Flow and Air Quality Modelling of a Car Cabin

OSKAR JOLÉRUS

June, 2019
Master’s Thesis, Master of Science in Engineering Physics, Umeå University.

*Flow and Air Quality Modelling of a Car Cabin*

Copyright © Oskar Jolérus, 2019.

Supervisors: Michael Mackenzie, Volvo Cars, Maria Bernander, Volvo Cars, and Martin Hübert, Volvo Cars.

Examiner: Eddie Wadbro, Department of Computing Science, Umeå University.
Abstract
Adverse health effects attributable to both short- and long-term exposure to air pollution have turned the focus on different microenvironments. The interior of vehicles is of relevance as road traffic emissions and re-suspension of road dust are major sources of pollutants associated with adverse health effects. Hence, the air quality inside vehicles deserves attention regarding human health. This thesis presents a new virtual methodology, using CFD, to study the distribution of fine particulate matter, PM2.5, inside a car cabin. In the CFD model, unsteady RANS and Lagrangian particle tracking were used to simulate particles entering from the exterior. In this study, a practical measurement of interior particle concentrations was also carried out as a first attempt to validate the CFD model. The objective was to find positions inside the cabin where elevated concentrations of PM2.5 are present. The results from the CFD simulations showed that significantly higher concentrations are present at head height in the front row. Due to a discrepancy in the investigated positions in the CFD model and the practical measurement, the simulation results could not be validated. Nevertheless, the simulation results in this study have provided guidelines for future measurements of interior particle concentrations.
Acknowledgements

First of all, I would like to address gratitude to my supervisors Michael Mackenzie, Maria Bernander and Martin Hübér. Thank you for your support, guiding and willingness to help me during this study.

I would also like to express gratitude towards everybody else at Volvo Cars that have helped me without hesitation. I could not have asked for better support whenever I needed help.

Finally, I want to thank my family for your unconditionally support during these 5 years.
Contents

1 Introduction ............................................. 1
   1.1 Background ........................................... 1
   1.2 Climate Control System ............................... 2
   1.3 Objective ............................................. 4

2 Theory ..................................................... 5
   2.1 Eulerian-Lagrangian Multiphase Flow .................. 5
      2.1.1 Eulerian Air Flow Model ......................... 5
      2.1.2 Lagrangian Particle Tracking Model ............... 7
      2.1.3 Turbulent dispersion ............................ 9
   2.2 Boundary layer ....................................... 9
   2.3 Wall Treatment ...................................... 10
   2.4 Particle-Wall Interaction ............................ 11

3 Method .................................................... 12
   3.1 Geometry ............................................ 12
   3.2 Mesh ............................................... 13
      3.2.1 Mesh Regions ..................................... 13
   3.3 Boundary Conditions .................................. 16
   3.4 Particle Injection ................................... 16
   3.5 Particle Concentration ............................... 17
   3.6 In Field Measurement ................................. 19

4 Results .................................................. 21
   4.1 In Field Measurement .................................. 21
      4.1.1 Particle Concentrations in Zone 2 ................. 22
      4.1.2 Particle Concentrations in Zone 3 ................. 24
   4.2 Simulations .......................................... 25
      4.2.1 Air flow 386 kg/h ................................ 26
      4.2.2 Air flow 183 kg/h ................................ 32
      4.2.3 Comparison Between Mass Flows .................. 35

5 Discussion & Conclusions ................................. 36
   5.1 Improvements of the Simulation Methodology ............ 37
   5.2 Future Work ......................................... 38
1 Introduction

1.1 Background

Air pollution is a global problem that represents one of the biggest environmental risks to human health. The World Health Organization estimates that around 4.2 million people die each year due to ambient (outdoor) air pollution, and including the indoor air pollution, the number would exceed 7 million [1]. Atmospheric pollution consists of substances, such as particulate matter (PM), reactive nitrogen substances and organic compounds [2]. Particles can be defined in different ways, and the parts that are regulated are PM10 and PM2.5, where PM2.5 is called fine particles. The classification of the particles is based on their aerodynamic diameter, where PM10 has a diameter smaller than 10 µm and PM2.5 has a diameter smaller than 2.5 µm. 92% of the world population lives in areas which have levels exceeding the WHO guideline for PM2.5, i.e., 10 µg/m³ as an annual mean and 25 µg/m³ in 24-hour mean [3].

Both short- and long-term exposure to PM2.5 is associated with adverse health effects including, lung cancer, respiratory and cardiovascular diseases [3]. In Sweden air pollution concentrations are among the lowest in Europe and the world, despite that, exposure to PM2.5 is estimated to cause around 3 500 deaths per year and people suffer from symptoms attributable to air pollution [4].

The distribution of air pollution is not evenly distributed around the world and not even locally. The highest concentrations can be found closest to the source of the pollutants. In a road environment, there are several sources of PM, where the major sources are exhaust emissions and re-suspension of road dust. Road dust arises mainly from the wear of the road surface, brakes, and tyres [4]. This means that cars are largely affected by elevated levels of PM2.5, therefore, car manufacturers need to prevent particles and other pollutants from entering the interior of cars. In some parts of Europe, the average daily driving time is between 1 and 2 hours [5], and in the US people spend on average 6% of their time inside vehicles [6]. Over a long term perspective improving the particle concentration within vehicles might not have a major impact on human health. However, several researchers have shown a short-term effect on lowering the particulate concentration, leading to improvements in cardiac and blood parameters [7, 8, 9]. Hence, good air quality inside vehicles deserves attention concerning human health.

There are no legal requirements regarding interior PM2.5 concentrations for car manufacturers. There are, however, requirements for some pollutants, such as carbon monoxide, nitrogen oxides, and hydrocarbons. For Volvo, good air quality in the cars is in line with their human-centric approach. Hence, requirements for
particulate concentrations within the vehicle have been established from research data along with the guideline values from the WHO for ambient concentrations. Measurements have been carried out looking at the difference between the concentration outside and inside the vehicle, an example can be seen in Figure 1, where it is shown that Volvo has a system which can remove above 95% of the PM2.5.

**Figure 1** – PM2.5 concentrations outside and inside a vehicle during a measurement of air quality.

### 1.2 Climate Control System

In Volvo cars, the cabin air intake is controlled by a climate control system. The purpose of the system is to ensure climate comfort and good air quality. It also defrosts and demist the windscreen and the windows. The system is very complex, hence this section is intended to only give an overview of how it works. In Figure 2, a simplified XC60 climate control system is shown. Covering the plenum cover, the HWAS (Heat, Water, and Air Separator), the HVAC (Heat, Ventilation, and Air Conditioning) unit, ducts, and vents.
Outside air enters the car through the plenum cover that is located at the base of the windscreen. The cover contains a water list that prevents water from entering the system and a grid that prevents, e.g., leaves from entering. Below the grid, the HWAS is located. In a brief description, it prevents hot air from the engine to enter the system, and it collects and drains water that manages to pass the water list. Inside the HWAS an AAC (Advanced Air Cleaner) is also positioned. The AAC ionize particles so that they are more easily filtered. The HWAS is in direct contact with the outside air intake in the HVAC unit.

The main function of the HVAC unit is to provide climate comfort according to the automatic settings or the settings set by the vehicle occupants. The HVAC unit first controls the amount of outside air (OSA) and recirculated air (REC) that reaches the cabin, then a blower draws all air through the multi-filter that minimizes the number of particles that enters the cabin. In addition, the filter contains active charcoal for volatile organic compounds, noxious gases, and odour control. Further, to ensure good air quality inside vehicles in polluted areas, there is an air quality sensor in the climate control system. The sensor measures relative changes in air quality. If the sensor measures a drop in air quality, e.g., when driving through a tunnel, the car automatically closes the OSA intake and switches to REC.
After the filter, air passes through an evaporator that cools down and dehumidifies it, and after the evaporator, air flows through heaters and/or bypass ducts. The amount of air that flows through the heaters and the bypass ducts is determined by the input temperature settings to the climate control system. The air is then mixed into a homogeneous temperature and distributed via ducts to the cabin vents. The air distribution between the various vents is determined by flap settings in the HVAC unit.

The main evacuation of air is through extractors that are located behind the rear wheels. A small amount of air also leaves through regions that are not 100% sealed, e.g., the doors. As air is drawn into the cabin, the cabin air pressure is above the exterior if doors and windows are closed. This prevents air from entering the cabin through other places than the climate control system, such as the extractors. If the cabin pressure drops below the exterior, there are mechanical flaps in the extractors that prevents air from entering through them.

1.3 Objective

The objective of this thesis is to develop a new virtual methodology using CFD (Computational Fluid Dynamics) to study particle flow inside a car cabin. A combined flow and particle model is created and applied to different scenarios to see how particles behave in the interior of a car. The model should be able to describe the particle distribution inside the cabin and find positions where elevated particle concentrations are present. The results from the simulations could be used to, e.g., optimize the positioning of sensors during measurements of air quality. The software STAR-CCM+ is used for the methodology development with an XC60 as a reference vehicle. A first attempt to validate the model is also performed with a measurement of interior particle concentrations.
2 Theory

2.1 Eulerian-Lagrangian Multiphase Flow

The numerical methodology used in this study is based on a Eulerian-Lagrangian modelling framework. The dispersed particles are treated as a discrete phase and solved in a Lagrangian reference frame while the air flow is described as a continuous phase and solved in a Eulerian frame. The motion of the particles is described by Newton’s second law of motion and the air flow is described by the Navier-Stokes equations. Interaction between the phases can be coupled via one-way or two-way coupling. With one-way coupling, the particles are influenced by the air flow and it is assumed that the particles have a negligible effect on the air flow, whereas two-way coupling includes the particles effect on the air flow as well. Two parameters used to determine the coupling interaction are the volume fraction of the dispersed particles and the particle Reynolds number. For low particle volume fractions and particle Reynolds numbers, less than $10^{-6}$ and $10^{3}$, respectively, it is expected that the one-way coupling assumption is sufficient [10]. If the volume fraction of the dispersed particles exceeds $10^{-3}$, particle-particle interactions become significant and four-way coupling, which also includes inter-particle interactions, should be used. However, in this study one-way coupling was used since the requirement for the assumption is fulfilled in the flows studied.

2.1.1 Eulerian Air Flow Model

To solve the air flow in this study the unsteady Reynolds-averaged Navier–Stokes equations (RANS equations) were used. The RANS equations are time-averaged equations of motion, they do not describe the details of turbulence only the effect of turbulence on the mean flow [11]. The concept behind the RANS equations is Reynolds time-averaging where any variable in Navier-Stokes equations is decomposed into a sum of a mean and a fluctuating part. The RANS equations for an incompressible Newtonian fluid can be written as

$$\frac{\partial u_i}{\partial x_i} = 0$$

(1)

and

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = - \frac{1}{\rho} \frac{\partial}{\partial x_j} \left[ p \delta_{ij} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \rho \langle u'_i u'_j \rangle \right],$$

(2)

where $u_i$ is the mean velocity, $u'_i$ is the fluctuating velocity, $p$ is the mean pressure, $\rho$
2.1 Eulerian-Lagrangian Multiphase Flow

is the density and \( \mu \) is the viscosity. For details of the derivation of the equations see [11, 12]. The RANS equations still contain fluctuating velocities in the Reynolds stress term, \( R_{ij} - \rho \langle u'_i u'_j \rangle \), and for closure of the equations the term needs to be modelled. In this study the realizable \( k-\epsilon \) turbulence model was adopted since it is commonly used in industrial applications and well suited for flows with strong streamline curvature and rotation [12]. The turbulence model provides closure to the RANS equations based on the concept of eddy viscosity. It is a two-equation model which contains two additional transport equations for the turbulent kinetic energy \( k \) and the turbulent dissipation rate \( \epsilon \). The equations are defined as follows

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho ku_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \tag{3}
\]

and

\[
\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S_\epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\nu \epsilon}} + C_1 \epsilon C_3 \epsilon G_b + S_\epsilon \tag{4}
\]

where \( C_1 = \max(0.43, \frac{\eta}{\eta + \sigma}) \); \( \eta = S^k \); \( S = \sqrt{2S_{ij}S_{ij}} \); \( S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \); \( \mu_t \) is the eddy viscosity, \( \sigma_k = 1.0 \), \( \sigma_\epsilon = 1.2 \), \( C_{\epsilon 1} = 1.44 \), and \( C_2 = 1.9 \) are model constants, \( G_k \) represents the generation of turbulence kinetic energy due to the mean velocity gradients, \( G_b \) is the generation of of turbulence kinetic energy due to buoyancy, \( Y_M \) represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate, and \( S_k \) and \( S_\epsilon \) are user defined source terms [13]. The transport equation for \( \epsilon \) is derived from an exact equation for the transport of the mean-square vorticity fluctuation, and the eddy viscosity is defined as

\[
\mu_t = \rho C_\mu \frac{k^2}{\epsilon}, \tag{5}
\]

where \( C_\mu \) is not constant as in the standard \( k-\epsilon \) model. The definition of \( C_\mu \) in the realizable \( k-\epsilon \) turbulence model can be found in [13].

In addition, the two-layer approach was used together with the realizable \( k-\epsilon \) turbulence model. In the approach, the turbulent dissipation rate and turbulent viscosity are specified as functions of wall distance within the viscosity dominated
near-wall regions. The transport equation for the turbulent kinetic energy, Eq. 3, is solved in the whole domain and the transport equation for the turbulent dissipation rate, Eq. 4, is solved in regions far from walls and blended with the near-wall values. The turbulent dissipation rate and turbulent viscosity as functions of wall distance can be found in [14].

2.1.2 Lagrangian Particle Tracking Model

The particles were assumed to be spherical and solid with constant density and the individual particle trajectories are solved from Newton’s second law defined as

\[ m_p \frac{du_{ps}}{dt} = F_i, \]  

where \( m_p \) is the particle mass, \( u_{ps} \) is the particle velocity and \( F_i \) are the forces exerted on the particle. The forces, \( F_i \), can be decomposed into

\[ F_i = F_{Di} + F_{gi} + F_{LSi} + F_{ai}, \]  

where \( F_{Di} \) is the drag force, \( F_{gi} \) is the gravity force, \( F_{LSi} \) is the shear lift force and \( F_{ai} \) are the additional forces exerted on the particle. The drag force is defined as

\[ F_{Di} = \frac{1}{2} C_D \rho A_p |u_{si}| u_{si}, \]  

where \( C_D \) is the drag coefficient of the particle, \( \rho \) is the air density, \( A_p \) is the projected area of the particle, \( u_{si} = u_i - u_{pi} \) is the particle slip velocity and \( u_i \) is the air velocity. The Schiller-Naumann correlation that is suitable for spherical solid particles was chosen to model the drag coefficient, the correlation is defined as

\[ C_D = \begin{cases} \frac{24}{Re_p} (1 + 0.15 Re_p^{0.687}), & Re_p \leq 1000 \\ 0.44, & Re_p > 1000 \end{cases} \]  

where \( Re_p \) is the particle Reynolds number defined as

\[ Re_p = \frac{\rho |u_{si}| d_p}{\mu}, \]  

where \( d_p \) is the particle diameter [15]. The gravitational force is defined as
2.1 Eulerian-Lagrangian Multiphase Flow

\[ F_{gi} = m_p g_i, \]  

(11)

where \( g_i \) is the gravitational acceleration. The shear lift force that is important where large velocity gradients in the air flow are present is defined as

\[ F_{LS_i} = C_{LS} \frac{\rho \pi}{8} d_p^3 \epsilon_{ijk} u_s \omega_k, \]  

(12)

where \( C_{LS} \) is the shear lift coefficient, \( \epsilon_{ijk} \) is the Levi-Civita symbol and \( \omega_k = (\nabla \times \mathbf{u})_k \) is the curl of the air velocity [16]. Further, the shear lift coefficient used is defined as

\[ C_{LS} = \frac{4.1126}{Re_s^{1/2}} f(Re_p, Re_s), \]  

(13)

where

\[
f(Re_p, Re_s) = \begin{cases} (1 - 0.3314 \beta^{1/2}) e^{-Re_p/10} + 0.3314 \beta^{1/2}, & Re_p \leq 40 \\ 0.0524(\beta Re_p)^{1/2}, & Re_p > 40 \end{cases}\]

(14)

where \( \beta = \frac{1}{2} \frac{Re_s}{Re_p} \), and the Reynolds number for a shear flow is defined as \( Re_s = \frac{\rho d_p^2 |\omega|}{\mu} \) [17].

Some of the additional forces exerted on the particles includes Basset force and virtual mass force caused by unsteady flows, and Brownian motion force. The Basset force is due to changes in the boundary layer and the Brownian motion force is due to molecular collisions. In this study, the Basset force and the virtual mass force can be neglected as the ratio of air density to particle density is small [18]. The Brownian motion force becomes prominent for particle size smaller than 0.5 \( \mu \)m, and neglecting the force will lead to errors [19], but since it is not available in STAR-CCM+ it was also neglected.

If the number of particles is high STAR-CCM+ uses a statistical approach to reduce the number of tracked particles. In the approach, parcels that represent particles with the same physical properties are defined and instead of following the evolution of each individual particle in the computational domain, the parcels are tracked.
2.1.3 Turbulent dispersion

As mentioned, the RANS-equations are time-averaged equations of motion, therefore turbulent dispersion of particles are not taken into account in the governing equations of the air flow. Therefore, to take into account the effect of velocity fluctuations in the air flow on the dispersion of the particles, the turbulent dispersion model that is available in STAR-CCM+ was used. The model is based on a random walk technique to synthesize the fluctuating nature of the turbulent velocity field in the continuous phase and the effect off the fluctuations on the particles [14].

2.2 Boundary layer

For a near wall fluid flow, the boundary layer is defined as the region closest to the wall where viscosity becomes dominant. The fluid flow in the boundary layer has different physical properties in different regions, therefore the boundary layer is divided into more layers. The boundary layer is first divided into an outer layer and an inner layer, and the inner layer is further split into a viscous sub-layer, a buffer sub-layer and a fully turbulent sub-layer, see Figure 3.

![Figure 3 – The sub-layers of the inner region within a boundary layer.](image)

To be able to describe properties of the fluid flow in these layers and the thickness of the layers a dimensionless distance is defined by

\[ y^+ = \frac{u_\tau y}{\nu}, \]

where \( y \) is the distance from the wall, \( \nu \) is the kinematic viscosity and \( u_\tau \) is the friction velocity defined as
2.3 Wall Treatment

\[ u_r = \sqrt{\tau_w/\rho}. \]  

(16)

The wall shear stress \( \tau_w \) is given by

\[ \tau_w = \mu \left( \frac{\partial u}{\partial y} \right)_{y=0}, \]  

(17)

where \( u \) is the flow velocity parallel to the wall. The thickness of the sub-layers is defined as: viscous sub-layer \( 0 < y^+ < 5 \), buffer sub-layer \( 5 < y^+ < 30 \), fully turbulent sub-layer \( 30 < y^+ < 400 \) \cite{12}. To describe fluid properties of the flow in these layers a dimensionless velocity is defined as

\[ u^+ = \frac{u}{u_r}, \]

where \( u \) is the fluid velocity. In the viscous sub-layer the dimensionless velocity is equal to the dimensionless distance from the wall,

\[ u^+ = y^+ \]

and in the turbulent sub-layer the flow is proportional to the logarithm of the distance

\[ u^+ = \frac{1}{\kappa} \ln(y^+) + B, \]  

(18)

where \( \kappa \approx 0.42 \) is the von Kármán constant and \( B \approx 5.0 \), which is referred to as the log law. These layers play an important role when defining the centroid of the first mesh cell closest to the wall.

2.3 Wall Treatment

To resolve the fluid flow across the boundary layer, there are three different wall treatments available in STAR-CCM+ (low-, high- and all-\( y^+ \)). The low-\( y^+ \) wall treatment resolves the viscous sub-layer, and it requires a sufficiently fine mesh with near wall cell centroid located at \( y^+ \approx 1 \). In the high-\( y^+ \) wall treatment the viscous sub-layer is not resolved, instead, wall functions are used to model the near-wall region. The requirement on the mesh resolution with the high-\( y^+ \) wall treatment is that the near wall cell centroid should lie within the log-law layer at
2.4 Particle-Wall Interaction

The main advantage of the high-$y^+$ wall treatment is, therefore, savings in computational time due to a lower number of mesh cells.

The only available wall treatment for the realizable k-ε turbulence model with the two-layer approach is the all-$y^+$ wall treatment, which is a combination of the low-$y^+$ and high-$y^+$. For intermediate resolutions, $1 < y^+ < 30$, a blending approach is used. However, avoiding intermediate resolutions is desired [14].

2.4 Particle-Wall Interaction

There are many factors that determine the outcome of a particle-wall interaction and it is therefore very complex to model. The interaction is among other things dependent on the material properties of the particle, the velocity of the particle at impact, the obliquity of the impact and the material properties of the wall surface [20, 21]. There exist analytical equations to determine a critical velocity of rebound, which is defined as the smallest velocity a particle rebound from a surface, for normal impact [21]. However, when it comes to PM2.5 the parameters needed to calculate a critical velocity are difficult to determine. Therefore, elastic bouncing was defined for all hard surfaces and a stick condition was defined for all soft surfaces. The stick condition means that if a particle hits a surface it sticks to the surface.
3 Method

In this chapter, the simulation methodology is presented. The methodology was developed in STAR-CCM+ version 13.04.010 and the pre-processing of the geometry was done in ANSA. A description of the measurement of interior particle concentrations that was carried out is also given. The results from the measurement was used as reference data for the particle concentrations in the simulations, and it was also used to compare with the simulation results.

3.1 Geometry

The geometry that was used in the simulations was a full cabin model of a Volvo XC60. Since the particle concentrations in the simulations were based on interior concentrations, the geometry inlet was defined after the multi-filter in the HVAC. Furthermore, the evaporator and the heaters were also not modeled in the geometry since only the distribution of the particles was of interest in this study. Thus, in the model they work as bypass ducts. The zone between the outlet vents, in the geometry, and the extractors was not included. In Figure 4, an overview of the geometry is shown.

![Figure 4](image)

Figure 4 – An overview of the geometry used in the simulation model.

Furthermore, the model contains all parts in a XC60 full cabin model including the ducts, the flaps, the vents and the nozzles that control the air flow distribution from the inlet to the cabin. In Figure 5, the vents where air flow and particles enters the cabin are highlighted in blue.
3.2 Mesh

The meshing is an important part in a CFD simulation since the numerical accuracy is dependent on the mesh. The denser the mesh, the more accurate the solution is, but a finer resolution also leads to an increased computational time. Hence, defining the mesh resolution is a trade-off between numerical accuracy and computational time. To cope with this, the geometry is often split into different regions where the resolution is based on the geometry and the flow characteristics within the regions. In some regions the resolution needs to be very fine to resolve the flow characteristics, e.g., within the boundary layer where large velocity gradients are present that can lead to large numerical errors [12]. Following this, the cabin geometry was split into different regions where the mesh resolution was finer/coarser.

3.2.1 Mesh Regions

In Figure 6, the defined mesh regions are shown. The regions are arranged from coarser to finer resolution. Some of the parts defined in Region 1 were the trunk, the carpet and the seat in the rear row. Regions 4 and 5 included the parts in the HVAC unit, the ducts, the vents and other parts that needed a fine resolution.
The mesh generation is divided into two steps where the first step is to create a surface mesh and the second step is to create a volume mesh. The resolution of the surface mesh needs to be fine enough to capture the geometry. Since the volume mesh is created with the surface mesh as a starting surface it will be affected by the resolution of the surface mesh as well. Hence, before creating the volume mesh, the surface mesh needs to be meticulously analyzed.

To create the initial surface mesh of the geometry a surface wrapper was executed. For the surface wrapper, the base size was set to 24 mm and the volume of interest was specified as largest internal, i.e., the wrapper extracts the largest internal self-contained volume from the input geometry surfaces [14]. Further, custom controls, specified as percentage relative the base size, was created for each mesh region. In Table 1, the values used in the custom controls are shown.

**Table 1** – The specified mesh sizes, relative to the base size, in all mesh regions.

<table>
<thead>
<tr>
<th>Region</th>
<th>Target Size</th>
<th>Minimum Size</th>
</tr>
</thead>
<tbody>
<tr>
<td>Region 1</td>
<td>50%</td>
<td>12.5%</td>
</tr>
<tr>
<td>Region 2</td>
<td>25%</td>
<td>6.25%</td>
</tr>
<tr>
<td>Region 3</td>
<td>12.5%</td>
<td>3.125%</td>
</tr>
<tr>
<td>Region 4</td>
<td>3.125%</td>
<td>1.5625%</td>
</tr>
<tr>
<td>Region 5</td>
<td>1.5625%</td>
<td>0.78125%</td>
</tr>
</tbody>
</table>
The values used in the surface wrapper provided an acceptable initial surface mesh of the geometry. Furthermore, to improve the quality of the surface mesh and to generate the volume mesh an automated mesh operation was executed. In the mesh operation the surface remesher and the automated surface repair mesher was used to improve the initial surface mesh. To generate the volume mesh the trimmed cell mesher was used. It creates a volume mesh with convex hexahedron cells that grows from the surface mesh.

To resolve the boundary layer in the near wall regions with relatively few cells, the prism layer mesher was also used. In the regions where the near wall flow affects the distribution of the air, and large near wall velocity gradients are present, i.e., in the HVAC unit, the ducts and in the vents, eight prism layers with near wall thickness 0.03 mm was defined. In the other regions where the near wall flow is not as important, two prism layers with near wall thickness 1.2 mm was defined. In Figure 7, the $y^+$ values in the simulation model are shown. The transparent regions has a lower $y^+$ than the lower limit of the color bar in the figures.

![Figure 7](image)

(a) The HVAC unit.  
(b) The ducts.  
(c) The cabin.

**Figure 7** – The $y^+$ values in the simulation model
3.3 Boundary Conditions

As mentioned in Section 2.4, a stick boundary condition was defined for all soft surfaces and a rebound condition was defined for the hard surfaces. The assumption is crude, but it was accepted since a deeper analyze on particle-wall interaction is beyond the scope of this study. In Figure 8, the soft surfaces are highlighted in green, including the front and rear seats, the carpet and the trunk. Further, the inlet was specified as a mass flow inlet and the outlet vents were specified as pressure outlets with atmospheric pressure.

![Figure 8](image) – The soft surfaces inside the cabin highlighted in green.

3.4 Particle Injection

This section describes how particles are introduced in the simulations. As mentioned in Section 2.1.2, particles with the same physical properties are grouped into parcels in STAR-CCM+. Parcels are introduced in simulations through injectors and there are several different types of injectors, e.g., part, point, surface and table. In this study a part injector, specified at the inlet, was used. As default, a part injector injects the same particle flow rate in each injection point. The injection points are defined according to the mesh faces of the input part to the injector. Parcels are also, as default, injected at each injection point and at every time step. This can be computational heavy in some cases, including this, since the computational cost is proportional to the number of parcels. Further, the number of particles injected in the simulations was based on relative differences between different particle sizes extracted from the measurement of interior particle concentrations. To be able to control the number of particles injected is a
key parameter in the simulation model.

For a part injector, there are three ways to control the number of particles injected: The particle flow rate, the number of parcel streams per injection point and the point inclusion probability. The particle flow rate controls the number of particles per parcel and when in the simulation the parcels are injected. To reduce the computational time, parcels were injected every 0.1 s in the simulations instead of every time step. For the same reason, the number of parcel streams per injection point was set to 1. Further, the point inclusion probability specifies the probability that an injection point is included. In Figure 9, two examples on how the point inclusion probability works for the specified injector are shown. The re-randomization was also enabled, which means that the included injection points are re-randomized between each injection.

![Figure 9](image)

**Figure 9** – Two examples on how the point inclusion probability works. In the left case the point inclusion probability is equal to 1, and in the right case the point inclusion probability is equal to 0.1.

### 3.5 Particle Concentration

In the methodology implemented, the particle concentration could not be determined for Lagrangian phases with particle distributions. Hence, the method is based on Lagrangian phases with constant particle diameter, i.e., one Lagrangian phase represents one particle size.

To determine the particle concentration at different positions inside the cabin, blocks were defined at the positions of interest. The blocks were only defined to be able to calculate particle concentrations, therefore, they were not a part of the simulation region and physics. The cabin was divided into four different zones, front row driver side (Zone 1), front row passenger side (Zone 2), rear row driver...
3.5 Particle Concentration

side (Zone 3) and rear row passenger side (Zone 4). In each zone, a block was created on a floor position, a seat position and a head position. In addition, a block was also defined behind the center stack in the rear row. In Figure 10, the defined blocks can be seen.

![Figure 10 – The defined blocks in the simulation model.](image)

Further, a volume data mapper was used to map the Lagrangian volume fraction from the simulation region to the block regions. The key when using a data mapper is to create the source mesh and the target mesh as similar as possible, since the mapping of the quantity of interest is based on interpolation. Hence, the positioning of the blocks was based both on the positions of interest and the simulation region mesh. Also, the dimensions of the blocks was based on the simulation region mesh. In Figure 11, the mesh of the head position block and the floor position block in Zone 3 are shown together with mesh planes for the simulation region.

![Figure 11 – The mesh of the blocks at the head position and floor position in Zone 3, together with mesh planes for the simulation region.](image)
The interpolation methodology used in the simulation model was the nearest neighbor interpolation. The method extracts values from the nearest source cell to the target cell in a specified region of proximity. The region of proximity for the mapping was in the model specified to be smaller than the size of the source and the target cells. Also, the mapping was specified to be from the interior of the source volumes. The particle concentration within a block was then calculated from

\[ n = \frac{1}{V_b V_p} \left( \sum_{i=1}^{N_c} \phi_i V_i \right), \]

where \( V_b \) is the volume of the block, \( V_p \) is the volume of the particle, \( N_c \) is the number of cells in the block, \( \phi_i \) is the Lagrangian volume fraction in cell \( i \), and \( V_i \) is the volume of cell \( i \).

### 3.6 In Field Measurement

The measurements of interior particle concentrations was carried out in Lundbytunneln in Gothenburg. The choice of location was based on the need for high ambient particle concentrations and in Gothenburg, where the air is clean, road tunnels are locations with elevated concentrations.

During the measurement, two Grimm Mini Wide Range Aerosol Spectrometers were used. The instruments detect airborne particles based primarily on the particle diameter and they operate on two principles. For particle diameters ranging from 10 nm to 200 nm, the principle used is electric detection by means of a Faraday cup electrometer. For diameters approximately larger than 200 nm the principal is based on optical detection by measuring scattered light intensity. The instruments measure particle sizes with diameters from 10 nm to 35 µm in 41 channels (10 electrical channels and 31 optical channels), and a measurement cycle is 1 minute [22].

Two instruments where used, they were placed in the front row on the passenger seat, and on the rear row seat, on the driver side. Thus, the measurement zones correspond to Zone 2 and 3 in the simulation model. To be able to measure on different positions within the zones, a radial symmetric sampling head was connected to each instrument via tygon tubes. As in the simulation model, a floor and a seat position was defined within each zone. A third position was also defined in the zones, but the position was located at a lower height than the head position in the simulation model. This was due to limitations in the height adjustment on the lab stands used to hold the sampling heads. Also, as in the simulation model,
a position behind the center stack was defined. In Figure 12, the measurement positions are shown.

![Figure 12](image)

**Figure 12** – The measurement points inside the cabin. A floor, a seat and a chest position were measured in the front row on the passenger side (Zone 2) and on the driver side in the rear row (Zone 3). In addition, a point behind the center stack was also measured.

During the measurement procedure, the particle concentrations were measured at the corresponding position in each zone simultaneously, i.e., each instrument was used to measure one zone. However, at the position behind the center stack both instruments were measuring at the same position. In the measurement each position was measured during 10-15 min. Further, the climate control system was regulated manually during the experiment to control the inflow and distribution of air.
4 Results

The climate control settings in the measurement and the simulations was full OSA (outside air) and distribution, i.e., air flow from all vents within the cabin. In this study, the particle distribution was studied for a low mass flow and a high mass flow of air, 183 kg/h and 386 kg/h, respectively. As mentioned in Section 3.6, the climate control system was regulated manually to control the inflow of air in the measurement. In the CFD simulations the mass flow was specified at the defined inlet. The mass flow and the flap positions within the HVAC unit, in the simulation model, were calculated from output data from the climate control system during the measurement. The data are estimated values from an experimental model based on fan speed, flap positions, car velocity etc. Further, a smaller particle size, 0.253 µm, and a larger particle size, 2.146 µm, were studied. The choice of particle sizes was based on the size channels in the measurement instrument and sizes of interest.

The results of interest in this study are relative differences in particle concentrations between different positions inside the cabin and different mass flows. Therefore, no values of particle concentrations are shown from the measurement. To be able to compare the results, the scaling on the axes in the figures compared are the same. One big difference between the measurement and the simulations is that in the measurement a background concentration of particles was present, which was not taken into consideration in the simulation model.

The relative difference in mean concentrations between different positions and/or mass flows was calculated as $RD_n = 100(n_{\text{high}} - n_{\text{low}})/n_{\text{high}}$, where $n_{\text{high}}$ is the higher concentration in the comparison and $n_{\text{low}}$ is the lower concentration.

4.1 In Field Measurement

In the measurement carried out, particle concentrations at different positions inside the cabin was measured for two different mass flows. Two instruments were used to measure one zone each (Zone 2 and 3). For the larger particle size there was a discrepancy between the two instruments when measuring at the same position, the center stack position, simultaneously. The difference between the instruments was about 24-44% for the two mass flows. This was due to the uncertainty in the measurement caused by low concentrations. Thus, comparing the larger particle size between the two zones are not relevant. The corresponding difference between the instruments for the smaller particle size was about 2%. However, the difference in concentrations between the same position in the two zones was below 10% for the smaller particle size. Instead, the main difference in particle concentrations was measured between the two mass flows.
4.1 In Field Measurement

4.1.1 Particle Concentrations in Zone 2

In Figure 13, the mean particle concentration at each position for the two particle sizes is shown. As seen, higher concentrations of the smaller particle size was measured than for the larger particle size. This is due to higher filter efficiency for the larger particle size, and lower exterior concentrations of the larger size. Further, also due to the filter efficiency, a higher mass flow rate results in a higher particle concentration of the smaller particle size.

![Figure 13](image)

(a) $\dot{m}_{\text{air}} = 183 \text{ kg/h}$

(b) $\dot{m}_{\text{air}} = 386 \text{ kg/h}$

**Figure 13** – The mean concentrations of the two particle sizes with standard deviation for the smaller size in Zone 2.

For the lower mass flow, the position with the highest mean concentration of the smaller size was the floor position. The corresponding position for the higher mass flow was the seat position. As shown, the mean concentrations of the second highest values are within the standard deviation of the highest concentrations. Thus, no position with significantly higher concentration was seen. The largest difference between the mass flows was measured at the seat position (63% difference) and the smallest difference was measured at the floor position (11% difference).
In Figure 14, the mean concentrations of the larger particle size are shown. The concentrations of the larger size inside the cabin was very low during the measurement, and no significant difference between the two mass flows was measured. Thus, a comparison between the mass flows is not relevant for the larger particle size.

(a) $\dot{m}_{\text{air}} = 183 \text{ kg/h}$

(b) $\dot{m}_{\text{air}} = 386 \text{ kg/h}$

**Figure 14** – The mean concentrations of the larger particle size with standard deviation in Zone 2.
4.1 In Field Measurement

4.1.2 Particle Concentrations in Zone 3

As shown in Figure 15, the same relation between the mass flows was measured for the smaller particle size in Zone 3. The largest deviation between Zone 2 and 3 was measured at the floor position for the lower mass flow. The difference between the mean concentrations was 8%. However, the difference was not significant.

![Figure 15](image)

**Figure 15** – The mean concentrations of the two particle sizes with standard deviation for the smaller size in Zone 3.

In Figure 16, the mean concentrations of the larger particle size in Zone 3 are shown. As in Zone 2, no significant differences were measured between the mass flows.

![Figure 16](image)

**Figure 16** – The mean concentrations of the larger particle size with standard deviation in Zone 3.
4.2 Simulations

In the simulations, 13 seconds of physical time was simulated for both mass flows. The time step used in the simulations was 0.0025 s and a second-order temporal discretization was used. The stopping criteria for the number of inner iterations within the time step was based on a point probe in the HVAC unit. The monitored value was the maximum value of the air velocity, and an asymptotic stopping criteria defined as 5 values with smaller difference than 0.01 m/s was used, i.e., 5 consecutive values with a smaller difference than 0.01 m/s. The number of mesh cells for both mass flows was ~74 million.

Further, the density of the air was defined as 1.225 kg/m$^3$ and the dynamic viscosity was defined as $1.85508 \times 10^{-5}$ Pa·s. For both particle sizes a density of 1000 kg/m$^3$ was used. The number of parcels injected in the simulations was based on the mean particle concentration at the center stack position from the measurement. Hence, the number of parcels injected of the smaller particle size was higher in the simulations. For the higher mass flow ~112 000 parcels and ~2 000 parcels were injected of the smaller particle size and the larger particle size, respectively. As the concentration of the larger particle size was very low in the measurement, only the larger particle size was studied in the simulation for the higher mass flow. The number of parcels of the smaller particle size was ~48 000 in the simulation for the lower mass flow. However, the particle concentrations in the simulations are only proportional to the measured concentrations, i.e., the concentrations in the simulations are not the same as the real concentrations measured inside the car cabin. In the simulation model the chest position from the measurement was replaced by a head position. Also, two additional zones were defined in the simulation model (Zone 1 and 4). The center stack position is used as reference value in all figures from the simulation results. Since no steady state concentrations were reached in the simulations the mean values was calculated the last three seconds, i.e., physical time 10-13 s. The interval was chosen as it was assumed that the last seconds in the simulations were closest to a steady state behavior, and to be able to capture fluctuations as well.
4.2 Simulations

4.2.1 Air flow 386 kg/h

In the CFD simulations the cabin was divided into four different zones, front row driver side (Zone 1), front row passenger side (Zone 2), rear row driver side (Zone 3) and rear row passenger side (Zone 4). In Figure 17, the concentrations of the smaller particle size in each zone, against physical time 1-13 s, are shown. As seen, the concentrations were higher in the front row than in the rear row. This is due to the mass flow in the first row is higher and that the particles reaches the positions in the first row faster. In the rear row, there is also no vents with nominal position directed directly or close to the defined head positions, as the front center vents in the front row.

![Figure 17](image)

Figure 17 – The concentrations of the smaller particle size over physical time for the higher mass flow (386 kg/h).
In Figure 18, the mean concentrations with standard deviation for the smaller particle size are shown. As seen, the head positions in the front row were the positions with the highest concentrations. The position with significantly higher concentration, than all other positions, was the head position in Zone 2. Further, there was a 36% difference between the head positions in Zone 1 and 2. Within Zone 2, the difference between the head position and the other positions was 79-91%. In the rear row, the concentrations on the driver side were higher than on the passenger side. The largest difference between the sides was at the floor position with 71% difference. The position with significantly higher concentration, in the rear row, was the head position on the driver side.

**Figure 18** – The mean concentrations of the smaller particle size with standard deviation for the higher mass flow (386 kg/h).
In Figure 19, the concentrations of the larger particle size over physical time are shown. As for the smaller size, the concentrations in the front row was higher than in the rear row. One difference between the particle sizes was that none of the larger particles reached the center stack position in the simulation.

![Figure 19](image)

**Figure 19** – The concentrations of the larger particle size over time for the higher mass flow (386 kg/h).
In Figure 20, the mean concentrations of the larger particle size with standard deviation are shown. As in the measurement, the concentrations of the larger size inside the cabin were low and the uncertainties were high. No position with significantly higher concentration than all other positions was found. However, the largest concentrations were seen at the head positions in the front row, as for the smaller particle size.

![Figure 20](image)

**Figure 20** – The mean concentrations of the larger particle size with standard deviation for the higher mass flow (386 kg/h).
In Table 2, the mass flow distribution for the simulation with the higher mass flow is presented. The highest and lowest distribution was in Zone 2 and 4, respectively. The difference in particle concentrations of the smaller size, between the sides in the rear row, is most likely due to the difference in mass flow distribution between the sides (20% difference).

Table 2 – The mass flow distribution for the higher mass flow (386 kg/h).

<table>
<thead>
<tr>
<th>Zone</th>
<th>m [kg/h]</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a) Zone 1</td>
<td></td>
</tr>
<tr>
<td>Front defroster (ds)</td>
<td>2</td>
</tr>
<tr>
<td>Rear defroster (ds)</td>
<td>19</td>
</tr>
<tr>
<td>Side defroster (ds)</td>
<td>3</td>
</tr>
<tr>
<td>Front floor (ds)</td>
<td>35</td>
</tr>
<tr>
<td>Front center vent (ds)</td>
<td>40</td>
</tr>
<tr>
<td>Front side vent (ds)</td>
<td>28</td>
</tr>
<tr>
<td>Total</td>
<td>127</td>
</tr>
<tr>
<td>(b) Zone 2</td>
<td></td>
</tr>
<tr>
<td>Front defroster (ps)</td>
<td>4</td>
</tr>
<tr>
<td>Rear defroster (ps)</td>
<td>26</td>
</tr>
<tr>
<td>Side defroster (ps)</td>
<td>4</td>
</tr>
<tr>
<td>Front floor (ps)</td>
<td>35</td>
</tr>
<tr>
<td>Front center vent (ps)</td>
<td>32</td>
</tr>
<tr>
<td>Front side vent (ps)</td>
<td>28</td>
</tr>
<tr>
<td>Total</td>
<td>129</td>
</tr>
<tr>
<td>(c) Zone 3</td>
<td></td>
</tr>
<tr>
<td>Rear center vent (ds)</td>
<td>20</td>
</tr>
<tr>
<td>Rear floor (ds)</td>
<td>26</td>
</tr>
<tr>
<td>B-pillar vent (ds)</td>
<td>25</td>
</tr>
<tr>
<td>Total</td>
<td>71</td>
</tr>
<tr>
<td>(d) Zone 4</td>
<td></td>
</tr>
<tr>
<td>Rear center vent (ps)</td>
<td>14</td>
</tr>
<tr>
<td>Rear floor (ps)</td>
<td>22</td>
</tr>
<tr>
<td>B-pillar vent (ps)</td>
<td>21</td>
</tr>
<tr>
<td>Total</td>
<td>57</td>
</tr>
</tbody>
</table>
Furthermore, there are differences in the nominal position of the front center vent and the front side vent between the driver and the passenger side. On the driver side the nominal position is directed directly at the head position and on the passenger side it is directed a little bit to the right, i.e., more to the middle of the cabin. This can be see in Figure 21a, where an isosurface of the velocity higher than 0.525 m/s is shown at physical time 1 s. However, a transient effect is seen, as the physical time passes, that shifts the air flow from the front center vent on the passenger side towards the head position, as seen in Figure 21b.

The difference between the concentrations at the head positions in the front row after ~10 s, as seen in Figure 17, is probably due to the difference in nominal position of the front side vent between the sides. It creates a difference in the air flow characteristics around the b-pillar, as seen in Figure 21b, where the flow is directed towards the head position on the passenger side, while on the driver side the flow follows the surface of the b-pillar.

![Figure 21](image_url)
4.2.2 Air flow 183 kg/h

In Figure 22, the concentrations of the smaller particle size are shown over physical time for the lower mass flow. One difference, compared to the simulation with the higher mass flow, was that the highest concentration was not seen at the head position in Zone 2. The difference between the mass flows could be due to that the physical time in the simulations was not long enough to see the same behavior as in the simulation with the higher mass flow.

![Graphs showing concentration over time for different zones](image)

**Figure 22** – The concentrations of the smaller particle size over physical time for the lower mass flow (183 kg/h).
In Figure 23, the mean concentrations with standard deviation for the smaller particle size are shown. The concentration at the head position in Zone 1 was significantly higher than at all other positions for the mass flow. The position with second highest concentration was the head position in Zone 3 (63% difference). As for the higher mass flow, the concentrations at the driver side in the rear row were higher than at the passenger side (21-65% difference).

Figure 23 – The mean concentrations of the smaller particle size with standard deviation for the lower mass flow (183 kg/h)
Regarding the mass flow distribution in the simulation with the lower mass flow, the main difference was between the driver and passenger side in the rear row, as in the simulation with the higher flow. The difference between the sides was 18%. In Table 3, the mass flow distribution for the simulation with the lower mass flow is shown.

Table 3 – The mass flow distribution for the lower mass flow (183 kg/h).

<table>
<thead>
<tr>
<th>Zone</th>
<th>( \dot{m} ) [kg/h]</th>
</tr>
</thead>
<tbody>
<tr>
<td>(a) Zone 1</td>
<td></td>
</tr>
<tr>
<td>Front defroster (ds)</td>
<td>1</td>
</tr>
<tr>
<td>Rear defroster (ds)</td>
<td>9</td>
</tr>
<tr>
<td>Side defroster (ds)</td>
<td>1</td>
</tr>
<tr>
<td>Front floor (ds)</td>
<td>17</td>
</tr>
<tr>
<td>Front center vent (ds)</td>
<td>20</td>
</tr>
<tr>
<td>Front side vent (ds)</td>
<td>14</td>
</tr>
<tr>
<td>Total</td>
<td>62</td>
</tr>
<tr>
<td>(b) Zone 2</td>
<td></td>
</tr>
<tr>
<td>Front defroster (ps)</td>
<td>2</td>
</tr>
<tr>
<td>Rear defroster (ps)</td>
<td>11</td>
</tr>
<tr>
<td>Side defroster (ps)</td>
<td>2</td>
</tr>
<tr>
<td>Front floor (ps)</td>
<td>16</td>
</tr>
<tr>
<td>Front center vent (ps)</td>
<td>16</td>
</tr>
<tr>
<td>Front side vent (ps)</td>
<td>14</td>
</tr>
<tr>
<td>Total</td>
<td>61</td>
</tr>
<tr>
<td>(c) Zone 3</td>
<td></td>
</tr>
<tr>
<td>Rear center vent (ds)</td>
<td>12</td>
</tr>
<tr>
<td>Rear floor (ds)</td>
<td>9</td>
</tr>
<tr>
<td>B-pillar vent (ds)</td>
<td>12</td>
</tr>
<tr>
<td>Total</td>
<td>33</td>
</tr>
<tr>
<td>(d) Zone 4</td>
<td></td>
</tr>
<tr>
<td>Rear center vent (ps)</td>
<td>11</td>
</tr>
<tr>
<td>Rear floor (ps)</td>
<td>6</td>
</tr>
<tr>
<td>B-pillar vent (ps)</td>
<td>10</td>
</tr>
<tr>
<td>Total</td>
<td>27</td>
</tr>
</tbody>
</table>
4.2.3 Comparison Between Mass Flows

In Figure 24, the mean concentrations of the smaller particle size with standard deviation for both mass flows are shown. The difference between the mean concentrations, comparing the same position, was in the range 2-98% (except from the center stack position where no particles reached in the simulation with the lower mass flow). The positions with significant difference were the seat positions in the front row, and the head and floor position in Zone 2. The largest difference was at the seat position in Zone 2 (98% difference). This could be due to that the physical time was not long enough for the particles to reach the seat position for the lower mass flow or that the stick boundary condition at the carpet and the seats prevents particles from reaching the positions. As seen, there was no significant difference in the rear row between the mass flows.

Figure 24 – The mean concentrations of the smaller particle size with standard deviation for both mass flows (386 kg/h and 183 kg/h).
5 Discussion & Conclusions

In this study, the simulations showed elevated concentrations of particles at the head positions in the front row. As mentioned, due to limitations in the measurement setup, they were not measured in the measurement carried out. Thus, the results in the simulations, with a significantly higher concentration of the smaller particle size at the head positions in the front row, could not be validated. Even though no steady state behavior was seen at the head positions in the simulations, they are recommended to measure in a future measurement. Furthermore, as the simulation results showed a significant difference between the head positions in the front row, the difference between the positions is also recommended to investigate in a future measurement.

However, as 85-95% of the PM2.5 can be removed by the climate control system and no background concentration was taken into consideration in the simulations, it could be the case that a dilution effect will be seen instead, i.e., lower concentrations at the positions with high concentrations in the simulations. The background concentration was neglected since we mainly wanted to analyze particles that enter the cabin from the exterior.

Furthermore, re-suspension of particles from, e.g., the carpet in the front and rear row was also not taken into consideration in the simulation model. It could be possible that the number of re-suspended particles is higher than the number of particles that deposit, especially in regions where the velocity of the air flow is relatively high. Thus, the stick boundary condition on the seats and the carpet in the simulation model could possibly affect the result in a way that does not represent a real scenario. Elastic bouncing on all other surfaces could also lead to a behavior that is inaccurate. In general, the accuracy and robustness of the simulation methodology developed in this thesis need to be investigated further, since some of the assumptions made could affect the results in an inaccurate way, as mentioned.

However, to validate the simulations are difficult with the measurement methodology used in this thesis, since the measurement was not performed in a controlled environment. Thus, the measurement results were dependent on parameters that could not be controlled, e.g., on the traffic in the road tunnel when the measurement was carried out. As the PM2.5 concentration in Gothenburg is low, it is also difficult to perform cabin measurements where enough particles enter to be able to measure. The low concentration and the uncertainty in the measurement of the larger particle size resulted in no significant differences between the positions inside the cabin. However, the measurement showed some interesting results for the smaller particle size. The difference between the measurement zone in the front...
and rear row was very low, less than 2% difference, at the corresponding position within the zones. Thus, the concentration inside the cabin was homogeneous at positions with the same height. As mentioned, no position with significantly higher concentration was measured. This could probably be explained by the presence of a background concentration. The sources of the background are materials inside the cabin, people, open doors, open windows etc.

Neglecting the background concentration in the simulations also makes it difficult to compare the simulation with the measurement. Ideally, either the measurement would be carried out without a background concentration in a controlled environment or a background simulation would be introduced in the simulations.

5.1 Improvements of the Simulation Methodology

There are several functionalities and methods that could be added to improve the simulation methodology. As a transient effect was seen in the air flow characteristics in the simulations, an improvement of the methodology would be to start from a steady state simulation of the air flow and importing the steady state solution as inlet data in the particle simulations. It would most likely increase the accuracy of the simulations.

Regarding the particle injection, the injection points in the model were dependent on the mesh on the inlet. Thus, as the density of the injection points was higher at the corners of the inlet, it could affect the distribution of the particles inside the cabin. To avoid an mesh independent injector, there are at least two possible methods that could be used. The first method is to force the mesh to be uniform on the specified injector part, and the second method is to use a table injector instead of a part injector. With a table injector it is possible to specify the coordinates and injection times for each injection point. However, as the point inclusion probability is not available with a table injector and creating a table injector is more time consuming than using a part injector, it is recommended to force the mesh to be uniform on the injector part and use a part injector in future simulations.

Furthermore, as the nominal position of the front center vent and front side vent is different between the driver side and the passenger side, the stick boundary condition on the front seats could have different effects between the sides, especially at head position. To investigate this, the incident mass flux of particles could be looked at. However, as the front seats were defined as one part in the model, it needs to be split into two parts to be able to do this in the future. The mass flux of particles through the vents could also be looked at by creating interfaces over them. Thus, the particle distribution between the zones could be explained by looking at the mass flux of particles instead of the mass flow of air. The difference
in mass flow, between the sides in the rear row, also needs to be investigated further since it is larger than acceptable. However, as the concentrations in the rear row were lower than in the front row, the difference in mass flow between the zones in the rear row did not affect the outcome of this study.

The methodology used to calculate the particle concentrations is based on a discrete number of positions. Hence, the positioning of the blocks could be optimized, to find positions with higher concentrations than found in this study. However, as the positioning of the blocks is mesh dependent, finding optimal positions are not as straight forward as desired. Thus, investigating different positions in a fast way is not possible. Despite this, there is a positions that should be investigated as a first step. The center stack block should be moved up so that the nominal direction of the rear center vents is directed directly to the block. In the model, the position of the center stack block is a little bit below the nominal position.

5.2 Future Work

The first step to evaluate the prospects of the simulation methodology developed in this study would be to measure the head positions in the front row. Optimally, the concentrations are significant higher/lower on the head position than all other positions during a future measurement. Further, one natural extension of this study would also be to continue the simulations over an increased physical time to see if any steady state behavior is observed. In commercial aspects, as results need to be produced as fast as possible, it would also be interesting to compare the results obtained with a steady state simulation.

Furthermore, one other area of use for this methodology is to investigate the background concentration. This can be done by using a table injector and specify the coordinates of the injection points inside the cabin. Injection points specified outside of the simulation region are not included in the simulation. Hence, creating an injection table to simulate a background concentration is relatively easily.

As the concentration of the larger particle size investigated in this study was very low inside the cabin in the measurement, future studies should focus on smaller particles than particles with diameter 2.146 µm.
References


REFERENCES


[14] “STAR-CCM+ - Documentation.”


