reduce interpolation error and this can be seen in figure 27 for the investigated cases.

Figure 27: The overset region in blue overlapping the background mesh in gray. The overset has approximately the same mesh size as the background mesh

As stated earlier in chapter 2.6 a zero gap function can be used when there is a possibility of the wall boundaries of the overset region coming in contact with the background region. This has been used for the floater since both the top and bottom part of the floater come in contact with the wall boundaries of the background region (Fig. 28). The minimum amount of layers, above which the solver will deactivate the overlapping cells, were set to the recommended value by STAR-CCM+[8] as 2 cell layers. Distance of deactivated region acting as a wall will thus depend on the cell sizes of in the proximity.
3.6.3 Post-processing

The volume of fuel in different regions during the simulations was used as a post-processing parameter. The volumes in the active versus passive side were measured as well as the volume inside the FLVV and were mainly used for the verification process of mesh and time-step (chapter 3.7). When investigating the LCO, the volume of fuel inside the liquid trap was tracked to see if and when fuel found its way past the floater in the FLVV. Another parameter used for the verification study was the center-of-mass coordinate of the fuel. This parameter as well as the volume of fuel in the active and passive side are forgiving parameters in the sense that it is evaluated over a large volume and small discrepancies will not be captured by these parameters. When these parameters are used for verification they should be used with that in mind.

The closing behaviour of the floater inside the FLVV was of interest to investigate for various conditions, especially if the purging of the canister can affect the floaters motion. Because of this the position was monitored during the simulations (Fig. 29) where the zero position is defined as the top, and closed, position of the floater.

Figure 28: The Zero Gap region activated as seen in the left image compared to the right where Zero Gap is deactivated

Figure 29: The definition the floater position where 0 imply that the floater is in its closed position
As it was of interest to identify if any fuel manages to leak out into the liquid trap a Schmitt trigger was implemented with a field function to track which points that had been in contact with fuel during the simulation. The field function can be applied to either surfaces or plane sections to identify after the simulation which areas that have been in contact with the fuel (Fig. 30). This function is here onwards referred to as the High Water Mark.

![High Water Mark on a plane section](image1)

(a) High Water Mark on a plane section

![High Water Mark on the floater surface](image2)

(b) High Water Mark on the floater surface

Figure 30: Two examples of the High Water Mark function. Left figure show a plane section where the blue indicates areas that have been in contact with the fuel during the simulation. Right figure show the surface of the upper floater from another simulation, also here the blue indicates areas that have been in contact with fuel

### 3.7 Spatio-Temporal Discretization Investigation

The verification study started with a base mesh size which was sufficiently fine to capture the geometrical details and also fulfilled the requirements for the near wall treatment model used (Tab. 7). The base time step used was based on the previous work that fulfilled the requirements for the CFL number and also the movement of the overset mesh region per time step and is further examined in section 3.7.1.

<table>
<thead>
<tr>
<th>Base size</th>
<th>TSS</th>
<th>MSS</th>
<th>SGR</th>
<th>PLTT</th>
<th>Cell count</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Background</strong></td>
<td>5.8E-3</td>
<td>100%</td>
<td>75%</td>
<td>1.3</td>
<td>50%</td>
</tr>
<tr>
<td>Venting house</td>
<td>25%</td>
<td>15%</td>
<td>1.3</td>
<td>50%</td>
<td>965k</td>
</tr>
<tr>
<td><strong>Overset</strong></td>
<td>2.2E-3</td>
<td>100%</td>
<td>75%</td>
<td>1.2</td>
<td>43%</td>
</tr>
<tr>
<td>Floater</td>
<td>75%</td>
<td>50%</td>
<td>1.2</td>
<td>43%</td>
<td>16k</td>
</tr>
</tbody>
</table>

**Total:** 981k

The recommendation by STAR-CCM+ is to have the average CFL below 0.5, at the free surface, which is an established best practice for the Second order temporal discretization combined with the VOF model. The CFL should be kept at 0.7 for the first order temporal scheme. If this is sufficient to just keep the average CFL below the limit for this particular problem is investigated and presented in the following sections.
Regarding the movement of the overset mesh, it is recommended by STAR-CCM+ to establish a time step which is small enough to allow for a movement of 0.5 cell size per time step. This is the requirement when using the second order temporal, for the first order this value can be set to 1. This requirement has its founding in the fact that the inactive cells that become activated when the body moves must be able to be an acceptor cell for at least one time-step, in case of first order temporal. This in order to ensure that a solution exists at the previous time step to obtain the rate of change term. For a second order temporal there must exist at least solutions for two previous time steps thus the 0.5 cell size movement per time step \[8\]. During all simulations the mass conservation inside the domain has been monitored to ensure that the mesh resolution and time-step are fine enough. The mass imbalance can increase due to bad interpolation between the overset mesh and the background mesh, i.e. a bad transition in mesh size between the overset mesh and the background mesh.

For the verification study the driving cycle used start at the 65 s mark of the Hällered test track data. The fuel level was set to 90\% of the full capacity. The time point was chosen based on that at this point some of the larger accelerations present in the data appear. The fuel level of 90\% full was chosen due to that the fuel levels investigated with regards to LCO were 95\% and 85\% full.

### 3.7.1 1st vs 2nd Order Temporal Discretization

As in the previous work the developed simulation method used exclusively first order temporal discretization \[3\]. An investigation was performed to evaluate if the problem was stable enough to be used with second order and if the method in that case would benefit from using it. The investigation was done with the initial mesh (table 7), with three different time-steps for both first order and second order temporal discretization.

The three different time-steps used were $3E^{-3}$ s, $1.5E^{-3}$ s and $7.5E^{-4}$ s. These were chosen based on the findings in the previous work \[3\] that had reasonable average CFL numbers of below 0.3 on the free surface. The time-steps was used to examine the differences between first and second order for both higher CFL, than the mentioned in the previous work, and lower. The essential findings are presented below (further data can be found in App. A).

#### 1st Order Temporal Discretization

From Tab. 8 the CFL data from the simulations performed with first order temporal discretization are shown. The average CFL on the free surface is below the limit of 0.7 for all time-steps. However the maximum CFL on the free surface ranges between 65 – 9 and the average percentage of cells above the limit is between 17 – 2 percent. In figure 31 the volume of fuel is plotted for the three time-steps and noticeable differences can be seen between them, especially in the time range 1 – 1.5 s and 2.5 – 3.5 s. The oscillations observed for the $3E^{-3}$ simulation are not present for the other two smaller time-steps, implying that the $3E^{-3}$ time-step, or rather an average CFL $\sim 0.5$, is too large for this flow problem for first order temporal discretization even though it is below the limit of 0.7.
Table 8: Obtained CFL numbers for the different time-steps with first order temporal discretization. Avg. CFL is the maximum average CFL on the free surface and max CFL is the maximum CFL on the free surface. The cells above CFL limit is the maximum percentage of cells that have been above the limit

<table>
<thead>
<tr>
<th>Time-step</th>
<th>Avg. CFL</th>
<th>Max CFL</th>
<th>Cells above CFL limit (0.7) [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.5E-4</td>
<td>0.13</td>
<td>9.07</td>
<td>2</td>
</tr>
<tr>
<td>1.5E-3</td>
<td>0.27</td>
<td>25.78</td>
<td>8</td>
</tr>
<tr>
<td>3.0E-3</td>
<td>0.52</td>
<td>64.64</td>
<td>17</td>
</tr>
</tbody>
</table>

![Graph](image1.png)

Figure 31: Plot of the volume of fuel inside the FLVV. Observe that the beginning has been cut out to highlight the interesting part

2nd Order Temporal Discretization

For the second order simulations oscillations for 3E-3 are seen, also for 1.5E-3 but smaller (Fig. 32). The oscillations for 3E-3 are more severe than the ones for first order simulation and differ more to the other two simulations. This can be coupled to that even the average CFL number (table 9) is above the second order requirement of 0.5. The 1.5E-3 show a much better agreement although oscillations are observed and a deviation can be seen around 1.5 s (Fig. 32).

![Graph](image2.png)

Figure 32: Plot of the volume of fuel inside the FLVV for the 2nd order temporal simulations. Observe that the beginning has been cut out to highlight the interesting part

No stability issues were observed for any of the simulations for second order suggesting that it could be used for this problem to minimize the truncation error. A simulation with a time-step of 5E-4 was also done to see the differences for an even lower CFL (Fig. 32). The conclusion was that this time-step gives similar result to the time-steps 7.5E-4 and 1.5E-3 implying that a time-step somewhere between
1.5E-3 and 7.5E-4 is suitable for this mesh size with second order temporal discretization. Implying that an average CFL on the free surface of below $\sim 0.3$ is needed.

Table 9: Obtained CFL numbers for the different time-steps with second order temporal discretization. Avg. CFL is the maximum average CFL on the free surface and max CFL is the maximum CFL on the free surface

<table>
<thead>
<tr>
<th>Time-step</th>
<th>Avg. CFL</th>
<th>Max CFL</th>
<th>Cells above CFL limit (0.5) [%]</th>
<th>CPU cost [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>5.0E-4</td>
<td>0.10</td>
<td>4</td>
<td>2</td>
<td>100</td>
</tr>
<tr>
<td>7.5E-4</td>
<td>0.14</td>
<td>18</td>
<td>4</td>
<td>70</td>
</tr>
<tr>
<td>1.5E-3</td>
<td>0.28</td>
<td>21.16</td>
<td>11</td>
<td>38</td>
</tr>
<tr>
<td>3.0E-3</td>
<td>0.53</td>
<td>54.37</td>
<td>24</td>
<td>19</td>
</tr>
</tbody>
</table>

Comparison

To see the differences between the first order and second order temporal discretization scheme in terms of accuracy a comparison was done (Fig. 33). It shows the simulation with time-step 7.5E-4 for the first order and the simulation with time-step 1.5E-3 for second order. As can be seen they agree well to one another and thus that the second order will be more accurate. A reduction of $\sim 50\%$ in computational cost is also achieved by using the second order with time-step 1.5E-3 compared to 1st order with time-step 7.5E-4.

![Comparison of the fuel inside the FLVV for first and second order temporal discretization](image)

Figure 33: Comparison of the fuel inside the FLVV for first and second order temporal discretization

### 3.7.2 Mesh Refinement

Iterations on the initial mesh were performed in order to find a suitable mesh in terms of accuracy and computational cost. The iterations of meshes investigated can be seen in table 10 with the time-step used for each one. The time-step was reduced for each mesh with about the same magnitude as the cell size in order to keep the CFL at the same level.
Table 10: The mesh setting for each refinement performed for mesh sensitivity study

<table>
<thead>
<tr>
<th>Cell count</th>
<th>Time-step</th>
<th>Background base size</th>
<th>Overset base size</th>
</tr>
</thead>
<tbody>
<tr>
<td>981k</td>
<td>1.5E-3</td>
<td>5.8E-3</td>
<td>2.2E-3</td>
</tr>
<tr>
<td>2M</td>
<td>7.5E-4</td>
<td>4.2E-3</td>
<td>2.1E-3</td>
</tr>
<tr>
<td>3M</td>
<td>6E-4</td>
<td>3.5E-3</td>
<td>1.8E-3</td>
</tr>
<tr>
<td>4.2M</td>
<td>5.2E-4</td>
<td>3.1E-3</td>
<td>1.75E-3</td>
</tr>
</tbody>
</table>

Figure 34: Comparison of the fuel volume inside the FLVV for the four different meshes

The results of the different meshes show similar trends but differences can be seen between them. Quantities like amount of fuel in passive and active side are very similar for the four meshes where the biggest difference is 0.5% between the 981k and 2M meshes. The volume of fuel inside the FLVV is more sensitive (Fig. 34), where distinct differences are seen. The coarsest mesh, 981k, shows a slightly different behaviour at the beginning of the simulation and has a lower fuel level at the end where the level stabilizes. Between the 2M and 3M meshes the volume is similar until the end where again the level is a bit lower for the 2M mesh compared to the 3M mesh. The finest mesh investigated, 4.2M, compared to the 3M mesh was very similar throughout the simulation with only a slight deviation at the 1.5 s mark which correspond to a difference of 3.8%. The good compliance between the 3M and the 4.2M meshes was deemed sufficient and the 3M mesh was used for the investigation of fuel leakage. To investigate how much of the differences, that are seen between the meshes, that can be related to the unsteadiness of the problem a repeatability was performed. Three simulations with the same setup, with the 3M mesh, were done and compared (Appendix B). The maximum standard deviation for the volume of fuel inside the FLVV was $8.98e^{-7}$ ml and compared to that the volume vary between 30−50 ml the deviation is small implying that the simulation has rather good repeatability.

3.7.3 Adaptive Time-step Control and Computational Cost

In order to reduce the computational cost of the model an investigation of using an adaptive time-step was conducted. The adaptive time-step control enables the time-step size to change during the simulation based on a given criterion, e.g. to keep CFL number within a given range. The purpose with this is then to speed up the simulation when possible. As for this particular flow problem the intensity of the sloshing will vary during the simulation time-period and if a constant time-step is used it will need to be chosen with respect to the most intensive sloshing time-period.
With an adaptive time-step it could be possible to use a larger time-step when the sloshing behaviour is calmer and thereby be able to reduce the computational cost.

The implementation of the adaptive time-step control was made by identifying an acceptable upper and lower limit for the average CFL number on the free surface. Later define the time-step size to increase if the CFL number hits the lower limit or decrease if it hits the upper limit. This was done with field functions and reports in STAR-CCM+. The upper limit was decided based on the findings from the time sensitivity study, see section 3.7.1, where an average CFL below \( \sim 0.3 \) on the free surface was deemed sufficient. The lower limit was also decided based on these findings that a lower average CFL than \( \sim 0.15 \) does not improve the accuracy with any significant amount. The used values can be found in table [11] where \( f \) is the factor with which the time-step size will increase or decrease. The \( \Delta t_{\text{max}} \) and \( \Delta t_{\text{min}} \) are the maximum and minimum time-step sizes allowed, introduced to prevent infinite increase or decrease of the time-step size. To avoid the time-step size to jump outside the given operational range the factor \( f \) was given a number (Eq. 44)

<table>
<thead>
<tr>
<th>Value</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>( CFL_{\text{max}} )</td>
<td>0.3</td>
</tr>
<tr>
<td>( CFL_{\text{min}} )</td>
<td>0.15</td>
</tr>
<tr>
<td>( f )</td>
<td>1.05</td>
</tr>
<tr>
<td>( \Delta t_{\text{max}} )</td>
<td>( \Delta t \times 1.25 )</td>
</tr>
<tr>
<td>( \Delta t_{\text{min}} )</td>
<td>( \Delta t \times 0.90 )</td>
</tr>
</tbody>
</table>

\[ 1 < f \leq \frac{CFL_{\text{max}}}{CFL_{\text{min}}} \quad (44) \]

The adaptive time-step control was used for all simulations performed except for the spatio-temporal discretization study. A comparison between a simulation performed with the adaptive time-step control and a constant time-step can be observed in App. [C]. The computational cost reduced by using an adaptive time-step control will vary depending on which time-interval of the driving cycle that is simulated. For the investigated time interval showed an improvement of around 50\% Another factor affecting the computational cost is the overset mesh. A typical simulation using the overset mesh had a computational time of \( 3.5 - 4.5 \) h per second of simulated time on 200 cores compared to when not using the overset mesh. Thereby also modeling the upper floater as rigid, resulted in a a computational time of \( \sim 1 \) h per second simulated time on 200 cores.
3.8 LCO Investigation and FLVV Performance

The parameters investigated in this study to conclude their effect on LCO are firstly an aggressive driving cycle where the acceleration in X and Y have been doubled. The purging of the canister has also been investigated described more thoroughly in section 3.8.1 with changed boundary condition. Longer simulations with a rigid floater was first conducted to identify possible interesting time windows to a lower cost, as the overset mesh could be disabled. This also made it possible to investigate the effects of including the moving floater or not. The effects of different fuel levels inside the tank, 85 % and 95 % has been also been investigated. All these cases have been run with the adaptive time-step control described in previous chapter.

3.8.1 Purge Resonance

As described in section 3.6.1, the outlet of the tank was set as a time varying pressure outlet when simulating purging. The simulations were conducted with a 50 % duty cycle resulting in a pressure at the outlet replicating PWM (Fig. 35). At $\sim$80 liters per minute (lpm) the pressure is $\sim$ -2.5 kPa for a new tank and -3.5 to -4.0 kPa in an aged tank. The change of mass inside the domain (Fig. 36) was monitored as in previous cases, spatio-temporal sensitivity study, to ensure mass conservation affected by mesh and time resolution. This can occur due to numerical dissipation and also high CFL number.

![Figure 35: The pressure variation at the outlet of the tank during purging with -4 kPa, at 80 lpm, as a peak pressure during an arbitrary time period of the driving cycle with 50 % DC](image)
3.9 Evaporation Model

A brief investigation of setting up an evaporation model was performed to see the increased computational cost with the added evaporation physics and if it is a viable choice to add evaporation to the method. The setup of the evaporation model was similar to the sloshing simulations, the difference being the added energy equation and evaporation model. The evaporation takes place at the VOF interface but the gaseous phase in this case is a mixture model of gasoline vapor and air. Due to major instabilities in the evaporation solver a simplified model was investigated without any overset region thus no moving floater. The simulated domain was a box with the same dimensions as the experimental 3D-printed tank (Fig. 37). The FLVV was added without the liquid trap to ensure further stability together with the extension at the top to account for the reversed flow at the interface. The domain is a closed volume with no inlets or outlets. The mesh used was the same as in the sensitivity study except some minor local changes inside the FLVV to counteract high residuals in those area. The domain was initialized with 20 % fuel vapor as recommendation from VCC. the vapor content in the gas was around 20-30 % during experiments depending on conditions.
A 55 second simulation was conducted to conclude the stability of the solver before implementing on the real tank geometry with an initial temperature of 300 K. The simulation was conducted with the same driving cycle as in the mesh and time sensitivity study. The conducted simulation required 1000 CPU hours per simulated second without overset mesh. This gives a 5 hour simulation time with the available 200 cores. The saturation pressure was set to be calculated according to Antoine’s equation. The constants used are specifically for Gasoline, tabulate according to tab. [12].

Table 12: The values set in the solver for the constants in Antoine’s equation

<table>
<thead>
<tr>
<th></th>
<th>Value</th>
<th>A</th>
<th>B</th>
<th>C</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>11.949</td>
<td>3978.205</td>
<td>-39.801</td>
<td></td>
</tr>
</tbody>
</table>
4 Results

In this chapter the results for investigating the LCO phenomena are presented for various conditions as well as the results from simulations of the simplified tank used to validate the method.

4.1 FLVV Motion Validation

The experiment conducted in order to validate the motion modelling of the upper floater shows good agreement to the corresponding simulation performed (Fig. 38). The data show that the floater starts to float at approximately the same time for both experiment and simulation. The same can be said for when the floater hits the upper, closed, position.

![Floater Position](image)

Figure 38: The movement of the floater for the simulation and also the two similar experiments performed during the 10 second of filling the jar

4.2 LCO investigation and FLVV Performance

4.2.1 Rigid Floater

Simulation with a rigid floater in its bottom position was performed to identify possible critical time windows with respect to LCO. Physical time simulated was 50 s with start at the 52 s mark of the Hällered test track data (Fig. 24). During this simulation no LCO was obtained but interesting time windows were seen around the 55 s to 56 s mark and at 73 s to 80 s (Fig. 39).

![Volume of fuel inside FLVV](image)

Figure 39: Plot of the volume of fuel inside the FLVV during the 50 s simulation with rigid floater
At 55 s a rapid increase in fuel volume inside the FLVV is observed as a result of a sharp decelerating right turn by the car. Further ahead at 73 s the volume of fuel inside the FLVV begin to increase and decrease repeatedly as a result of quick left and right turns. These two time windows were investigated further with a moving floater for various conditions as well as with a rigid floater and accelerations in x- and y-directions multiplied by two. The essential results of these are presented in the following sections.

4.2.2 Aggressive Driving Cycle

Different driving styles, i.e. amplitude of accelerations, were investigated to see how it affect the amount of LCO when driving aggressively or not. This was implemented by doubling the acceleration amplitude in x- and y-direction to replicate a more aggressive driving. A 30 s simulation was done with a rigid floater to compare to the previous one and as well identify any interesting time windows (Fig. 40 and 41). Overall a higher amount of fuel is present inside the FLVV during most of the simulation for the aggressive case compared to the normal case (Fig. 40). LCO was discovered during this case in contrast to the previous simulation (Fig. 41 and 42). The first appearance of LCO occur at the 56 s mark, corresponding to the rapid increase of fuel inside the FLVV identified previously, with an LCO amount of approximately 2 ml. The next increase of LCO occurs between the 68 s mark and the 73 s mark with an increase of 16 ml. This occurs when the car goes from a left turn straight into a sharp right turn. After the 73 s mark the LCO volume decrease as the car alternates between small left and right turns by finding its way back through the FLVV outlet hole.

![Figure 40: Volume of fuel inside the FLVV for the two simulations with a rigid floater. The solid black line represents simulation with acceleration input multiplied by two. Overall a higher fuel volume is obtained for the case with higher accelerations](image-url)
Figure 41: The amount of LCO inside the FLVV for a simulation between 54–61 s for aggressive driving (2 times larger acceleration vectors in X and Y) with a rigid floater.

Figure 42: A snapshot of the FLVV with the liquid trap during the aggressive simulation with rigid floater at time of peak LCO, the LCO is represented by the contours showing the cells that contain fuel.

Based on these findings simulations were performed with a moving floater for the time window 54 s–61 s with both the normal, refereed to as baseline, and aggressive accelerations. The volume of fuel inside the FLVV is increased for the aggressive case compared to the baseline case between 54 s and 59 s (Fig. 43). This can be related to the position of the floater that is at, or close to, its upper most position for a longer duration for the aggressive case compared to the baseline case (Fig. 44).

Figure 43: The amount of fuel inside the FLVV for a simulation with moving floater between 54–61 s for baseline (normal acceleration) and aggressive driving (2 times larger acceleration vectors in X and Y)
For the aggressive case LCO of 0.7 ml is obtained during this time window (fig. 45) which is approximately half of the LCO obtained for the rigid case in the same time window. During the increase of LCO the floater is observed to oscillate in its upper most position. For the baseline case still a small amount of 0.03 ml is obtained in contrast to the corresponding rigid case where no LCO was discovered.

4.2.3 Purge Resonance

To determine the possible influence purging has with regards to LCO two different purging simulations were conducted with a 50% duty cycle. The results of these are presented below.

Purging 2.5 kPa 50% Duty Cycle

The purging at 2.5 kPa and the baseline simulation results show the same amount of FLVV during the driving cycle from 54 s to 61 s from the Hällered test track (Fig. 46). The fuel inside the FLVV increases till 56 s and plateaus at around 55 ml till 58 s to later decrease and oscillate up and down.
The movement of the floater during purging at 2.5 kPa starts from top position (0 m) to be released and drop to bottom position at $-0.012 \text{ m}$ and then rise to the top again and oscillate up and down between 56 s and 57.5 s when purging is active (Fig. 47). The floater stays at top position for 0.5 s to later drop to bottom position and has some minor movement upwards. During the second purging period the floater stays at the bottom position most of the time. The sudden drop at around 58 s corresponds to the decreased fuel volume inside the FLVV at the same time (Fig. 46) the movement upwards after 59 s can be correlated to the slight increase in fuel in the FLVV.

**Purging 4 kPa 50% Duty Cycle**

The amount of fuel inside the FLVV during purging at 4 kPa corresponds well to the amount during baseline simulations. The amount of fuel increases rapidly to 42 ml and slowly increasing to 54 ml till 58 s and later decreasing and having some oscillations between 30 and 48 ml.
The movement of the floater during purging simulations at 4 kPa shows a release from the top position (0 m) and dropping to -0.012 m (Fig. 49). The floater start rising towards the top position again a bit after 55 s and start oscillating around -0.006 m up and down during the purging of the tank. The floater stabilizes at the top after the first purging session to later drop down to the bottom again. At the beginning of the second purge session the floater start rising again till -0.006 m to later drop down again. The floater stay at the bottom during most of the second purging session. In the same manner as results for the purging at 2.5 kPa can the amount of fuel be correlated to the movement of the FLVV.

**LCO with Purging**

The amount of LCO during purging at 2.5 kPa is around 0.7 ml whilst LCO at 4 kPa purging is around 0.4-0.35 ml and almost no LCO for the baseline case (Fig. 50). The amount of LCO increases during the same time period all three cases at around 56 s till 58 s to later plateau but the rate of LCO at that time period is the highest for purging at 2.5 kPa. The increasing LCO during that time period corresponds to the same time period at which the FLVV is at its top position oscillation up and down (Fig. 47 and 49) during the first purging session. The location of the liquid trapped inside can be seen to be consistent between the simulation (Fig. 51). As seen for case C the floater at its top most position forces liquid out into the liquid trap.
4.2.4 Fuel Level

The amount of fuel inside the FLVV at baseline plotted against 85 % of 60 l. The baseline peaks at around 60 ml at 58 s whilst the lower fuel level peaks at 36 ml just before time period 57 s (Fig. 52). The lower fuel level has a more constant fuel level in the being and after increasing it returns to a stable value whilst the baseline values have a rate of change during the simulation. The baseline has been initialized with no fuel inside the FLVV but the lower fuel level case starts with around 18 ml inside. The simulation with the lower fuel level does not give rise to floater movement upward after the initial drop at simulation start (Fig. 52 and 53). But the higher fuel level in base line keeps the floater at its top position as long as the fuel volume is above about 44 ml. No LCO was discovered with a lower fuel level.
4.3 Evaporation

The results from the amount of vapor change show that the evaporation increases linearly after the first few seconds without showing any major decrease in the rate of evaporation (Fig. 54). The evaporation rate is high in the first few seconds to later decrease and stabilize at a constant level giving a steady increase in vapor inside the domain. The vapor percentage is 20% at the beginning to increase to just below 21%.
5 Discussion

5.1 Numerical Method

Modelling set-up

As the VOF model is an interface capturing multiphase model the mesh resolution is of high importance in areas where the flow is dispersed. Since the flow is highly irregular it is not possible to refine certain parts of the domain thus some details will be lost outside the venting region due to large element size. However, inside the venting house the domain is fine enough to capture droplets of around 1 mm.

An interesting aspect with regards to the accuracy of the solver is the mass conservation at the interface between the overset mesh and background mesh due to the linear interpolation between the meshes. The need of interpolation is due to the fact that the overset mesh at its new position requires old variable values for the unsteady terms at the new position. This will essentially lead to non conservation which affects the pressure-correction equation. This will cause a mass imbalance and will contribute as a net mass source in the equation. There are two different options for reducing this mass imbalance but due to time limitations they have not been investigated in regards to added computational cost and the stability of the solver [22]. The overset mesh itself has shown to increase the overall computational cost with approximately four times, which is why the longer simulations performed were done with a rigid floater to save computational cost with the objective to find interesting time intervals.

The modelling of the closed floater at its top position is as stated earlier done by deactivating two prism layers overlapping each other on the floater and the upper walls. That distance is in fact a further 0.3 mm in a physical FLVV during an experimental study. This together with the fact that the part of the floater which seals the hole is manufactured in a flexible rubber with some deflection gives a rise to the numerical model not being fully representative of the physical FLVV. The flexible rubber will have a significant deformation especially during purge where the lower pressure sucks the floater and pushes it harder against the wall. This rubber sealing will also be a source absorbing most of the kinetic energy of the upper floater when coming in contact with the wall. This energy absorption is not modelled either instead as stated earlier a damping length is used which slows down the floater when the distance is lower than $3E^{-4}$.

Another interesting aspect of the modelling is that the bottom floater is not being modelled as a moving body which depending on conditions does effect the distance the upper floater can drop before hitting the bottom floater. If the bottom floater was to be modelled the $0.012 \, m$, distance the upper floater can travel, would actually vary thus effecting the closing performance of the upper floater especially in the case of purging. The bottom floater is not going to float just from the buoyancy force contribution thus it is supported by a small spring which together can make the bottom floater move and subsequently make the upper floaters lowest position change depending on the bottom floaters position. To be noted, a modelling of the bottom floater will not only introduce a new overset region with DFBI but also...
two extra zero gap walls. Thus with the resources at hand and the limitations of STAR-CCM+ in regards to scalability (simulation time to amount of cores) it is a wise course of action to not model the bottom floater as of now.

The purging of the fuel tank is reproduced using a pressure outlet with a negative pressure of a few kPa to evacuate the hydrocarbons inside the canister. This is not fully representative of the physical tank due to lack of evaporation modelling and thus no pressure build up prior to inducing purge. Thus it is unknown how the pressure build up effects the floater motion and whether it enhances the pumping effect and subsequently LCO introduced by the purging action (Fig. 47 and 49). This can be clearly seen during the first purging session. In order for the purge to affect the floater movement the floater has to be close to its top position so that the pressure drop acts as a suction effect through the small gap left before the floater closes completely. If the floater is close to its bottom position purging won’t be able to effect the floater translation as seen in the second purging session (Fig. 47 and 49).

An important factor effecting the amount of LCO is the contact angle which effects the surface tension. As described earlier the contact angle determines how the liquid sticks to the surface (in this case the upper floater) and thus how much of the liquid stuck on top of the upper floater finds its way into the liquid trap. The same can be said about the droplets inside the FLVV which according to Hu et. al [23] is significantly effectied by the contact angle especially during spreading and recoil process when hitting a surface. Thus a dynamic contact angle can be used for more accurate modelling. The contact angle of the fluid is modelled with a static preset value obtained from experiments. This will lead to a constant contact value even at times when the flow is moving in different directions along a surface. Dynamic modelling is a way to model the contact angle dependent on the velocity of the triple line [24]. One method available in STAR-CCM+ is the Kistler correlation which was not investigated due to limited time and resources.

Model Verification

The spatial and temporal sensitivity study approach was based on finding the suitable time-step for the coarsest mesh and then refine the mesh. This by keeping the CFL number constant to that was found to be sufficient, until an appropriate spatial resolution was found. Another approach that could have been used is to keep the CFL number at recommended values (max 0.5 for second order temporal discretization) and refine the mesh until it is fine enough to capture the wanted flow features. After that investigate if a bigger time-step can be used. This way the difference in the results due to mesh resolution will be minimized when comparing different time-steps. The reason for the chosen approach was time and resource limitations. A suggestion would be to investigate an even finer timestep for the concluded 3M mesh domain to see if it shows any significant impact on the results. The investigation of the repeatability of the simulation showed that for the parameters used the repeatability is good. However other parameters that are not as forgiving as volumes might show a greater deviation. This is due to the unsteadiness of the problem and needs to be considered depending on the accuracy that is required when using the method. The inner loop iterations were kept at five for all simulations and no
investigation regarding if the chosen mesh, 3M, and time-step could benefit from using fewer inner iterations has been made.

**Evaporation**

Even though LCO is an occurring phenomena in the simulations it does not necessarily mean that it will effect the performance of the canister. The performance is effected only if the liquid state finds its way to the canister. This based on the fact that the ratio of surface area to fuel amount will be large and thus the fuel inside the liquid trap will evaporate fast. This is something that is of interest to investigate further with CFD.

A brief investigation was conducted (Fig. 54) with a simplified tank (Fig. 37) due to numerical instabilities of the evaporation solver for a real tank. As can be seen the rate of evaporation is slow thus to be able to simulate a long enough period until reaching steady state might require weeks of simulation. The steady state pressure values could then be used as a pressure conditions inside the domain. Those resources are not available as of today. This evaporation model was later implemented on the real fuel tank (same settings and domain set-up) but due to numerical instabilities no results were concluded. As stated earlier evaporation could be implemented only in parts of the domain to reduce computational cost.

**Simulation Results**

As expected the fuel level inside the FLVV increases with an aggressive driving cycle (Fig. 40). By comparing the LCO during an aggressive driving condition with a rigid floater (Fig. 41) to the same driving condition with a moving floater one can conclude the importance of the floater. The maximum amount of fuel passing through to the liquid trap is 20 fold higher the maximum amount with a moving floater. This can be compared to the baseline case with normal driving conditions where almost no LCO was detected in rigid nor moving floater. One needs to bear in mind though that the aggressive driving conditions are somewhat exaggerated to be able to provoke LCO. Thus the simulated 7s is not necessarily the worst case scenario but only what has been detected to be the worst case scenario from a section of the test track.

The reason behind the higher LCO in the case of purging at 2.5 kPa compared to 4 kPa and baseline at 50% DC (Fig. 50) is that the floater in the 2.5 kPa case has a more frequent movement up and down (Fig. 47) compared to the 4 kPa case (Fig. 49). This induces a pumping effect and forces more fluid through. Even though the simulations show that the floater has a pumping effect during purge the consequences of the floater not being there is vital (Fig. 41). This is especially important in situations where the FLVV is submerged in liquid at around 60 ml and above (Fig. 40). The conducted simulations with 50% DC was based on assumptions of at what DC the floater would have a pumping behaviour. To ensure that the simulated DC is the worst case scenario in regards to pumping effect a thorough parametric study has to be done which due to resource limitations was not performed. One major factor effecting LCO is the fuel level inside the fuel tank. A fuel level of 85% compared to 95% not only shows a very low volume of fuel
inside the FLVV during the entire simulation (Fig. 52) but also no movement of the floater. Thus to use this method for investigation of LCO it is preferred to start with a fuel level of 95% and above.

All cases of pronounced LCO can further be investigated by introducing different baffles inside the tank to change the fluid sloshing behaviour. One can also take the re-positioning of the FLVV into consideration since the developed method can easily be used to readjust the position of the FLVV. By studying the flow structure and behaviour one can simply reorient the FLVV and thus position it in such way that the venting holes on the side and bottom are less likely to be hit by the fluid during different driving sequences such as turning, braking, accelerating etc.

One needs to take into consideration that the results obtained in this study do not include evaporation which can have a profound effect on the floater movement and behaviour of the fluid. For instance if the floater is at its top position sealing the outlet for a longer period of time the pressure build up could keep the floater attached even if the buoyant forces are not present due to the pressure difference compared to the venting region.

5.2 Experimental set-up and Validation

The experimental setup of the FLVV to validate the numerical setup of the overset mesh and DFBBI had some room for improvement to completely match the simulated domain. Some systematic errors and some random errors were present when conducting the experiments. The FLVV was glued to the container at the bottom but was not completely sealed leaving some small opening where the fluid could leak in. In the simulation the only place where the fluid can get into the FLVV was at the opening done for visual inspection of the floater. This induces some error to the amount of fluid inside the FLVV at different time level compared to the simulation which will effect at what time level the floater starts moving. The flow rate from the filler pipe was measured ocularly and timed giving another source of error. This value was later set as the inlet boundary condition as mass flow inlet. To reduce random errors in the experiments such as how well the fluid level or the floater position has been measured, more repeated measurements should be performed. The floater movement was also measured ocularly inducing further error to the experiment.

A physical difference between the simulation model and experimental FLVV is the guiding pins on the FLVV walls keeping the upper floater locked in X and Y axis and only keeping the movement in Z axis. The friction between the floater and the guiding pins is not being modelled thus one of the reasons for the gradient of the experimental data being different compared to the simulation (Fig. 38). The increased resistance due to friction makes the floater start and stop at a slower pace.

An error in the validation method in the numerical setup was the assumption of same contact angle for all three materials glass, rubber and Polyoximetylen (POM) inside the domain. Previously performed test at VCC showed a contact angle of 60° for water on POM which was used for glass and rubber as well due to limited time and resources. This will have a minor impact on the obtained simulation results and not fully representative of the physical setup. However the performed validation
displayed good agreement between the simulation and experiment. The difference between the two is partly due to errors in the experimental setup as described earlier.

5.3 Model Usage and Further Model Development

The high complexity of this problem containing a few moving rigid bodies as well as different physics such as sloshing and evaporation during driving makes the development of a single model too computationally expensive to be used in the industry. To be able to use a fully developed model with both floaters as moving bodies together with evaporation is not feasible as of today for early design study due to the high computational cost. A suggestion would be to simulate a case with evaporation and by studying the obtained parameters and physics evaluate how that will effect the simulations without evaporation. The results obtained from those simulation might also indicate if modelling of evaporation or both floaters moving not giving any significant impact on the end results simulated without evaporation and only one moving floater.
6 Conclusions

A CFD method has been developed with purpose to be able to investigate the performance of a venting valve inside a fuel tank. The method incorporates multiphase modeling, by use of the VOF-method, and Dynamic Fluid Body Interaction, in conjunction with a overset mesh to model the movement of the venting valve. Modelling of the upper floater as a moving body showed significant impact on the LCO. Therefore it is of high importance to include it as a moving body. The motion modeling has been validated with experiment by comparing the position of the floater with respect to time showing good agreement. Validation of the fuel sloshing inside the tank is to be conducted to be able to further strengthen the validity of the developed method.

The conducted simulations showed a substantially larger computational time when introducing overset mesh to the VOF simulation thus an optimization of the overset region together with the VOF is of high importance. By utilizing a rigid floater and reducing computational cost, longer driving periods can be simulated to highlight the most critical part of the driving. The isolated critical driving sequence can later be simulated with a moving floater.

The results showed that aggressive driving condition and high fuel level gave rise to increased LCO. It also showed that a fuel level at 85% and lower does not provoke LCO in the current cases. The purging of the fuel tank also gives rise to LCO when the floater moves up and down inducing a pumping effect but a further elaboration is to be done to investigate the most crucial purging conditions.
7 Future Work

The methodology developed to evaluate the performance of the FLVV has been verified and validated to some extent. By using the developed and manufactured 3D-printed SLA tank the final step of the validation study can be performed to validate the fuel sloshing behaviour inside the tank.

To be able to fully replicate the physics evaporation modelling has to be added to account for the created fuel vapors inside the tank and how they affect the behaviour of the floater. It is also recommended to replicate the purging phenomena by using a flow out inside of pressure outlet and investigate the difference. It is also of high importance to create an experimental rig specifically for validation of any developed evaporation model. A test scenario of acceleration with worst case scenario can be also be developed to be used as a basis for all simulations where the most vital movements of the vehicle are added instead of using the Hällered test track data.

Using other fuel tank where LCO has been reported to be present during driving can be used to further investigate the causes of it and how to prevent it.

A criteria for the model to be used is a low computational time especially with added evaporation modelling thus an optimization of the simulation setup to reduce computational cost is of interest.
8 Perspectives

At VCC the utilization of CFD is growing at a fast pace due to the higher costs of experimental testing especially in the fuel department where the main objective is reducing the emission from the fuel system and fulfilling the emissions regulations. The use of CFD also reduces the emissions during the development of a new part by reducing the production of physical test objects. Even though CFD could be utilized in this area one has to bear in mind the overall negative effect of an internal combustion engine using fossil fuel on the environment releasing greenhouse gases such as carbon dioxide. To reduce the costs of development of new parts during an early design stage CFD can act as a good CAE tool. As of today there are some limitation to CFD to completely replace experimental testing but in the future further development in the this area could lead to more advanced and effective modelling.
References


A Sensitivity Study Data

Data for parameters used to investigate first and second order temporal discretization scheme. The data are sampled for large volumes for these parameters thus insensitive for small differences in the results and should be interpreted with this in mind.

![Graph showing volume of fuel over time for first order discretization simulations](image1)

Figure 55: Fuel in the passive side of the fuel tank for the first order temporal discretization simulations

![Graph showing center of mass coordinate over time for first order discretization simulations](image2)

Figure 56: The center of mass coordinate of the fuel in the y-direction for the first order temporal discretization simulations

![Graph showing volume of fuel over time for second order discretization simulations](image3)

Figure 57: Fuel in the passive side of the fuel tank for the second order temporal discretization simulations
Figure 58: The center of mass coordinate of the fuel in the y-direction for the second order temporal discretization simulations

Figure 59: Plot of the average CFL number on the free surface for the simulations done with 2nd order temporal discretization
B  Repeatability Data

Data for three simulations with same setup used to investigate the repeatability of the used method.

Figure 60: Plot of the volume of fuel inside the FLVV for three simulations with the same set-up

Figure 61: The standard deviation for the volume of fuel inside the FLVV for the three simulations

C  Adaptive time-step data

Data for a comparison between simulation with an adaptive time-step and a constant time-step.

Figure 62: Plot of the volume of fuel inside the FLVV for simulation with adaptive time-step and a constant time-step
Figure 63: Plot of the time-step size for simulation with adaptive time-step and a constant time-step