CFD and Experimental Study of Refuelling and Venting a Fuel System

Aditya Naronikar, Linköping University
Anton Riström, Luleå University of Technology
CFD and Experimental Study of Refuelling and Venting a Fuel System

Aditya Naronikar, Linköping University
Anton Riström, Luleå University of Technology

Academic supervisor: Roland Gårdhagen
Industrial supervisors: Ehsan Yasari, Anders Pihl
Examiner: Matts Karlsson
Abstract

In 1999, California Air Resources Board (CARB) implemented a regulation that required all gasoline cars sold in California be fitted with an Onboard Refueling Vapor Recovery System (ORVR). The ORVR system is designed to prevent Volatile Organic Compounds (VOCs) from escaping into the atmosphere during refuelling by storing the gas vapours in a carbon canister. Due to the complex nature of the fuel system, making design changes could have large implications on the ORVR performance of the vehicle. It is therefore desirable to develop a CFD model that can predict the effects of design changes, thereby reducing the need to perform physical tests on each design iteration. This master thesis project was performed at the Fuel Systems department at Volvo Cars in order to help reduce project lead times and product development costs by incorporating CFD as a part of the fuel system development cycle. The CFD results obtained were validated through experimental tests that were also performed as part of this project.

In this master thesis project, a CFD model was developed to simulate the refuelling of gasoline for a California specification Volvo XC90 with an OPW-11B pump pistol. The model was set up in STAR-CCM+ using the Eulerian Volume of Fluid model for multiphase flow, the RANS realizable $k-\varepsilon$ turbulence model and the two layer all $y^+$ wall treatment. The effects of the carbon canister were modelled as a porous baffle interface in the simulations where viscous and inertial resistances of the porous media were adjusted to obtain a desired pressure drop across the canister. This method proved to be a suitable simplification for this study. The effects of evaporation as well as a chemical adsorption model for the carbon canister have been excluded from the project due to time limitations.

It was found that the CFD simulations were in good agreement with the experimental results, especially with respect to capturing the overall behaviour of the fuel system during refuelling. It was found that resolving the flow spatially (and temporally) in the filler pipe was a crucial part in ensuring solver stability. A pressure difference between experiment and simulation was also observed as a consequence of excluding evaporation from the CFD model.

After the CFD model had been verified and validated, changes to different parts of the fuel system were investigated to observe their effects on ORVR performance. These included changing the recirculation line diameter, changing the carbon canister properties and changing the angle of how the pump pistol was inserted into the capless unit. It was found that the recirculation line diameter is a very sensitive design parameter and increasing the diameter would result in fuel vapour leaking back out into the atmosphere. Similarly, increasing the back pressure by swapping to a different carbon canister would result in the leakage of fuel vapour. On the other hand, insignificant changes in system behaviour were observed when the fuel pistol angle was changed.
Acknowledgements

This master thesis project was performed at the fuel storage group of the Fuel System department at Volvo Car Corporation in Gothenburg during the spring of 2019. We would like to thank our supervisors Ehsan Yasari and Anders Pihl for counselling and guidance during the thesis work. They have been instrumental in helping us both understand the technical aspects of this project as well as solve day-to-day problems and issues we faced - from getting set up at Volvo Cars all the way to the final stages of the project.

We would like to thank Christofer Karlberg, Advanced Engineering Leader at the fuel systems department, for his efforts in guiding and helping us in progressing with our project. We appreciate his commitment and dedication in making our stay at Volvo Cars a memorable experience. We would also like to thank Anders Aronsson for always offering help and valuable input with his great expertise in material science, physics, chemistry and overall knowledge of the fuel system.

During the spring, several other master thesis projects were performed at the Fuel Systems department. Sharing information between the groups have ensured the best possible results for all thesis works. Therefore, we would like to thank Jakob Dahlqvist, Gulled Faisal, Sadegh Fattahi, Philip Månsson, Kavyaa Somasundaram, Oscar Sundell and Hassan Zafar for their input and thoughts whenever we needed them.

Aditya would additionally like to thank Roland Gårdhagen and Matts Karlsson from Linköping University for supervising and examining the thesis work, respectively. Aditya would like to acknowledge Roland for his constant guidance and support not only during the master thesis project, but through the entirety of the master’s studies. A sincere thank you goes to Marcus Lång, Aditya’s study partner, colleague and friend from his time at university, with whom he has spent countless hours and late nights working on coursework. Aditya would like to thank Marcus for being there even through the most turbulent of times.

Anton would like to thank Gunnar Hellström from Luleå University of Technology for supervising and examining his work. Additionally, Anton would like to thank his friends from his time at the university with whom he has shared his best and worst moments. Thank you for fantastic camaraderie and support Johannes Ekelund, Jesper Haglöf, Victor Hasselfors, Hanna Marklund, Torkel Skoog and Christoffer Stenberg.

Aditya Naronikar and Anton Riström
Göteborg
June 2019
## Nomenclature

### Abbreviations and Acronyms

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Meaning</th>
</tr>
</thead>
<tbody>
<tr>
<td>BDF</td>
<td>Backward Differentiation Formula</td>
</tr>
<tr>
<td>CAE</td>
<td>Computer Aided Engineering</td>
</tr>
<tr>
<td>CARB</td>
<td>California Air Resources Board</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CV</td>
<td>Control Volume</td>
</tr>
<tr>
<td>DDES</td>
<td>Delayed Detached Eddy Simulation</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>EVAP</td>
<td>Evaporative Emissions</td>
</tr>
<tr>
<td>FDM</td>
<td>Fuel Delivery Module</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Method</td>
</tr>
<tr>
<td>FLVV</td>
<td>Fill Limit Vent Valve</td>
</tr>
<tr>
<td>FVM</td>
<td>Finite Volume Method</td>
</tr>
<tr>
<td>HDPE</td>
<td>High Density Polyethylene</td>
</tr>
<tr>
<td>HRIC</td>
<td>High Resolution Interface Capturing</td>
</tr>
<tr>
<td>HRS</td>
<td>High Resolution Scheme</td>
</tr>
<tr>
<td>ICV</td>
<td>Inlet Check Valve</td>
</tr>
<tr>
<td>IDDES</td>
<td>Improved Delayed Detached Eddy Simulation</td>
</tr>
<tr>
<td>LDP</td>
<td>Leak Detection Pump</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>NVD</td>
<td>Normalized Variable Diagram</td>
</tr>
<tr>
<td>ORVR</td>
<td>Onboard Refuelling Vapour Recovery</td>
</tr>
<tr>
<td>PHEV</td>
<td>Plug-in Hybrid Electric Vehicle</td>
</tr>
<tr>
<td>PSO</td>
<td>Premature Shut Off</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds Averaged Navier Stokes</td>
</tr>
<tr>
<td>RKE</td>
<td>Realizable K-Epsilon</td>
</tr>
<tr>
<td>ROW</td>
<td>Rest of the World</td>
</tr>
<tr>
<td>RST</td>
<td>Reynolds Stress Tensor</td>
</tr>
<tr>
<td>RVP</td>
<td>Reid Vapour Pressure</td>
</tr>
<tr>
<td>SIMPLE</td>
<td>Semi-Implicit Method for Pressure Linked Equations</td>
</tr>
<tr>
<td>SPA</td>
<td>Scalable Platform Architecture</td>
</tr>
<tr>
<td>SST</td>
<td>Shear Stress Transport</td>
</tr>
<tr>
<td>URANS</td>
<td>Unsteady Reynolds Averaged Navier Stokes</td>
</tr>
<tr>
<td>VCC</td>
<td>Volvo Car Corporation</td>
</tr>
<tr>
<td>VOC</td>
<td>Volatile Organic Compounds</td>
</tr>
<tr>
<td>VOF</td>
<td>Volume of Fluid</td>
</tr>
<tr>
<td>WMLES</td>
<td>Wall Modelled Large Eddy Simulation</td>
</tr>
</tbody>
</table>
## Latin Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$A$</td>
<td>Surface Area</td>
<td>$[m^2]$</td>
</tr>
<tr>
<td>$p$</td>
<td>Static Pressure</td>
<td>$[Pa]$</td>
</tr>
<tr>
<td>$u$</td>
<td>Velocity in X direction</td>
<td>$[ms^{-1}]$</td>
</tr>
<tr>
<td>$v$</td>
<td>Velocity in Y direction</td>
<td>$[ms^{-1}]$</td>
</tr>
<tr>
<td>$w$</td>
<td>Velocity in Z direction</td>
<td>$[ms^{-1}]$</td>
</tr>
<tr>
<td>$V$</td>
<td>Velocity $(u, v, w)$</td>
<td>$[ms^{-1}]$</td>
</tr>
<tr>
<td>$e$</td>
<td>Internal energy due to molecular motion</td>
<td>$[J]$</td>
</tr>
<tr>
<td>$\dot{q}$</td>
<td>Heat flux due to thermal conduction</td>
<td>$[Wm^{-2}]$</td>
</tr>
<tr>
<td>$k_c$</td>
<td>Thermal conductivity</td>
<td>$[Wm^{-1}K^{-1}]$</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature</td>
<td>$[K]$</td>
</tr>
<tr>
<td>$n$</td>
<td>Outward normal vector</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$t$</td>
<td>Time</td>
<td>$[s]$</td>
</tr>
<tr>
<td>$S_\phi$</td>
<td>Source term</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$U$</td>
<td>Freestream velocity</td>
<td>$[ms^{-1}]$</td>
</tr>
<tr>
<td>$L$</td>
<td>Characteristic length</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$\bar{u}, \bar{v}, \bar{w}$</td>
<td>Average components of velocity in X, Y and Z</td>
<td>$[ms^{-1}]$</td>
</tr>
<tr>
<td>$\bar{p}$</td>
<td>Average component of pressure</td>
<td>$[Pa]$</td>
</tr>
<tr>
<td>$u', v', w'$</td>
<td>Fluctuating components of velocity in X, Y and Z</td>
<td>$[ms^{-1}]$</td>
</tr>
<tr>
<td>$y^+$</td>
<td>Dimensionless wall-distance</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$u_f$</td>
<td>Friction velocity</td>
<td>$[ms^{-1}]$</td>
</tr>
<tr>
<td>$u^+$</td>
<td>Dimensionless velocity</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$k$</td>
<td>Turbulent kinetic energy</td>
<td>$[m^2s^{-2}]$</td>
</tr>
<tr>
<td>$C_{\mu}$</td>
<td>RKE model coefficient</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$f_{\mu}$</td>
<td>RKE model damping function</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$V$</td>
<td>Volume</td>
<td>$[m^3]$</td>
</tr>
<tr>
<td>$N$</td>
<td>Number of phases</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$\bar{v}$</td>
<td>Mass-averaged velocity for a multi-phase mixture</td>
<td>$[ms^{-1}]$</td>
</tr>
<tr>
<td>$C_\alpha$</td>
<td>Sharpening factor</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$d_f$</td>
<td>Diameter of the filler pipe</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$l_p$</td>
<td>Position in the recirculation line</td>
<td>$[m]$</td>
</tr>
<tr>
<td>$L_r$</td>
<td>Recirculation line total length</td>
<td>$[m]$</td>
</tr>
</tbody>
</table>
### Greek Letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Units</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\rho$</td>
<td>Density</td>
<td>$[kgm^{-3}]$</td>
</tr>
<tr>
<td>$\mu$</td>
<td>Dynamic viscosity</td>
<td>$[kgm^{-1}s^{-1}]$</td>
</tr>
<tr>
<td>$\nu$</td>
<td>Kinematic Viscosity</td>
<td>$[m^2s^{-1}]$</td>
</tr>
<tr>
<td>$\phi$</td>
<td>Scalar field function</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$\Gamma$</td>
<td>Diffusion coefficient</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>Turbulent dissipation rate</td>
<td>$[m^2s^{-3}]$</td>
</tr>
<tr>
<td>$\alpha_i$</td>
<td>Volume fraction of phase $i$</td>
<td>$[-]$</td>
</tr>
<tr>
<td>$\theta$</td>
<td>Contact angle</td>
<td>$[degree]$</td>
</tr>
</tbody>
</table>
Contents

1 Introduction 1
  1.1 Background .................................................. 1
  1.2 Purpose of the study ........................................... 2
  1.3 Aims and Objectives ........................................... 2
  1.4 Limitations .................................................... 3

2 Theory 4
  2.1 The Automotive Fuel System ................................. 4
    2.1.1 Fuel Storage System ...................................... 5
    2.1.2 EVAP system .............................................. 7
  2.2 Computational Fluid Dynamics ............................... 7
    2.2.1 Governing equations ...................................... 7
    2.2.2 Models of the Flow ....................................... 9
    2.2.3 Turbulence Modelling .................................... 10
    2.2.4 RANS Turbulence Modelling ............................... 11
    2.2.5 Law of the Wall and Wall Treatment Approaches ... 12
    2.2.6 Realizable $k - \varepsilon$ Model ....................... 14
    2.2.7 Multiphase Modelling .................................... 15
  2.3 Literature Review/Previous Work ........................... 18
    2.3.1 Experimental Investigations/Flow Physics ............. 18
    2.3.2 Previous CFD Research .................................. 19

3 Method 21
  3.1 CFD Study .................................................... 21
    3.1.1 Preparation of the OPW 111B CAD Model ................ 22
    3.1.2 Surface Preparation and CAD Clean-up .................. 23
    3.1.3 Boundary Conditions ..................................... 25
    3.1.4 Solver Setup ............................................. 27
    3.1.5 Meshing Strategy and Verification Study ................ 28
  3.2 Validation Study ............................................. 36
    3.2.1 Experimental Setup ...................................... 36
    3.2.2 Experimental Procedure .................................. 37
  3.3 Investigating Changes to the Fuel System .................. 38
    3.3.1 Recirculation Line Diameter ............................. 38
    3.3.2 Pump Pistol Nozzle Angle ................................ 39
    3.3.3 Carbon Canister Pressure Drop .......................... 40

4 Results 41
  4.1 Experimental Results and Comparison to CFD ............. 41
  4.2 Flow Phenomena Observed from CFD .......................... 43
  4.3 Design Change Investigation ................................ 46
    4.3.1 Recirculation Line Geometry .............................. 46
    4.3.2 Change in Fuel Pistol Angle ............................. 48
    4.3.3 Changing the Canister .................................... 49
## 5 Discussion

- **5.1 Solver Setup** ................................. 51
- **5.2 Model Validation** ......................... 52
- **5.3 Flow Physics** .............................. 52
  - **5.3.1 Top fill case** ......................... 52
  - **5.3.2 Bottom fill case** ..................... 53
- **5.4 Design Changes** ......................... 54
  - **5.4.1 Recirculation Line** ................. 54
  - **5.4.2 Nozzle Angle** ....................... 55
  - **5.4.3 Canister Pressure** ................. 55
- **5.5 General Comments** ...................... 55

## 6 Future Work ................................. 57

## 7 Conclusion .................................. 58
1 Introduction

1.1 Background

In times where social responsibility, sustainability and climate awareness are important factors, the legal requirements and regulations on automotive emissions are getting stricter and car manufacturers have to follow. One such emission particularly with gasoline vehicles is that of volatile organic compounds (VOC) escaping into the atmosphere during the refuelling process.

In order to reduce these VOC emissions, the California Air Resources Board (CARB) implemented a regulation in August 1999 that all gasoline cars sold in California be fitted with an ‘Onboard Refuelling Vapor Recovery System’, or ORVR, to capture the VOCs and store them in the carbon canister instead of allowing them to escape into the environment. These VOC reductions are made possible through the ‘Evaporative Emission Controls for On-Road Motor Vehicles’ program which contains regulations and standard test procedures [1, 2] that the vehicle must satisfy to be able to be sold in the market. The regulations were last amended in September 2015 and state that a maximum of 0.2 grams of vapour can be emitted for every gallon of gasoline refuelled. [2]

In the ORVR system, the hydrocarbons are adsorbed by the carbon canister which contains a bed of carbon pellets. The hydrocarbons are afterwards ‘purged’ through the carbon canister and are sucked into the intake manifold where they are mixed with fresh incoming air and thereafter burnt in the engine. This benefits fuel efficiency while avoiding the previously mentioned VOC emissions.

A Computer Aided Engineering (CAE) model that can simulate and predict the effects of design changes and their effects on ORVR system performance would significantly reduce the need to perform physical tests on each design change investigation. This master thesis project was performed at the Fuel Systems department at Volvo Car Corporation (VCC) in order to help reduce project lead times and product development costs by incorporating CFD as a part of the fuel system development cycle.

In 2018, Volvo Car Corporation passed their previous yearly sales volume and sold around 650 000 cars worldwide with 15% of these sales being represented by the United States, second only to China where 20% of the sales were reported. [3] The US market therefore becomes very important for the company thereby requiring them to invest in R&D to develop the US market cars, even if the certification regulations are quite different from the rest of the world (ROW). An important point to be noted here is that VCC does not sell any diesel vehicles in the US market - the sales are from gasoline vehicles and PHEV (Plug-in Hybrid Electric Vehicles) only. Any mention of 'fuel' in this report henceforth refers to gasoline, unless otherwise stated. One of the most significant design changes to be included in the US market cars when it comes to the fuel system is the inclusion of the ORVR system and its supporting sub-systems.

The ideal refuelling process can be divided into a few general steps. These are:
1. Open external filler door
2. Open filler cap, skipped if there is no cap
3. Insert the pump pistol
4. Refuel and stop at automatic shut-off
5. Withdraw pump pistol
6. Close filler cap, skipped if there is no cap
7. Close filler door

1.2 Purpose of the study

At the time, the Fuel System department at VCC had a working ORVR system that had been designed with the aid of extensive experimental testing. However, the exact sensitivities of the design parameters on the performance of the ORVR system were unknown. If any design change were to be investigated, physical experiments were performed which in turn involved the manufacture of special parts, a test rig as well as the cost of time and labour - all of which resulted in increased product development lead times and costs. From a longer term perspective, this project aimed to increase the use and reliability of Computer Aided Engineering (CAE) to investigate design changes in the Fuel System department, thereby hoping to reduce lead times and overall development costs.

1.3 Aims and Objectives

The overall goal of this master thesis project was to establish, verify and validate a Computational Fluid Dynamics (CFD) method using the commercial CFD solver STAR-CCM+ to simulate the refuelling process on a US market Scalable Platform Architecture (SPA) platform automobile fuel system that included the ORVR system. The SPA platform includes the Volvo XC60 and XC90, S60 and S90 and the V60 and the V90 models (including the cross-country variants).

The objectives of the study were identified as follows:

- To develop a CFD method using a RANS turbulence model to reliably and accurately simulate the refuelling process for the Volvo Cars SPA platform fuel system.
- Perform experimental tests on the refuelling procedure to validate the results obtained from CFD.
- Compare and contrast the flow physics behaviour observed between CFD and experiments, and provide suggestions for future work to improve the CFD model.
- If time permits, investigate the effect and sensitivities of changing selected design parameters by performing CFD simulations with the previously established model. The design parameters of interest include recirculation line diameter, fuel pistol insertion angle and canister back pressure.
1.4 Limitations

- Evaporation has not been included in the CFD study. It is well known that the evaporation of fuel has a significant effect on the refuelling process but has been neglected due to the increase in computational cost associated with evaporation modelling. Attempts were made to investigate an evaporation model but the chemical properties of gasoline and its various components as well as its mixing with air proved to be too complicated to include within the scope and time frame of this thesis work.

- The opening and closing of the inlet check valve, the roll over valves and the fill limit vent valve were not modelled. If the movement of valves were to be modelled, it would involve the use of suitable motion capturing meshing techniques such as overset mesh, which would increase both the complexity and the computational cost of the CFD model.

- The physics involved with the automatic shut-off of the fuel pistol have not been included. This would involve the implementation of the movement of the floater in the fill limit vent valve as well as some sort of modelling of the pressure sensor in the fuel pistol, both of which were deemed to be outside the scope of this thesis work.

- The model to be set up must be able to produce useful results within a simulation run time corresponding to 5 days on 200 cores. This is to ensure that the CAE tool can actually save time over experimental testing, based on the available resources (at the time) at the Fuel System department at Volvo Car Corporation.
2 Theory

2.1 The Automotive Fuel System

The fuel system consists of a number of components that can be divided into two major groups, fuel storage and EVAP system. Fuel storage can be divided into different parts, all which play an important role in refuelling and ORVR-performance. Figure 1 shows the 3D CAD model assembly of the different components that together form the fuel system on the car investigated in this study, the Volvo XC90.

In order to better understand the various components and their functions, Figure 2 shows a simplified, schematic view of the various components that make up the fuel system.
Where,

1. Fuel pistol and nozzle assembly
2. Fuel filler neck, includes capless filler inlet
3. Filler pipe
4. Inlet Check Valve (ICV)
5. Saddle design fuel tank
6. Fill Limit Vent Valve (FLVV)
7. EVAP (Evaporative Emissions) Line
8. Carbon canister
9. Purge line, goes to intake manifold
10. Leak detection pump
11. Clean air, released into environment
12. Recirculation/leak detection line
13. Fuel feed line; from Fuel Delivery Module (FDM) in tank to engine

2.1.1 Fuel Storage System

Capless Filler Unit
The capless filler unit is the part that comes in contact with the fuel pistol nozzle and is found when the refuelling door is opened. The capless unit replaces the traditional filler cap and allows the user to insert the pump pistol without having to unscrew the filler cap. In addition, the capless unit has built-in features which prevents the customer from filling the car with the wrong fuel. The main reason why the capless unit affects the ORVR-performance has to do with its geometry and how it holds the fuel nozzle which distributes the fuel into the filler neck. It has been shown that the angle of which the fuel jet enters the filler neck has a great impact on how the pattern of the spray moves through the filler pipe and into the fuel tank. [4]

Filler Pipe
The filler neck and filler pipe transport the fuel from the pump pistol into the tank. The design of the filler neck and pipe is crucial with regard to spit back and premature shut-off (PSO), both of which are unacceptable from a customer satisfaction point-of-view. Spit back refers to the condition where fuel is forced back up the filler pipe and spits out of the capless filler unit and onto the hands of the person refuelling the car. PSO is the phenomenon where automatic shut-off of the fuel flow though the pistol is triggered even though the fuel level in the tank is not at maximum capacity. The geometry of the first bend is crucial in how the fuel propagates throughout the filler pipe and hence the refuelling performance. [4]

Inlet Check Valve
Before the fuel enters the fuel tank, it encounters the inlet check valve (ICV). The function of the ICV is to let fuel into the tank in a controlled and secure way. It also prevents fuel from travelling back up the fuel filler pipe which helps in avoiding PSO and spit back.
The ICV is held closed by a relatively weak spring and hinge mechanism.

**Fuel Tank**
The fuel tank is the container that holds the fuel for the pump to deliver to the engine. The tank also holds a number of internal parts such as the fuel delivery module (FDM), the vapour venting valve, fuel level float indicators and baffles plates, which are components which reduce the effects of sloshing. The saddle type fuel tank in the SPA platform cars is generally described as having two halves that are split at the saddle. The half that houses the FDM, is termed as the active side, whereas the other half is called the passive side. The active side also houses the ICV and is the part of the tank where fuel flows in from the filler pipe.

**Fill Limit Vent Valve**
The function of the fill limit vent valve (FLVV) is to indirectly control fuel flow shut-off after the tank has reached its maximum volume capacity. As the fuel level approaches full tank, gasoline will rush through the ducts in the FLVV, carrying a moving check valve up due to buoyancy and immediately cutting off the escape route for the fuel vapours out of the fuel tank. This in turn causes a sharp increase in pressure in the fuel system which is sensed by the pump pistol and the flow is automatically shut off. From an ORVR perspective, the FLVV also serves to protect the carbon canister from liquid fuel carrying over from the fuel tank into EVAP line.

**Recirculation/Leak Detection Line**
The recirculation line, or leak detection line, is a stainless steel pipe that runs from the filler neck to the tank and has two functions. During refuelling, the fuel flowing into the tank displaces the air and fuel vapour that was previously inside the tank. These gases flow into the LCO box and subsequently the carbon canister through the EVAP lines. The recirculation line provides a alternate path for the gases to flow by allowing them to recirculate between the tank and the filler neck (where they flow down back to the tank again through the filler pipe). This helps reduce the mass flow into the canister, thereby reducing canister loading, which is highly desirable for fuel system design. Canister loading is the phenomenon where the hydrocarbons in the gases are adsorbed onto the activated carbon in the canister.

The recirculation line serves as a leak detection line at all other times except during refuelling. Legal requirements state that leaks must be identified in the entire fuel system, including both the EVAP lines and fuel storage system. To achieve this, a leak detection pump is connected to the carbon canister which in turn is connected to both the fuel tank and to the purge lines leading to the engine. When the car is turned off after driving, the pump will start sucking air out of the fuel system, through the carbon canister, creating a lower pressure in the entire fuel system. The pump can sense this lower pressure and diagnose whether there is a leak or not.
2.1.2 EVAP system

The EVAP system, or evaporative emissions system, monitors the leak detection and purging of the carbon canister. The system is mostly concerned with the transport of evaporated fuel vapour from the gasoline in the fuel tank. The EVAP system can be divided into the following key components.

**EVAP and Purge Lines**
The EVAP and purge lines are lines leading from the tank to the carbon canister and from the carbon canister to the intake manifold at the engine, respectively. It is through these lines that the VOCs travel and it is crucial that there is no leak in the system due to legal requirements.

**Carbon Canister**
Vapours from the tank which escape during refuelling are carried through the EVAP lines to the carbon canister. The carbon canister is a plastic shell containing chambers full of carbon pellets. This carbon captures hydrocarbons in the VOCs and allows clean air to pass through into the atmosphere. When the car is running, the air flow in the canister can be reversed using the suction from the intake manifold and the canister is ‘purged’. The hydrocarbons are sucked into the intake manifold through the purge line and then burnt during combustion.

**Purge valve**
The purge valve is responsible for opening and closing the purge line leading from the carbon canister to the engine. When the purge valve opens, fuel vapours rush into the engine due to the lower pressure at the intake manifold.

### 2.2 Computational Fluid Dynamics

Computational Fluid Dynamics involves the implementation of numerical methods to solve the governing equations of fluid flow in order to predict the behaviour of a fluid in motion. The applications of CFD are numerous, ranging from vehicle and aerospace engineering to weather prediction to blood-flow pattern computations in the human heart. CFD therefore becomes an integral CAE tool where the common goal irrespective of the industry is to simulate fluid flow conditions that would otherwise be expensive, if not impossible to replicate through physical testing.

The development of CFD codes and solvers over the years has been directly dependant on the availability of affordable high-power computing solutions. This is primarily due to the complexity of the numerical solution procedures involved in solving the governing equations of fluid flow.

#### 2.2.1 Governing equations

There are two main approaches employed in CFD to be able to track and visualize a moving fluid. The first approach is to follow the fluid particles as they move through the continuum in space and time. This method is called the Lagrangian approach. The
second method, called the Eulerian approach, is to consider the change in fluid properties in a fluid element at a fixed position in space and time. The fluid domain is therefore divided into a number of fluid elements and the governing equations are solved for each element. The Eulerian approach is more commonly used than the former and is also the flow model used in this study. The governing equations described below are therefore based on the Eulerian approach.

The governing equations of fluid flow are based on the following three physical principles:

- Conservation of mass
- Newton’s second law
- Conservation of energy

The conservation of mass through a fluid defines the mass balance for a given fluid element such that the rate of increase of mass in the fluid element is equal to the net rate of flow into the fluid element. This leads to the unsteady, three-dimensional continuity equation as given in equation 1.

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

Equation 1 is the differential, conservation form of the continuity equation and includes the effects of compressibility. However, as a rule of thumb, compressibility effects in the domain can be excluded when the Mach number of the flow does not exceed a value of 0.3. Mach number is the ratio of the velocity of the flow to the local speed of sound. This implies that density \( \rho \) is constant in space and time and the continuity equation therefore reduces to:

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{2}
\]

The second physical principle, Newton’s second law, gives the momentum equations in the X, Y and Z directions. The underlying principle is that the rate of change of momentum of a fluid element is equal to the sum of the forces of the fluid element. These forces can be further categorized as surface forces and body forces, where pressure forces and viscous forces constitute surface forces and and gravity forces, centrifugal force and electromagnetic force are body forces.

With these physical principles and neglecting body forces, the momentum equations in X, Y and Z can be derived for a given fluid element as given in equations 3, 4 and 5 respectively. These three equations are known as the Navier-Stokes equations, in the honour M. Navier and G. Stokes, both of whom obtained these equations independently in the nineteenth century. It must be noted that equations 3, 4 and 5 represent the Navier-Stokes equations for an incompressible, Newtonian fluid with constant viscosity.

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \tag{3}
\]

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \tag{4}
\]
\[
\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \nu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right)
\] (5)

The energy equation is based on the conservation of energy, i.e. the first law of thermodynamics, which states that the rate of change of energy of a fluid element is equal to net rate of heat added to the fluid element plus the net rate of work done on the fluid element. When this principle is applied to an infinitesimal fluid element, the energy equation for an incompressible, Newtonian fluid can be derived in the conservation form in terms of internal energy as described in equation 6.

\[
\frac{\partial (\rho e)}{\partial t} + \nabla \cdot (\rho e V) = \rho \dot{q} + \frac{\partial}{\partial x} \left( k_e \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( k_e \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left( k_e \frac{\partial T}{\partial z} \right) + \\
\mu \left[ 2 \left( \frac{\partial u}{\partial x} \right)^2 + 2 \left( \frac{\partial v}{\partial y} \right)^2 + 2 \left( \frac{\partial w}{\partial z} \right)^2 \right]
\] (6)

2.2.2 Models of the Flow

The governing equations described in section 2.2.1 must be solved at all locations in the flow domain over time to be able to obtain a solution for the given flow problem. In this study, this is accomplished by using the finite volume method (FVM), where the entire domain is divided into a number of discrete control volumes (CV). In the FVM, the numerical method implementation assumes that the nodal points are at the centres of each CV, as opposed to the finite element method (FEM) where the nodal points form the corner vertices of each element.

The governing equations can all be described by the general transport equation, which represents the transport of a scalar property \( \phi \) through the domain of fluid flow. The general transport equation in the differential form is then integrated over a CV using Gauss’s divergence theorem to obtain the integral form as follows: [6]

\[
\frac{\partial}{\partial t} \left( \int_V (\rho \phi) dV \right) + \int_A \mathbf{n} \cdot (\rho \mathbf{V} \phi) dA = \int_A \mathbf{n} \cdot (\Gamma \nabla \phi) dA + \int_V S_\phi dV
\] (7)

This integral form of the transport equation is then used to obtain the discretized set of governing equations. Discretization allows for the application of the governing equations to solve for the flow field at each CV in the whole flow domain. The discretization methods, or schemes, directly affect the accuracy of the solution such that higher order schemes imply higher accuracy at the cost of increased computational time. [6] The discretized equations can be solved either using a segregated flow solver or a coupled flow solver.

The segregated flow solver, which is the solver implemented in this study, solves the integral forms of the governing equations sequentially one after the other for the field variables
The segregated flow solver makes use of a pressure-velocity coupling algorithm to be able to handle the non-linearity of the governing equations while ensuring that the calculated velocity and pressure fields satisfy the continuity and momentum equations. On the other hand, the coupled flow solver solves all the integral forms of the equations simultaneously. Segregated flow solvers result in comparatively lesser costs in terms of computing power and solution times and was the solver chosen for the current study.

The pressure-velocity coupling used in STAR-CCM+ v12.06.010 is a Rhie-Chow interpolation based pressure-velocity coupling combined with a SIMPLE-type algorithm on a collocated grid. A collocated grid implies that all the flow field variables are all stored at the same locations, as opposed to a staggered grid where the scalar quantities are stored at the centres of the CVs and the velocity variables are stored at the CV faces.

2.2.3 Turbulence Modelling

Almost all engineering fluid dynamics problems observe seemingly random, chaotic variations in the fluid flow motion. This irregular state of fluid flow motion is termed as turbulence. Modelling the turbulent behaviour of the fluid accurately is therefore one of the most significant challenges in the CFD field. The most common method of defining whether a given flow is laminar or turbulent is by calculating the flow’s Reynolds number \( Re \), which is a non dimensional parameter that represents the ratio of inertial forces to viscous forces in the fluid.

\[
Re = \frac{\rho U L}{\mu}
\]  

Where, \( L \) is the characteristic length of the object/geometry whose flow characteristics are to be studied. Depending on the type of flow (internal or external) and the shape of the geometry, there is a critical Reynolds number at which the flow is expected to undergo transition from the laminar state to the turbulent regime. For internal pipe flow, such as the flow of fuel through the filler pipe, the characteristic length \( L \) is the inner diameter of the pipe and the corresponding critical \( Re \) is approximately 4000.

There have been several turbulence models developed over the years, with each new model improving upon the previous to improve the accuracy and robustness of CFD codes in general. However, there has so far not been a single turbulence model that is universally applicable to all types of flow problems. Therefore, in any CFD problem, it is up to the user to carefully examine the flow problem at hand and make an informed decision when selecting a turbulence modelling approach. Turbulence models can be broadly classified as follows:

- Reynolds Averaged Navier Stokes (RANS) Models
- Scale Resolving Simulations

Figure 3 shows where the different turbulence models stand in terms of turbulence modelling/resolution and computational cost.
2.2.4 RANS Turbulence Modelling

As described previously, turbulence involves random variations of field variables in the flow. The field variable $\phi$ can be decomposed into a steady, mean value $\bar{\phi}$ and a fluctuating component $\phi'$, as shown in equation (9).

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

This is called the Reynolds decomposition and forms the base for the Reynolds Averaged Navier Stokes (RANS) turbulence modelling approach. The mean or time-averaged field variable is defined as shown in equation (10).

$$\bar{\phi} = \frac{1}{\Delta t} \int_{0}^{\Delta t} \phi(t) \, dt$$

The decomposed field variables are inserted into the Navier-Stokes equations to obtain the time-averaged Navier-Stokes equations, also known as the RANS equations. Equations (11), (12) and (13) represent the RANS equations in X, Y and Z directions, respectively.

$$\frac{\partial \pi}{\partial t} + \text{div}(\pi \nabla) = -\frac{1}{\rho} \frac{\partial \pi}{\partial x} + \nu \text{div}(\nabla \pi) + \frac{1}{\rho} \left[ \frac{\partial (-\rho u'^2)}{\partial x} + \frac{\partial (-\rho u' v')}{\partial y} + \frac{\partial (-\rho u' w')}{\partial z} \right]$$

$$\frac{\partial \pi}{\partial t} + \text{div}(\pi \nabla) = -\frac{1}{\rho} \frac{\partial \pi}{\partial y} + \nu \text{div}(\nabla \pi) + \frac{1}{\rho} \left[ \frac{\partial (-\rho v'^2)}{\partial x} + \frac{\partial (-\rho u' w')}{\partial y} + \frac{\partial (-\rho v' w')}{\partial z} \right]$$

$$\frac{\partial \pi}{\partial t} + \text{div}(\pi \nabla) = -\frac{1}{\rho} \frac{\partial \pi}{\partial z} + \nu \text{div}(\nabla \pi) + \frac{1}{\rho} \left[ \frac{\partial (-\rho w'^2)}{\partial x} + \frac{\partial (-\rho u' w')}{\partial y} + \frac{\partial (-\rho w' w')}{\partial z} \right]$$

Figure 3: Examples of turbulence models and the trade-off between resolution and computing cost
Where, $\overline{u^2}$, $\overline{v^2}$, $\overline{w^2}$, $\overline{uv}$, $\overline{vw}$, $\overline{uw}$ are the extra terms introduced from the time-averaging and correspond to six additional stresses, called Reynolds stresses. They are together defined in a 3x3 symmetric tensor known as the Reynolds stress tensor (RST). This results in an extra unknown variable (the RST) versus the number of equations available for solving (i.e. the RANS equations). This is the closure problem associated with the RANS equations. In order to address this issue, the RANS turbulence modelling approach makes use of the Boussinesq hypothesis which states that there is an analogy between viscous stresses and Reynolds stresses. This allows for the introduction of the eddy viscosity concept, a ‘pseudo’ viscosity, that would model the Reynolds stresses. Therefore, the Reynolds stresses can be expressed as functions of the mean rates of deformation (and therefore velocity components) with the eddy (turbulent) viscosity as the proportionality constant. This effectively eliminates the unknown Reynolds stress tensor from the system of equations. [10]

2.2.5 Law of the Wall and Wall Treatment Approaches

Geometrical walls in the flow domain have a significant effect on the flow physics. They are regions of the flow where there is significant turbulence production due to high gradients of flow quantities. Therefore, it is crucial that the near-wall region and the flow physics in boundary layer be accurately modelled to ensure accurate flow predictions. A non-dimensional distance, $y^+$, is typically used to quantify the distance from the wall and define the extents of the boundary layer. It is defined as:

$$y^+ = \frac{y u_\tau}{\nu}$$

(14)

Where, $y$ is the absolute wall distance, $u_\tau$ is the friction velocity (calculated as the square root of the ratio of the wall shear stress to fluid density) and $\nu$ is the kinematic viscosity. Another useful non-dimensional parameter is $u^+$, which is a dimensionless velocity given by:

$$u^+ = \frac{U}{u_\tau}$$

(15)

The Law of the Wall states that in the near-wall region, the flow velocity only depends on the distance from the wall, fluid density, viscosity and wall shear stress and is independent from the free stream parameters. [6] In other words,

$$u^+ = f(y^+)$$

(16)

The boundary layer can be divided into three layers as follows: [11] [8]

- Viscous sublayer
  This is an extremely thin layer of fluid that is in direct contact with the wall where, at the wall surface, the fluid is stationary. Here, the viscous effects dominate over the turbulent effects and as a result it can be said the flow in the viscous sublayer is laminar. The viscous sublayer corresponds to a $y^+$ wall height of less than 5.

- Buffer layer
  The buffer layer is the region of the boundary layer between the inner viscous layer and the outer log-law layer and lies between $y^+$ values of approximately 5 to 30.
• Log-law layer

The log-law layer corresponds to a region in the boundary layer where the turbulent effects dominate. This region corresponds to $y^+$ values between 30 and 500.

![Figure 4: Subdivisions of the boundary layer.](image)

**Wall Treatment Approaches**

Turbulence models include different near wall modelling approaches and are commonly referred to as wall treatments. The implementation and nomenclature of these wall treatments differ slightly between CFD solvers but employ more or less the same basic concepts. The low-Reynolds approach (or low $y^+$ approach) resolves the viscous sublayer and solves for the flow field variables as in the free stream. This requires a near-wall mesh with sufficient cells (minimum of 10) to capture the physics in the viscous sublayer, i.e. the first cell height must be around a $y^+$ value of 1.

High-Reynolds (or high $y^+$) wall treatments include the use of wall functions to model the effects of the viscous sublayer instead of resolving it. This implies that the first cell height of the mesh must correspond to a $y^+$ value of 30 or more. This approach is less computationally expensive than the low $y^+$ approach since there are less cells in the near-wall region. It is for this reason that this wall treatment is generally preferred for industrial applications, where computational time and resources are limited.

The STAR-CCM+ CFD solver additionally has an 'all $y^+$' hybrid wall treatment option, where the solver is compatible with both low-Re and high-Re meshes. The solver detects near-wall cells that fall within the buffer region, i.e. $5 < y^+ < 30$ and applies a blending function to calculate turbulent production and dissipation. It also automatically switches between low-Re and high-Re wall treatments wherever the mesh allows for it. This is especially useful for domains of varying geometrical detail where it might be crucial to completely resolve the viscosity affected near-wall layer only in some regions. [8]
The two-layer model was first suggested by Rodi as a viable alternative to low-Re $k - \varepsilon$ models in terms of obtaining the same if not more accuracy with the benefit of lowering the computational cost. The goal of this wall treatment method is to be able to resolve the viscous sublayer and the buffer layer while still using the $k - \varepsilon$ model in the bulk flow region. The two-layer model essentially makes use of a one-equation model to resolve the viscous-affected near-wall layer and blends into the two-equation $k - \varepsilon$ model towards the free stream. The one-equation model solves for the turbulent kinetic energy but models the dissipation rate and eddy viscosity as functions of wall distance and length scale. The one-equation model by Wolfshtein is a common choice for two-layer formulations and has shown good performance to capture the boundary layer physics in turbulent internal flow. [12, 13]

2.2.6 Realizable $k - \varepsilon$ Model

The $k - \varepsilon$ model is a RANS turbulence model originally developed by Jones and Launder [14] that estimates the eddy viscosity through two additional transport equations - one for turbulent kinetic energy ($k$) and another for turbulent dissipation rate ($\varepsilon$). It is arguably the most widely used RANS turbulence model for industrial applications and has undergone several improvements over the years. [8] One of such improvements is the Realizable $k - \varepsilon$ (RKE) model, developed by T.-H. Shih et al. at the NASA Lewis Research Center in 1994. [15] It includes an improved transport equation for the turbulent dissipation rate. The $\varepsilon$ equation in the RKE model is formulated based on the dynamic equation for fluctuating vorticity. [15]

The standard formulation for eddy viscosity ($\mu_t$) is as follows:

$$\mu_t = \rho C_\mu f_\mu \frac{k^2}{\varepsilon}$$

(17)

Where, $C_\mu$ is a model coefficient and $f_\mu$ is a damping function. The RKE model includes a revised formulation for the eddy viscosity to ensure the realizability of the solution. Realizability refers to the ability of the model to refrain from producing non-physical results due to negative normal normal stresses and the violation of Schwarz’s inequality for shear stresses. These violations are typically caused due to the large mean strain rates in the flow. The RKE model satisfies these constraints by defining $C_\mu$ as a variable that is a function of turbulence quantities ($k, \varepsilon$) and mean flow deformation. [15, 8, 16]

It must be noted that selecting a turbulence model is one of the most significant choices for a given CFD investigation. Each turbulence modelling approach has its respective strengths and weaknesses with respect to capturing flow behaviour, examples of which include flow separation, internal pipe flow, external aerodynamics, etc. The RKE model has been chosen for this study after considering the project objectives and similar research work done previously. The motivation for the RKE turbulence model is discussed further in section 2.3.2.
2.2.7 Multiphase Modelling

Multiphase flow refers to the flow condition where there are several types of fluids present in the domain of interest. Multiphase flow could include gas-solid, gas-liquid or liquid-solid phase interactions. Some examples of multiphase flows include air bubbles rising in water, the process of boiling, bubble formation in fluidized beds and erosion through solid particles in liquid flow. This study includes the liquid and gaseous states of matter in terms of liquid gasoline and gaseous air.

When performing numerical calculations of flows with several phases, additional complexity is added in comparison to single phase flow as the model now needs to take into account the interface between the two phases and the varying intricacy of this interaction. For phase interactions where the two phases are well mixed, such as transportation of powder in gas or bubbles in liquid, the flow is dispersed. When the there is a prominent free surface in the flow, it is said to be stratified, which is the case for this research. The difference between dispersed and stratified flow is illustrated in [Figure 5]. The Volume of Fluid (VOF) method is a well suited method for simulating stratified multiphase flow and has been used previously in similar researches [4, 8, 17, 18].

![Figure 5: The different regimes of multiphase flow.](image)

**Volume of Fluid method**

The VOF method was first published by C.W. Hirt and B.D. Nichols and is a simple but powerful method based on the concept of a fractional volume of fluid within each cell. It is useful for treating complicated free boundary configurations but assumes that the mesh resolution is sufficient enough to resolve the interface position and shape between the two phases [Figure 6]. [19] [8] The implementation of the VOF multiphase model in STAR-CCM+ is an interface-capturing method that is able to predict the movement and distribution of the interface of immiscible phases [8].

To avoid modelling errors that occur when the interface breaks and bubbles or droplets are formed, a suitable grid is required. To sufficiently resolve small bubbles or droplets, a grid where the droplet or bubble is in contact with at least three cells in each direction is required [8], see [Figure 6]. Due to air being entrained into the fuel when entering the filler pipe, it is of importance to retain this grid resolution where these air entrainment bubbles might occur.
The volume fraction of fluid phase $i$ is defined as:

$$\alpha_i = \frac{V_i}{V}.$$  

(18)

Where $V_i$ is the volume of phase $i$ in the cell and $V$ is the volume of the cell itself. When the volume fraction of phase $i$ is 0, the cell is completely empty of the phase. When $\alpha_i = 1$, the cell contains only phase $i$. For values of $\alpha_i$ between $0 < \alpha_i < 1$, the cell consists of a mixture of both phases and is managed by the interface model chosen for the VOF method. Per definition, the volume fraction of all phases in a cell must add up to one.

$$\sum_{i=1}^{N} \alpha_i = 1.$$  

(19)

Where, $N$ is the number of phases present in the cell.

The material properties in each cell are determined by a number of equations. The density, $\rho$ and dynamic viscosity, $\mu$ are determined through the following equations [8]:

$$\rho = \sum_i \rho_i \alpha_i$$  

(20)

$$\mu = \sum_i \mu_i \alpha_i.$$  

(21)

The mass conservation equation governs the transport of the volume fraction $\alpha_i$. Equation \ref{eq:mass_conservation} represents the volume fraction transport equation used by the VOF model. In STAR-CCM+, the volume fraction is calculated differently depending on whether there are two or more phases present [8]. If there are two phases present, equation \ref{eq:mass_conservation} is solved only for the first phase. The second phase volume fraction is adjusted depending on the volume fraction of the first phase to satisfy equation \ref{eq:sum_volume_fractions}. For the case of three or more phases present, equation \ref{eq:mass_conservation} is solved for all phases and is therefore, more computationally demanding.

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

(22)

Where, $\mathbf{v}$ is the mass-averaged velocity for the mixture, $\mathbf{a}$ is the surface area vector $\mathbf{v}_{d,i}$ is the diffusion velocity, $S_{\alpha_i}$ is a source term for phase $i$ and $\frac{D\rho_i}{Dt}$ is the Lagrangian derivative
of the phase \(i\). The VOF model therefore essentially solves an additional transport equation to be able to track the movement of the different fluid phases in the domain. \[19\]

**Surface Tension, Contact angle and High Resolution Interface Capturing**

It is essential in most multiphase flow cases that the fluids in the analysis are separated by an interface of reasonable quality. For such flow cases, the high resolution interface capturing (HRIC) scheme can be implemented which introduces a sharpening factor \((C_\alpha)\) and adds another term to the transport equation, as shown in Equation 23.

\[
\nabla \cdot (\mathbf{v}_{ci} \alpha_i (1 - \alpha_i)).
\]

(23)

Where,

\[
\mathbf{v}_{ci} = C_\alpha \times |\nabla \alpha_i|.
\]

(24)

The sharpening factor \(C_\alpha\) in Equation 24 is user specified and can be used to reduce numerical diffusion. The values that can be used range from 0 to 1, with 0 being the default setting where there is no reduction in numerical diffusion. Setting the \(C_\alpha\) value to 1 results in no numerical diffusion and a very sharp interface but a higher computational cost.

The HRIC scheme is a High Resolution Scheme (HRS) and is based on the normalized variable diagram (NVD). It is designed to outperform higher order schemes like the central differencing and second order upwind scheme in terms of estimating the convective transport of immiscible fluids of different volume fractions. It ensures that the field variables stay bounded and preserves monotonicity, i.e. minimises oscillations. This is achieved by switching between higher order schemes to reduce numerical diffusion and lower order schemes (in this case, the First Order Upwind scheme) that are inherently bounded. \[8, 20\]

Surface tension is the cohesive force exerted between the liquid molecules on the surface of a liquid. Modelling the surface tension accurately therefore contributes to a more accurate representation of the multiphase fluid interface in the CFD study. While the interface between phases in reality is defined by a sharp boundary, specifying large values for the sharpening factor has the possibility to introduce non-physical alignment of the free surface with the grid lines. \[8\]

The contact angle quantifies the wettability of a surface by a liquid and varies based on material. Contact angles above 90\(^\circ\) signify that the liquid is phobic to the surface, while a contact angle below 90\(^\circ\) signify that the liquid wets the surface. Surface tension and contact angle are both interdependent liquid properties and as discussed previously, have a significant effect on the interface capturing ability of the CFD model.
2.3 Literature Review/Previous Work

There have been several studies over the years with the common goal of understanding the complex multi-phase flow interactions during the automotive refuelling process. The previous researches that were of particular interest for this study can be divided into those that investigated the refuelling process through mathematical models and/or experiments [21, 22, 23] and those that implemented CFD models to analyse the refuelling process [18, 4, 24, 25, 17, 26].

2.3.1 Experimental Investigations/Flow Physics

The refuelling process can be divided into three distinct segments as described by Mastroianni et al [22]. Segment 1 corresponds to the initial increase of pressure inside the tank. Segment 2 observes a constant pressure and represents the steady filling process where vapour is continuously vented out of the fuel tank. The final segment begins when the FLVV closes due to the rising fuel level which results in a pressure rise inside the tank since the vapour cannot escape. The pressure peak corresponds to the fuel level reaching the sensing port on the nozzle, after which fuel supply from the dispenser is cut off.

Mastroianni et al. [22] additionally performed an extensive experimental study to understand the sensitivities of the vent tube diameter, Reid Vapour Pressure (RVP) and fuel fill rate on the refuelling performance. They found that as the vent tube diameter increased, the peak pressure in the tank during Phase 1 reduced. Additionally, they reported that an increase in RVP and hence volatility of the fuel increased the peak tank pressure during the first segment of the refuelling process. This indirectly implies that evaporation has a significant effect and therefore must be included in the CFD model to analyse ORVR performance in more detail.

Persson and Stahm performed experimental investigations at Volvo Car Corporation to investigate the effects of air entrainment, fuel temperature and pre-existing fuel level on...
Canister loading refers to the amount of hydrocarbon vapour that leaves the tank during refuelling and enters the carbon canister and hence can be directly associated with ORVR performance. They reported that the canister loading increases with an increase in nozzle entrained air, while also suggesting that this entrained air becomes fully saturated with fuel vapours as it exits the tank through the venting port. This experimental observation was consistent with the mathematical model predictions by Reddy [21], as well as the research performed by Banerjee et al. [24]. Banerjee et al. additionally concluded that the air entrained directly depends on the filler pipe geometry and fuel flow rate through the nozzle, with an increased risk of reversed flow of air with flow rates above 34 l/min for their filler pipe design. This reversed flow was reported in another research by Banerjee et al. which showed that for a fuel flow rate of 45 l/min, there was a net outflow of air from the filler pipe mouth back into the surroundings. They concluded that this was due to the fact that the path through the tank and venting line offered more resistance to flow as compared to travelling back up the filler neck. This implies that the pressure at the venting box outlet is a crucial parameter to ensure that the vapour escapes the tank only through the carbon canister and not back through the filler pipe. In the case of the SPA fuel system investigated in this study, this pressure is controlled by changing the properties of the activated carbon in the canister which essentially alters its porosity and creates a different pressure at the boundary. Efforts have been made in this study to include the effects of the porous nature of the activated carbon.

Another observation that was consistent with the studies performed by Reddy and Persson and Stahm was the effect of 'top-fill' and 'bottom-fill'. Top fill refers to the filling condition where the level of pre-existing fuel in the tank is below the tank inlet and bottom fill implies that the fuel level in the tank is above the tank inlet. In the case of the top fill condition, the fresh incoming fuel mixes with the vapour in the tank whereas in the bottom fill condition, there is no direct contact between the dispensed fuel and the vapour. These two filling scenarios as well as the temperatures of the dispensed fuel and tank vapour can cause different vapour generation behaviour inside the tank and have been identified as critical time windows during the refuelling process. Based on these findings, it was decided to establish the CFD model in the present study for these two critical time windows. Details about the time duration simulated and temperatures investigated are discussed further in section 3.1.

2.3.2 Previous CFD Research

As mentioned previously, there have been a number of CFD studies within the automotive refuelling research area. However, these studies have mostly focused on the different parts of the fuel system individually by either simplifying or neglecting the remaining components of the fuel system. A recent study by Dake et al. [18] revealed that the level of detail of the nozzle geometry included in the CFD model significantly influences the fuel spray pattern and therefore influences the performance of the ORVR system. They also investigated the applicability of the RKE turbulence model for VOF fuel flow simulations and found that the CFD results were within an error of 10% when compared to experimental results. The previously mentioned research by Banerjee et al. [24] included a multi-phase evaporation model based on continuous thermodynamics to study the flow interactions in
the filler pipe. In addition to their observations on air entrainment, they recommended not using fluids other than gasoline for ORVR performance research as the flow physics differed significantly.

Gunnesby performed VOF simulations to compare the differences between the URANS Shear Stress Transport (SST) turbulence model and Delayed Detached Eddy Simulation (DDES) model for the filler pipe with diesel as the working fluid. A conclusion from this research work was that the URANS modelling approach offered good efficiency in terms of computational cost versus accuracy. The study however showed that the SST \( k-\omega \) model appeared to predict late separation in the filler pipe. Gunnesby also reported that the first bend in the filler pipe is an important part of the domain and must be resolved sufficiently in terms of grid spacing.

Eklund and Kreuger [17] recently performed a CFD study where they compared the Improved DDES (IDDES), SST and RKE turbulence models for tank sloshing using the VOF model. Tank sloshing refers to the phenomenon where the fuel in the tank is thrown and/or splashes around as a result of the G-forces experienced by the car (and hence the fuel tank) during driving. The flow patterns in such a case are highly irregular and transient. They found that all models showed satisfactory agreement with experimental results. That being said, the RKE model imposed significantly lower mesh requirements and therefore proved to be a less computationally demanding option for the CFD method that was set up.

Halfway through the project, a new research paper by Dake et al. [26] investigating automotive refuelling was published. They developed a CFD model using the RKE model for the refuelling process and performed experimental tests to validate the simulation results. The study was performed with Stoddard solvent to exclude the effects of evaporation. While the CFD and experimental results show good agreement, the meshing strategy and boundary conditions implemented in the CFD solver are unclear. The research paper did not include a verification study and claimed to utilise a mesh with an overall cell count of only 2.2 million cells, which for an entire automotive refuelling system might be questionable. Also not included in the paper were details about the valve physics required to be able to model the pressure peak that corresponded to automatic shut-off.

With the main objective of this study being the reduction of product development time, it was decided to set up the CFD model with the RKE turbulence model with the Two-Layer All \( y^+ \) wall treatment in order to obtain reasonably fast computational time without sacrificing significant accuracy. The Two-Layer model is particularly useful for the regions in the domain that are quite intricate (i.e. higher geometrical detail), such as in the recirculation pipe, venting pipe and the fuel nozzle internal geometry while the wall function approach can be used in the other regions of the domain that would allow it. Details about model setup and meshing strategy are discussed in the following sections.
3 Method

3.1 CFD Study

Figure 8 shows the three-view of the computational domain used for the CFD study while Figure 9 shows the different parts of the domain.

The refuelling process for the SPA 71 l fuel tank takes approximately 112 s with a fuel pistol/pump flow rate of 37.8 l/min. This flow rate corresponds to 10 gallons per minute,
which is the maximum allowable pump dispensing rate in the USA. At the time, it was impractical to simulate the entire 112 s of the refuelling process as that would have resulted in a total simulation run time of over a month. In order to capture the most significant events during the refuelling process, two critical time windows were identified:

1. Top fill: fuel level is below the inlet check valve
2. Bottom fill: fuel level is above the inlet check valve

![Figure 10](image.png)

Figure 10: The two initial fuel volume cases that were tested.

(a) Phase I, top fill  
(b) Phase II, bottom fill

Figure 10a and b show the initial fuel level for the top fill and bottom fill cases, respectively. The top fill case is henceforth referred to as 'phase I' while the bottom fill case will be referred to as 'phase II'. The initial fuel level in the phase I case was set to 7.1 l, which corresponds to 10% of the total fuel volume while for the phase II case, the fuel volume was set to 34 l. The initial fuel volume for the phase II case was selected with the intention of simulating the effects of the fuel spilling over from the active side of the fuel tank to the passive side, which at the time had not been studied in detail before at Volvo Cars.

3.1.1 Preparation of the OPW 11B CAD Model

In order to be able to include all the internal details of the OPW 11B fuel nozzle into the CFD model, it was necessary to first construct a detailed CAD model of the nozzle. An existing OPW 11B fuel pistol was physically dismantled following which the nozzle section was carefully cut along the middle. The individual parts were then measured using a digital caliper that had a least count of 0.01 mm to obtain measurements which were then used to construct a CAD model of the nozzle in CATIA v5. A cross-section view of the nozzle is shown in Figure 11.

![Figure 11](image.png)

Figure 11: Cross section view of the OPW 11B fuel pistol nozzle CAD model constructed in CATIA v5.
The fuel nozzle was then included into the fuel system CAD file by inserting it into the capless unit and ensuring that the angle of flow into the filler pipe was as close as possible to that observed in the experimental setup.

3.1.2 Surface Preparation and CAD Clean-up

There are many parts in the fuel system that are both detailed and complex in their shapes. Increased geometrical detail in the model implies higher computational cost. This means that there is always a balance between level of detail (i.e. flow resolution) and computational efficiency. CAD clean-up is the process of removing and simplifying the smaller details in the geometry while still retaining the prominent geometrical features that are crucial to the flow characteristics. For comparison, the fully detailed CAD model of the fuel system initially contained 367 individual parts which was reduced to 26 over the CAD clean-up process. During the process, several parts were removed, simplified as well as combined to prepare the geometry for meshing.

One such part is the capless unit, shown in Figure 12a, which contained a high level of geometrical detail that was not critical for the flow of air or fuel. All these details were removed and/or simplified to obtain the capless unit shown in Figure 12b, which was used for the study.

![Figure 12: (a) The capless unit before CAD clean-up. (b) The capless unit after CAD clean-up.](image)

Deciding which geometrical details to preserve and those to remove is crucial to the physics that the simulation would be able to capture. For the capless unit, a few such critical areas which were expected to have a significant impact on the solution were identified. The first is the size and shape of the opening around area 1 (the pink coloured surface), as seen...
in Figure 13. This is the main obstruction to the air flowing into the capless unit which means that its shape and size has an immediate effect on how much air is allowed into the filler pipe. The second area, which was in fact left unchanged, is the rail which the pump pistol nozzle rests on when fully inserted into the capless unit. The angle with which the pump pistol nozzle is inserted into the capless unit affects the flow pattern in the filler pipe and in an effort to accurately capture that angle, the rail is left unchanged. This rail is denoted by area 2 in Figure 13. Area 3 in Figure 13 also remains unchanged. The red flap seen in the figure is directly in front of the hole where the recirculation line attaches to the filler neck. It is necessary to keep this unchanged due to the importance of the recirculation line in regards to refuelling performance.

![Figure 13](image_url) The green circles outline the crucial areas of the capless unit geometry.

The geometry of the ICV portion of the filler pipe was modelled as a straight pipe leading from the filler pipe into the tank. The flap was opened by rotating it around the pin holding the hinge by 90°. Since the spring holding the flap is fairly weak, it was assumed that it is completely pushed open by the fuel rushing into the tank. The pin, hinge and spring were removed completely from the geometry. The cleaned ICV geometry area can be seen in Figure 14.

![Figure 14](image_url) The cleaned ICV with the hinge and spring mechanism removed and the flap left open.
In the bulk volume of the fuel tank, details that were deemed unnecessary for the purpose of this study were removed and/or simplified. These changes included completely removing the internal pipes and fuel lines (except the line connecting the FLVV and LCO-box) as well as smoothing out small bumps and sharp edges in the fuel tank. The FLVV pipe was preserved in the geometry in order to allow gases to escape out of the tank through the pipe and into the LCO box as the fuel level rises. The FLVV, roll-over valves and baffle on the passive side of the fuel tank were reduced to cylinders for the sole purpose of occupying volume in the fuel tank. Details were preserved in regions of interest such as the baffle on the active side of the fuel tank. Figure 15 shows the cleaned tank with the list of its internal components specified in Table 4.

![Image](image.png)

**Figure 15:** The cleaned tank and its internal components.

<table>
<thead>
<tr>
<th>Number</th>
<th>Component</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>ICV</td>
</tr>
<tr>
<td>2</td>
<td>Active Side Baffle</td>
</tr>
<tr>
<td>3</td>
<td>FDM Body</td>
</tr>
<tr>
<td>4</td>
<td>Roll-over Valves</td>
</tr>
<tr>
<td>5</td>
<td>FLVV</td>
</tr>
<tr>
<td>6</td>
<td>Passive Side Baffle</td>
</tr>
<tr>
<td>7</td>
<td>Passive Side Pickup</td>
</tr>
<tr>
<td>8</td>
<td>FLVV Pipe</td>
</tr>
<tr>
<td>9</td>
<td>LCO Box</td>
</tr>
<tr>
<td>10</td>
<td>LCO Box Venting Outlet</td>
</tr>
</tbody>
</table>

### 3.1.3 Boundary Conditions

The CFD solver settings and boundary conditions correspond to the options available in the commercial CFD solver STAR-CCM+, version 12.06.010. The boundary conditions
enforced on the different parts of the domain are listed below:

- **Fuel pistol inlet:**
  A mass flow inlet type boundary with a flow rate of 0.473 kg/s was specified at the inlet face of the fuel pistol. This mass flow rate corresponds to the maximum allowable fuel dispensing rate in the USA, which is 10 gallons per minute (or \(\sim 38 \text{ l/min}\)). The fuel flow was specified to ramp linearly from zero to maximum over the first 0.1 seconds, which was estimated to be the approximate time required to manually depress the fuel pistol trigger. \[4\]

- **Atmosphere surrounding the capless unit:**
  A cylinder of diameter \(3d_f\) and length \(10d_f\) (where \(d_f\) is the diameter of the filler pipe) was created and placed at the inlet face of the capless in order to mimic the effects of the atmosphere surrounding the fuel pistol and refuelling cap on the car. It was found from initial test simulations that this cylinder size was sufficient to not affect the results. The pressure on the surfaces shaded with light blue in Figure 9 were set to a stagnation inlet with atmospheric pressure while the dark blue shaded surface was set as a no slip wall, intended to replicate the body of the car.

- **Solid wall surfaces**
  All the solid surfaces such as the fuel nozzle, filler pipe, tank, etc. were set as no-slip walls with fluid contact angles obtained through an experiment. A drop of gasoline was placed on a plate of 304L stainless steel and then on a flat piece of high density polyethylene (HDPE). The steel material represented the filler pipe and recirculation line whereas HDPE is the plastic material used to manufacture the tank. For simplicity, it was assumed that all plastic materials in the computational domain were of the same material as the tank. A picture was taken level with the surface of the test pieces to measure the contact angle. The gasoline was found to completely and instantly spread, thereby wetting the surfaces of both materials. This implied that the contact angle for both materials was \(0^\circ\).

- **Venting box outlet**
  The venting box outlet pipe geometry was modified to include an infinitesimally thin ‘porous baffle interface’ surface, which was then used to obtain a pressure drop across the pipe and therefore mimic the effect of the flow resistance caused by both the EVAP line as well as the carbon canister. The pressure drop across the carbon canister was calculated from experimental measurements to be 0.8175 kPa. The pressure drop through the EVAP line was simplified and estimated using the Darcy-Weisbach equation for pressure loss in pipes \[9\] as in equation \[25\].

\[
\Delta p = f_b\frac{\rho l_{EVAP} V_{EVAP}^2}{2d_{EVAP}} \tag{25}
\]

Where, \(l_{EVAP}\) and \(d_{EVAP}\) are the length and diameter of the EVAP line, respectively, \(V_{EVAP}\) is the velocity of the fluid through the EVAP line and \(f_b\) is the Blasius approximation for friction coefficient for turbulent pipe flow \[9\] as given by equation \[26\].

\[
f_b = 0.316 Re^{-0.25} \tag{26}
\]
The sum of the pressure drops due to the canister and the EVAP line were then used in equation [27] to perform an iterative calculation to find the values of $\alpha$ and $\beta$, which are the porous inertial resistance and the porous viscous resistance of the baffle interface, respectively.

\[ \Delta p = -\rho \left( \alpha \left\| v_n \right\| + \beta \right) v_n \tag{27} \]

Where, $v_n$ is the velocity of the fluid normal to the baffle interface surface and in this case was the velocity of the flow in the EVAP line $U_{EVAP}$, as obtained from experimental measurements. The porous inertial and viscous resistances are the parameters that affect the permeability of the baffle surface. The inertial resistance considers the bulk flow losses such as expansion, contraction and bends through the pore channels whereas the viscous resistance takes into account the effects of viscosity and its related friction losses. The calculated values of $\alpha$ and $\beta$ were then used as inputs in STAR-CCM+ to obtain the desired pressure drop. Using a porous baffle in the outlet pipe allows for a more realistic outlet boundary condition rather than applying a constant pressure from time $t = 0$, since the flow velocity through the outlet rises over time and is not constant.

### 3.1.4 Solver Setup

A basic overview of the solver settings implemented in this study is shown in Table 5.

<table>
<thead>
<tr>
<th>Setting</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flow Solver</td>
<td>Eulerian, Segregated URANS</td>
</tr>
<tr>
<td>Time Model Solver</td>
<td>Implicit Unsteady</td>
</tr>
<tr>
<td>Pressure-Velocity Coupling</td>
<td>Rhie-Chow interpolation based SIMPLE*</td>
</tr>
<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>Eulerian Multiphase Model</td>
<td>Volume of Fluid (VOF) Method</td>
</tr>
<tr>
<td>Thermal Solver</td>
<td>Isothermal</td>
</tr>
</tbody>
</table>

* Default setting in STAR-CCM+ v12.06.010 for segregated flow solvers

The refuelling process has been previously found to cause highly unsteady and chaotic flow patterns through the filler pipe [4]. Based on these findings as well as initial test simulations, a second order temporal discretization method was employed in order to adequately capture as much flow detail as possible. The second order implicit temporal discretization scheme uses the Backward Differentiation Formula, or BDF2, where the 2 implies that it uses the solution at the current time step as well as the previous two time steps.

Test simulations in the initial stages of the project showed that time-step selection played an extremely important factor in the stability of the CFD solution. This was found to be due to the fact that the fuel flows at Reynolds numbers in the vicinity of 200 000 through the fuel pistol and filler pipe, which are by far the most geometrically intricate and detailed regions of the domain (thereby requiring a finer mesh in those regions). This resulted in a finer requirement on the time step-size, which was subsequently chosen to
be $3.33 \times 10^{-5}$ s as the base value for all mesh verification simulations. It was estimated that selecting such a low time step would invariably be well within what would be required for this flow case. Once the mesh had been verified, an adaptive time step control was implemented based on the average and maximum CFL numbers on the liquid-gaseous interface, which is discussed further in the following sections. These recommended values for VOF multiphase simulations correspond to an average CFL value being lower than 0.5 with the maximum CFL value never-exceeding a value of 10. [8] It was found that 8 iterations per time-step proved to be the ideal setting for this study to ensure solver stability as well as solution convergence. Simulations were set up to run for a total of 3 seconds of flow time, since it was observed that the flow quantities such as static pressure and mass flow appeared to steady out after approximately 2 seconds.

Second order schemes were chosen for all the equations solved in the VOF model. Additionally, the HRIC scheme has been used to ensure that the interface between the liquid and gaseous phases was adequately captured while ensuring the solution remained stable over time. Gasoline and air were the two immiscible fluids specified as the Eulerian multiphase materials, with gasoline being the primary phase and air being the secondary phase for the VOF-VOF Phase Interaction. The density and dynamic viscosity of gasoline were representative of the properties of the LEV III certification test fuel used in the experimental part of this study. Air was specified as an ideal gas, which implied that the density was allowed to vary only with pressure, since the CFD analyses were all isothermal. Initial attempts were made with air specified as a constant density fluid but it appeared to induce severe instabilities in the solver.

Additional settings include the ‘Cell Quality Remediation’ model in STAR-CCM+ and the inclusion of gravity in the negative Z direction. The Cell Quality Remediation model identifies elements of poor quality by checking the mesh for quantities such as skewness angle and volume change. The model then modifies the computation of the gradients in these elements so as to improve the solver stability and robustness. [8]

### 3.1.5 Meshing Strategy and Verification Study

The meshing strategy followed is arguably the most critical part of a CFD study since the mesh directly defines the spatial limits up to which the flow physics will be captured. In other words, the CFD solver is capable of resolving flow structures only if the cell size is adequately small enough. The meshing process for this study was particularly challenging and time-consuming due to the large size of the domain coupled with the relatively extreme variation in geometrical detail, while simultaneously attempting to obtain a mesh size that was (and will in the future be) reasonable to run on the available computational resources. The following section provides an overview on the meshing strategy as well as the steps taken to verify mesh independence during this study.

The meshes investigated in this study were generated using polyhedral elements for the whole domain. As mentioned previously, the flow pattern through the filler pipe was found to be highly irregular and chaotic. The use of polyhedral elements is beneficial over a hexahedral (or even trimmed) mesh in cases such as the one in this study where the flow is not aligned with the mesh grid. The advantage is mostly due to the fact that numerical
diffusion in the case of a polyhedral mesh is isotropic, whereas it is not in the case of a hexahedral mesh. Polyhedral meshes also offer significant advantages over tetrahedral meshes. Polyhedral meshes require approximately four times lesser elements and are able to compute solutions 5 to 10 times quicker than tetrahedral meshes and still provide the same level of accuracy. [8]

STAR-CCM+ was used to generate both the surface mesh and volume mesh. A combination of several surface and volumetric controls were implemented to refine the mesh in the regions of interest in different parts of the domain. The individual surface and volume controls were made to vary as a percentage of the set 'Base Size', which implied that as the base size was reduced, the entire domain was globally refined in accordance with the pre-defined ratios. This allowed for the overall refinement steps between meshes to be consistent and ensured a fair comparison during the verification study. The cell size in the domain in the various controls was varied from a minimum of 15% to a maximum of 100% of the base size. Prism layers were specified so as to maintain the $y^+$ values within the recommended range of 30 to 120 except in the recirculation line and LCO box (venting line) outlet where the $y^+$ was less than 5 for all meshes. This was due to the smaller cross sectional area of these specific pipe sections, which therefore required a finer mesh. It was in these regions that the Two-Layer All $y^+$ wall treatment approach proved to be useful.

A combination of volumetric refinement controls and surface controls were used to control the cell size in the regions of interest. The volumetric refinements used in and around the capless unit are shown in [Figure 16]. It was found that the spatial discretization in the filler pipe played a critical role in determining solution stability. A refinement box was placed at the interface between fuel and air. For the phase I case, the box was created such that the height of the box would correspond to the approximate fuel level rise during the entire simulation. For the phase II case, the refinement region not only covered the initial interface in the active side, but also covered the saddle slope and the bottom face of the passive side, where fuel was expected to flow. The interface refinement boxes for the phase I and phase II cases are shown in [Figure 17].

**Figure 16:** The volumetric refinement boxes for the pump pistol body and the capless unit. The green zones are refinement volumes for the atmosphere and filler pipe respectively.
The verification study was performed only for the phase I case. Three mesh sizes were investigated as part of the verification study, as shown in Table 6. The mesh sizes were dictated by the individual controls that were applied. The surface and volumetric control settings were chosen such that the coarsest mesh sufficiently captured all the geometrical details of the domain, even in the most intricate and detailed regions. This consequently resulted in an element count of approximately 3.8 million cells for the coarsest mesh. The base sizes were then adjusted to obtain finer meshes such that the overall mesh sizes increased by a factor of 1.5 between refinement steps.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Base Size $[\text{mm}]$</th>
<th>Element Count</th>
<th>CPU Hours for 1 s of Flow Time</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh 1</td>
<td>7.5</td>
<td>3 800 000</td>
<td>12 000</td>
</tr>
<tr>
<td>Mesh 2</td>
<td>6</td>
<td>5 700 000</td>
<td>17 622</td>
</tr>
<tr>
<td>Mesh 3</td>
<td>5</td>
<td>8 200 000</td>
<td>18 889</td>
</tr>
</tbody>
</table>

Figure 18 and Figure 19 provide visual representations of the volume mesh for the coarsest mesh, i.e. Mesh 1, in the tank and capless regions, respectively. Figure 18 also shows the surface mesh on the FDM body, active side baffle, ICV and roll over valve.
The quantities measured for the verification study are listed below:

1. Static pressure in the tank calculated as the average surface pressure on the FDM cap.
2. Mass flow rate through the recirculation line, measured as a surface average on a cross sectional plane in the line near the tank.
3. Time averaged value of the pressure drop along the recirculation line over the last 1 second of flow time.
4. Surface averaged static pressures at different cross sections along the length of the filler pipe. Values were time averaged over the last 1 second of flow time.
5. Time averaged mass flow rate through the outlet over the last 1 second of flow time.

Quantities 1 and 2 were also measured over the duration of the simulation. The differences in static pressure in the tank and recirculation line mass flow rate between mesh sizes are shown in Figure 20 and Figure 21 respectively. The data set ‘Mesh 2 TS’ corresponds to the results obtained from an automatic time step control that was implemented once the spatial discretization settings had been verified and will be discussed in a further part of this section.

![Figure 19: Cross section view of the coarsest volume mesh around the capless unit.](image)

![Figure 20: Gauge static pressure measured in the tank for different meshes.](image)
Table 7 and Table 8 show the average and maximum changes in the measured quantities that were time averaged over the last 1 second of the simulation, where the flow pattern and fluid behaviour were observed to steady out.

**Table 7:** Average change in quantities between mesh sizes over the last 1 second of simulated flow time.

<table>
<thead>
<tr>
<th>Measured Quantity</th>
<th>Mesh 1 to Mesh 2</th>
<th>Mesh 2 to Mesh 3</th>
<th>Mesh 2 to Mesh 2 TS</th>
</tr>
</thead>
<tbody>
<tr>
<td>(p_{\text{tank}})</td>
<td>1.18%</td>
<td>1.40%</td>
<td>1.05%</td>
</tr>
<tr>
<td>(\dot{m}_{\text{out}})</td>
<td>0.79%</td>
<td>1.01%</td>
<td>0.37%</td>
</tr>
<tr>
<td>(\dot{m}_{\text{rec}})</td>
<td>2.60%</td>
<td>2.23%</td>
<td>0.81%</td>
</tr>
</tbody>
</table>

**Table 8:** Maximum change in quantities between mesh sizes over the last 1 second of simulated flow time.

<table>
<thead>
<tr>
<th>Measured Quantity</th>
<th>Mesh 1 to Mesh 2</th>
<th>Mesh 2 to Mesh 3</th>
<th>Mesh 2 to Mesh 2 TS</th>
</tr>
</thead>
<tbody>
<tr>
<td>(p_{\text{tank}})</td>
<td>3.50%</td>
<td>2.85%</td>
<td>2.36%</td>
</tr>
<tr>
<td>(\dot{m}_{\text{out}})</td>
<td>2.43%</td>
<td>2.05%</td>
<td>1.26%</td>
</tr>
<tr>
<td>(\dot{m}_{\text{rec}})</td>
<td>5.58%</td>
<td>4.64%</td>
<td>2.68%</td>
</tr>
</tbody>
</table>

It was observed from Figure 20 and Figure 21 that all three meshes appeared to reach similar values towards the end of the simulation whereas the gradient with which Mesh 1 reached that steady value was noticeably different from Mesh 2 and Mesh 3. Meshes 2 and 3 showed steeper slopes in the first 1.5 seconds and steadied out before Mesh 1. As seen in Table 7, the average percentage change over the last 1 second is around the same for both mesh increments (i.e. Mesh 1 to 2 and Mesh 2 to 3). However, from Table 8, the absolute maximum percentage change was found to decrease as the mesh was refined.

Figure 22 shows the relative static pressure along the recirculation line, represented as a time averaged quantity over the last 1 second. The pressure drop was calculated from the surface average static pressures collected over 23 different section planes placed along the length of the recirculation line. The planes were placed before and after each bend in the pipe. The horizontal axis in Figure 22 represents the position \(l_p\) in the recirculation line,
expressed as a ratio of the total line length $L_r$. The position $l_p/L_r = 0$ corresponds to the point where the recirculation line attaches to the tank whereas $l_p/L_r = 1$ corresponds to the attachment point in the capless unit.

![Graph showing pressure drop over recirculation line for different meshes. Measured values were time-averaged over the last 1 second of flow time.](image)

**Figure 22:** Pressure drop over recirculation line for different meshes. Measured values were time-averaged over the last 1 second of flow time.

It was observed from **Figure 22** that there were no significant differences between the different meshes for the pressure drop over the recirculation line. The large sudden drop at $l_p/L_r = 0.03$ corresponds to the 90° bend at the tank attachment point, which has also been consistently captured by all meshes.

**Figure 24** shows the surface averaged static pressures at different downstream locations in the filler pipe, measured as a factor of the filler pipe diameter $d_f$. All cross section planes were measured in the downstream direction from the reference plane in the capless unit, as shown in **Figure 23**. For reference, planes $10d_f$, $20d_f$ and $30d_f$ correspond to the first, second and third bends in the filler pipe. Plane $40d_f$ represents the point where the filler pipe connects to the tank, i.e. it marks the end of the filler pipe.

![Diagram showing location of the reference plane $0d_f$ in the capless unit from where all filler pipe cross section planes were measured from.](image)

**Figure 23:** Location of the reference plane $0d_f$ in the capless unit from where all filler pipe cross section planes were measured from.
Figure 24: Surface average static pressure on cut planes time averaged over the last 1 second of flow at different locations in the filler pipe. Locations were measured with reference to the capless face D0.

From Figure 24 it was observed that there were no major differences in average filler pipe pressure between mesh refinements. There was however a slight discrepancy observed for the pressure values at the second bend, i.e. at \(20d_f\). The discrepancies between meshes in regards to filler pipe flow patterns are more pronounced in Figure 25, which shows the instantaneous volume fraction contour plots at different cross sectional planes in the filler pipe for the three meshes, extracted after exactly 2.5 seconds of flow time had elapsed. A volume fraction value of 1 corresponds to 100% liquid fuel, while a value of 0 implies that the region is 100% gaseous air. It is clear from Figure 25 that the meshes consistently capture the bulk flow structures but show discrepancies when it comes to capturing the smaller secondary flow structures that arise as a result of the swirling motion of the fuel through the filler pipe.

Figure 25: Instantaneous volume fraction contours at 2.5 s at different downstream locations in the filler pipe for (a) Mesh 1 (b) Mesh 2 (c) Mesh 3.

Figure 26a and Figure 26b show the maximum and average CFL numbers monitored on the interface of the liquid fuel and gaseous air. STAR-CCM+ recommendations state that for a VOF flow problem such as the one in the present study, it is desirable to maintain an average CFL value of less than 0.5 and a maximum CFL number of equal to or less than 10. It was found that the maximum number of cells that were above a CFL value of 0.5 corresponded to around 0.8% of the total cell count on the domain for the finest mesh. This value was observed to be 0.3% and 0.2% for Mesh 2 and Mesh 1, respectively.
With the findings presented thus far, Mesh 2 was deemed to be sufficient to proceed with the remaining investigations in this study. It was decided for the purpose of this study that verifying the bulk system behaviour was more important than the finer details such as secondary flow structures in the filler pipe, which were known to be slightly inconsistent between mesh refinements. This is further discussed in section 5. As observed from Tables 7 and 8, the average and maximum errors reduced as the mesh was refined. While all three meshes tended to plateau to similar values of pressure and mass flow towards the end of the simulation (see Figure 20 and Figure 21), there were smaller differences in the behaviour for the first 1.5 s between Mesh 2 and Mesh 3 as compared to Mesh 1 and Mesh 2.

In order to save solution time as well as identify the sensitivity of the time step, an automatic time step control was implemented in STAR-CCM+. The field function was defined to adjust and calculate the time step (at every time step) based on both the average and maximum CFL numbers to be within the VOF method recommendations provided by STAR-CCM+. The field function also included a limit on the maximum and minimum possible time step values, with the upper limit being fixed at 1E-4 s and the lower limit being 3.33E-5 s. The results for the automatic time step control are presented as the 'Mesh 2 TS' data set in all the figures from Figure 20 to Figure 26. It was found that the results from the automatic time step case do not vary significantly from those obtained.
from Mesh 2 with a fixed time step of 3.33E-5 s. However, the computational time was reduced by almost 60%. It was found that the solution required a lower time step for the first 0.1 to 0.2 s, where it stayed close to the lower limit of 3.33E-5 s. The time step then quickly ramped up to an approximate value of 9E-5 s and remained at that value for the remainder of the simulation. This time step value resulted in the maximum CFL plateauing at a value of 10, as seen in Figure 26. It was found that when changing the time stepping from 3.33E-5 s to the adaptive time step, the iterations required for running 3 seconds of physical flow time (for Mesh 2) reduced from 721 000 iterations to 263 000 iterations.

3.2 Validation Study

Both phase I and phase II cases were validated with experimental test results. The phase I CFD cases were initialised with 7.1 l in the fuel tank and were run for a total time of 3 s. The phase II simulations were set to start from a tank volume of 34 l and were allowed to run for a longer flow time of 5 s. This was so that the CFD simulation for phase II adequately captured the effects of the fuel flowing over the saddle and spilling over into the passive side.

3.2.1 Experimental Setup

The experimental rig used is shown in Figure 27. The rig structure and a few sensors were recycled and re-purposed from a previously used test rig at the fuel testing laboratory at Volvo Cars. Pressure sensors were fitted so as to measure the static pressures in the tank, filler neck and canister inlet. An additional pressure sensor was used to measure the atmospheric static pressure during each experimental trial. The pressure sensors used were IPETRONIK CANpressure pressure transducers with an operating range of 0 to 2 bar (absolute). The four pressure transducers were routed to a 4 channel CSM CANMiniModule, which was then connected to a computer where the quantities were logged and saved using the ETAS INCA software tool. The measured quantities were logged at a frequency of 100 Hz.

Figure 27: The experimental rig. The red arrows show the locations of the pressure sensors.
The list of pressure sensors used as seen in Figure 27 are specified in Table 9.

<table>
<thead>
<tr>
<th>Number</th>
<th>Pressure Sensor</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Filler neck</td>
</tr>
<tr>
<td>2</td>
<td>Fuel tank</td>
</tr>
<tr>
<td>3</td>
<td>Ambient</td>
</tr>
<tr>
<td>4</td>
<td>Canister inlet</td>
</tr>
</tbody>
</table>

### 3.2.2 Experimental Procedure

After identifying the critical time windows discussed in section 3.1, the refuelling experimental procedure was split into three distinct segments to match the top-fill and bottom-fill cases and are listed as follows:

1. Pre-fill the tank with 10% of nominal tank volume.
2. Fill at maximum flow rate up to 34 litres.
3. Fill at maximum flow rate up to automatic shut-off.

Three trials of the experimental test procedure were performed over the same day to ensure repeatability and eliminate any anomalies from the test results. Performing the experiments over the same day ensured that the ambient conditions in the test cell remained more or less constant. Each test trial was performed with three different canisters, all of which are regular production-spec Volvo SPA carbon canisters. Each canister was purged for 2.5 hours before performing the experiment. Purging in this case refers to the process of passing fresh air through the canister in a controlled manner in order to desorb the trapped hydrocarbons from the carbon bed. The fuel in the storage tank in the test cell was conditioned before each experimental trial in order to match the temperature of the dispensed fuel to the ambient temperature of the test cell.

The first segment of the test procedure was the pre-fill, where 10% of the nominal tank volume (7.1 l) was filled in order to match the conditions of the ORVR test procedure. The pre-fill was performed with a spare ‘slave’ canister connected to the outlet of the EVAP line. After the pre-fill, the fuel in the tank was allowed to settle for 5 minutes before replacing the slave canister with the actual test canister. For the second segment, fuel was pumped in at the maximum flow rate until the total fuel volume reached 34 l. This part of the refuelling experiment corresponded to the top-fill case. That is, when the initial fuel level is below the ICV. At this point, the refuelling process was paused for 90 seconds in order to let the fuel settle as well as to allow the pressures in the different parts of the fuel system equalise. This fuel volume was chosen so that the third segment started at a fuel level that would capture the behaviour of the bottom-fill case and corresponded to a physical fuel level slightly lower than the tank saddle. This fuel volume could also easily be replicated in simulations. The end of the third segment corresponded to the automatic shut-off of fuel flow.
3.3 Investigating Changes to the Fuel System

As mentioned in section 1.2, the aim of the thesis project was to increase the use and reliability of CAE to investigate design changes in the fuel system. After the verification and validation studies, the simulation model was used to investigate changes to the fuel system.

Three different design parameters of interest were chosen to be investigated:

1. The recirculation line diameter
2. The angle of which the nozzle (and hence fuel pistol) is inserted into the capless unit
3. The pressure drop over the canister and EVAP-line

All three design studies included starting the simulations with an initial fuel volume of 34 l and running them for a total of 3 seconds physical flow time each.

3.3.1 Recirculation Line Diameter

Due to the recirculation line being an important part in the refuelling performance of the system [22], it was also in the interest of Volvo Cars to investigate and understand the effects of changing its diameter. The recirculation line itself varies in diameter through the pipe where the widest part is 8.5 mm in diameter and the narrowest section measures 4.7 mm in diameter. The widest part, 8.7 mm, is found directly where the line connects to the fuel tank. This diameter reduces to 4.7 mm before the first bend and is kept at a constant diameter for another 74 mm of pipe. Over a clip-on connector, the diameter then increases to 6.0 mm and is constant through the entire pipe up to where it connects to the filler neck. The dimensions and how they change can be observed in Figure 28.

The recirculation line investigation was twofold. The first was to investigate the effects of pinching the existing recirculation line to create a constriction of diameter 2 mm. The second part was to understand the effect of varying the recirculation line diameter on refuelling performance. In order to do so, simulations were performed with recirculation lines with constant diameters of 4 mm, 6 mm and 8 mm over the entire length of the pipe.

The first recirculation line case investigated included modifying the 4.7 mm wide section to include a 10 mm long choke with a diameter of 2.0 mm which had the effect of restricting
the flow through the recirculation line. This solution provides a cheap and easy way to implement a flow restriction in the recirculation line and could be practical to include in production. All of the new designs are presented in Figure 29.

\[\text{Figure 29:} \ (a) \text{ Shows the 2mm choke inserted into the 4.7 mm section of the recirculation line. (b), (c) and (d) show the 4 mm, 6 mm and 8 mm wide recirculation lines respectively.}\]

### 3.3.2 Pump Pistol Nozzle Angle

Up to this point in the project, all simulations have been performed with the pump pistol angle in the same position. In order to understand the effect of the customer changing the fuel pistol angle, an extra simulation was run with a modified fuel nozzle position.

The two cases correspond to two typical fuel pistol positions that could occur during refuelling. The "original", which is the position that has been used so far, represents the scenario where the customer holds the pistol, pushes it as deep as possible into the capless unit and continues to hold it there through the rest of the refuelling process. The "new" angle corresponds to the customer inserting the nozzle into the capless and then taking their hands off the pistol handle such that the top of the nozzle rests on the bottom part of the capless unit due to the weight of the pistol.

In the original nozzle angle, the two points of contact between the capless unit and the pump pistol were at the bottom of the capless unit at the part which extends out of the car and into the atmosphere and the rail in area 2 of Figure 13. This meant that the nozzle pointed down into the bottom surface of the top part of the filler pipe. For the new angle, the nozzle was repositioned in such a way that the pistol now was in contact with the top part of area 1 instead of the rail as was the case for the old angle. This implied that the fuel jet from the pump pistol nozzle now entered the filler pipe with a 3.4° higher incline. Figure 30 shows the positioning of the pump pistol nozzle in its different configurations.
3.3.3 Carbon Canister Pressure Drop

Investigating the pressure drop across the canister was and still is of interest from an ORVR point of view. Through previous experiments at Volvo Cars, it was found that with a higher canister-induced back pressure, the adsorption rate of hydrocarbons in the canister would be lower. This is preferable and would be beneficial in reducing overall VOC emissions during refuelling. Therefore, it was decided to investigate how the system would behave if the viscous and inertial resistance coefficients in equation [27] were changed such that the corresponding back pressures would be 2.5 kPa and 4 kPa respectively, as compared to the original case which was around 0.9 kPa.

Figure 30: The nozzle positions in its original and new configuration.
4 Results

4.1 Experimental Results and Comparison to CFD

Figure 31 and Figure 32 show the static gauge pressure in the tank over three experimental tests for the first and second phases, respectively. As mentioned previously, the first phase corresponded to the top fill case, where the initial fuel level in the tank was below the inlet check valve. Phase II represented the bottom fill case, which implied that the fuel level in the tank was above the inlet check valve.

![Figure 31](image1)  
**Figure 31:** Tank pressure values over three experimental trials for phase I of the refuelling process.

![Figure 32](image2)  
**Figure 32:** Tank pressure values over three experimental trials for phase II of the refuelling process.

It was observed that the test results were consistent, with a slight difference in the exact time instant where the automatic shut off peak occurred. Another observation was that the pressure in the tank remained more or less constant through the entire refuelling process, including during the spill over from the active side to the passive side. The pressure values in the tank and the filler neck obtained from the three experimental tests were subsequently averaged and compared to the results from the CFD simulations. These results are shown in Figures 33, 34, 35, and 36.
Figure 33: Comparison of phase I (top fill) tank pressure obtained from experimental tests and CFD.

It was found that the CFD results were in good agreement with the experimental test results, with the same trends and overall behaviour being adequately captured. For the top fill case (Figure 33), it was seen that CFD consistently under predicted the static pressure in the tank by an offset margin of approximately 0.15 kPa. The prediction of pressure in the filler neck was however found to be more accurate (Figure 34) with slight deviations in the initial 0.4 s.

Figure 34: Comparison of phase I (top fill) filler neck pressure obtained from experimental tests and CFD.

Figure 35: Comparison of phase II (bottom fill) tank pressure obtained from experimental tests and CFD.
As seen from Figure 35 for the bottom fill case, the CFD results initially under predict the experimental values of static pressure by the same margin that was seen in the top fill case. However, the experimental and CFD curves intersect at approximately 3.7 seconds following which the CFD results tend to over predict the experimentally measured static pressure. This time instant corresponded to the fuel flowing over the saddle from the active side and covering the bottom surface of the passive side of the fuel tank. Figure 36 shows that the overall behaviour of the static pressure in the filler neck was adequately captured by the CFD model. However, the CFD model failed to predict the negative pressure peak that occurred at 0.5 s.

4.2 Flow Phenomena Observed from CFD

Figure 37 shows the fuel flow through the filler pipe at a few time instants during the first 0.18 s. The fuel flow was visualised as an iso-surface on the interface between the liquid and gas phases, i.e. the interface between fuel and air.

As mentioned in previous sections, the path and quantity of air entrained into the fuel
system were important parameters that were studied during this research. The path and direction of air flowing between the atmospheric cylinder and the capless unit were visualised using velocity streamlines that were superimposed on a contour plot of static pressure, as shown in Figure 38. The cut-plane that was used to generate the contour plot and streamlines resulted in a top view of the capless unit, filler neck and recirculation line with the atmosphere part of the domain on the right side of the figure. The four windows (a), (b), (c) and (d) in Figure 38 correspond to the same time instants shown in the windows (a), (b), (c) and (d) in Figure 37.

It was observed that the air flowed back out of the capless unit and into the atmosphere for the first 0.14 seconds before reversing direction and flowing into the capless unit and filler pipe. The air flow that leaked back into the atmosphere was mostly the air that was stagnant inside and around the capless unit, since the streamlines show that the flow tended to travel inwards into the tank further downstream in the filler neck. The red region in the middle portion of each image corresponds to the fuel flowing at maximum flow rate. At 0.16 seconds, a slight negative pressure in the filler neck was observed. This negative pressure increased at 0.18 seconds and propagated further downstream, which indicates that the fuel flowing through the filler pipe induces a suction effect on the air entrained from the surrounding atmosphere.

The behaviour of the system for the top fill case can be observed clearly in Figures 39 and 40 which show the mass flow rate of air through various sections and regions of the domain and the static pressure in the filler neck over a longer time frame, respectively. The quantities are presented for the first 1.5 seconds of flow time, since it was found that there were no significant changes in the measured quantities between 1.5 seconds and the end of the simulation (i.e., at 3 seconds).
As mentioned previously, the locations of the cross sectional planes in the filler pipe are expressed as factors of the filler pipe diameter, $d_f$, from the reference plane ($0d_f$) in the capless unit (Figure 23). Locations $6d_f$ and $40d_f$ correspond to cross section planes just before the first bend and the point where the filler pipe connects to the tank, respectively. The data set ‘Rec’ corresponds to the net mass flow through the recirculation line. Negative mass flow rates at the filler pipe planes imply that the air leaks back out of the capless unit and into the atmosphere. A positive mass flow rate for the Rec data set implies that air flows through the recirculation line from the tank to the filler neck. It is interesting to note here that reversed flow of air occurred at both $0d_f$ and $6d_f$, but not at $40d_f$. The mass flow rate through the recirculation line increased steadily over time before plateauing at approximately 0.75 seconds.

Figures 41 and 42 show exactly the same quantities as Figures 39 and 40 respectively, except that the former correspond to the results from the bottom fill case. Similar behaviour was observed for the mass flow rate of air through the different sections, with the most significant difference being the length of time for which reversed flow through the capless unit appeared to occur. In the bottom fill case, reversed flow at $0d_f$ and $6d_f$ was observed over a span of approximately 0.3 seconds, around double that observed for the top fill case. However, the absolute maximum peaks of air mass flow - both leaked and
entrained were found to be lower for the bottom fill case than the top fill case.

Figure 41: Mass flow rate of air through different sections of the filler pipe for the bottom fill case.

Figure 42 shows the gauge static pressure in the filler neck for the bottom fill case. A significant difference from the top fill case was that the negative pressure peak was considerably smaller than the top fill case, with a value of approximately -0.0025 kPa. The pressure subsequently recovered and stabilized at atmospheric pressure after 1 second of flow time.

4.3 Design Change Investigation

4.3.1 Recirculation Line Geometry

The gauge static pressures in the tank for different recirculation line geometries are shown in Figure 43, while the mass flow rate of air through plane 0$d_f$ is shown in Figure 44. It was observed that the pinch caused an overall increase in both tank pressure and entrained air mass flow. When comparing the recirculation lines of different diameters, it was found that increasing the diameter contributed to a lower pressure in the tank, with the larger diameters (6 and 8 mm) also resulting in a negative mass flow through 0$d_f$ over the whole simulation. The results for the 6 mm case were the closest to the original geometry which was expected since the original geometry makes use of 6 mm piping from after the elbow connection at the tank, as shown in Figure 29.
Table 10 shows the time averaged volume flow rates (in l/min) through a few of the different sections of the fuel system for the different recirculation line geometries. As expected, the mass flow rate through the recirculation line drops drastically when the pinch is incorporated into the original geometry, a result of which was an increased rate of flow through the outlet. For the 4, 6 and 8 mm cases, it was observed that the air that leaked back into the atmosphere increased substantially as the diameter increased. It was also observed that as the diameter increased, the flow rate through $6d_f$ increased while the flow rate through the outlet decreased.

Table 10: Volume flow rate in litres per minute through different sections in the fuel system, time averaged over the last 1 s of flow time.

<table>
<thead>
<tr>
<th>Recirculation Line Geometry</th>
<th>Plane 0$d_f$</th>
<th>Plane 6$d_f$</th>
<th>Through Recirculation</th>
<th>Through Outlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>0.03</td>
<td>10.14</td>
<td>10.03</td>
<td>37.05</td>
</tr>
<tr>
<td>2 mm pinch</td>
<td>3.47</td>
<td>9.41</td>
<td>5.91</td>
<td>40.41</td>
</tr>
<tr>
<td>4 mm</td>
<td>3.93</td>
<td>8.17</td>
<td>4.08</td>
<td>41.24</td>
</tr>
<tr>
<td>6 mm</td>
<td>-0.93</td>
<td>9.08</td>
<td>9.86</td>
<td>36.06</td>
</tr>
<tr>
<td>8 mm</td>
<td>-5.73</td>
<td>10.66</td>
<td>16.42</td>
<td>31.27</td>
</tr>
</tbody>
</table>
Figure 45: Pressure drop over the different recirculation line geometries; calculated as time averaged gauge pressures at different axial sections over the last 1 second of flow time.

The pressure drop over the length of the recirculation line for the different geometries is shown in Figure 45. The total pressure drop decreased as the diameter of the piping increased from 4 mm to 8 mm.

4.3.2 Change in Fuel Pistol Angle

Changing the fuel pistol angle resulted in a change in the impingement of the jet of fuel leaving the nozzle. This behaviour is outlined in Figure 46, which shows the volume fraction of gasoline at different cross sectional planes along the filler pipe for both the original case as well as the new angle investigated. A volume fraction value of 1 implies that the region comprises of 100% liquid fuel, while a value of 0 implies that the region is 100% gaseous air.

Figure 46: Time averaged contours of volume fraction of gasoline at different axial planes in the filler pipe for (a) the original fuel pistol angle and (b) the new fuel pistol angle.

From Figure 46a, it can be seen that for the original angle, the fuel jet impinged the bottom of the filler pipe before swirling around and flowing downstream into the filler pipe. In the case of the new angle (Figure 46b), it was found that the fuel jet flowed straight through the initial section of the filler before impinging against the outer radius of the
first bend and thereafter swirling downstream.

Although the flow patterns did not appear to be consistent when the fuel pistol angle was changed, Figure 47 and Figure 48 show that the measured bulk flow quantities (pressure in tank and mass flow rate of air through $0d_f$, respectively) did not vary significantly.

![Figure 47: Measured gauge pressure in the tank for the two different fuel pistol angles.](image)

![Figure 48: Mass flow rate of air through the capless unit for the two different fuel pistol angles.](image)

### 4.3.3 Changing the Canister

The effect of changing the canister (i.e. changing the porous interface properties) can be seen in Figures 49 and 50, which show the gauge static pressure in the tank and the mass flow of air through $0d_f$, respectively. As previously stated, the 2.5 kPa and 4 kPa canister cases were allowed to run for a total flow time of 5 seconds to be able to reach stable values of pressure in the tank. However, as can be observed from Figure 49, it was found that the 2.5 kPa case plateaued to a tank pressure value of 2 kPa whereas the 4 kPa case did not appear to reach a stable value even after 5 seconds of simulation.
Although the static pressure values in the tank did not stabilize even after 5 seconds, the trends in air entrainment for increasing canister back pressure can be seen in Figure 50. It was found that an increase in canister back pressure resulted in significant leakage of air through the capless unit into the atmosphere.

Figure 49: Measured gauge pressure in the tank for the two different canister back pressures.

Figure 50: Mass flow rate of air through the capless unit for the two different canister back pressures.
5 Discussion

5.1 Solver Setup

As mentioned previously in section 3.1.5, the flow patterns in the filler pipe (Figure 25) were observed to be inconsistent with each mesh refinement. Initial test simulations (results not presented) showed that the mesh in the filler pipe was crucial to maintaining a stable solution. This is because the filler pipe is the region of the domain where the velocity of the fuel is the highest. As a result of this high velocity flow through the relatively small diameter filler pipe, there is a relatively violent mixing of air and fuel which then gets transported down into the tank. In addition to the mixing, the geometry of the filler pipe itself induces a strong swirling motion to the air-fuel mixture as it flows down into the tank. These physical phenomena have been observed and documented with transparent filler pipes over several internal experimental tests at Volvo Cars and have even been a topic of research previously. [4] This information led to strong speculations during the current study on whether such a flow problem could even be addressed with a RANS modelling approach and if the isotropic assumption for turbulence modelling could be used. Gunnesby previously evaluated the applicability of the DDES model on filler pipe flow and found that no major differences were observed between RANS and DDES. [4] However, the DDES simulations were performed using the same mesh setup as for the RANS case which meant that the mesh resolution ($\Delta x$, $\Delta y$ and $\Delta z$ values) might have been insufficient. This shows that multiphase filler pipe fuel flow could possibly be investigated further, preferably with a scale resolving simulation (SRS) model to resolve some of the turbulence. At the same time, it is important to note that this would involve further research in hydrocarbon based automotive fuel systems, when the automotive industry itself as a whole is striving towards research and development in electric propulsion.

An initial test simulation was performed using constant density gas as the fluid model for air, with all other solver settings remaining the same. It was found that the solution was inherently unstable and tended to produce non-physical results. Previous studies so far have not specified nor documented any information about the model that has been used for the gaseous phase, i.e. whether the ideal gas or constant density model had been used. While the exact cause for these instabilities is unknown, there are some speculations that the porous baffle interface in the venting tube outlet could be the cause for these instabilities. At the time, theory documentation on the implementation of the porous baffle interface for multiphase VOF simulations was unfortunately limited, except for a general guideline on how one is expected to use the feature in STAR-CCM+.

The verification study was performed with a fixed time step of 3.33E-5 s. This was done to ensure that the spatial discretization in the solution could first be verified before testing different time steps to verify the temporal discretization. Selecting a low enough time step, such as 3.33E-5, more or less ‘ensured’ that the solution was independent of time step before attempting to optimise the setup for faster run times through an adaptive time step control. As such, it was found that the solution required a low time step (in the order of 5E-5 s) for the first 0.3 seconds in order to maintain stability before ramping up to an approximate value of 9E-5 s for the remainder of the simulation. The first 0.3 seconds of
the simulation corresponds to the fuel still flowing through the filler pipe and represents a point in time where the flow has still not reached a stable flow pattern yet.

5.2 Model Validation

Comparing the experimental results with the CFD results show a discrepancy in the behaviour of the CFD predictions between the trends observed for phase I and phase II. In Figure 33 it can be observed there is a constant offset such that the CFD underestimates the static pressure through the full 3 seconds of refuelling. It could be argued that the difference in pressure is present due to the vapour pressure created by the gasoline vapours in the tank. Vapour pressure is an effect of evaporation and is not captured by the CFD due to the exclusion of an evaporation model in the simulation. Similarly, Figure 35 shows how the pressure indicated by the CFD simulation overestimates the experimental results between 3.8 - 5.0 seconds of refuelling. It could be argued that the decrease in pressure occurs in the experimental test as a result of slightly warmer fuel coming in contact with the cooler surface that is the passive side of the tank. As the heat transfers from the fuel to the fuel tank and cools down, the fuel vapours could condense and create a lower pressure. While care was taken to ensure that the fuel and ambient tests cell were maintained at the same temperature, there is a possibility that the tank might have been at a slightly lower temperature.

5.3 Flow Physics

5.3.1 Top fill case

During the first 0.1 seconds, the flow rate ramped up from zero to maximum and there was no fuel in the filler pipe yet. This caused a slightly higher pressure in the capless unit and filler neck area which created an adverse pressure gradient for the fresh air coming in from the atmosphere into the capless unit. The adverse pressure gradient can be seen from Figure 40 where the highest pressure in the filler neck was observed at 0.1 seconds. The air that was already in the capless unit then followed the path of least resistance and flowed back out from the capless unit and leaked back into the atmosphere, as seen in Figure 38.

The peak of mass flow through $0d_f$ was of larger magnitude than $6d_f$ in Figure 39 which suggests that the air that leaked out was only the air in the close vicinity of $0d_f$ and in the capless unit itself but not from further down in the filler pipe. There was a slight reversed flow observed through $6d_f$ at 0.1 seconds, which could be due to the air in the capless and filler neck region once again finding the path of least resistance, i.e. through the capless unit and back into the atmosphere.

At 0.1 s, the fuel exited the nozzle and maximum flow rate was achieved. Here, the air mass flow rate through $0d_f$ and $6d_f$ rapidly increased to the same peak value, approximately 1 g/s at exactly 0.2 seconds. This point also corresponded to the maximum air flow into the tank ($6d_f$), which implied that the fuel flow pushed the air that was previously in the filler pipe down into the tank. The air mass flow switched direction and flowed inwards while
the pressure switched from a positive value to a negative pressure at 0.14~0.15 seconds, which corresponded to the point where the fuel just reached the first bend of the filler pipe. The time window between 0.1 to 0.2 seconds corresponded to the flow exiting the nozzle and passing the first bend. The second half of this window was where there was air entrained into the system.

Air entrainment after 0.2 seconds decreased steadily over time and was because the pressure in filler neck continued to rise. This can be attributed to the pressure in the tank rising steadily over time, thereby causing a resistance to flow. After the flow settled at approximately 1.2 seconds, the mass flow rate of air through the filler pipe was purely due to the recirculated mass flow rate through the recirculation line. The oscillations can be attributed to the highly unsteady and chaotic motion of the two-phase flow in the filler pipe.

From these results, it can be observed that for this specific case with this filler pipe geometry, nozzle position, fuel flow rate, etc., air entrainment occurs only during the first 1.3 seconds of the refuelling process, where the first 0.15 seconds correspond to leaking air from the capless unit into the surroundings. One might argue that the neglection of the inlet check valve might be the cause for the air leakage during the first 0.1 seconds. However, on observation of the mass flow rates, the mass flow rate through $40d_f$ was observed to remain positive from the beginning to the end which implied that no air leaked backwards from the tank, through the filler pipe and out into the atmosphere. This proves that the ICV can be neglected for refuelling simulations such as the one in this study, where the modelling of automatic shut off is not investigated.

5.3.2 Bottom fill case

The overall air entrainment behaviour of the bottom fill case was more or less the same as the top fill case, but there were still a few differences that were observed. It is important to note that in this case, the initial fuel level corresponded to the active side being full just before the height where fuel would spill over from the active to the passive side. This implied that there was also a certain amount of fuel in the bottom part of the filler pipe, i.e. up to a height lower than the saddle.

As observed with the top fill case, there was an initial pressure increase in the capless unit up to 0.1 seconds, where consequently the reversed flow out of the capless unit into the atmosphere was also the highest. The pressure in the neck then immediately began to decrease when the fuel flow rate reached maximum flow and subsequently created a suction pressure which caused air to be entrained into the filler pipe. The negative pressure occurred at approximately 0.31 seconds, which corresponded to the fuel flowing past the first bend in the filler pipe. The magnitude of the negative pressure peak created was almost 4 times smaller than that for the top fill case. This could be due to the increased resistance at the end of the filler pipe caused by the stagnant fuel, which did not allow the freshly dispensed fuel to create a strong enough suction. As a result, the maximum mass flow of air entrained is also significantly lower than that in the top fill case.
The time window at 0.42-0.45 seconds corresponded to the dispensed fuel and air mixture reaching the ICV and flowing into the tank. The air mass flow rate through $40d_f$ immediately peaked at this time instant and subsequently decayed over time. It is interesting to note that this time instant also corresponded to the negative pressure peak observed in the filler neck, which showed that the suction pressure in the filler neck occurred as the fuel completely filled the filler pipe. As in the top fill case, the consequent air mass flow through the filler pipe was observed to be only from the recirculating air being fed back into the filler pipe from the recirculation line, i.e. there was approximately no air entrainment through $0d_f$.

5.4 Design Changes

5.4.1 Recirculation Line

It was found that the original recirculation line geometry resulted in an air entrainment through $0d_f$ which oscillated around 0 g/s, as can be seen in Figure 44. These results are considered acceptable but not ideal from an ORVR performance point of view. It is ideal, or rather a safe design, to have an air entrainment consistently above 0 g/s in order to completely eliminate the risk of allowing hydrocarbons to escape into the atmosphere through the filler head during refuelling. The 2 mm pinch solution offers a cheap and easy alternative to the current design of the recirculation line as it shows good air entrainment capabilities. However, the 2 mm pinch also induces a higher tank pressure and mass flow rate through the outlet (Table 10) due to the increase in air entrainment. As a result, there is a higher risk of PSO and higher canister loading, respectively. The 4 mm design shows similar trends to the 2 mm pinch design, however, replacing the entire length of the recirculation line is arguably more expensive from a mass production point of view and hence less optimal than using the 2 mm pinch design to improve the fuel system refuelling performance. It could also be observed from the results in Table 10 that a decrease in mass flow out of the system directly correlated to air leaking back into the atmosphere. In Figure 45, a drastic pressure drop can be observed for the 2 mm pinch design. This behaviour is caused due to the sudden contraction caused by the 2 mm orifice in the recirculation line.

When comparing the 4 mm, 6 mm and 8 mm recirculation line designs, the same trends in measured tank pressure could be observed in the CFD results as have been found experimentally by Mastroianni et al. [22]. As the recirculation line diameter increases, the mass flow rate through the recirculation line increases which in turn pushes gases out of the filler pipe and into the atmosphere instead of pulling it back down into the fuel tank. The tank pressure also decreases with the larger diameters.

Consequently, the results obtained in the recirculation line design investigation strongly signify the importance of the part in the fuel system itself as it greatly affects the performance of the whole system. It is therefore imperative to find a good balance that ensures a continuous air entrainment while minimising canister loading and risk for PSO.
5.4.2 Nozzle Angle

The nozzle angle investigation provided an understanding, to a limited extent, in whether the customer could influence the refuelling performance by holding the pump pistol in different configurations. If the customer holds the pump pistol during the entire refuelling and forces the nozzle as deep as possible into the capless unit, the angle would resemble the 'original' case. If the customer uses the automatic fill function and releases the pump pistol, allowing the weight of the pump pistol and hose to pull the pump pistol nozzle higher up into the capless unit, the 'new' angle is achieved. It could be observed that the angle of the pump pistol affects the flow pattern into the filler pipe (Figure 46). However, no changes in refuelling performance was found in terms of air entrainment and tank pressure behaviour (Figure 48 and Figure 47). Therefore, during normal refuelling conditions, it can be inferred that the fuel system is independent of customer uncertainty with regards to holding the pump pistol versus allowing it to sit under its own weight in the capless unit.

5.4.3 Canister Pressure

After allowing the canister pressure investigation simulations run for 5 seconds of physical flow time, it could be observed that the tank pressures were not yet stabilised. It was deemed impractical to allow the simulations to run further due to time limitations and were hence aborted. Since the design study aimed to provide a general understanding of the trends associated to making changes to the carbon canister, the tests were deemed successful. It can be observed in Figure 50 that with increasing canister resistance, air leakage through $0d_f$ increases. This can be directly correlated to the increase in resistance caused to the flow travelling into the tank. It is interesting to note that just a 1 kPa increment in canister back pressure (the 2.5 kPa case) immediately resulted in air leakage through the capless unit, which shows that canister back pressure is another design parameter that must be carefully calibrated to achieve optimum performance.

5.5 General Comments

It was observed that there was a significant pressure drop over the length of the recirculation line, with the $90^\circ$ bend at the connection to the tank causing a significant drop in static pressure. Dake et al implemented a field function based boundary condition that was calibrated based on experiments to include the effects of this pressure drop. The result of doing so was that the physical geometry of the recirculation line was excluded from the CFD model. While the exact implementation of this method is unknown, it is believed that in order to investigate design changes of the recirculation line, it is necessary to include the physical geometry and capture the associated flow physics as well.

[26] This study excluded the effects of the ICV opening and closing as the fuel exits the filler pipe and enters the fuel tank. However, on closer observation of Figure 39 and Figure 41 it can be seen that the mass flow of air through $40d_f$, which is the plane at the tank inlet, is always positive. This implies that the initially observed reversed flow through the capless unit back into the atmosphere is independent of the ICV being included in the system. However, in order to be able to model and investigate automatic shut-off and spitback, it
is estimated that the movement ICV will need to be modelled since it is one of the main components that helps avoid the fuel flowing back up the filler pipe and causing spit-back. Other aspects to consider to be able to model automatic shut-off would be to include the physics of FLVV and the buoyancy effects of the floater inside it, as mentioned previously.

One shortcoming of the present study is the simplified carbon canister model, which was achieved through the use of a porous baffle interface in STAR-CCM+. It would be ideal to instead include a carbon bed with a calibrated adsorption model to accurately predict the change in pressure at the outlet of the venting box. However, the implementation of such a model would also require the modelling of the evaporation of fuel, which further increases computational costs.

By observing the results obtained for the recirculation line and carbon canister design investigations, it would be an interesting subject to further investigate to combine the two design changes. As seen in Figure 50, with increasing canister back pressure, there is an increased amount of air leaking out of the filler pipe and into the atmosphere. Additionally, it was found that with the implementation of a 2 mm pinch in the recirculation line, the air entrainment into the system could be increased (Figure 44). It could be argued that if both these design changes were combined, a solution could be obtained where air would be entrained into the system while simultaneously decreasing hydrocarbon adsorption through the canister by implementing a higher back pressure.
6 Future Work

This thesis study has laid the groundwork for possible future developments of the model. While the current fuel system evidently is performing well, there are several areas which would be interesting to investigate further in order to improve its performance. Including a model for evaporation and condensation could be useful when using the model to investigate the movement of gasoline vapours in detail. It would also be useful when further investigating the behaviour of the pressure in the tank, as the results from the current CFD model and experimental tests show a constant offset that has been speculated to be due to the absence of evaporation modelling.

As discussed in section 5, Gunnesby [4] has investigated the flow through the filler pipe using the IDDES turbulence modelling approach, however, the mesh used was not modified from the RANS investigation and could in such be insufficient. It is therefore suggested as future work to perform an extensive study of the filler pipe flow using suitable SRS models.


7 Conclusion

Through the course of this study, a experimentally validated CFD model to simulate the refuelling process of gasoline has been developed using the RKE turbulence model and the Eulerian VOF multiphase model. Due to computational limitations, including a full 112 second refuelling simulations was discarded and the study focused on two critical time windows instead. These included simulating the fuel flow for a few seconds, starting at 7.1 l and 34 l of initial fuel volume. During the mesh verification study, it was found that refining the mesh over the entire length of the filler pipe was crucial. After performing the mesh verification study, an adaptive time step function was implemented which controlled the time step based on the average and maximum CFL values at the fluid interface. This was proven to reduce the number of iterations necessary for a 3 second simulation from 721 000 iterations to 263 000 iterations.

From the design study, it can be concluded that the recirculation line is a very important component of the fuel system. It directly contributes to the air entrained and therefore the amount of fuel vapour that leaks back out into the atmosphere. It was found that the fuel pistol angle does not have a significant effect on the performance of the fuel system and as a result removes a certain degree of uncertainty caused due to the customer. Changing the canister to induce higher back pressures than the stock setup would result in fuel vapour leaking out to the atmosphere through the capless unit. Alternatively, it would be interesting to study how the system would perform if changing both the canister and the recirculation line in the same geometry. As restricting the recirculation line increases the air entrained into the system and increasing canister back pressure has the opposite effect, it could be argued that implementing both design changes simultaneously could show an overall improvement of the ORVR performance while keeping the air entrainment around 0 g/s.

While the CFD model developed in this project is not in itself enough to eliminate the need for physical testing, the aim to help reduce development lead times and costs was achieved as the model can be used to investigate design changes in an early development stage. As such, the Fuel System department at Volvo Cars will have the ability to discover shortcomings with early designs, make changes to said design and consequently find a working model before moving onto prototype manufacturing and testing.


References


